

# ORCAD/PSPICE Tutorial

Dr. George L. Engel  
Department of Electrical and Computer Engineering  
Southern Illinois University Edwardsville  
Edwardsville, IL

September 3, 2003  
Version 1.0

Students may obtain a student version of ORCAD and SPICE by visiting Dr. Engel's homepage at [www.ee.siu.edu/~gengel](http://www.ee.siu.edu/~gengel). Click on the **Teaching** link and then on **ECE327**. Download the zip file and unzip it. Install the software by double clicking on the Setup icon. When asked if you wish to install *Schematic* or *Capture*, select *Capture*.

After downloading and installing the demo version of ORCAD, create a shortcut to the Capture program. Place the shortcut on your desktop to make it easy to launch the application. Also, create a folder somewhere on your drive where you can save your schematics and your PSPICE simulation files.

For example, here is the shortcut on my desktop. I called it ORCADdemo

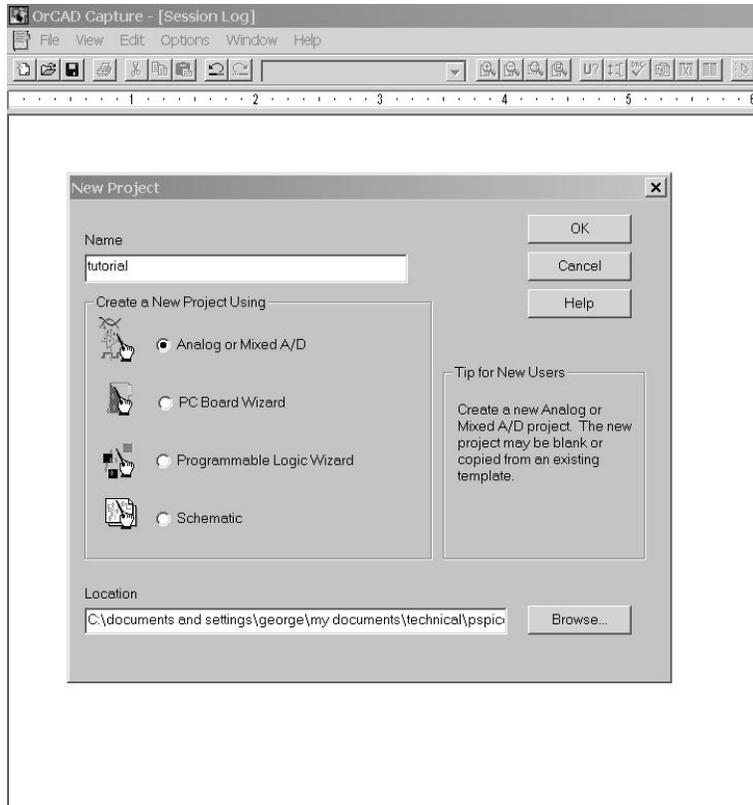


ORCAD is a schematic editor. In other words, it is a program that allows you to draw circuit schematics. The demo version that you have downloaded comes with a nice collection of parts libraries with parts that you are likely to use in student assignments and student laboratory exercises.

PSPICE is an electrical simulator that allows you to simulate the operation of the circuits whose schematic you prepared using ORCAD. This allows students to check for correct circuit operation even before they actually construct the circuit in lab. The demo version of ORCAD allows you to run PSPICE without even leaving the ORCAD user interface.

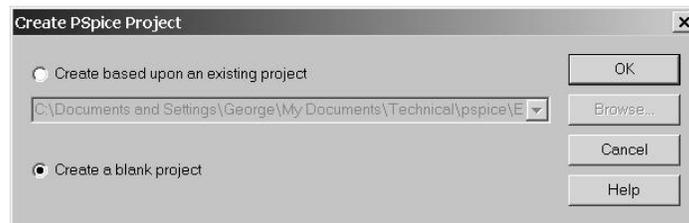
Launch the ORCAD schematic editor by double-clicking on the *ORCADdemo* icon.

Click on **File** from the main menubar and then select **Open New Project**. You should then see the following dialog box.

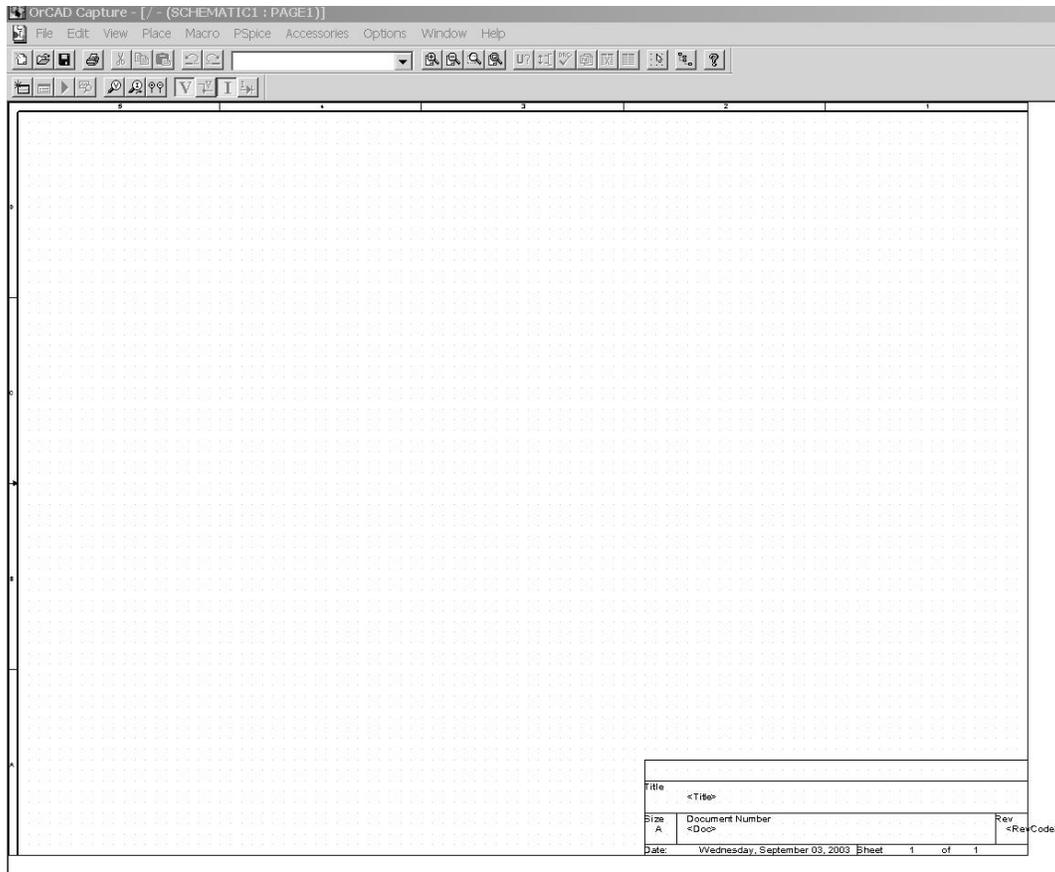


I decided to name my project "tutorial". Notice I have entered the *Location* of the folder where the schematics and my simulation results will be saved. Use the *Browse* button to select the location where you would like to save your files. Make sure the **Analog or Mixed A/D** option has been selected. You may then click the OK button.

The following dialog box will appear.

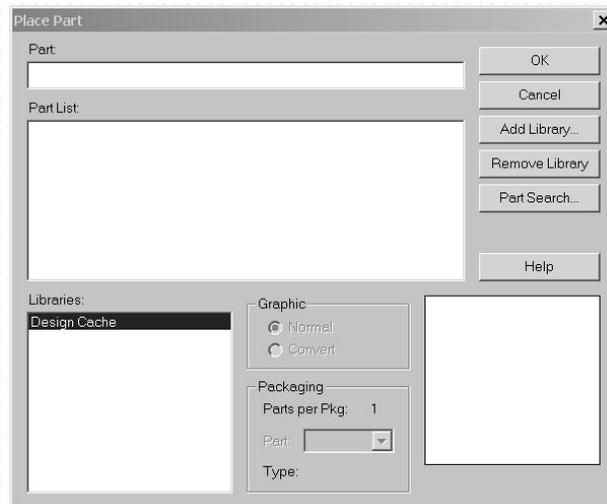


Make sure to select **Create a blank project** and then click the OK button.

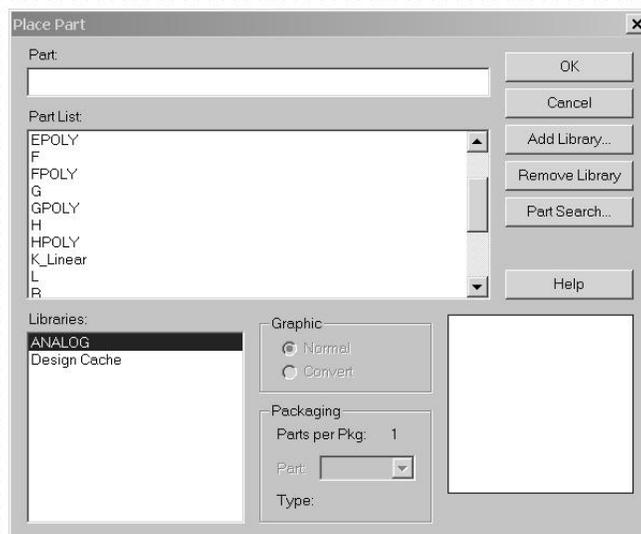


Let's create a simple voltage divider i.e. two resistors, each with a value of  $10\text{K}\Omega$ . We will drive the series combination with a sinusoid and observe the voltage at the node between the two resistors. Voltage division principles tell us that the voltage should be half of the input voltage.

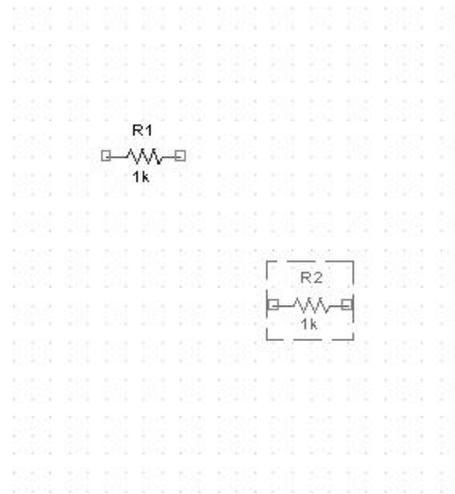
From the menu at the top, select **Place** and then **Part** (**Shift-P** if you wish can be used as a shortcut). The following dialog box will appear.



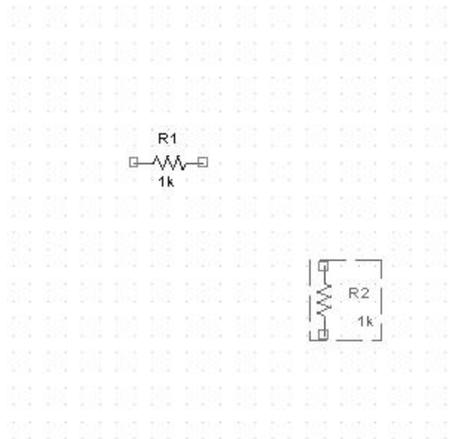
Click on the **Add Library** button and choose the ANALOG library. You will see the following.



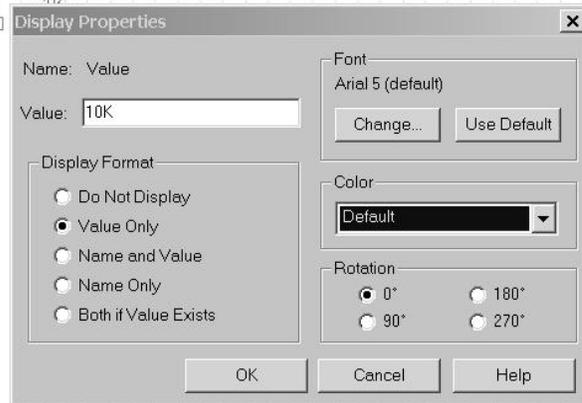
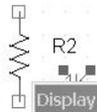
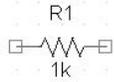
Select the R *i.e.* resistor component. This can be done by double-clicking on the R. Place the component by clicking somewhere near the middle of the schematic page. Click again and place R2. You should then hit the *Esc* key to exit the place mode. The resistor R2 should still be selected as is shown on the next page.



With resistor R2 selected, right mouse click on it. A menu will appear. Select the **Rotate** option. You should observe the following.



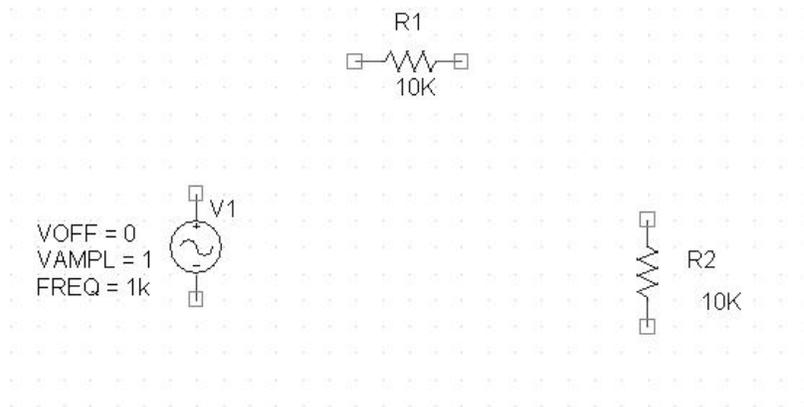
Right mouse click on the 1k value on the R1 component. A menu will appear. Select **Edit Properties**. The dialog box shown below will appear. Change the value to 10K.



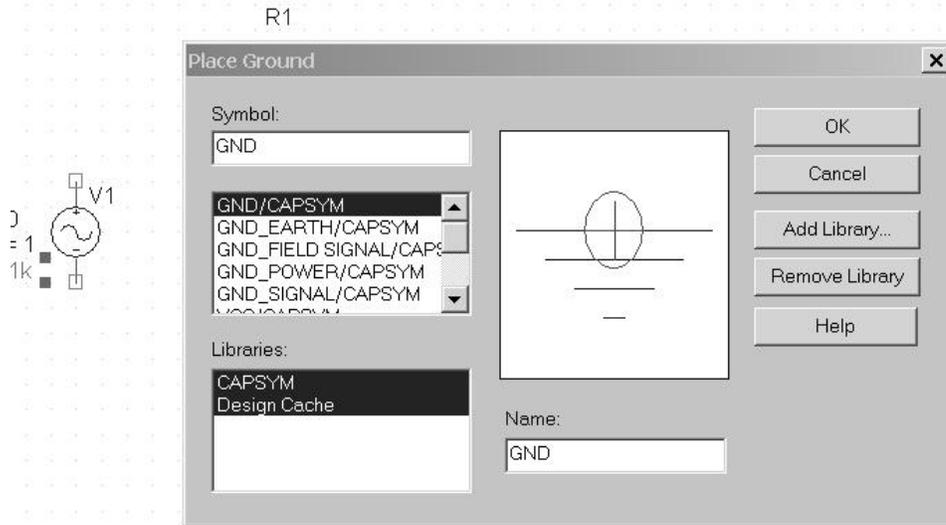
Do the same for the R2 component.

At this point we should probably save our work. This can be done by either typing Ctrl-S or by clicking on the diskette icon in the toolbar or by going to the **File** menu and selecting **Save**.

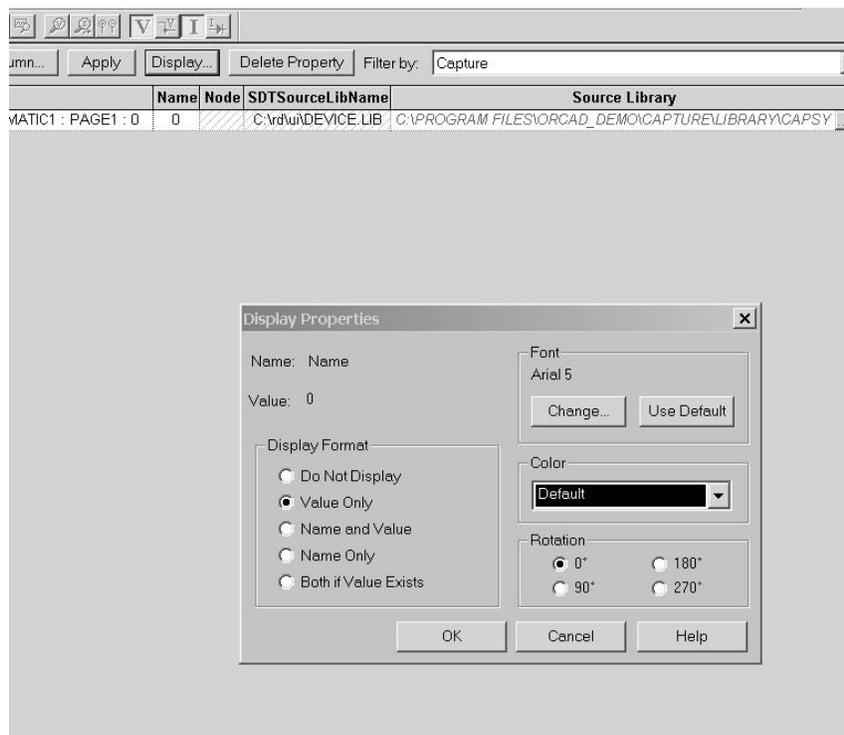
Do a **Place Part** again. This time add the *Source* library, From the *Source* library, select **VSIN**. Place the sinewave generator and change the parameter values. VOFF should be 0. VAMPL should be set to 1. FREQ should be set to 1k.



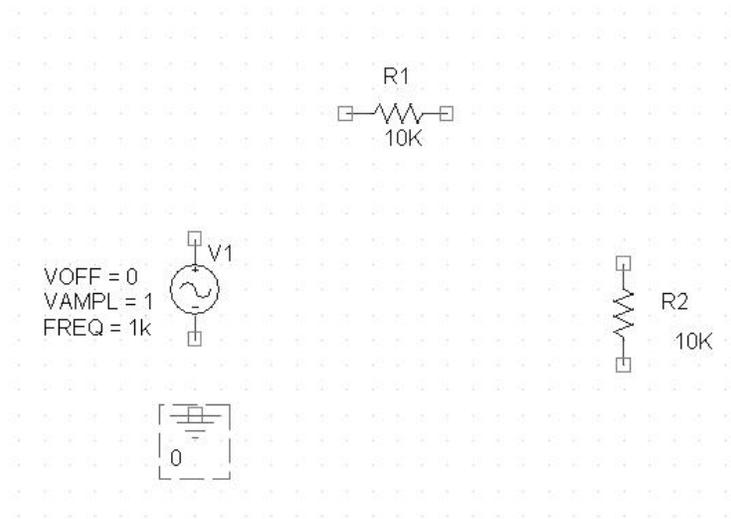
Now place the ground symbol. Choose **Place** and **Ground**. Let's use the GND/CAPSYM symbol. Any of the grounds would work but this is my personal favorite!



In order for PSPICE to simulate the circuit correctly, we must change the name of the GND symbol from GND to 0. This is because in all versions of SPICE, circuit ground must always be assigned a nodename of 0. We can do this by selecting the GND symbol. Now, right mouse click on the symbol. A menu will pop up. Select the **Edit Properties** option. In the *Name* field where it says GND, change this to 0 as I have done below. Use a right mouse click on the Name field to highlight the field and click on display. The *Display Properties* dialog box will appear.



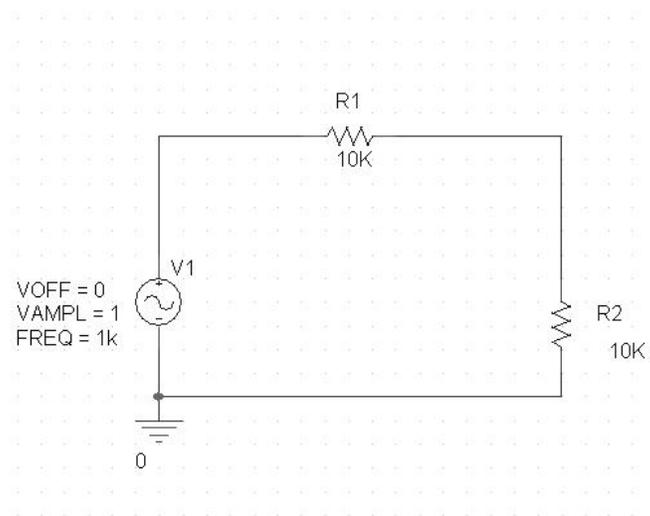
Your circuit should look like the figure below.



It is now time to wire the components to one another. This can be done by placing a wire. **Place Wire** and Shift-w is the shortcut! Click on a pin then drag to mouse to another pin and click again.

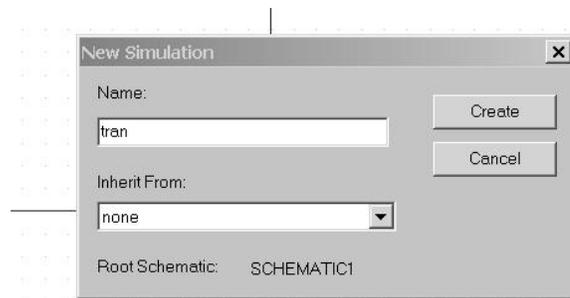
When the components are all wired together, leave the wiring mode by hitting the *Esc* key.

Your circuit should look like mine below.

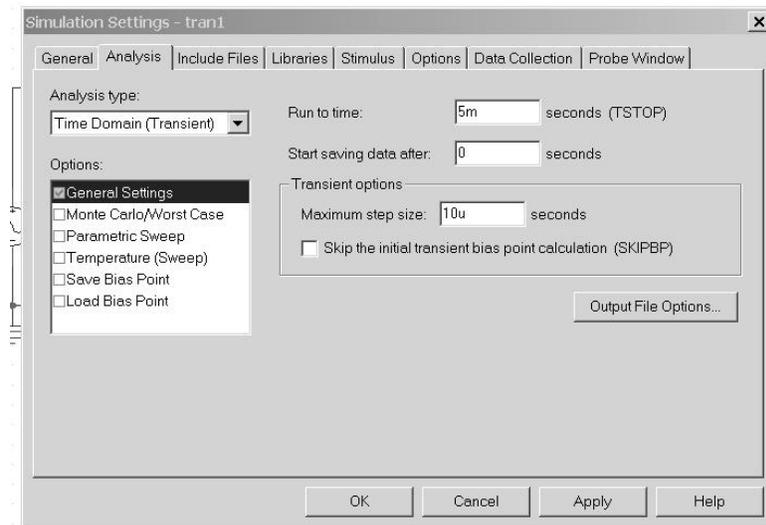


We now are ready to simulate the circuit. We must first prepare a simulation profile where we describe how we want the simulation to be performed. To do this select **PSpice** and then **New Simulation Profile**.

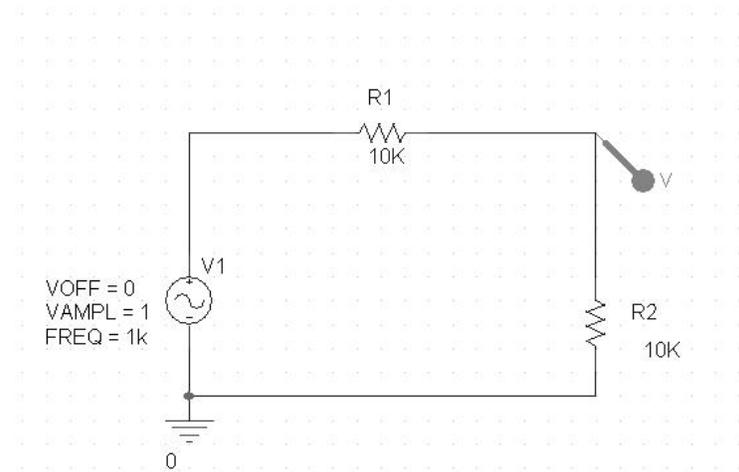
The following dialog box appears. I have entered a name for the simulation. I decided to call it tran. You may call it anything you wish. Click on the **Create** button to create the profile.



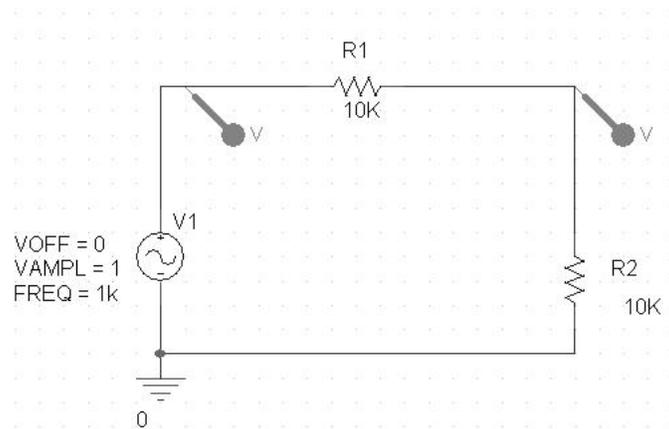
After clicking on the **Create** button, the following dialog box will appear. Since the input frequency is 1 kHz which has a corresponding period of 1 ms, I have decided to run for 5 ms *i.e.* 5 periods of the waveform. I also desire 100 data points per period of the sinusoid so I have indicated that the maximum step size be 10  $\mu$ s ( $1 \mu$ s / 100 = 10  $\mu$ s)



We also need to tell PSPICE which nodes we would like to plot. We do this by going to the main menubar and selecting **PSpice**. We then select **Markers** and **Voltage**. We place the marker on the output of the voltage divider as shown below.

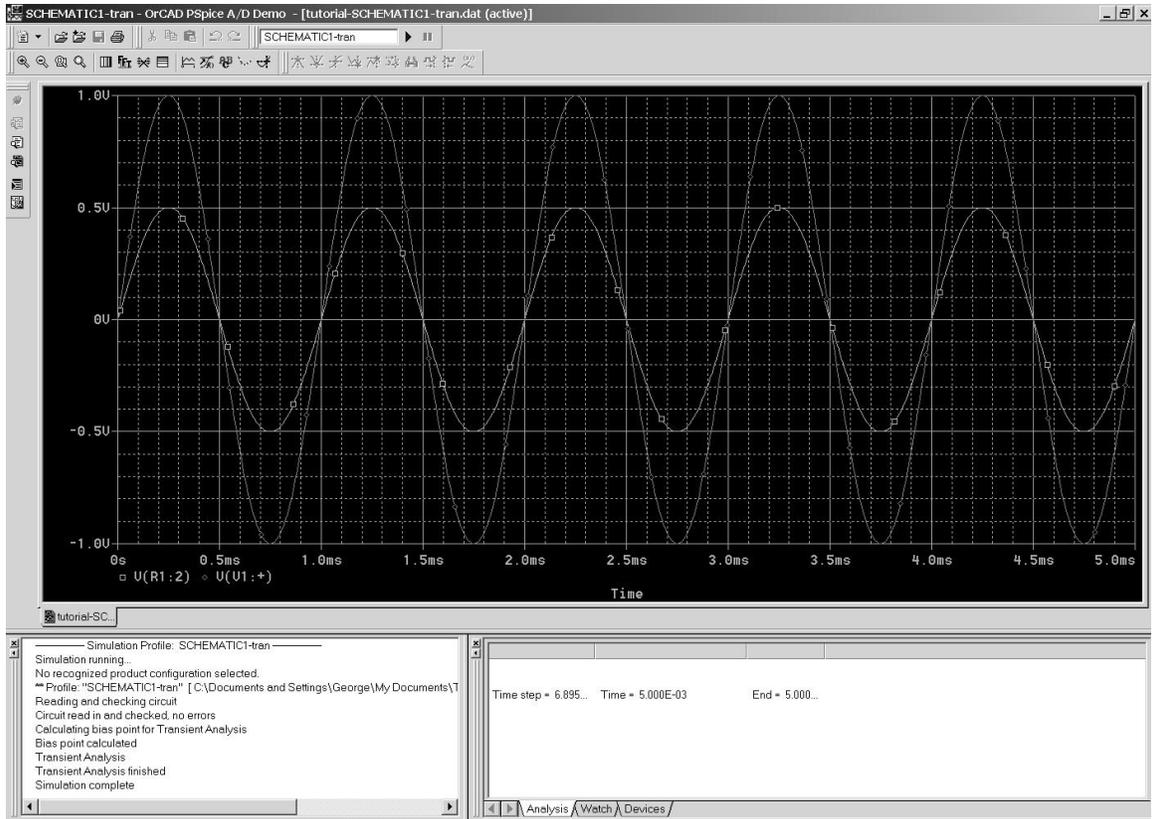


For comparison purposes, let's also place a marker on the input node as illustrated in the figure below.



At this point we should probably save our work. This can be done by either typing Ctrl-S or by clicking on the diskette icon in the toolbar or by going to the **File** menu and selecting **Save**.

We are now ready to run the PSPICE simulation. We do this by selecting **PSpice** from the main menubar and then **Run**. We could also click on the arrow icon on the toolbar. After a few seconds another window will open and you will see the following simulation traces.



Notice the green trace (output) has one-half the amplitude of the red trace (input) as we predicted.

Congratulations!!! You have successfully run PSpice to simulate a simple voltage divider.