

Fields and Waves I
ECSE-2100 Spring 2006

PSpice Analysis

Since transmission lines can be modeled using PSpice, you can do your analysis by downloading the student version of this excellent program.

PSpice can be downloaded from the following sites.

The newest version is at the OrCAD website:

<http://www.orcad.com/download.orcaddemo.aspx>

This is quite large, but it is the most up-to-date.

There is an smaller, older, usually reliable, but unsupported version available at

<http://hibp.ecse.rpi.edu/~connor/education/91pspstu.exe>

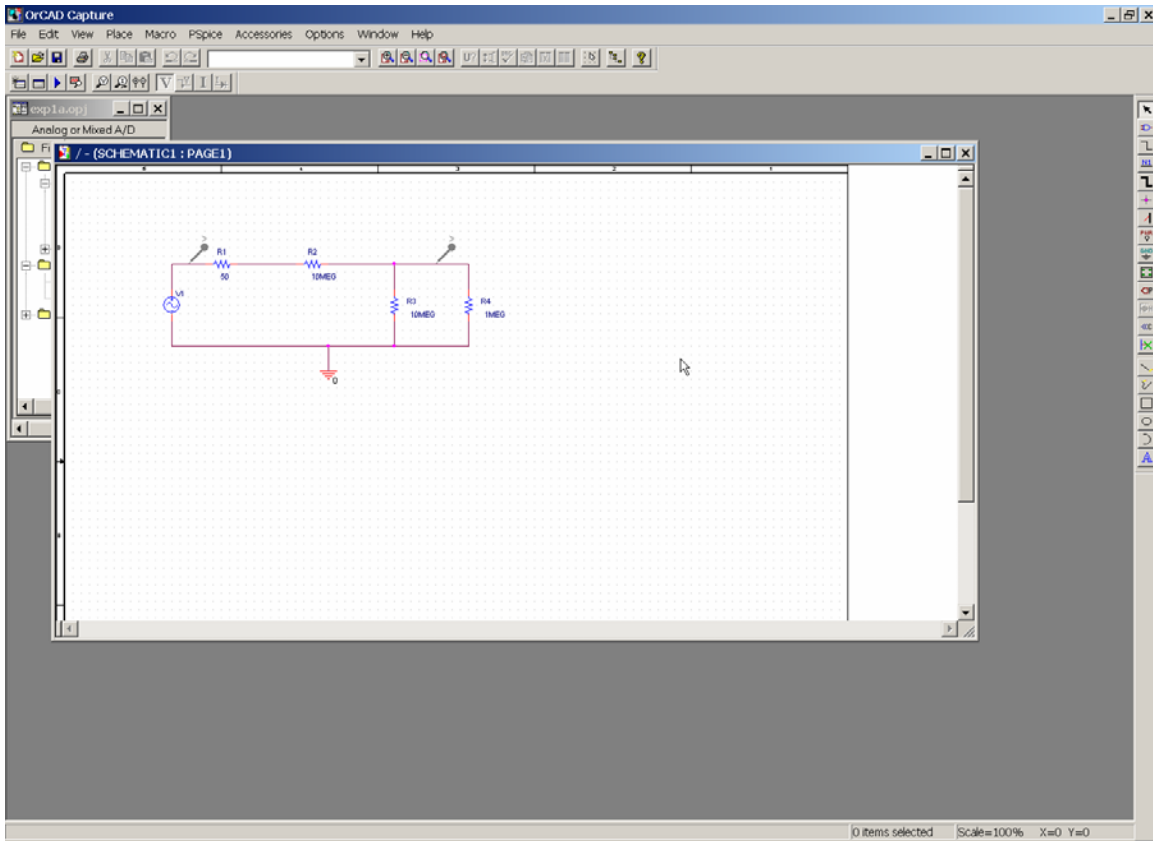
Once you have downloaded and installed the program, you must use the Capture program to draw your circuit and do the analysis.

Simulations with PSpice

The software we will be using to simulate the operation of circuits in this course is called *PSpice*. Actually we will be using a combination of two programs, *Capture* and *PSpice*. We will use the first to set up the circuit problem and the second for the analysis. These are loaded on the studio computers, so one looks for something called *Capture* and then clicks on it. Once you have done this, you should eventually see a window that looks something like the one below, except that neither of the smaller windows may be showing in it. In general, we will use the command buttons to make PSpice do our bidding. By moving the mouse arrow until it is located over any button you will be able to determine its function. To get a feel for how this works, we will draw the simple circuit: a combination of resistors and a sinusoidal voltage source.

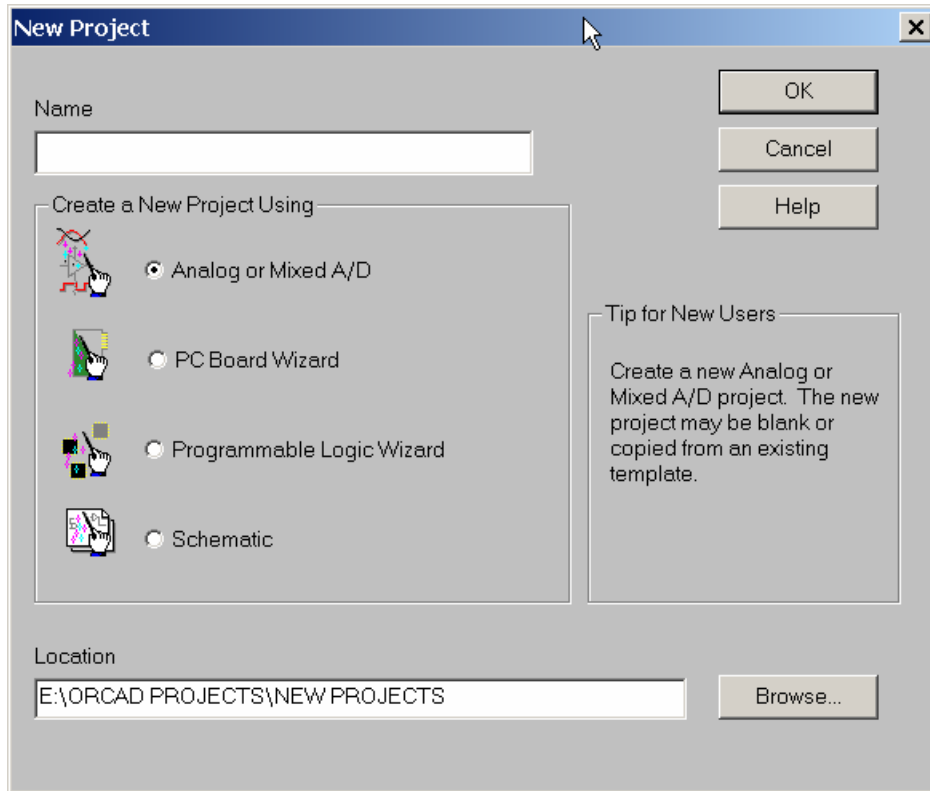
Fields and Waves I

ECSE-2100 Spring 2006



Before we can draw our circuit, we must set up the problem architecture for *Capture*. First click on the *File* pull-down menu and select *New Project*. You will see a new window named *New Project* shown on the next page. Be sure that the *Analog or Mixed-Signal Circuit Wizard* is selected, since that is the one that allows us to set up circuits for *PSpice* using *Capture*. You will use this window to give this project a name and choose a location where you wish to store it. In the box at the top of this window, give your project a recognizable name, such as *CKT-1*. You should then see the two small windows above, but with no circuit showing in the larger one. We will next see how to draw a circuit.

Fields and Waves I
ECSE-2100 Spring 2006



Drawing a Circuit

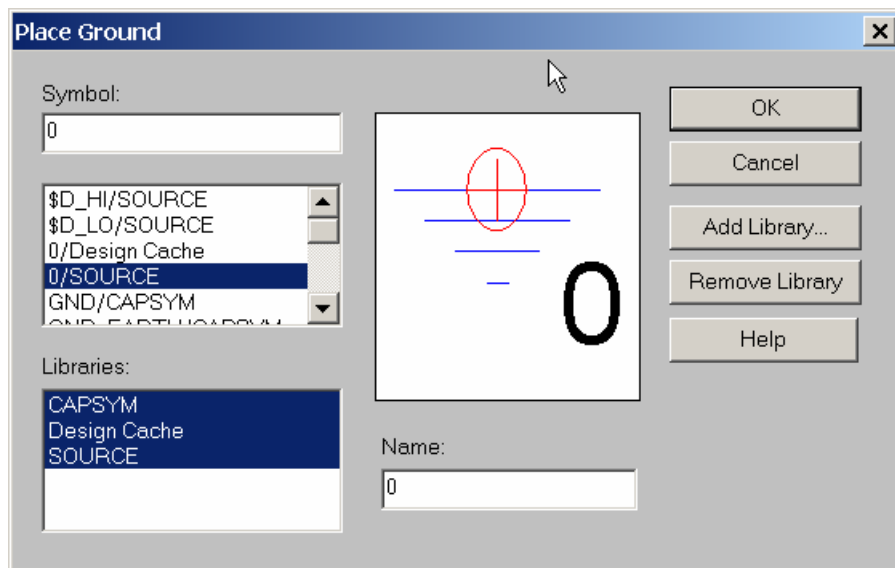
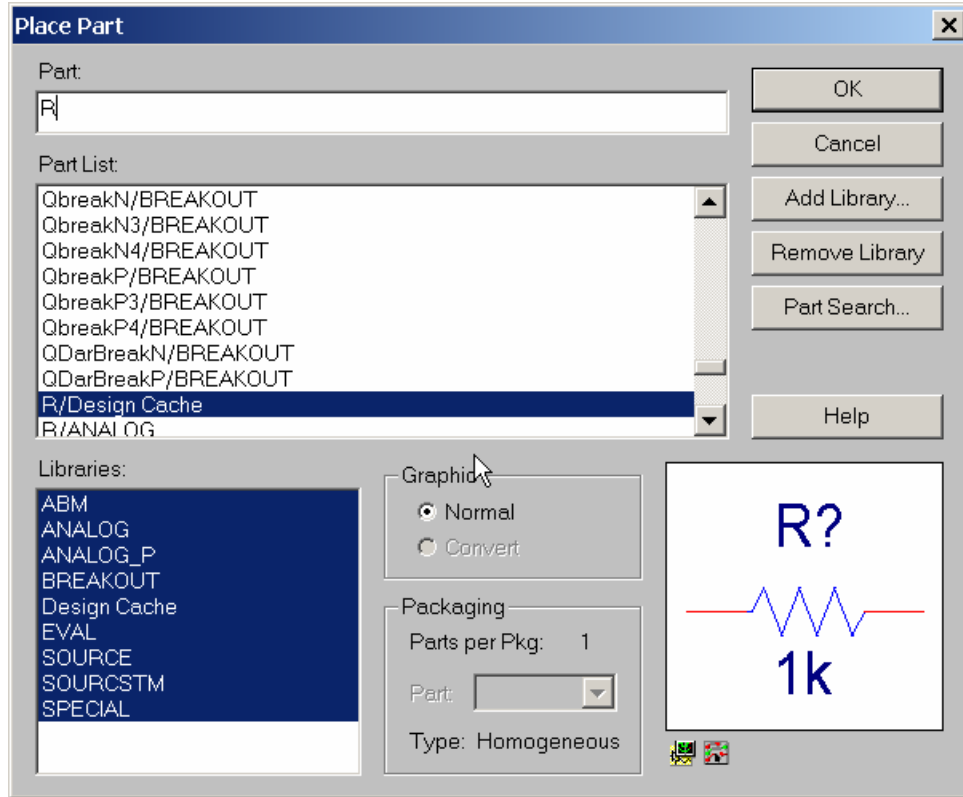
In the circuit shown, we have some resistors, a sinusoidal voltage source, a ground and some wires. To create this diagram, we will use the command buttons. For the resistors and the voltage source, we will click on the second button from the top that has the device with the three little wires sticking out of it (*Place Part*). You can also do this by using the *Place* menu or by hitting shift-P. The components and their PSpice names are:

Components	Resistor	Sinusoidal Voltage Source
Name	R	VSIN

To get a resistor, click on this button and then type an *R* in the space at the upper left marked *Part*, as shown below. The diagram below shows what the *Place Part* window looks like after a resistor has been selected. Then click on *OK* and you will be back to the main window and can place the resistors where you want them by moving the mouse around and clicking at the appropriate location. You will notice that there is a default condition for each component. For the resistor, the value will be 1k ohm and it will be horizontal. Since our circuit involves both horizontal and vertical resistors, you will need to rotate the symbol before placing it. To rotate the symbol, you need to use the keyboard command *R* for rotate. Once you have the resistor in the proper orientation, you can place it in your selected position. Just place all the resistors first. We will change their values later. When you have finished placing resistors, hit the *Esc* key or just click on the *Parts* button again. Then, to get the voltage source, click on the *Place Parts* button and type *VSIN* in the parts box and place the voltage source where you want

Fields and Waves I
ECSE-2100 Spring 2006

it. Again, you will have to rotate it first. Finally, all circuits need a ground reference so we know where the voltage is equal to zero. Click on the *Place Ground* button, which is about half way down on the right. You want to choose the *0/SOURCE* ground from the *Place Ground* window, since it is the only one that works with *PSpice*.

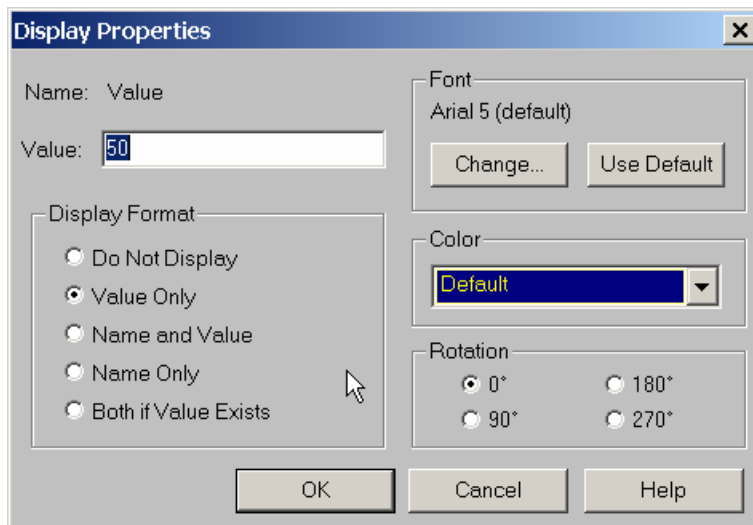


We now have all the components and must connect them with wires. For the wires we will click on the third button down on the right with the thin line that steps down on it

Fields and Waves I ECSE-2100 Spring 2006

(*Place Wire*). The mouse symbol will become a little cross. You will note that each component comes with little connecting wires on it. To connect two components together, just click the little cross near the boxes at the end of the little connecting wires. Make sure you click near the boxes. You can draw a wire almost anywhere, but you can only connect to the device at the ends of its connecting wires. 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. Then the dot will not appear. We now have drawn the circuit and need to indicate on it where we want to determine the voltage. On a physical circuit, we would connect a multimeter or an oscilloscope at these points. In our simulation we will do the same thing using the *Voltage Level Marker* button, which should be obvious, since it shows something that looks like a little probe with a *V* on it. (This button is located in the bottom row at the top of the main window.) Click on that button and place the probe to display the voltage at the upper right corner of the circuit (across the 1MEG resistor). You may want to rotate it into a convenient position. When the circuit is analyzed, a plot will be produced that shows the voltage at this location.

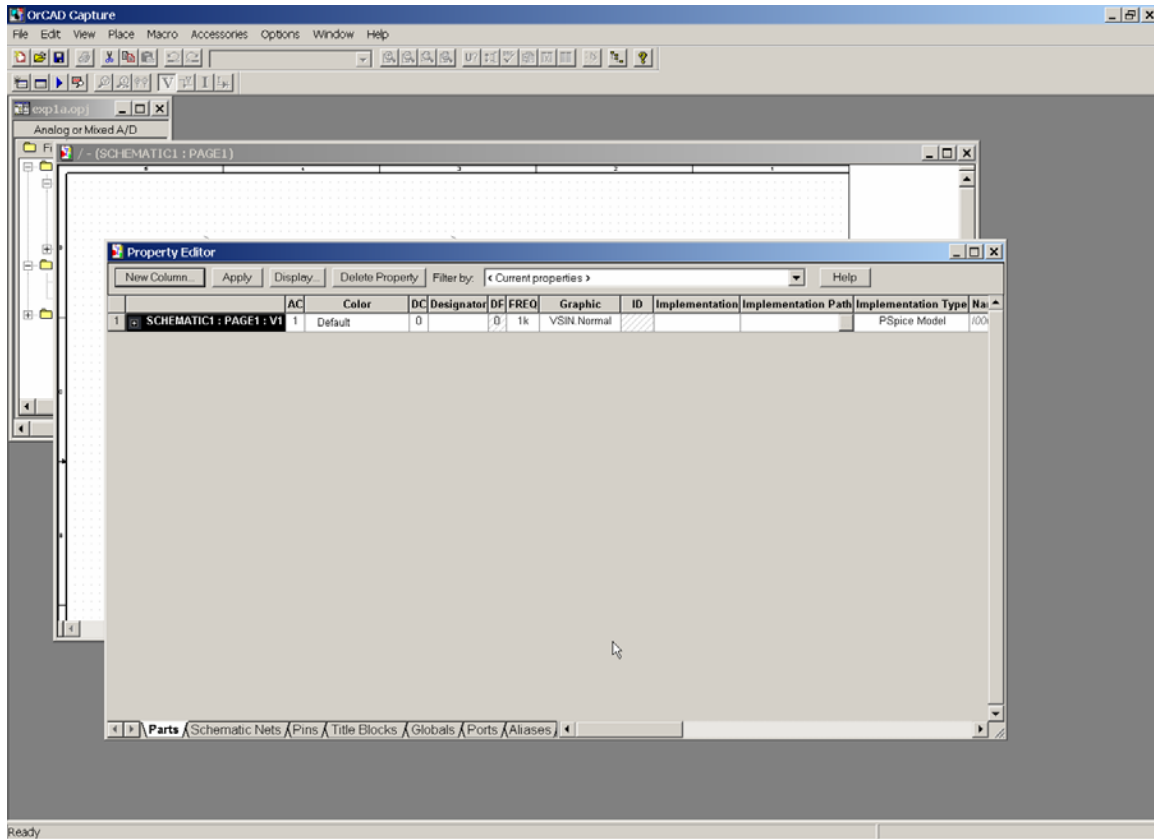
To complete the schematic, we have to change the component values. Each resistor was given a name in the order it was placed on the diagram. Thus, your resistors may not have the same names as shown above. However, for simplicity, they will be referred to by the name shown here. To change R1 to 50 ohms, double click on the value *1k* and you should get a window with the name *Display Properties*. Change the *1k* to *50*. If you find that the number is in a hard to read position on the circuit diagram, you can single click on it and then move it with the mouse. The next resistor brings up a very important issue since it has a value of 10 M ohms. When you type the value in the *Display Properties* window, you must type *10MEG* with no spaces. Note that you have to type *MEG* since PSpice uses *M* to mean 10^{-3} and *MEG* for 10^6 . It is not



case sensitive. After you have changed all the resistor values and moved them to readable positions, you must set up the voltage source. To do so you must click on the part itself, not the name. You should see a spreadsheet in which all the properties of the

Fields and Waves I ECSE-2100 Spring 2006

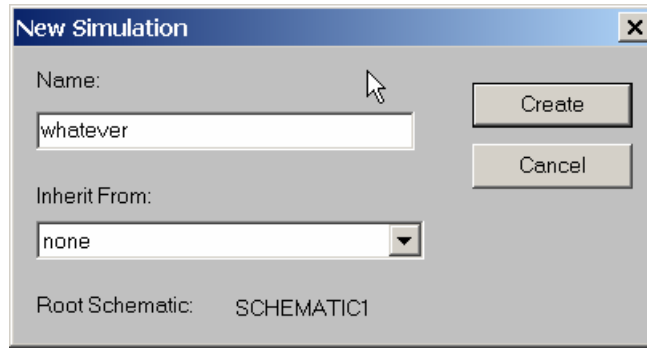
voltage source are stored, except that there will be no values for the last six parameters. These sources can be very flexible and can be used in a variety of ways. We need an amplitude and a frequency, so everything else except the last item can be zero. We will not be using the *ID* so we can leave that blank. Set the frequency (*FREQ*) to 1k and the amplitude (*VAMPL*) to 100mV. Note again that there is no space between the number and the *m* or *k*. Since we will often be using the source for more than one kind of analysis, it might be good to get into the habit of setting the AC source (*AC*) value to be the same as the amplitude. If you always do this, it will make life a little simpler in the future. Now the schematic is complete and each symbol stands for the correct part.



Setting Up the Analysis

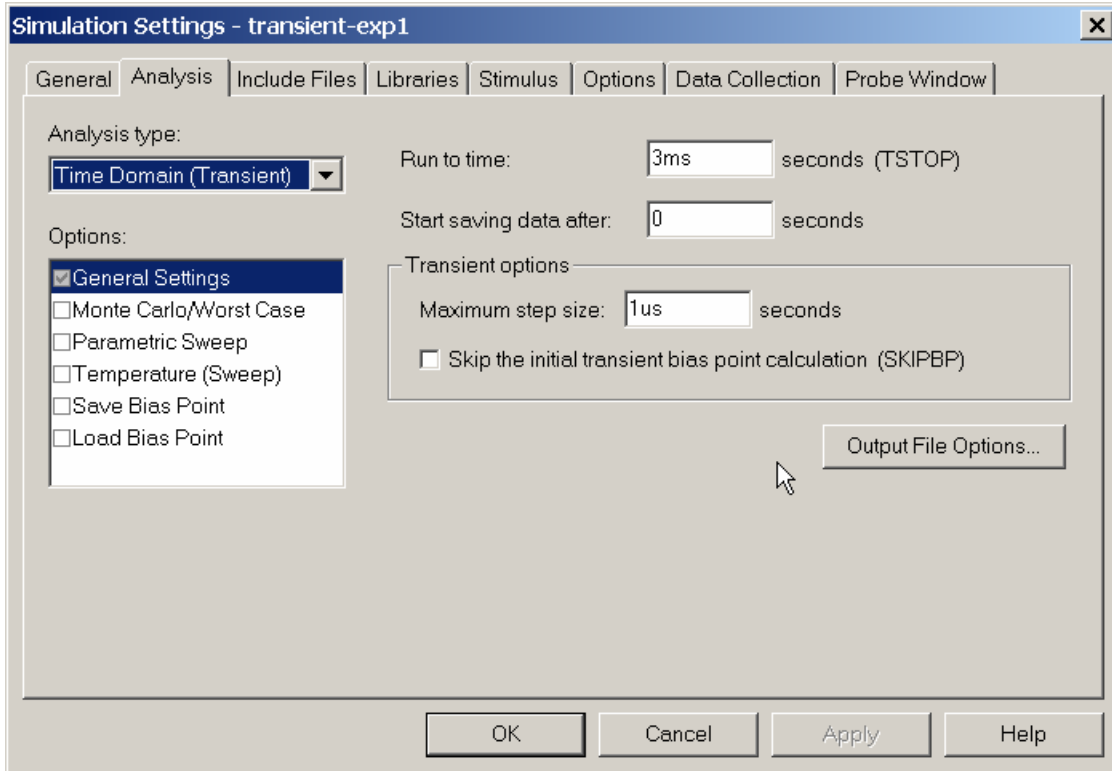
After we have defined all the components, wired them up and changed their values appropriately, we are ready to do some analysis. Find the *New Simulation Profile* button and click on it. (Hint: It looks like a little window.) You will get the window shown below, in which you must give a name to the file where the specifications for the analysis will be stored. Click on the *Create* button when you have chosen the name for your profile.

Fields and Waves I
ECSE-2100 Spring 2006



We will be doing *Transient Analysis* since that will produce a plot that is similar to what we see on the 'scope. The analysis options are found in the pull-down menu at the upper left of the *Simulation Settings* window, as shown below. Since this is the default option, you will not have to do anything this time. There are three boxes in which times need to be indicated. At least two and usually three numbers have to be put into these boxes. The final time (*Run to time:*) has been set at *3ms* because the period of a 1kHz sine wave is 1 msec. This allows us to see three periods. The *Maximum Step Size* has been 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. Note again that there should never be any spaces between the number and its units. Click on the *OK* button when you are finished putting in the numbers. When you have finished, you should notice that the button that looks like an arrow in the second row at the top of the main window, now has become active. This is the *Run* button that we will use to run our simulation. All the buttons in this row should now be active.

Fields and Waves I
ECSE-2100 Spring 2006

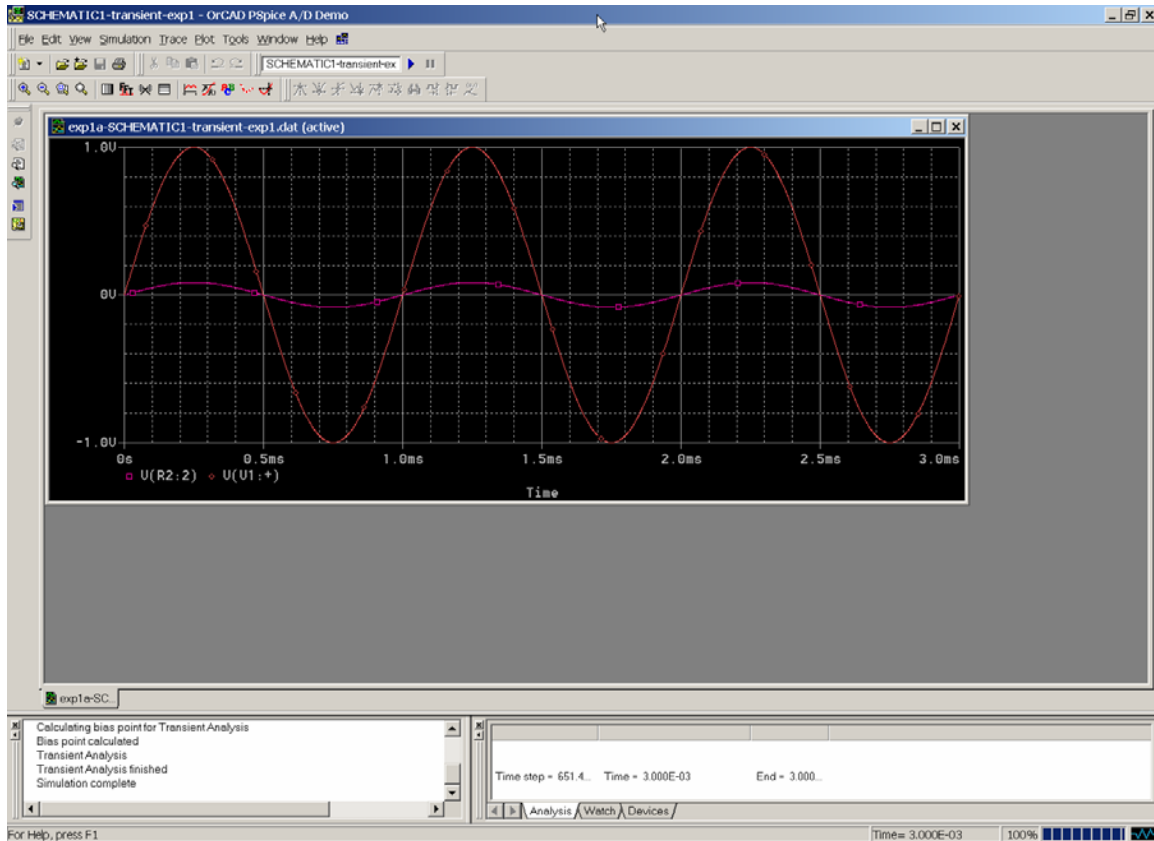


Transient Analysis

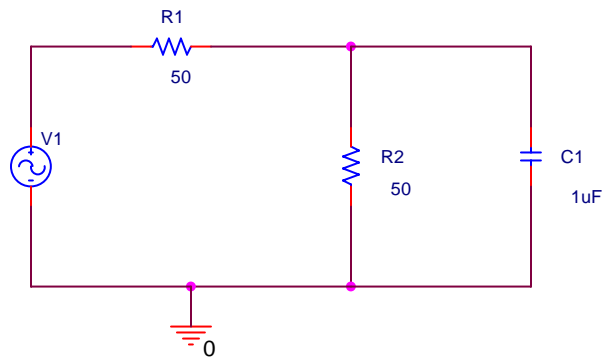
You are now ready, we think, to do the simulation. Click on the *Run* button. It is possible to set up many kinds of analysis using the *Simulation Settings* window. Since you have already told PSpice where you want to know the voltage, it will produce a plot with the signal you have asked for. You should now go back and add another voltage level arrow at the location that represents the output of the function generator. 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. You should get something like the window shown below.

Fields and Waves I

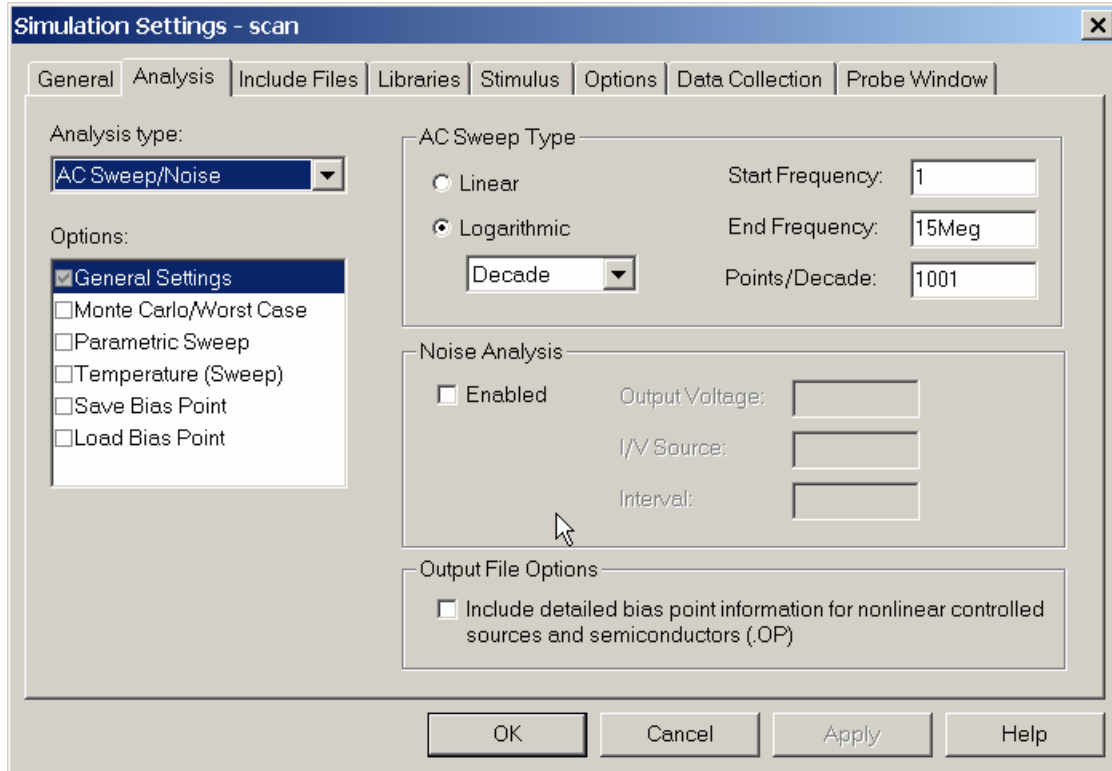
ECSE-2100 Spring 2006



There are other types of analysis that can be done. To determine the frequency response of a circuit, we can use AC analysis. Use PSpice to configure the circuit below (we need some capacitors or inductors to demonstrate frequency dependence). Click on the *Edit Simulation Settings* button. Select the *AC Sweep/Noise* analysis and set it up as shown.



Fields and Waves I
ECSE-2100 Spring 2006

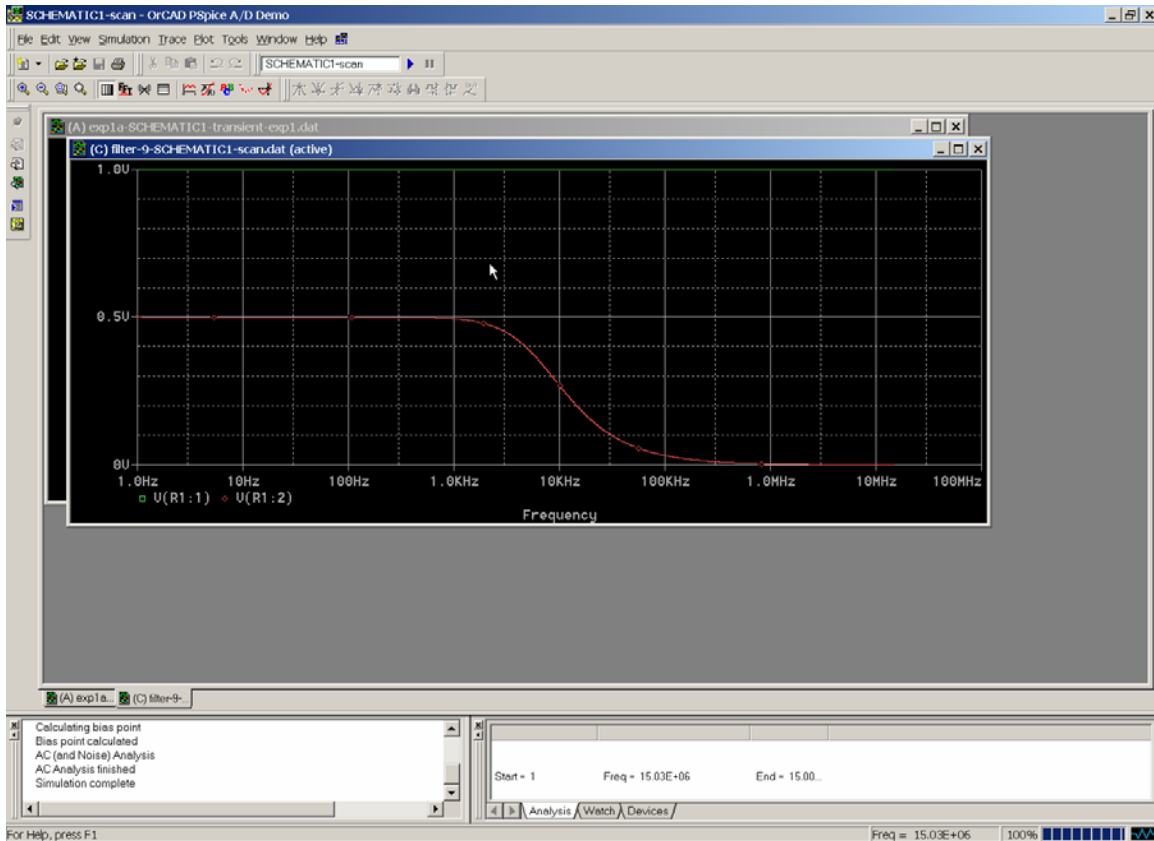


The AC Sweep Type has been chosen as *Logarithmic - Decade* since frequency effects usually only become obvious when we change orders of magnitude. This generates a log scale for frequency. The start frequencies and end frequencies are chosen to cover an interesting range. Usually this range is selected 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 the voltage source V1 again to bring up its 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.

PROBE will produce a plot consisting of a flat line and a line that is flat for lower frequencies but falls to zero for higher frequencies. These lines are the amplitudes of the sine waves at the two nodes where we have placed voltage markers.

Fields and Waves I

ECSE-2100 Spring 2006



Some of the Parts Included in the Student Version Libraries

R, L, C are obvious

VSIN is the sinusoidal source – specify VAC, VDC, frequency, VAMPL and VOFF

VPULSE is a pulsed source – V1 is V off, V2 is V on, TD is time delay before pulse starts, TR is rise time, TF is fall time, PW is pulse width and PER is period

VDC is a DC source

T is a lossless transmission line – there are two ways to specify the transmission line parameters.

- TD, the time delay or transit time for the line and Z0, the characteristic impedance
- F, the frequency and NL, the relative wavelength

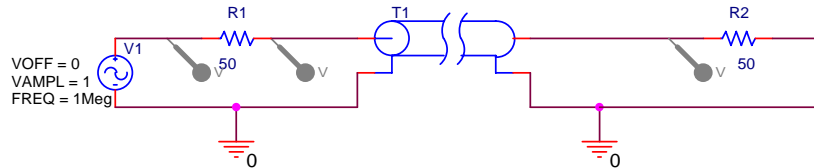
TLOSSY is a lossy transmission line – the parameters of the line are the usual R, L, G and C per unit length and LEN the actual length (in meters) of the line. The student version will not run with more than 4 lossy transmission lines. A common error made by many students is to input the characteristic impedance for R. *Remember that R is the term that represents the loss in the copper of the conductors and G is the term that represents the loss in the insulator between the conductors. Neither of these terms is the characteristic impedance. PSpice figures that out from the values for R, G, L, and C that you give it.*

Fields and Waves I
ECSE-2100 Spring 2006

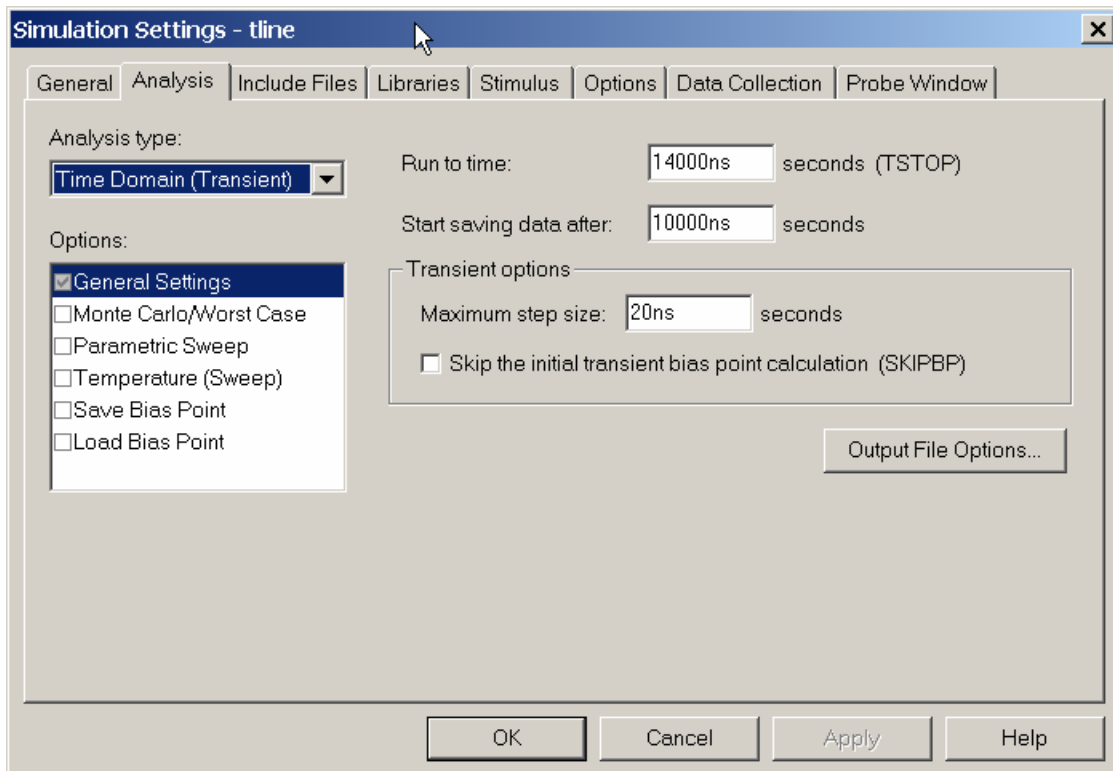
Transmission Line Modeling

The two transmission line models available are quite good. To do a simple study of a standard transmission line, there are several approaches available.

Using the Lossless Line – A function generator driving a matched 50 ohm line with a 50 ohm load looks as follows.



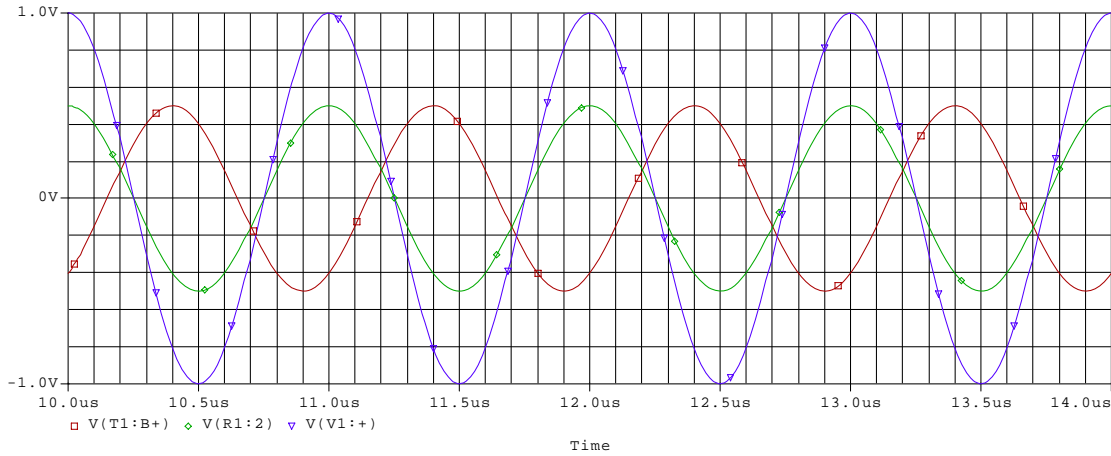
Note that the source is VSIN and that the offset voltage is 0, the voltage amplitude is 1 and the frequency is 1MHz. This image was produced by highlighting the entire circuit in the capture window and then copying and pasting it here. The transmission line parameters chosen are TD=400ns and Z0=50. Thus, it corresponds roughly to the spool of coax used in the Fields and Waves I class. The transient analysis of this circuit was set up as follows. Note that it is necessary to have the simulation run for a while before plotting the output to allow steady state conditions to be achieved.



Fields and Waves I

ECSE-2100 Spring 2006

which results in the following voltages at the three points shown.



This plot was produced from the capture program using the copy to clipboard command found under the window option.

This same result can be obtained using the lossy (TLOSSY) transmission line, if $R = G = 0$, $L = 0.25\mu\text{H}$ and $C = 102\text{pF}$. The line length also needs to be 80 meters. The phase of the output is a little different (probably an artifact of the code) but the voltages are the same.

