

Introduction to OrCAD PSPICE

OrCAD Capture 9.2 - Lite Edition - 9.2.448 - D

PSPICE Rules

1. All circuits must have a reference node. In PSPICE, this node is usually marked with the number "0." Note: The conventional OrCAD ground symbol is not appropriate for PSPICE simulation.

This is a proper ground: 

2. All component leads must be connected. Unconnected wires and leads will cause the simulation to fail.
3. All components must have a value. The following suffixes are used to modify values:

k, K	= 1K = 1 x 10 ³
m, M	= 1 m = 1 x 10 ⁻³
MEG, meg	= 1 M = 1 x 10 ⁶
u, U	= 1 μ = 1 x 10 ⁻⁶
p, P	= 1 p = 1 x 10 ⁻¹²
f, F	= 1 f = 1 x 10 ⁻¹⁵

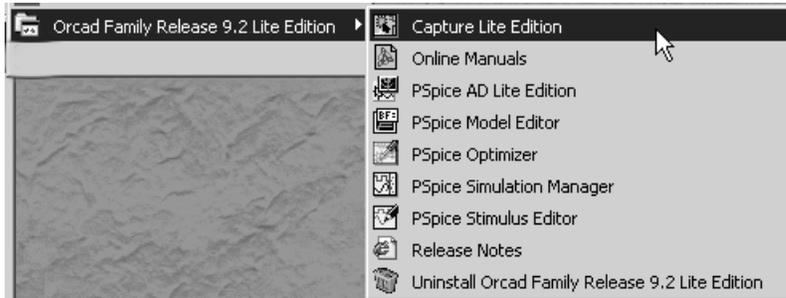
Caution: Note that "m" and "M" mean *milli* to PSPICE, not *meg*!

4. All components must have a unique designator. For example, there can't be two instances of a component reference "R1" within the same circuit.
5. There must be no space between a number value and its modifying suffix. For example, the value "1 k" for a resistor or "1 V" for a voltage source will cause an error.

The correct way of specifying these parts is "1k" and "1V" (upper and lower case make no difference.)

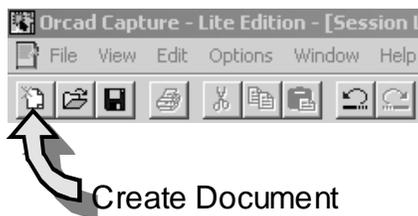
DC Static Analysis Example

1. Start OrCAD Lite as shown below:

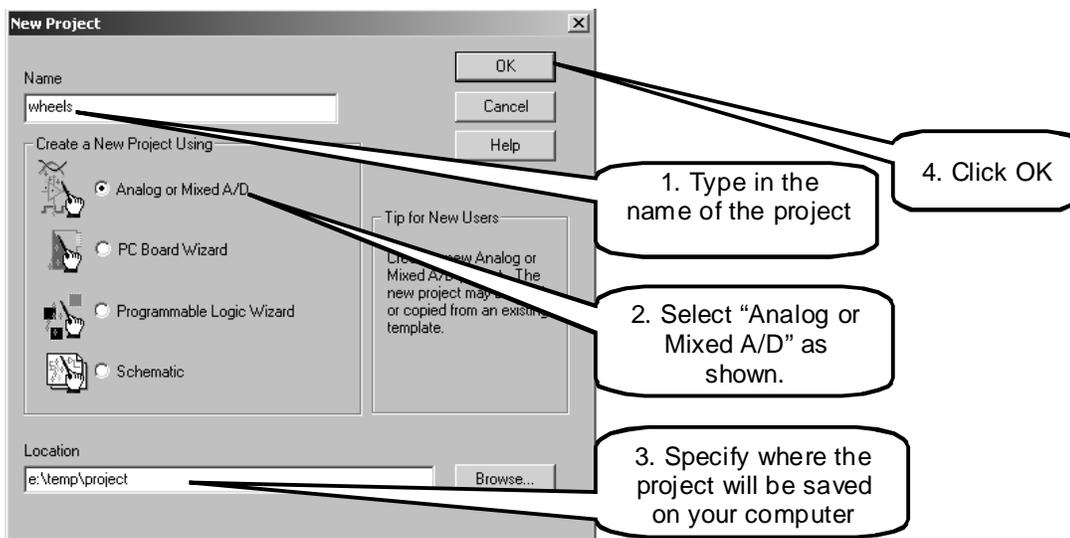


Note that the only program you need to start is “Capture Lite Edition.” The other programs will run automatically as they’re needed.

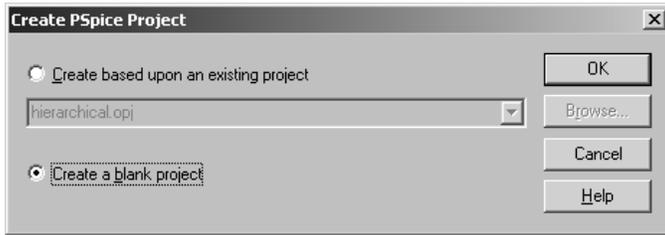
2. Click the “Create Document” button on the toolbar.



3. The following dialog will appear. Fill in the information as shown.

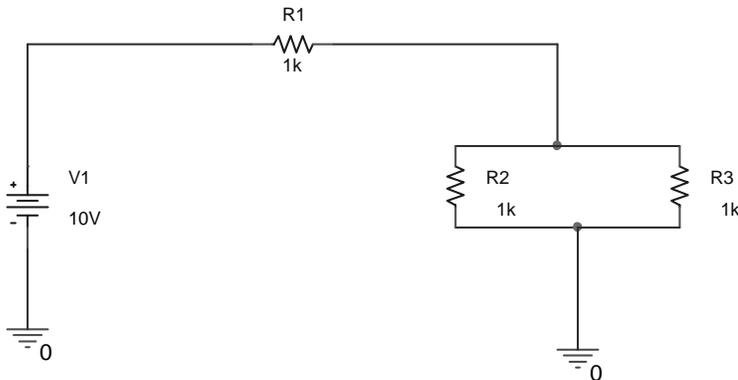


4. The following dialog will appear:



Choose "Create a blank project" and click OK.

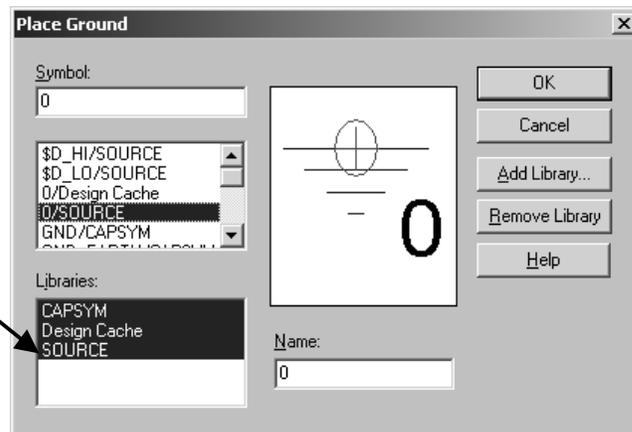
5. The schematic editor will open. Enter the following schematic into the program:



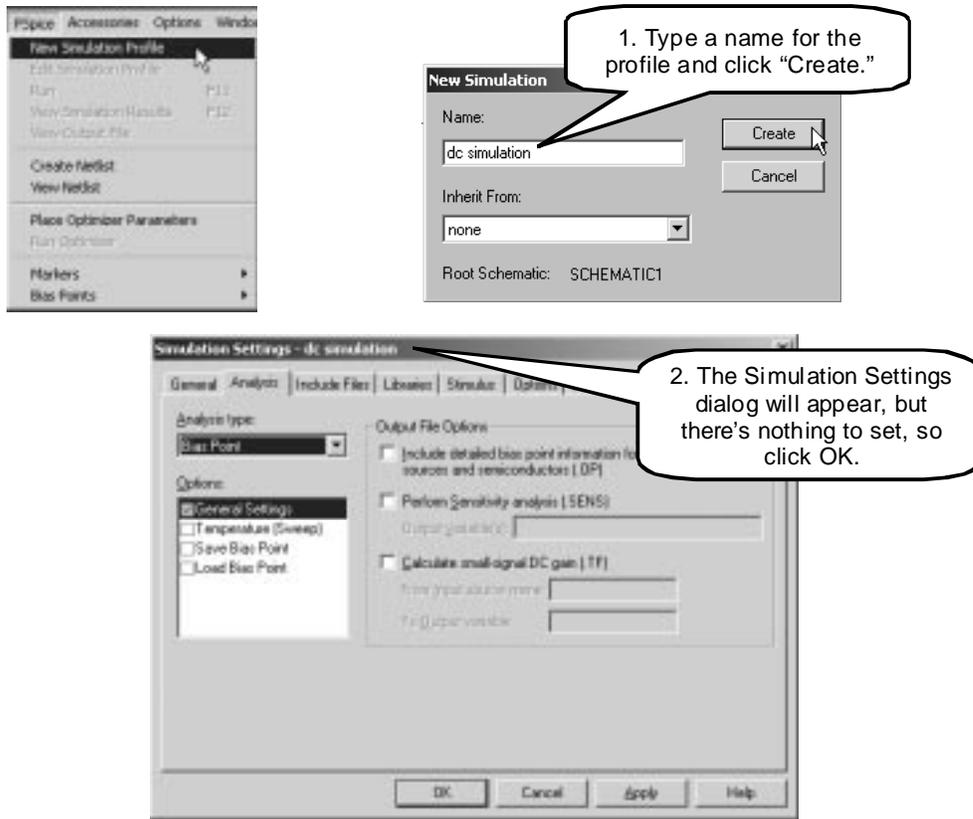
Note the following:

- Parts are placed by SHIFT-P
- The voltage source is called "VDC" in the parts library
- The resistors are called "R"
- The ground is obtained by clicking the ground button  and choosing part "0/SOURCE."

SOURCE library should show here. If not, press "Add Library" button.



6. Create a new simulation profile for the circuit by going to the PSPICE menu and choosing “New Simulation Profile” as shown:



You can have multiple profiles for a given project. This allows you to analyze a circuit in different ways without having to change settings back and forth.

7. Click the toolbar buttons for displaying the bias-point voltages and currents so that they are depressed.

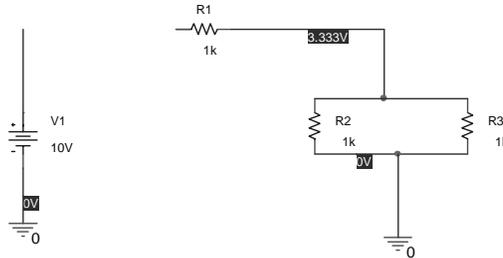


8. Press F11 to run the simulation. (This is the same as going to the PSPICE menu and choosing *Run*). The screen will show the DC voltages and currents directly on the schematic.

Debugging PSPICE Simulation Errors

Sooner or later (probably sooner), you'll get a nasty error message from the simulator. Because this program uses a PSPICE core to do the actual work, you need to be able to locate and decipher the actual error within the OUTPUT file, then correct it in your circuit.

1. Let's "break" the circuit. Delete one of the conductors and press F11 to run the simulation again. I broke mine like this:



Depending on how your version of PSPICE is configured, you'll get either a dialog box with an error, or a full screen dump of the output file, which shows the error.

If you don't get the output file, select "View Output" from the PSPICE menu to see it. It might look like this:

```
** Creating circuit file "wheels-schematic1-dc simulation.sim.cir"
** WARNING: THIS AUTOMATICALLY GENERATED FILE MAY BE OVERWRITTEN BY SUBSEQUENT
SIMULATIONS

*Libraries:
* Local Libraries :
* From [PSPICE NETLIST] section of C:\Program Files\OrCAD\PSpice\PSpice.ini
file:
.lib "nom.lib"

*Analysis directives:
.PROBE V(*) I(*) W(*) D(*) NOISE(*)
.INC ".\wheels-SCHEMATIC1.net"

**** INCLUDING wheels-SCHEMATIC1.net ****
* source WHEELS
V_V1      N00893 0 10V
R_R1      N00812 N00197 1k
R_R2      0 N00197 1k
R_R3      0 N00197 1k

**** RESUMING "wheels-schematic1-dc simulation.sim.cir" ****
.END

ERROR -- Less than 2 connections at node N00812
```

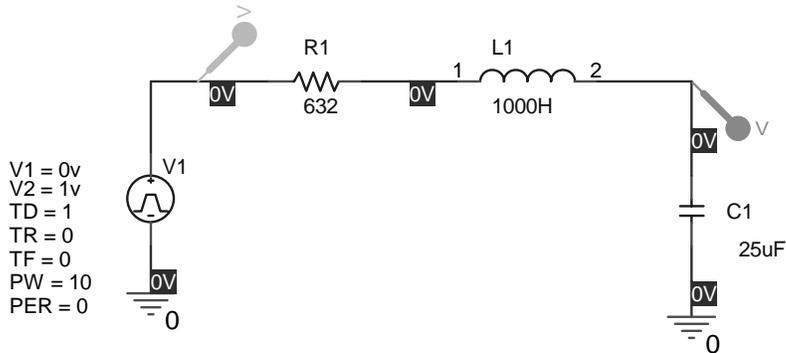
Node N00812 belongs to R1.

2. You can see the ERROR. At node N00812 there's a problem. By looking at the netlist portion of the listing, you can see that node N00812 is associated with R1.

Transient Analysis

In PSPICE, the transient analysis option is used to do both steady and transient analysis of circuits. This can involve both AC and DC signal sources. Steady-state AC time-domain results are obtained by the same setup as for transient analysis. Let's try it out.

1. Start a new project called "boing" and build the following schematic into it. (Make sure to completely close the previous project).

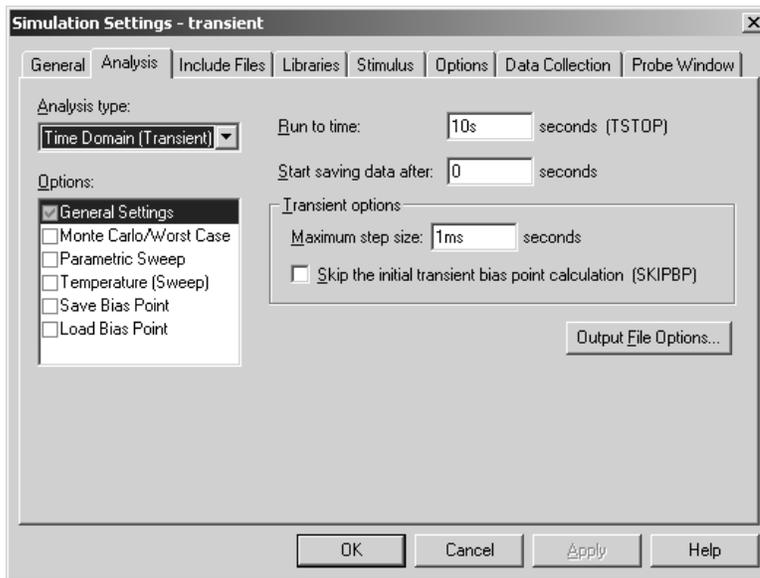


Place the two voltage probes by clicking the probe button on the toolbar.  Note the *color* of each probe; it will match the color of the trace in the PROBE readout during simulation.

The source used is called "VPULSE" and the parameters on the source have the following meanings:

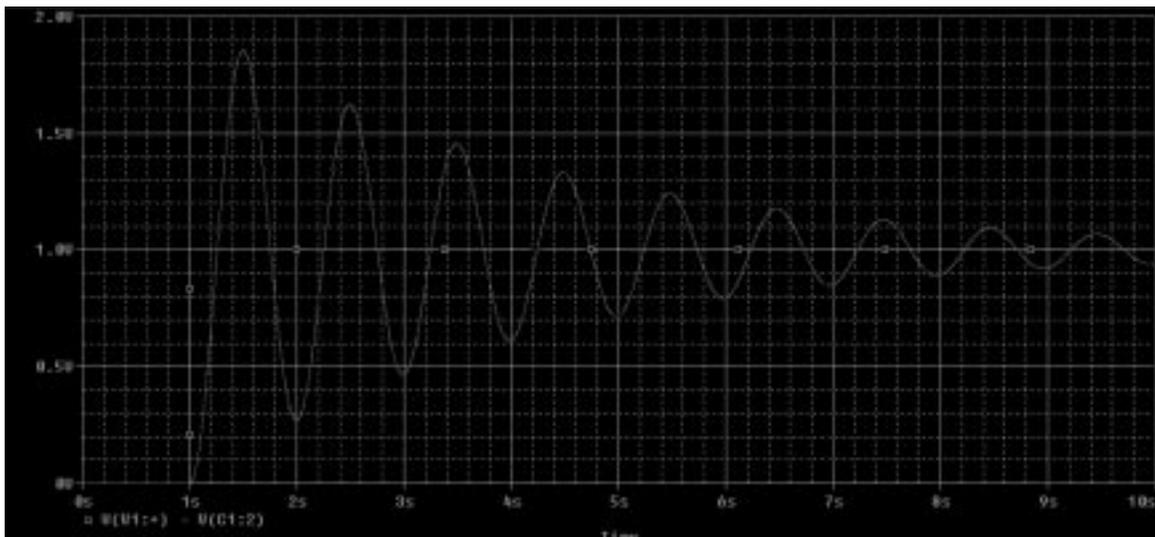
- V1 is the "off" voltage
- V2 is the "on" voltage
- The simulation begins with the source in the OFF state. TD is the time delay in seconds before the pulse turns on.
- TR and TF are the rise and fall times of the step function.
- PW is the pulse width (how long the source stays "on" after TD has passed.)
- PER is the period. Enter 0 for a one-shot pulse, or a non-zero period for the pulse to be periodic.

2. Build a new simulation profile called “transient” and make it look like this:

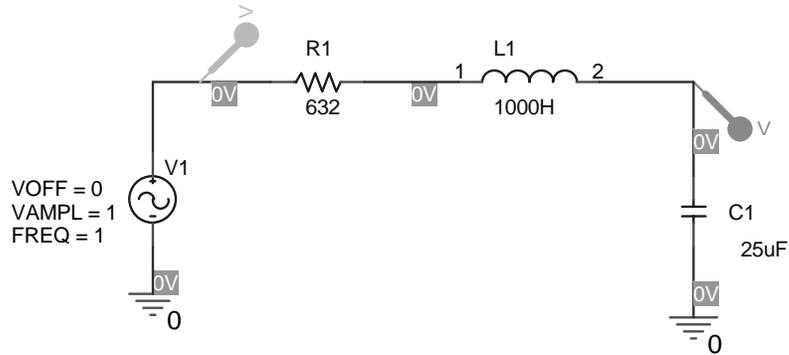


- “Run to time” says how long to run the simulation. 10 seconds is forever in the world of circuits, but this one is special.
- “Start saving data after 0 seconds” means to use *all* simulation results starting with time = 0. Entering a non-zero value causes initial data points to be discarded.
- “Maximum step size” is a very important setting. It specifies how often PSPICE will recalculate the circuit parameters during simulation. If you’re expecting sinusoids to look good, you need at least 100 or more calculations for each sine wave period.

3. Press the F11 key. (Warning: Stand several feet back from the circuit when doing this!)
You should get the following display:



4. Try replacing the DC pulse source in the circuit with an AC signal source and repeat the analysis again. You might want to change the “Run to time” in the analysis profile to 20 seconds. The circuit looks like this:



For the VSIN signal source, the parameters have the following meanings:

- VOFF is the DC offset
- VAMPL is the *peak* voltage of the sine wave
- FREQ is (you guessed it), the frequency of the signal in Hz.

You can set other parameters of the VSIN source by double-clicking the body of the part. You'll get a complete listing of the settable parameters and their values.

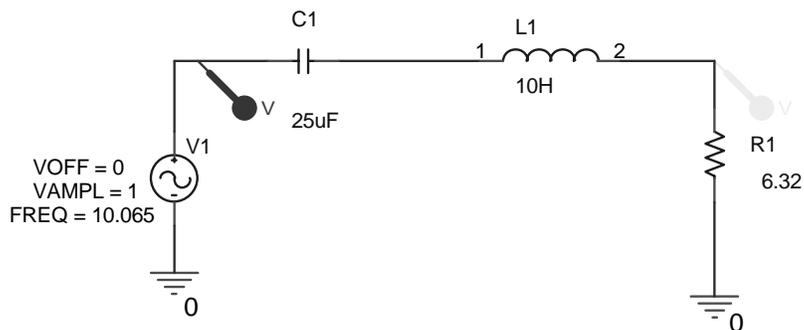
AC Parametric Sweeps

In order to learn about the frequency response of circuits, an AC parametric sweep can be performed. This is a different type of analysis than transient analysis as far as PSPICE is concerned, because instead of calculating the instantaneous values for voltages and currents, PSPICE instead calculates the steady-state effective currents and voltages.

Any source with an AC parameter can be used as a swept-frequency AC voltage source. The object “VSRC” is a generic AC signal source. The “VSIN” device also has the capability to be used during AC sweeps. It is usually more convenient to use “VSIN” since the same signal source can be used for both steady state and transient analysis.

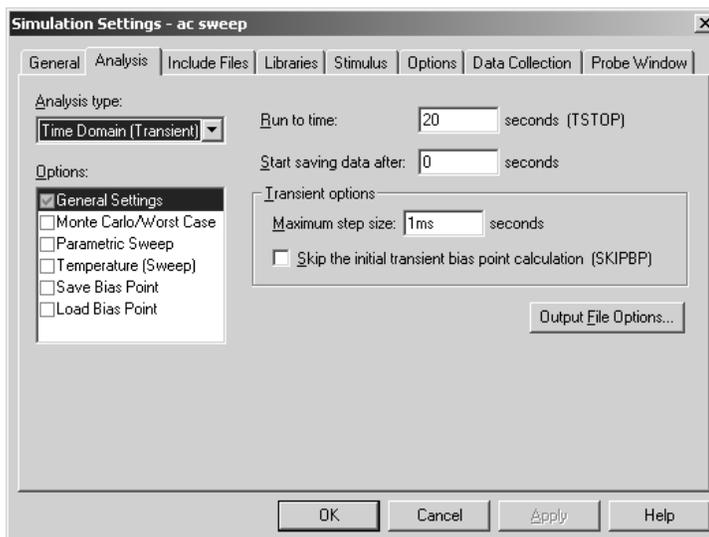
PSPICE can present the results of an AC sweep in many useful ways. We’ll first look at raw voltage data, then have PSPICE display the data in decibel form.

1. Change the schematic diagram of the project to look like this:



This modification raises the resonant frequency of the circuit to 10.065 Hz, and the Q becomes 100.

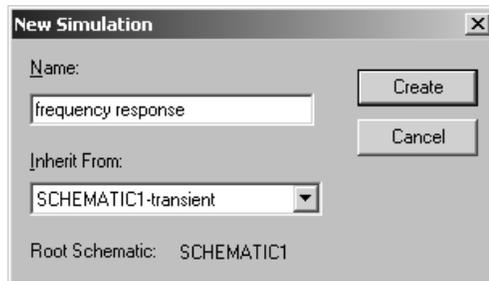
2. Change the simulation profile to look like this:



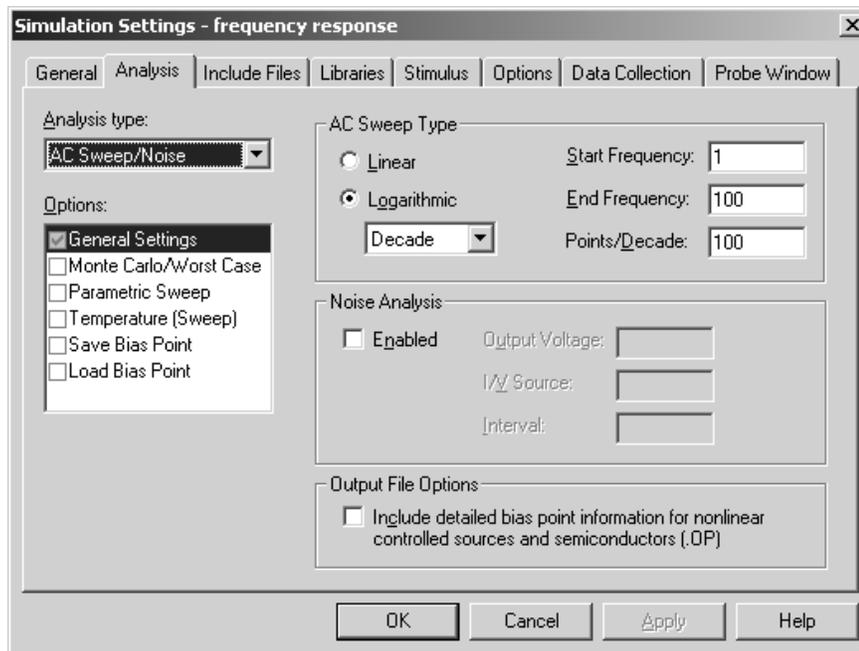
3. Run the simulation. Notice that the circuit takes a little while to build its voltage up due to the high Q of the tank circuit!

4. Let's find the frequency response of this filter circuit. First, let's create a new simulation profile (remember, a project can contain multiple simulation profiles.)

- At the PSPICE menu, choose "New Simulation Profile"
- Fill in the resulting dialog box as follows and click the CREATE button.

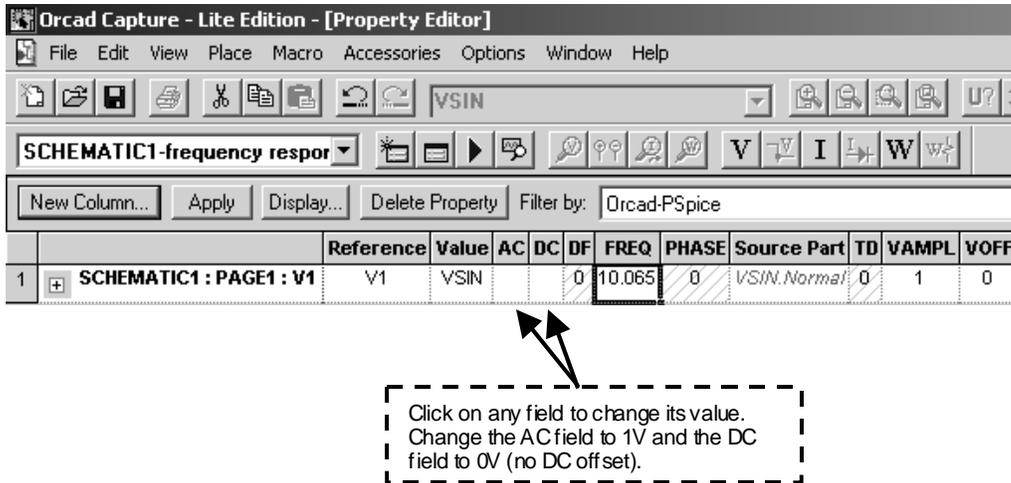


- A new simulation profile will appear. Set it up as follows:



This profile will sweep any AC sources in the circuit from 1 Hz to 100 Hz (two decades), with 100 points for each decade.

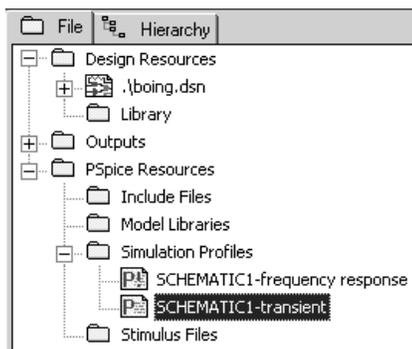
- To do an AC frequency sweep, there must be at least one signal source with an “AC” parameter in the circuit. Double click the body of the voltage source and you’ll get a detailed screen like this one:



This screen is available for all components in a design. Not all of the parameters display on your drawing because the display isn’t turned on for them. To fix that, highlight one of the items, then click the “Display...” button. Choose “Name and Value” from the dialog to make the parameter show on the drawing.

To exit the property editor, click the *lower* “X” button at the upper-right portion of the display.

- The circuit should now simulate. Press the F11 key to do so. But wait, there’s a problem -- why doesn’t the output voltage rise all the way to 1 V (Vin) at resonance? *The circuit doesn’t work!* (Try more points per decade in the simulation setup -- 1000 is a good number.)
- You can easily switch back to the previous simulation profile. Simulation profiles are stored with the project.
 - From the Window menu of the schematic editor, choose “boing.opj”. You’ll see the following display:

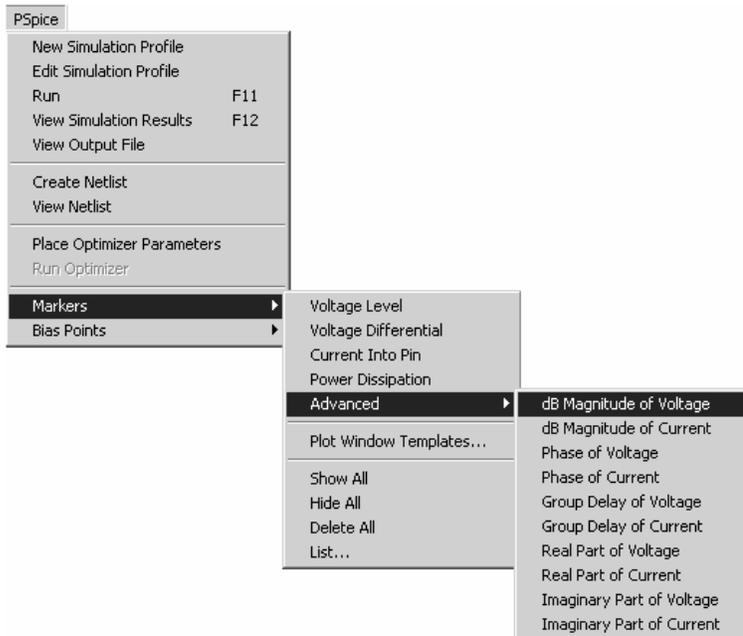


- By expanding the “Simulation Profiles” folder, you can see that your project has two different profiles.
 - You can enable any profile by highlighting it, then selecting “Make Active” from the “PSPICE” menu.
 - The active profile is always highlighted with a red “P!” (for Probe).
8. Let’s look at the data with a decibel display. Switch back to the schematic (making sure the “frequency response” profile is active).

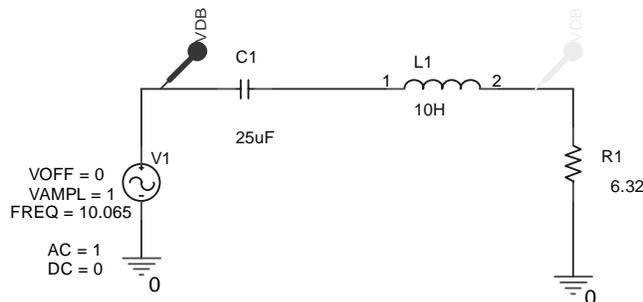
PSPICE provides special probes that read in dBV units. By definition:

$$dBV = 20 \log \left(\frac{V}{1 \text{ Volt}} \right)$$

These special probes are available by following the menus as follows:



Place two of these probes onto the circuit (and delete the other voltage probes). The circuit should look like this:



- With these special probes in the circuit, run the analysis again. Notice that we played a dirty little trick. By setting the input voltage to 1 V (0 dBV), we get output voltage magnitude readings that correspond to the voltage gain.

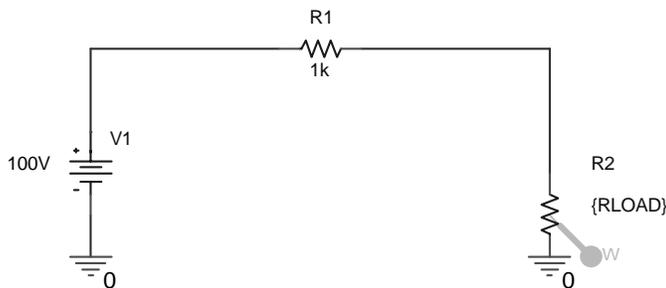
It is possible to mathematically compute gain by adding a trace to the plot. See if you can figure out how it works.

DC Parametric Sweeps

It is also possible to vary the parameters in a DC-swept fashion. Common examples of parameters that can be varied include resistance, capacitance, transistor Beta, and so on.

A good example of this is to demonstrate the maximum power transfer theorem, where the load resistance of a circuit must equal the source resistance in order to get maximum power into the load.

- Create a new project called “power” and add a schematic to the project as follows. Note that the value of R2 is {RLOAD}, which will refer to a global parameter.



- Place a new part on the drawing called “PARAM” (from the SPECIAL library.) Place it on the schematic, then double-click its body to open its property sheet.

Within the property sheet editor, make the following changes to the PARAM block:

- Select the “OrCAD-PSpice” Filter to make it easier to see the important fields.



- Click the “New Column” button and add a column called RLOAD. Set its value to 1k, as shown below.

		Reference	Value	RLOAD	Source Part	
1	+	SCHEMATIC1 : PAGE1 : 4	4	PARAM	1k	PARAM.Normal

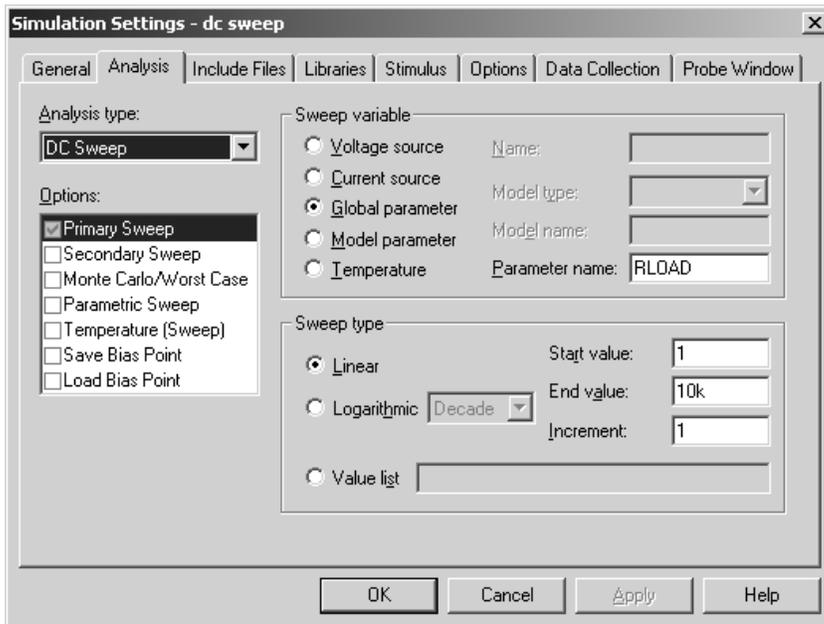
- Make sure to set RLOAD visible by highlighting it and clicking the “Display” button.

- Close the property sheet. Your drawing should now have a parameter block that looks something like this:

PARAMETERS:
RLOAD = 1k

The value of 1k has no effect on a DC swept analysis, but will instead be used for bias point (steady-state) calculation (if that is required).

3. Edit the analysis profile to look like this:



The sweep will vary the global parameter you named -- RLOAD -- in accordance with the rules set forth under “Sweep type.”

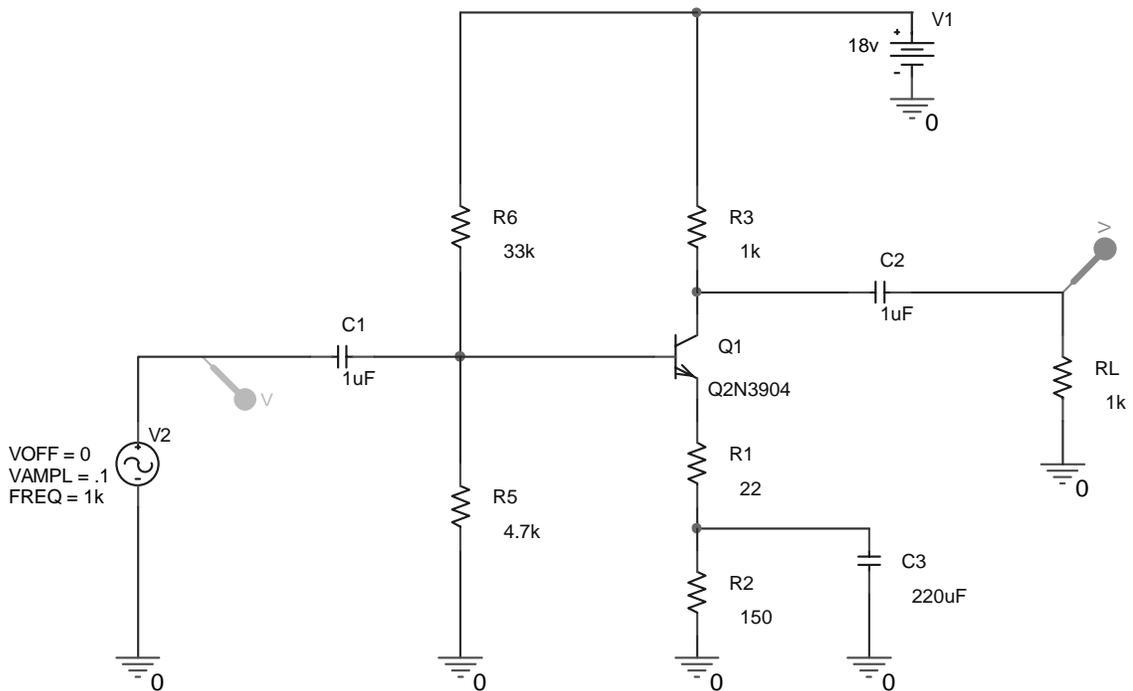
4. Run the analysis with F11.

Solid State Circuit Analysis

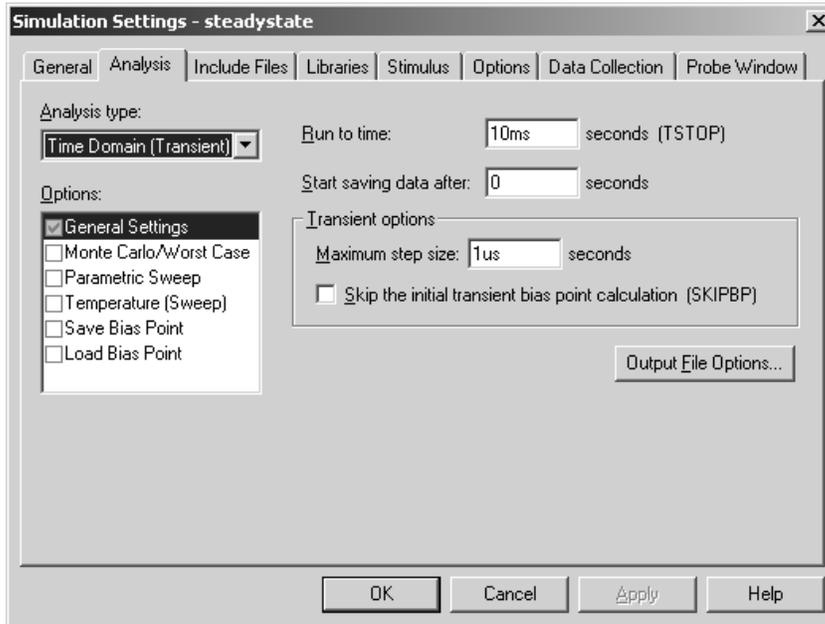
OrCAD PSPICE can model circuits with mixed analog and digital components. Below is a common-emitter amplifier. Using PSPICE simulation we can determine:

- What are the DC operating voltages of the amplifier?
- What is the voltage gain and compliance of the amplifier?
- What is the frequency response of the unit?

1. Start a new project called “transistor” and insert the circuit of the amplifier below.



2. Create a new simulation profile and set it as follows:



Note that the VSIN source in the circuit is set to 1 KHz, and that the analysis steps are 1 μ S -- so there will be 1000 calculations for each sine wave cycle. With the TSTOP setting of 10 mS, we should see 10 cycles of the waveform on the display.

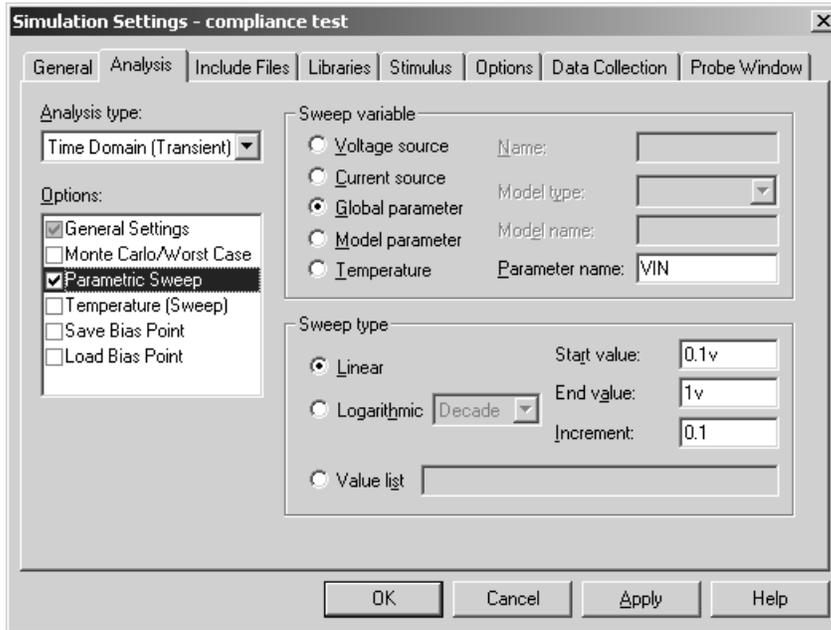
3. Run the simulation. To see the DC voltages and currents, click the appropriate toolbar buttons.

Compliance Evaluation

To perform a compliance test of an amplifier, we gradually increase the input signal amplitude until clipping is observed at the amplifier output. We can “sweep” the VSIN amplitude in our circuit to achieve this.

1. Create a new simulation profile called “Compliance Test.” Inherit the profile from the original profile in your project.

2. Modify the simulation profile to look like this:



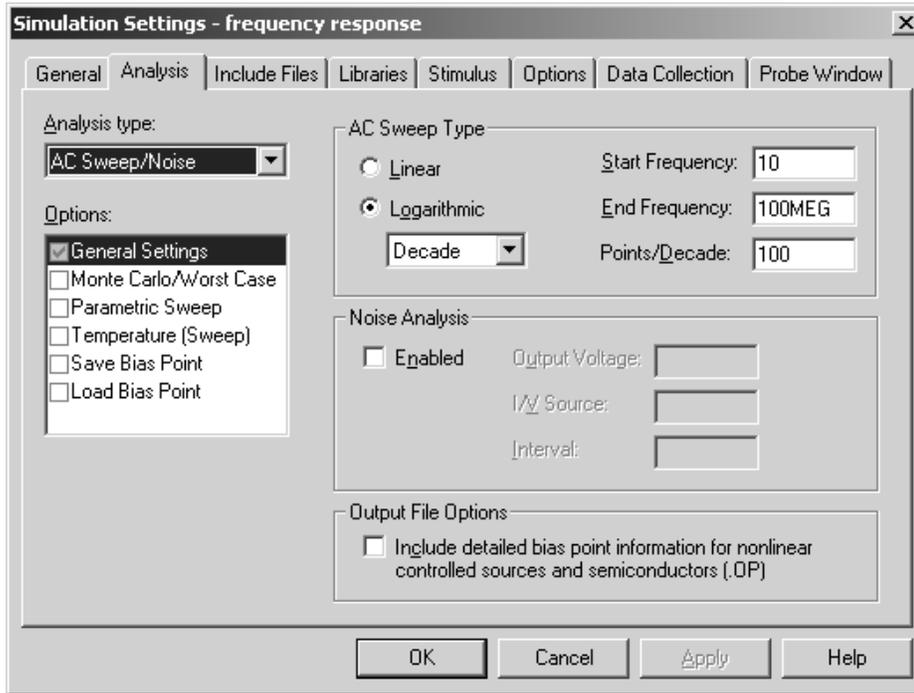
To set this up, *first* click on the checkbox by “Parametric Sweep” in the Options window. Then fill in the fields on the right as shown.

3. Add a part called “PARAM” to the schematic page. Double-click the PARAM block body and add a field called “VIN.” Set the value of VIN to 0.1 V (for static analysis) and turn its display on.
4. Modify the signal source VAMP parameter to be “{VIN}”, which will cause the signal source amplitude to ramp up in step with the parametric sweep.
5. Run the simulation. You’ll now see a family of curves for Vin and Vout. You can now tell when the amplifier will clip.

Frequency Response of the Amplifier

1. Create a new simulation profile called for the circuit called “Frequency Response.” Inherit this profile from the base profile “steadystate.”

2. Do an AC sweep on this profile as follows:



3. Recall that the VSIN component is a dual-purpose signal source - it can be used for both transient and AC sweep analysis - you just have to enable the AC parameters. Double-click the body of the VSIN source and set the AC value to 0.1v (RMS), and the DC value to 0v. Turn on the display of these items so that you can see them on the schematic.
4. Run the simulation. You may want to exchange the Vin and Vout voltage markers for VDB markers.