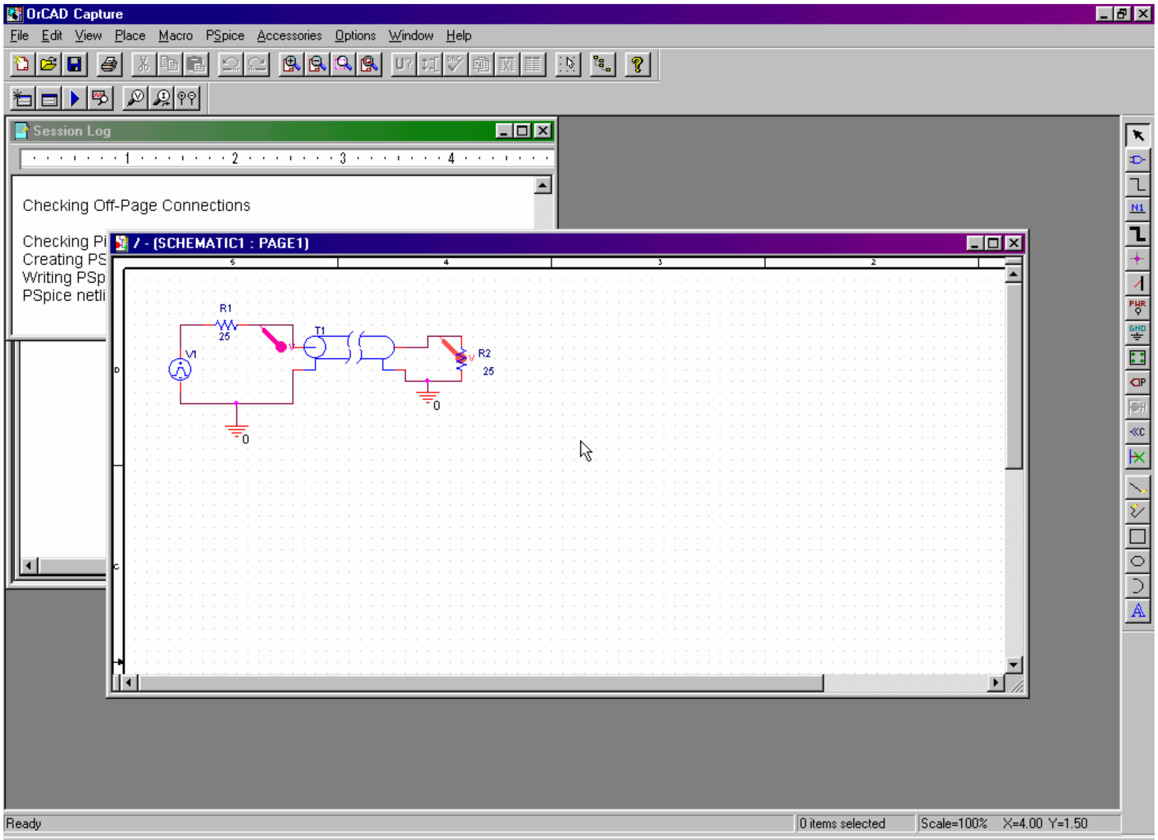
Using PSpice to Simulate Transmission Lines

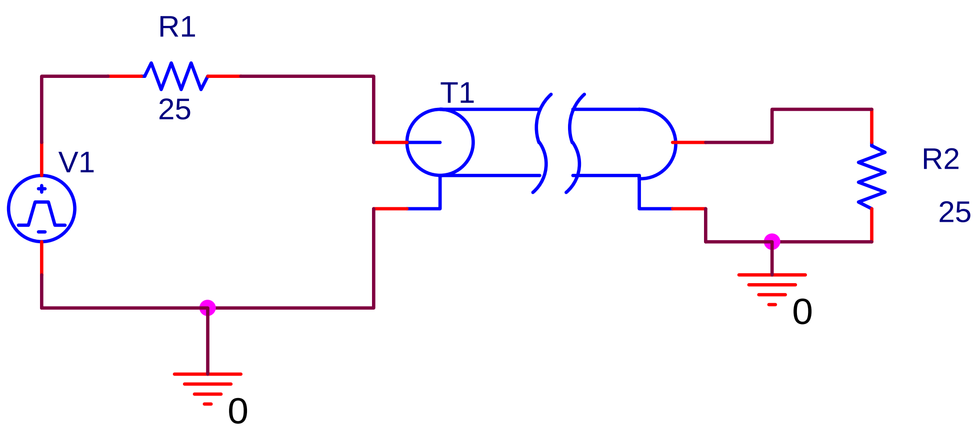
K. A. Connor

Summer 2000

Fields and Waves I



We want to produce the image shown above as a screen capture or below as the schematic of this circuit.



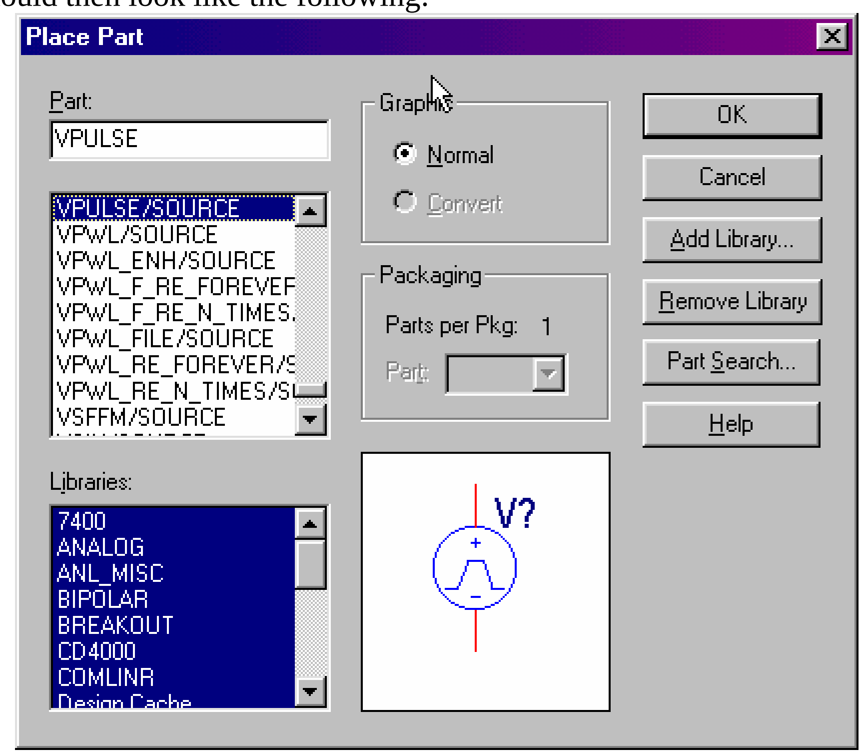
1. Begin by starting the program *Capture* from ORCAD.

2. Click on *File*, *New*, *Project*.

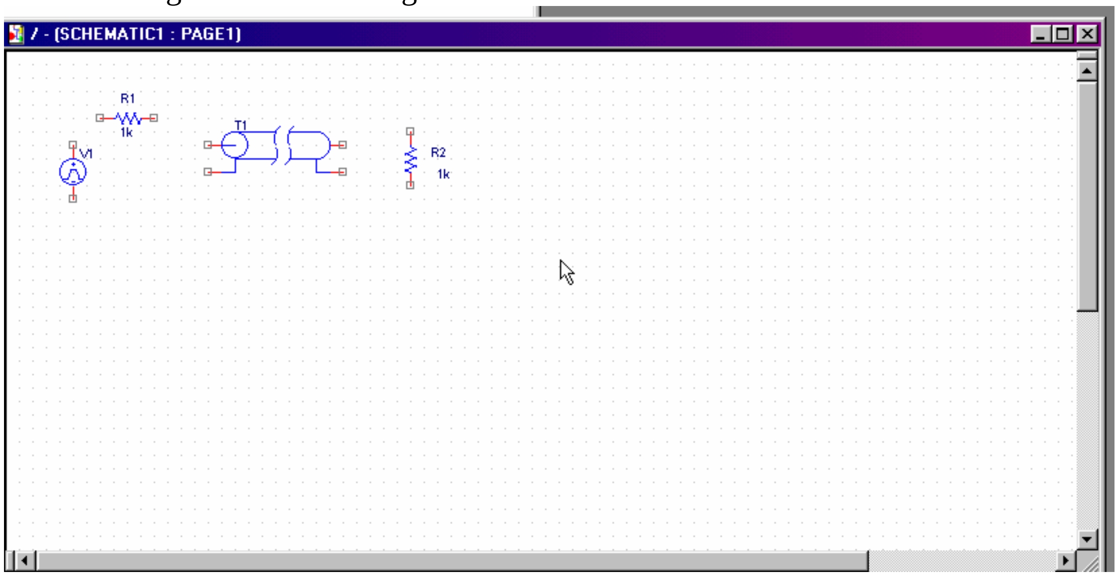
3. A new window will appear entitled *New Project.* You must pick a name for your project. It does not matter what you call it, but give it a name you can remember. The name goes in the space marked *Name*. (I know that was obvious, but I want to put in all of the steps.) Hit enter or click on *OK.* A new window will appear asking you what libraries you want to include in your project. Just click on the *Finish* button. You will be accepting the standard set of libraries, which is all you will need.

4. A new window appears called something like *SCHEMATIC1*. This is the window where you will draw your circuit. Look to see if there is a set of menu buttons along the right side of the screen. If not, click on the bar at the top of the new window. It should change from green to blue and the buttons should appear. You will need to use a small number of these buttons. You should see something like the first figure above, except that there will be no circuit components shown yet. It takes several steps to create the circuit diagram, prepare and run the simulation.

5. The first button you will need to use is the second from the top and looks like a logic gate symbol. Click on this button and you will see a new window with the name *Place Part.* To choose a part, you type its name in the space provided. We will begin by simulating the function generator and select a pulsed voltage source *VPULSE.* After you type this word in, you should see the pulse at the top of the list right below where you typed *VPULSE.* You must now select *VPULSE/SOURCE* from this list. The first word for each part is its name, the second word is the name of the library in which it is found. After you have selected the voltage source, its symbol should appear in the box. Your window should then look like the following:



Click on the *OK* button. You can then place the symbol for the source anywhere on the schematic window. The other parts we will need are *R* for two resistors and *T* for transmission line. Once you have selected and placed these parts, your window should look something like the following:



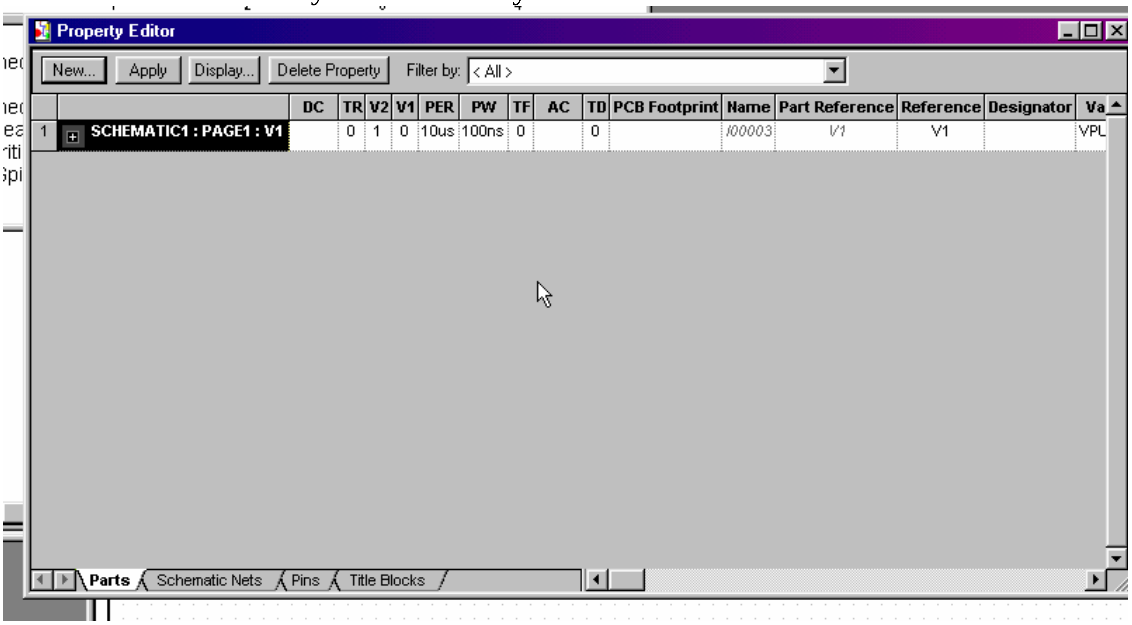
When you first place the resistors, you should see that they are positioned horizontally. To rotate the load resistor at the right, click on it and then hit the *R* key in your computer keyboard. You can also choose *Edit, Rotate* to accomplish the same thing.

6. The next button we need is the wire button, which is the third button, just under the parts button. After you click on it, you will be able to draw wires between the connection points at the end of each of the parts.

7. The next button is the ground button, which is the ninth button and looks like a ground symbol. Because of the model used for the transmission line (which is discussed in Paul, Whites and Nasar), it is necessary to ground the bottom connection on both sides of the transmission line. The ground you need to choose is called *0/SOURCE*, because PSpice needs some nodes set to zero volts to work properly. There are several other choices which look good, but don't work. Once you select and place your grounds, you will have to wire them to the circuit using the wire button.

8. The next step is to specify the characteristics of the devices in our circuit. The resistors are easy. You only need to double click on their values (which should be the default value of 1k) and a new window will come up. Change the values of these resistors to 50, so we can begin by looking at matched source and load conditions. Next you must double click on the voltage source. Be careful to hit the source and not the wires connected to it. This program pretty much allows you to change everything, so be sure that you have selected the correct part. What should come up then is a new window called the *Property Editor*. This has the form of a spreadsheet in which the part properties are specified. To obtain a pulsed source with an amplitude of 1 volt, a duration of 100ns and a period of 1us (PSpice uses u for micro), fill the table as shown. You will have to

use the horizontal slider to get to these parameters in the table. The window shown below shows how it looks after you have made your choices.

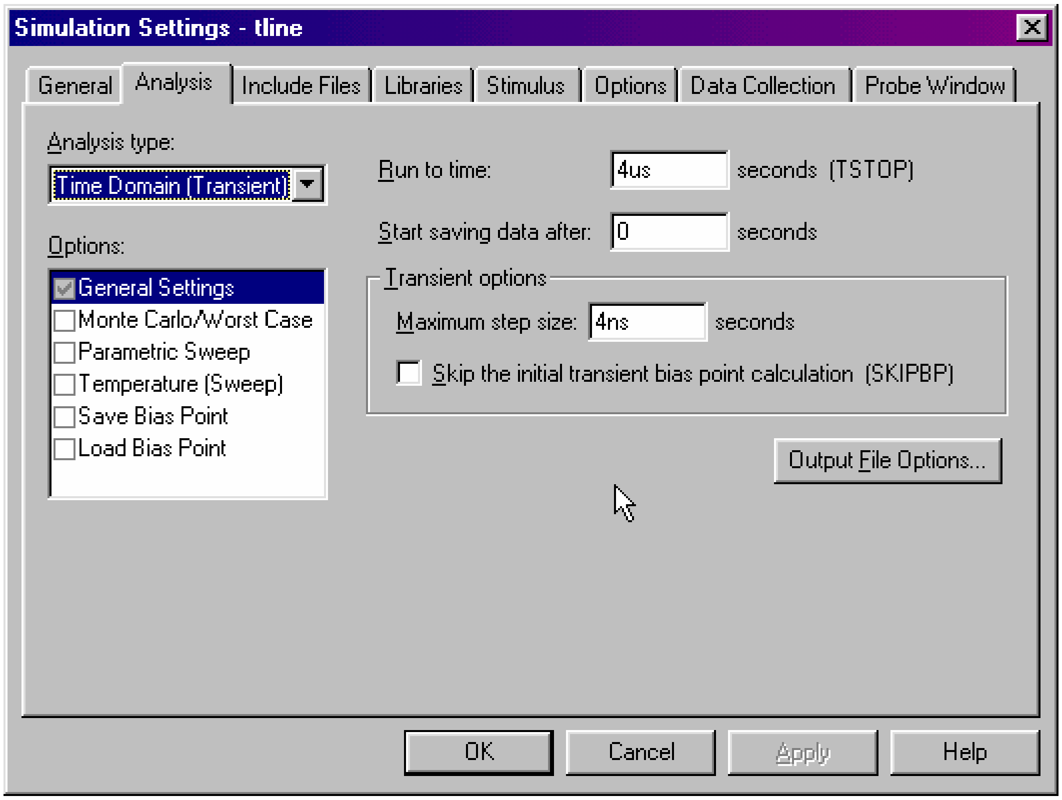


When you have finished, click on *Apply* and then close the window. Note that you should set the fall time, rise time and delay times to zero. Also, the AC and DC can be left blank. Next, double click on the transmission line and set its characteristic impedance to 50 and time delay to 400ns.

Please Note: Those of you who have used PSpice will recall that you cannot leave any spaces between the number and the units. Also, *m* and *M* are used for *milli-*, so you must use *Meg* for *meg-*.

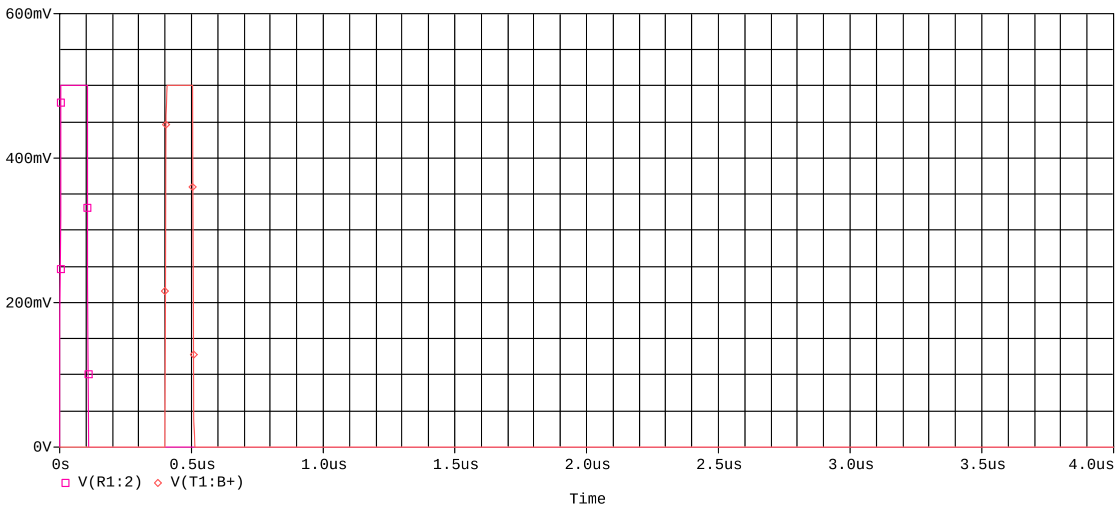
9. The circuit is now complete. You must now specify the simulation you wish to run. You have several choices. We want a transient simulation that allows us to see the pulses should they reflect back and forth a few times on the line. Thus, we should simulate up to about 4us. To set up the simulation, you must click on the button at the upper left (not on the right any more) that looks like the symbol for a window. It is located below the *File* menu button. After you click on it you will be asked for the name of your simulation. Again, you can choose anything you wish. After you choose a name, you will see the following window, which has already been filled in with the information we need.

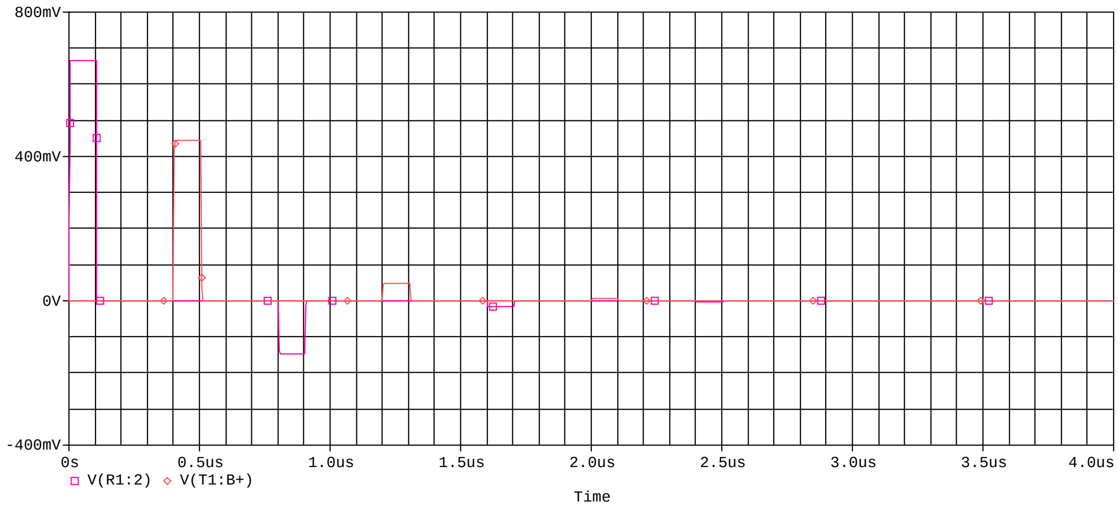
Note: Just like most other programs, you can see what each button does by positioning your mouse over it and waiting a second or two. Then a short description of the purpose of the button will appear. The first button we want is called *New Simulation Profile*.



To run this simulation, click on the arrow button (two to the right of the simulation button and called *Run*). If everything has worked correctly, you will get a new window which will contain the plots of the voltages and currents in the circuit.

10. We want to display the input and output voltages, just like when we do an experiment. This can be done graphically by returning to the schematic window. Two buttons to the right of the arrow button is the voltage level button which looks like a V in a circle and an arrow. Click on this button and then place a voltage marker at the input and output of the transmission line. You will notice that the voltage markers will have distinct colors. If you look at the plot window now, you will see two voltage plots, color coded to match your markers. Note that in this case, we have a matched load and source, so the pulse appears only once at the input and once at the output and then nothing else happens. This boring situation is exactly what we want to happen. If we change the source and load resistances to 25 ohms, we get the next plot.


600mV
400mV
200mV
0V
0s
0.5us
1.0us
1.5us
2.0us
2.5us
3.0us
3.5us
4.0us
V(R1:2)
V(T1:B+)
Time


800mV
400mV
0V
-400mV
0s
0.5us
1.0us
1.5us
2.0us
2.5us
3.0us
3.5us
4.0us
V(R1:2)
V(T1:B+)
Time