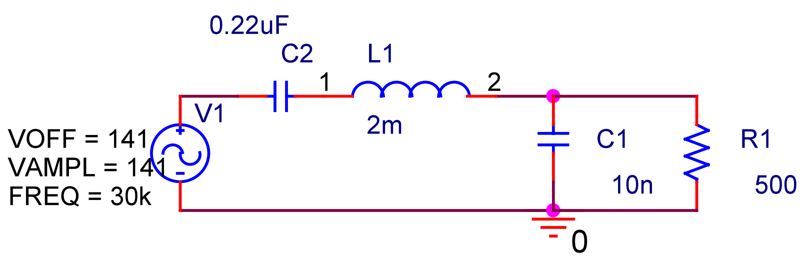
Using Cadence/Orcad PSpice II
AC and Transient Analyses
ENGS 32, Winter 2002

In the previous handout, we did two kinds of analysis, dc bias point. and dc sweep. Neither one takes into account any dynamics. (dc sweep just does a set of dc analyses with some parameter different, and plots all the results.) In this handout, two dynamic measurement methods are described: AC analysis and Transient analysis. First, however, a note on something that has given several people trouble:

Finding the correct ground
The correct ground to use is in the source library, and is just "0" or "Source/0". When you click on the ground icon, you may not have the source library available. In that case, you need to add that library by clicking on "Add Library…".

Transient analysis.
Transient analysis runs a simulation of the circuit, using something similar to the Runge-Kutta method used in MATLAB's ode45. This simulation includes the full nonlinear dynamic behavior of the circuit. Sometimes you just want to see the transient behavior of a circuit when it is first turned out, but another common use for this is to find out the response of the circuit to some time-varying input.

For an example, consider the fluorescent-ballast problem (#5) in homework 2. Here's the circuit drawn in PSpice:

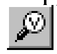


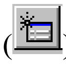
There are two changes relative to the original problem. First, the source includes some dc offset (141 V). That might happen if, for example, the circuit generating that voltage ran on a single power supply (of about 300 volts), rather than a split +/- 150 V supply. To avoid applying that dc voltage to the lamp, we've added C2, a dc blocking cap.

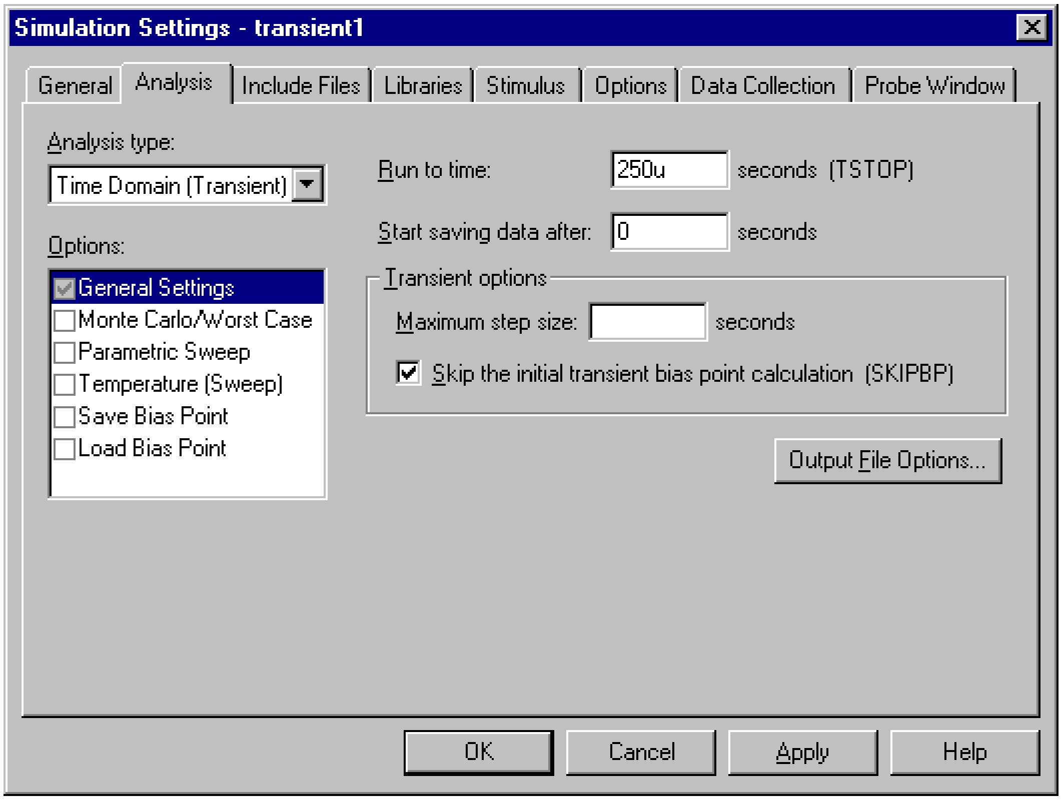
Voltage sources in transient analysis: There are many time-varying voltage sources you can use in transient analysis, in addition to the same dc voltage sources you use in dc analysis. These include :

- VEXP (exponentially varying voltage)
- VPULSE (a periodic pulse)
- VPWL (piece-wise linear voltage—you can construct arbitrary complicated waveforms with this.)
- VSIN (sinusoidal voltage).

Note that 'VAC' does ***not*** work for transient analysis. It is for ac analysis only. It doesn't really produce a voltage, but just specifies an input for a transfer function. More on that later. If you want an ac voltage, you must pick a waveform (probably sinusoidal) and use VSIN. VOFF in VSIN is the dc voltage component, and VAMPL is the zero-to-peak amplitude.

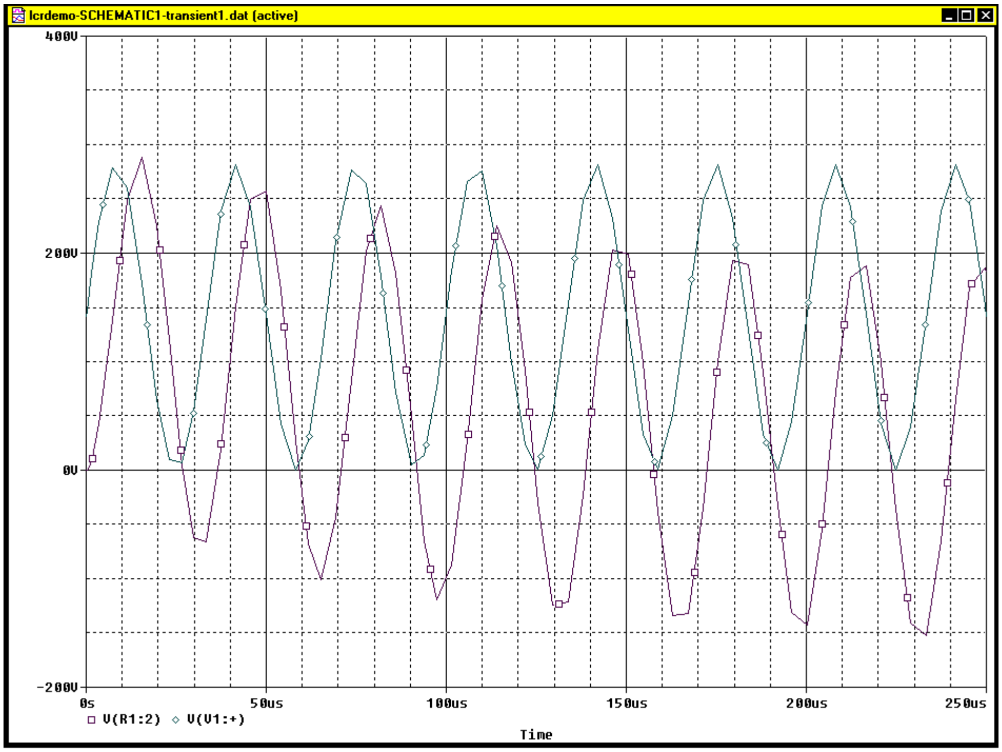
Before running a transient analysis, it is a good idea to apply markers to the voltages of interest. Either look in the *PSpice: Markers* menu, or use the  icon.

Then create a new simulation profile (), and select Time Domain (transient) (the default). It is very important to select the run time. If you select too long a run time, your simulation will take a very long time; too short and you won't see anything happening. We are using a 30 kHz signal, which has a period of 33 usec. So to get a few cycles, I'll use 250 usec.

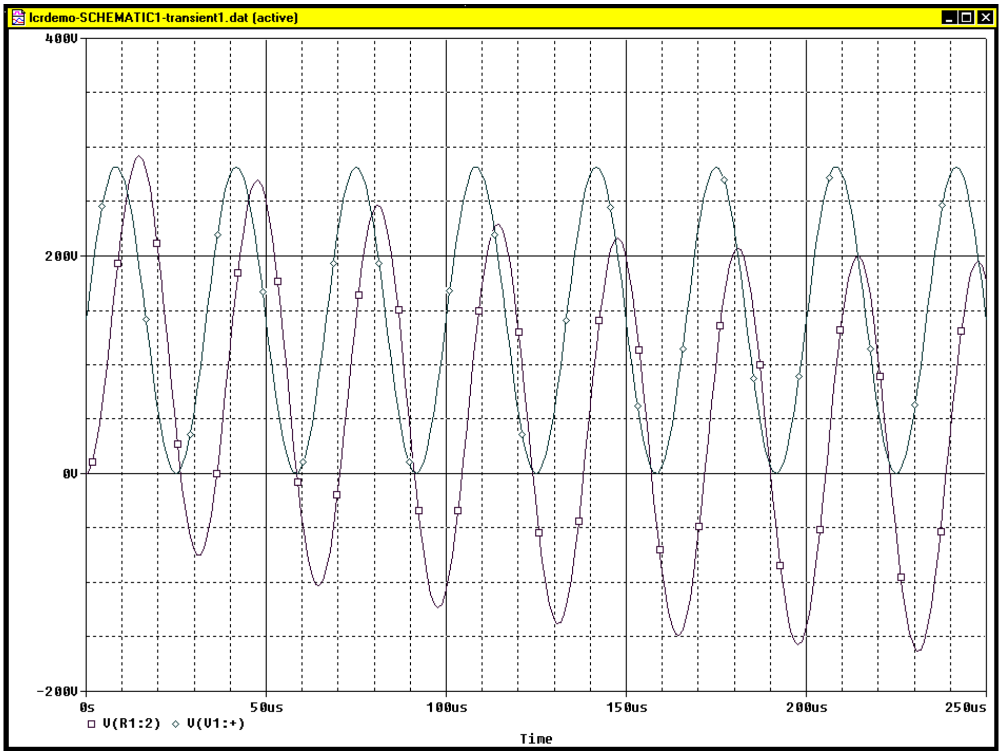


If you want to find out what happens when you first turn the system on, you want to "Skip the intital transient bias point calculation" However, if you want to see what happens if a transient hits after your system is already on, you can uncheck this, and use the initial bias point calculation.

The result of running the simulation, with voltage markers on the input voltage and the output voltage across the resistor is on the next page. We see the output shifted in phase, gradually settling down to a pure ac voltage as the dc blocking cap charges to its steady-state dc value. But is it ugly—not smooth. That is because it computed few points. The analysis is correct, but the plot, with straight lines drawn between points, is misleading. To get more points, and a smoother waveform, I changed the maximum step size parameter to 1usec. The result is the second graph on the next page.



First Transient Analysis.



Repeated with smaller maximum step size

If we wanted to find out the sinusoidal steady state response of that circuit, we could look at the phase and amplitude at the last cycle, after it has settled to a steady state.

AC analysis.
Suppose we want to find out the sinusoidal steady-state response at a number of different frequencies, either to select one to use for our system, or to find out how the system will respond to different inputs. We could run a series of transient analyses and look at the final steady state behavior of each. We might hope that PSpice could do that automatically, and plot the results vs. frequency. SPICE and PSpice's ac analysis does ***not*** do that. Instead, it gets you what is usually the same result by calculating a transfer function from input to output, and then using the transfer function to calculate frequency response, and using the frequency response result to predict the output.

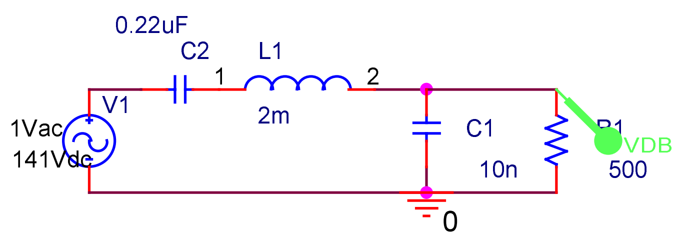
Why would you care how it calculates it, as long as it gives you the answer? Because transfer functions and frequency response calculations assume the system is linear. Thus, the ac analysis result does not tell you what the system will really do—it only tells you what the system *would* do *if* it were linear. That might seem unfortunate, but when a system is behaving nonlinearly, the concept of frequency response is not very well defined anyway, so you really need to do a transient simulation to find out what really happens.

Often what you want to know about your system is the gain and phase response. You may not know exactly what the input amplitude from, say, a microphone will be. But you know it will be small (mV) and you want to multiply it by a gain of 100. There's an easy shortcut to find out gain: just use an input signal of 1 V. Then the number you get for output is actually the gain. Very often the voltage numbers one gets are unrealistic. For example, with a gain of 1, you get 100 V. For a circuit with a 15 or 5 or 3.3 V power supply, that would not be possible. But SPICE assumes everything is linear and gives you the same gain of 100 that your circuit actually only gives with an input voltage below a few tens of mV.

Thus, I recommend always using a 1 V input in ac analysis in SPICE, because:

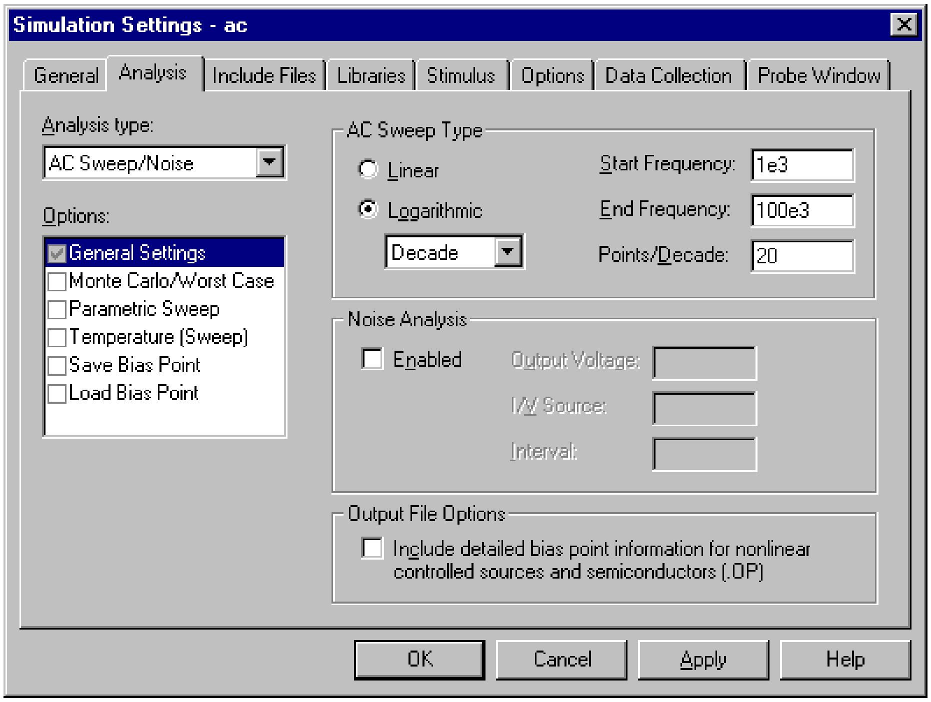
- You'll get the gain directly, without having to bother with additional calculations.
- You'll be less likely to fool yourself into thinking that you are really simulating the circuit. You are not—you are just calculating what the transfer function would be if the circuit were linear.

For the circuit we are using as an example, our model is linear (although the actual fluorescent lamp is not), so we can use ac analysis with no problem.



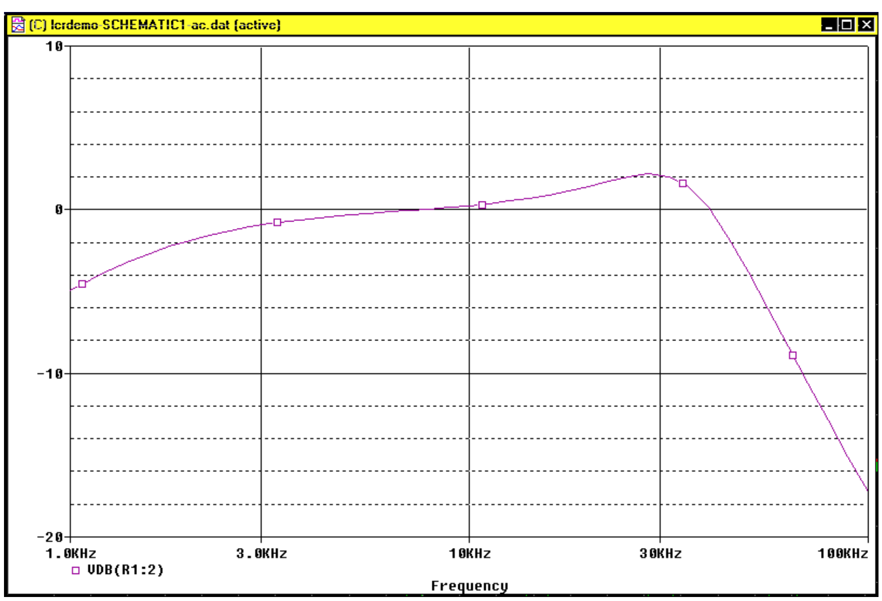
To do this, we need to delete the VSIN voltage source, and replace it with a VAC voltage source. The VAC voltage source defines the input point we use to calculate a transfer function.

Next, modify the simulation profile, or create a new one. I recommend a new one, because that allows you to keep the settings of your transient simulation in case you want to go back to that. The settings window is shown on the next page. You must select a start frequency and an end frequency. The default logarithmic sweep will give you something like a standard bode plot format.



After setting up a simulation profile for an ac simulation profile, the menu under *PSpice: Markers*, includes *Advanced,* under which you will find VDB and VPHASE—the easiest ways to generate dB magnitude and phase plots. Often it is best to run the two plots separately, because the scale you want for each is likely to be different.

Here's a plot of dB magnitude of the lamp circuit. Because this input voltage was one, this is really a plot of gain.



Summary:

Transient analysis vs. AC analysis

- Use transient analysis whenever you are interested in initial transients or response to something other than a sine wave.
- Use transient analysis whenever you are interested in finding out the effect of nonlinearities in your circuit.
- Use ac analysis when you want to know the final steady-state behavior over a range of frequencies, and you are happy ignoring any nonlinear effects.

Transient analysis

- Use VSIN for sinusoidal inputs.
- Set smaller maximum step size if the graph looks choppy where it should be smooth.
- If you want to use zero initial conditions, be sure to skip the initial bias point calculation (check the box).
- Think about the time period you want to run the simulation for; otherwise it might take a long time, or not show you anything.

AC analysis

- I recommend always using 1 V input; this gives you gain as the output.
- This does not simulate the circuit—it just calculates a transfer function and assumes that works.