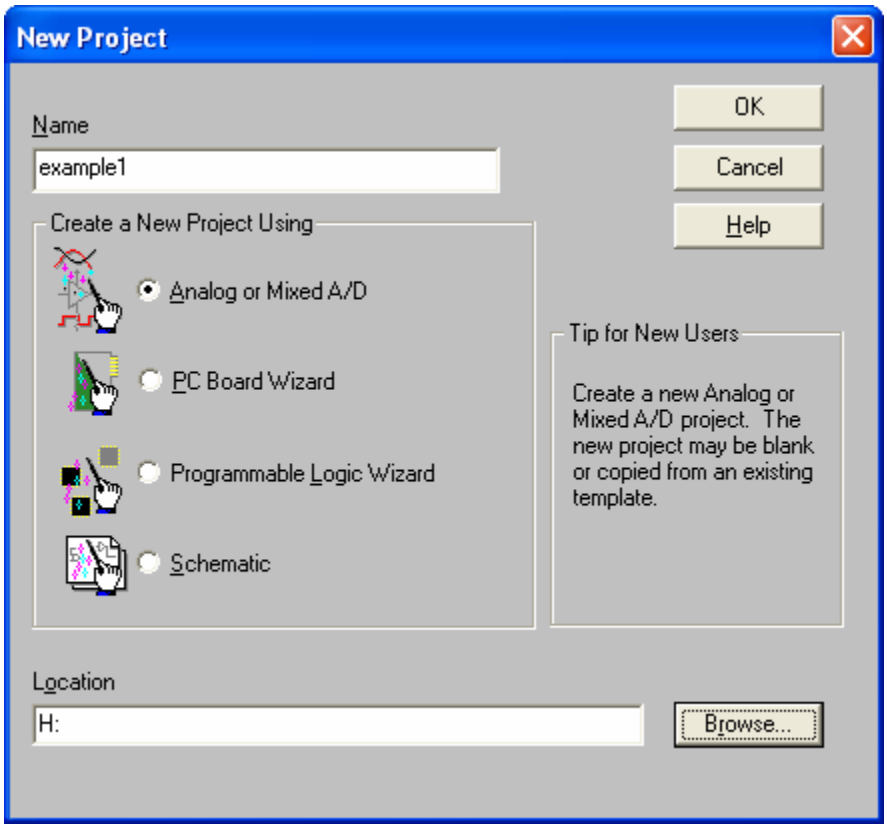
PSpice with Orcad 10

1. Creating Circuits Using PSpice Tutorial
2. AC Analysis
3. Dependent Sources
4. Variable Phase VSin Source

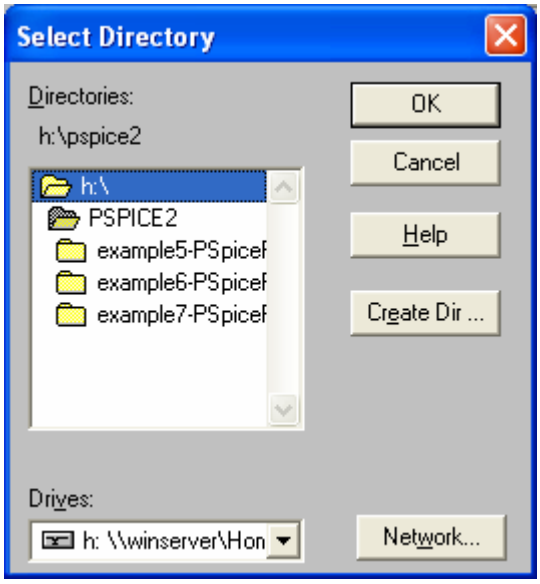
Creating Circuits using PSpice

Start → Orcad 10.5 → Capture CIS Demo

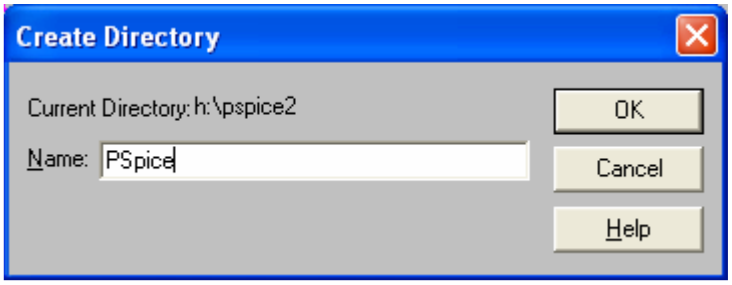
File → New → Project



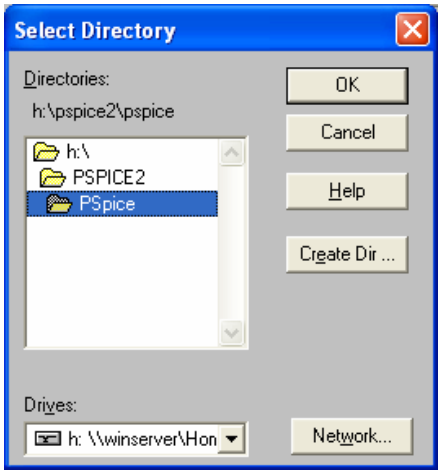
Type in example1in the **Name** field.
Select **Analog or Mixed A/D.**
Click the **Browse…** button



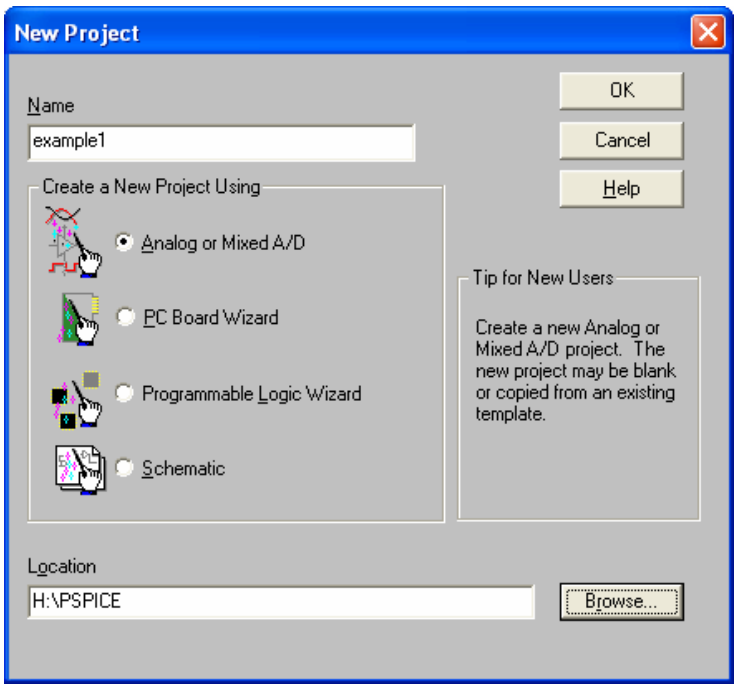
Select the **h:** directory and click on **Create Dir…**



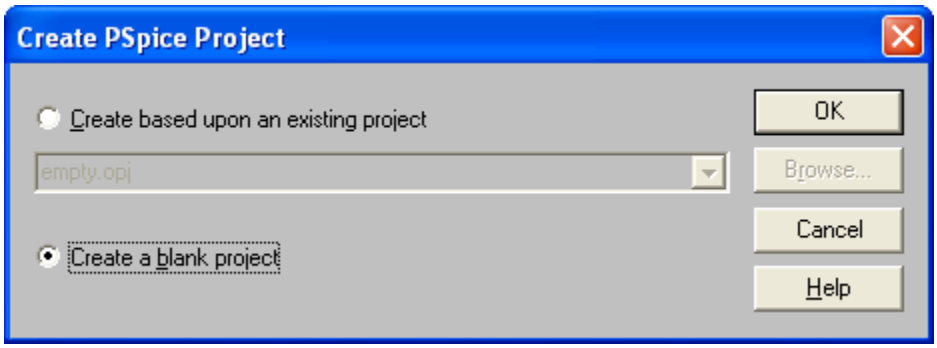
Enter PSpice in the **Name:** field. Click **OK**.



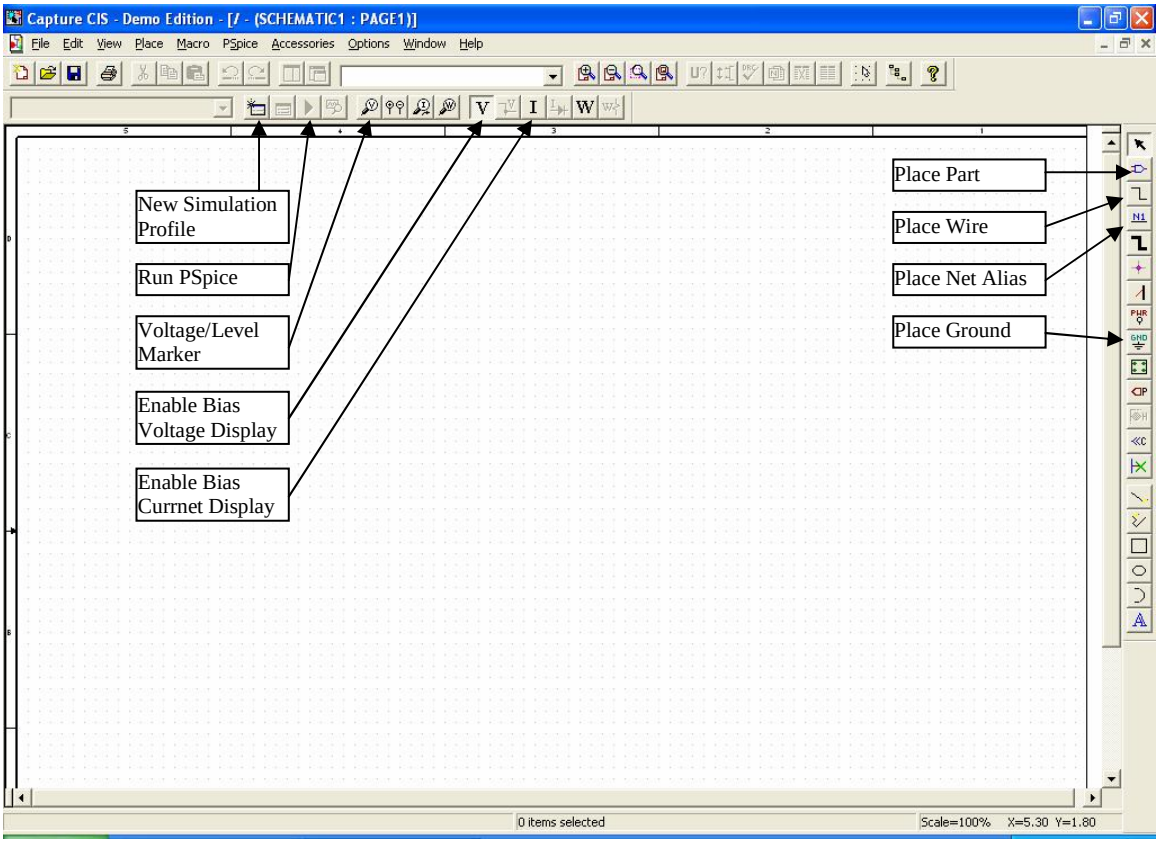
Double click on **PSpice** to select it. Click OK. That will take you back to the original **New Project** dialog box shown below.



Click **OK** to create the new project.

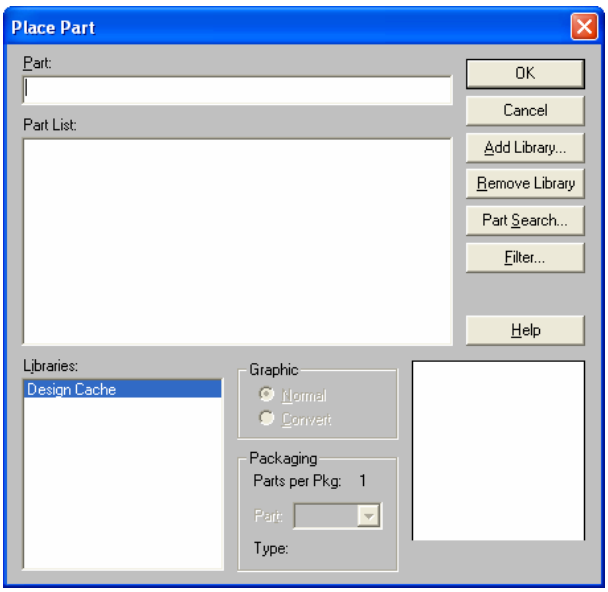


Select **Create a blank project**. Click OK. This will open the schematic editor.

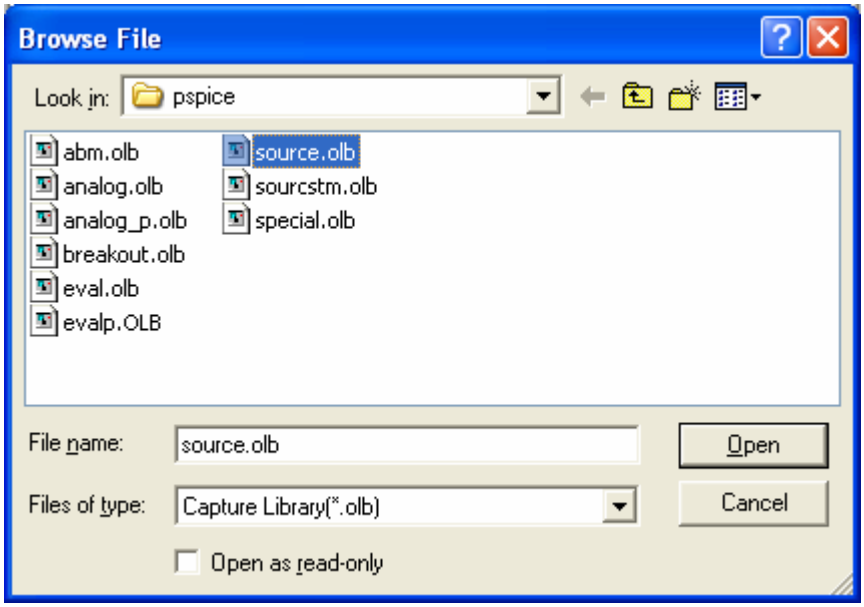


Schematic Editor with all the buttons used in this tutorial labeled.

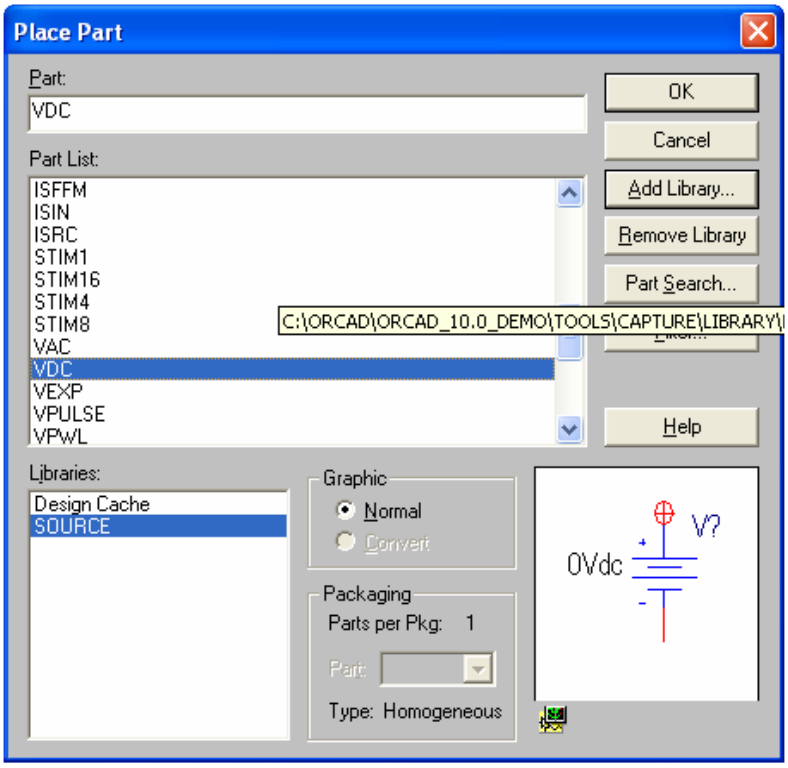
After the schematic editor opens, select the Place Part tool by clicking on the second button on the vertical toolbar on the right side shown above.



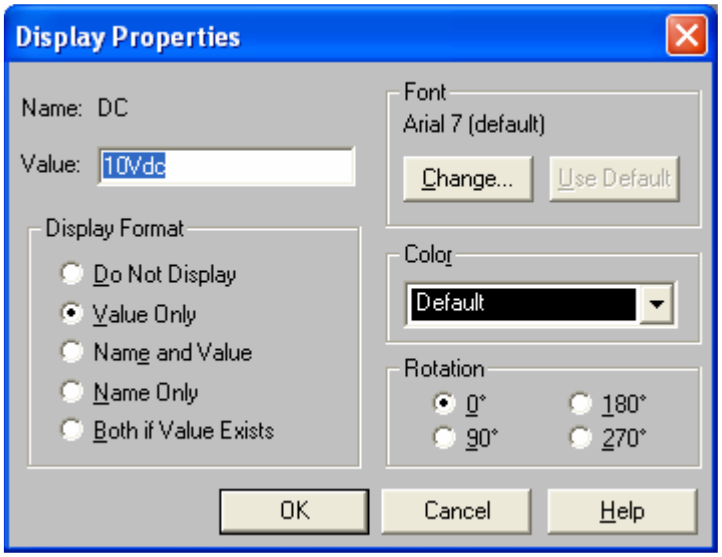
Select **Add Library…**



On the screen select **Source.olb** and click **Open**.

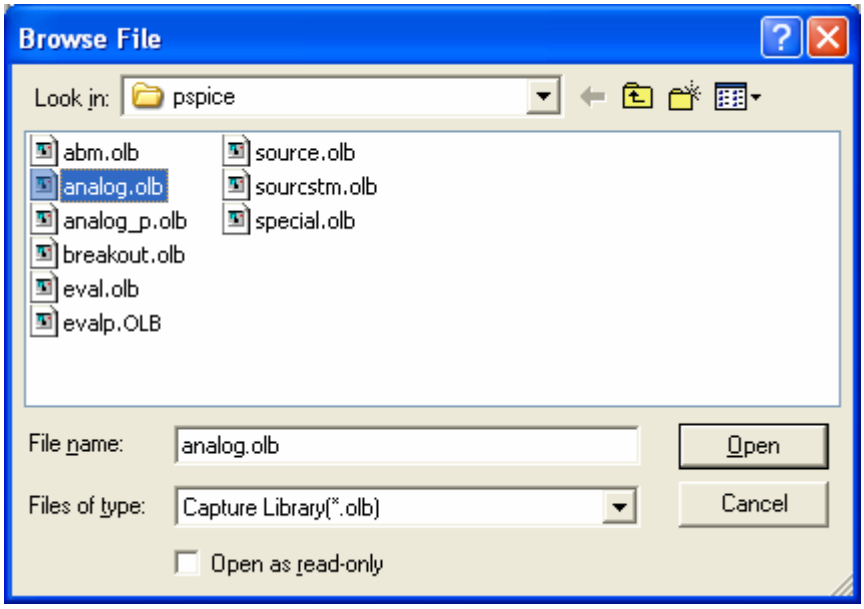


On this screen select a battery (VDC) then **OK**. Place the battery on the work area by clicking on the blank schematic page and click ESC to stop placing DC sources. Then double click on 0VDC to change the DC voltage. The following dialog box will be displayed:

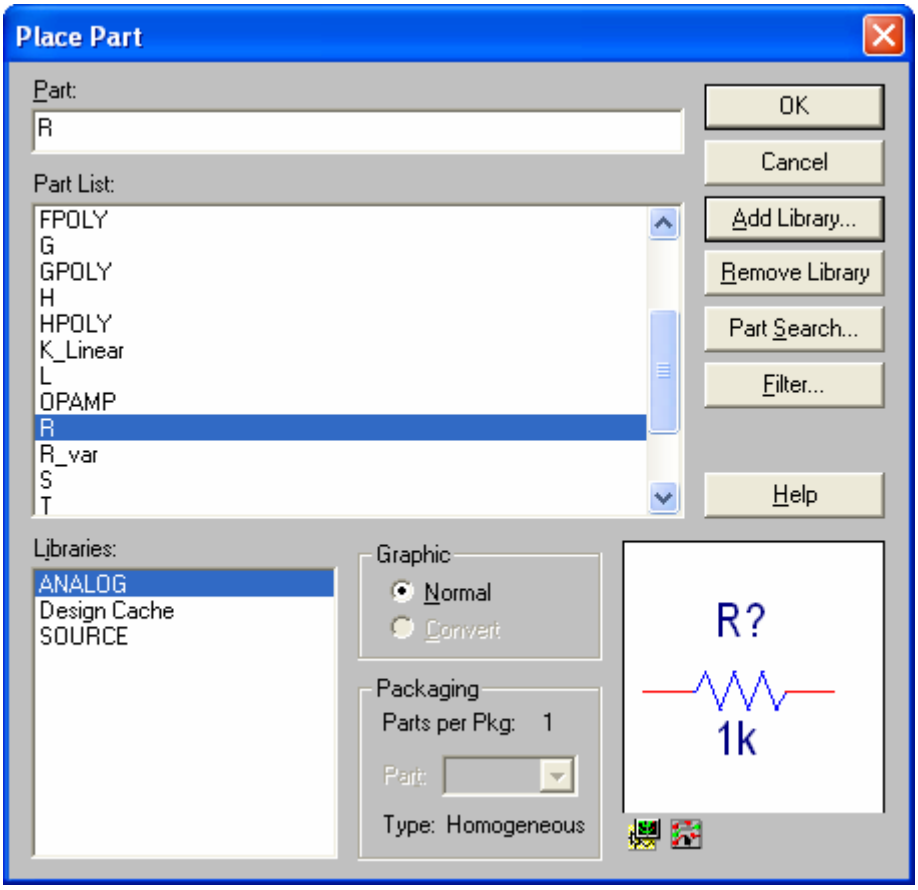


Change value to 10VDC and then click **OK**.

Click on the **Place Part** button, then the **Add Library…** button.



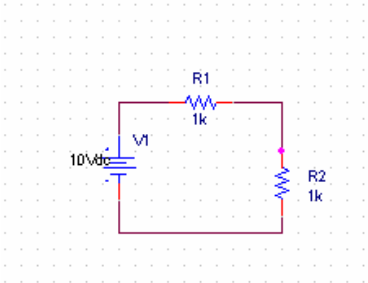
Select **analog.olb** and press **Open**.



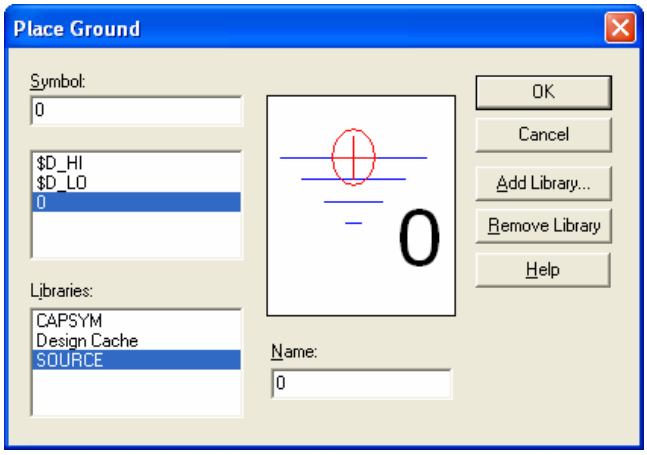
Select the **ANALOG** library and select **R** from the **Part List**. Click **OK**.

Place two resistors in the work space. To stop placing parts, press ESC or right click an item and select "End Mode". To rotate an part, select the part and press 'r' or right click and select rotate from the drop down menu. If you need to delete a part, select the part and press the Delete key.

To connect the items in the circuit, select the **Place Wire** tool by clicking on the third button from the top on the vertical toolbar on the right side. Drag the mouse between the terminals of your placed parts to connect them.

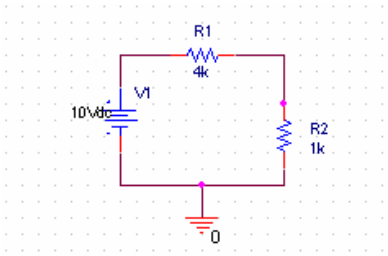


In order for PSpice to simulate your circuit, it must have a "zero" node for a ground. To add this ground, select the **Place Ground** tool by clicking on the 9th button from the top on the vertical toolbar on the right side. The following dialog box will appear.

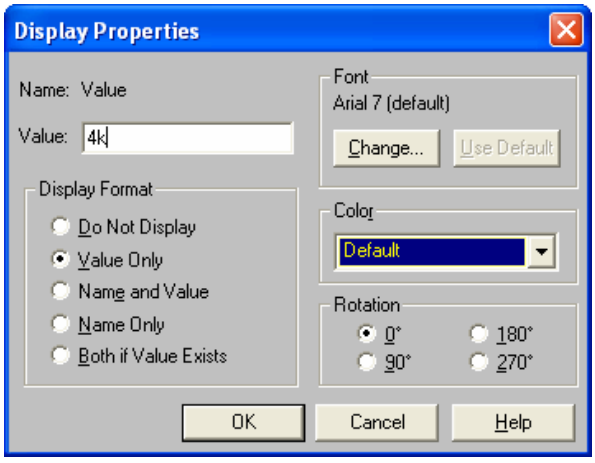


Select the **SOURCE** Library and the **0** Symbol followed by **OK**.

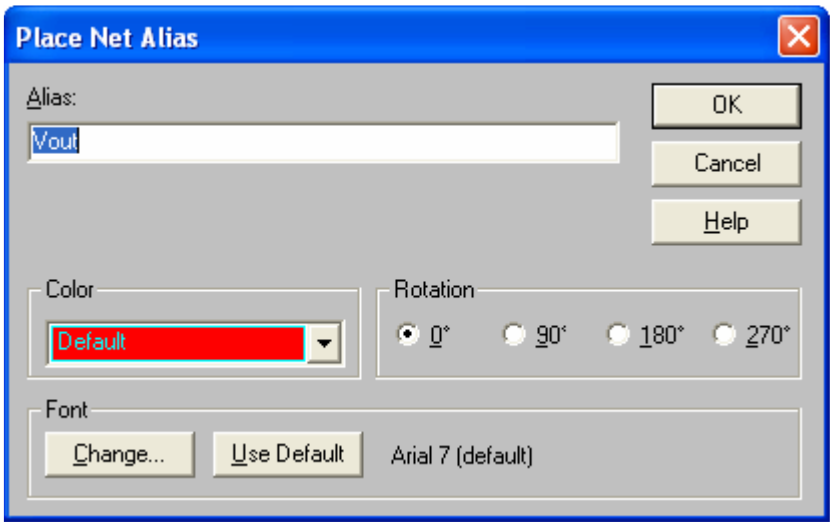
Place the ground for your circuit by clicking on the schematic page and connect it with the **Place Wire** tool. Your diagram should look like this:



To change the value for an item, double click the value you want to change and enter the value you want on the dialog box that appears. Double click on the **1k** value of the horizontally placed resistor and change the **Value** to 4k as shown below.

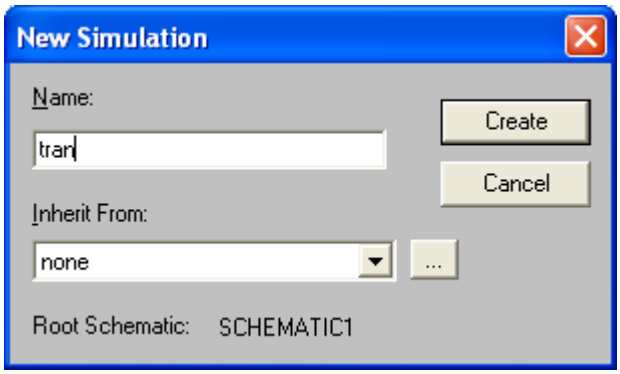


It is important to name the nodes you want to plot in PSpice so you can find them easily. To name a node, select the **Place Net Alias** tool and the following dialog box appears. Change the **Alias** to Vout and click **OK**.

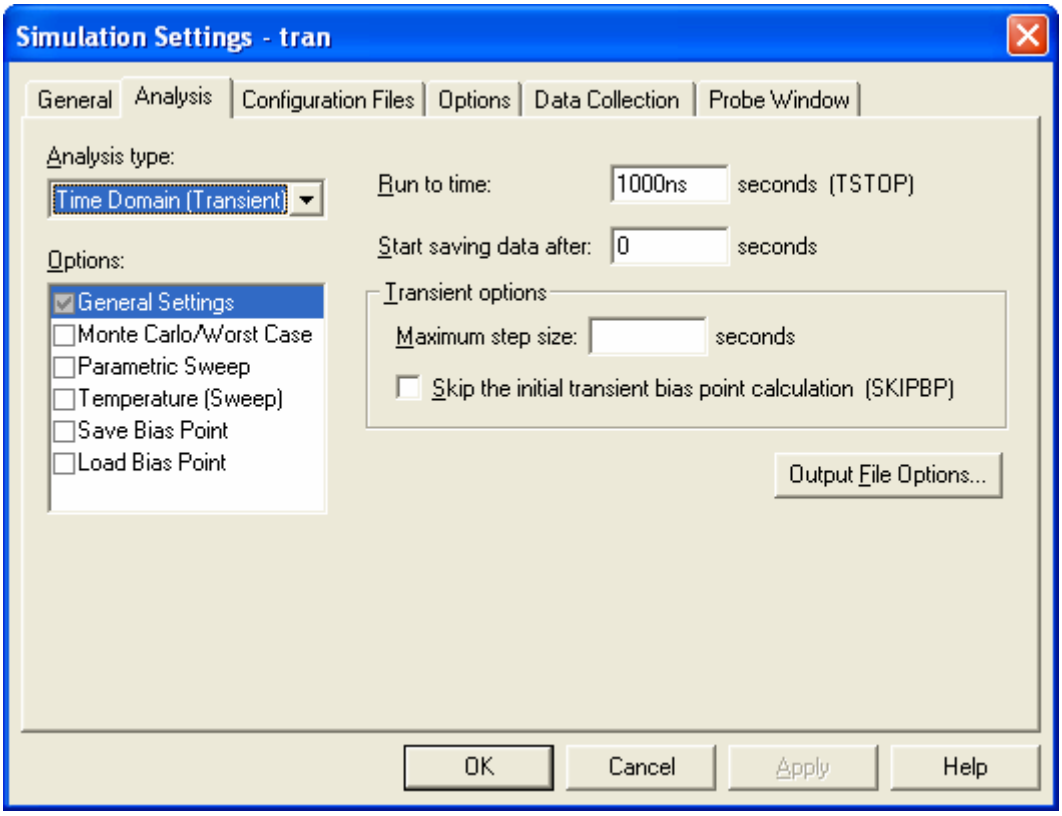


Then place the alias on the desired node for Vout (i.e. the junction of the 2 resistors).

Next, configure the simulation by clicking the **New Simulation Profile** button on the top toolbar. Enter the **Name** tran as shown below.



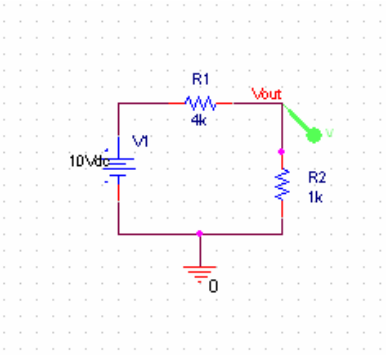
Click **Create** and the following dialog box will appear.



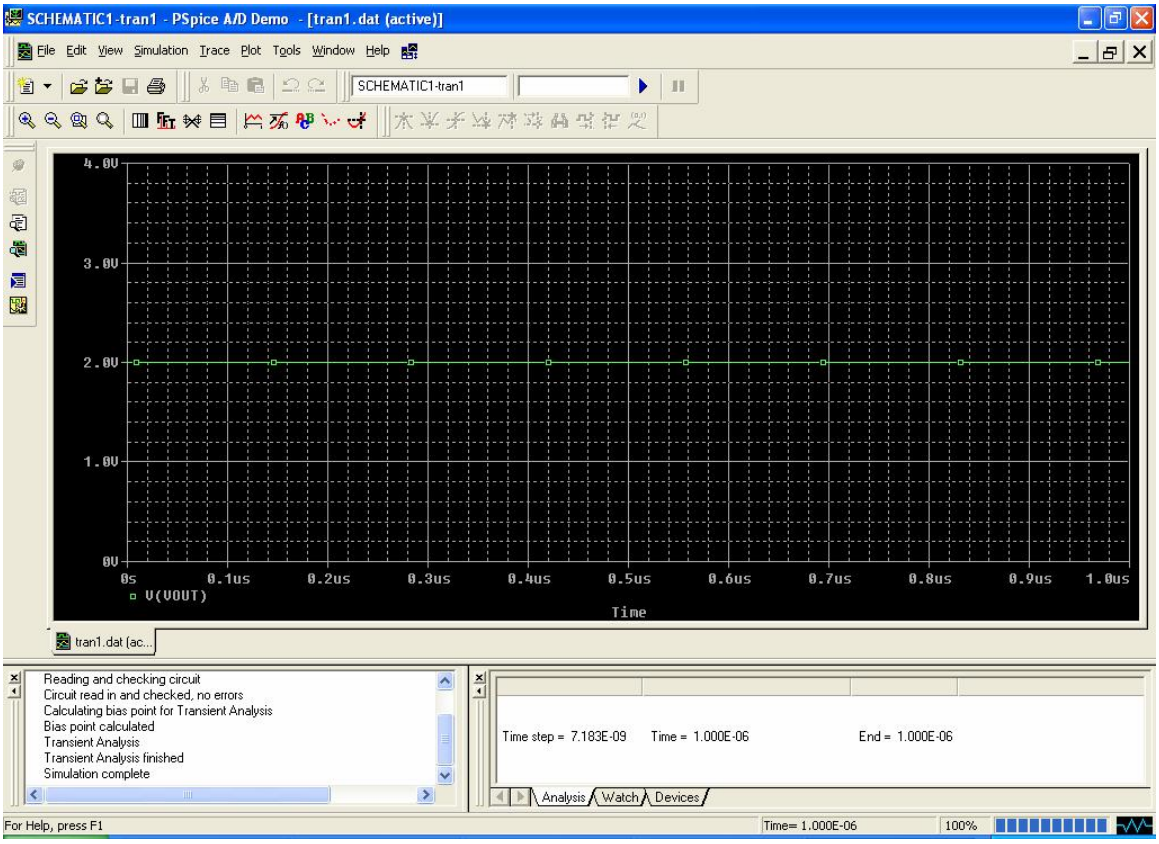
Click **OK** to accept the default settings of the **Time Domain (transient) analysis.**

From the top menu select
PSpice → Create Netlist . This is only necessary so that you can add the **Voltage Level Marker.**

Click on the **Voltage/Level Marker** button to add a marker to the Vout node by clicking on it as shown below.

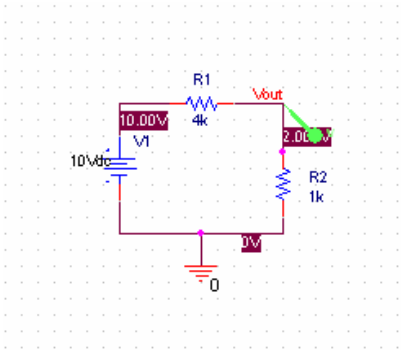


Click the **Run PSpice** button and the PSpice Analysis results will appear as shown below.



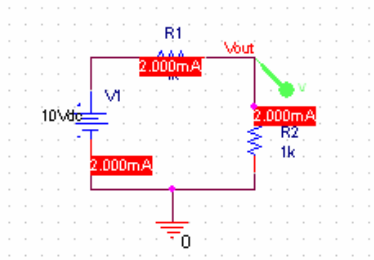
This automatically plots nodes with Voltage (or Current) markers on them. In this case, the voltage divider gives you 2V for Vout.

Close the PSpice window. On the schematic, click the **Enable Bias Voltage Display** button to see all the DC voltages in the circuit as shown below.



Circuit with Bias Voltage Display Enabled

On the schematic, click the **Enable Bias Voltage Display** button to toggle it off. Then, click on the **Enable Bias Current Display** button to see all the DC current(s) in the circuit as shown below.

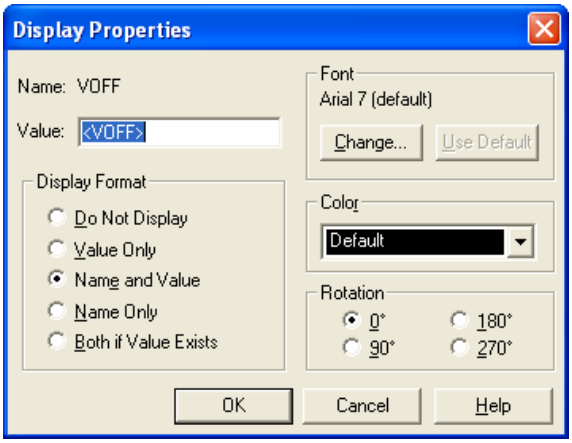


Circuit with Bias Current Display Enabled

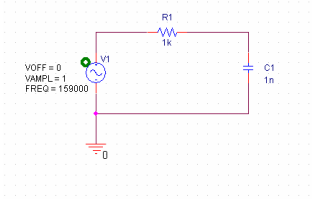
AC Circuit Analysis

Transient Analysis:

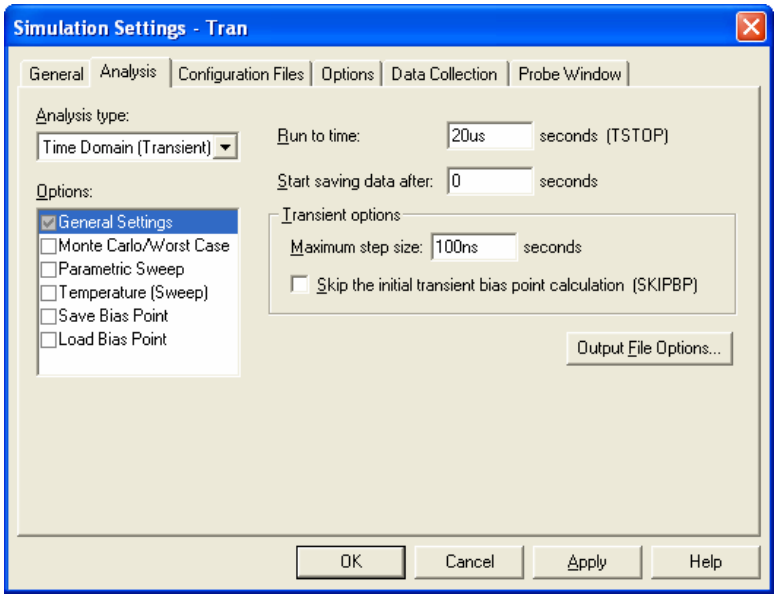
For this circuit, select and wire these components: Capacitor (Place Part C in ANALOG library), Resistor (Place Part R in ANALOG library) , Sinusoidal Source (Place Part VSIN in SOURCE library), and Ground (Place Ground 0 SOURCE library). Double click on the VOFF attribute of the VSIN component. The dialog box shown below should appear.



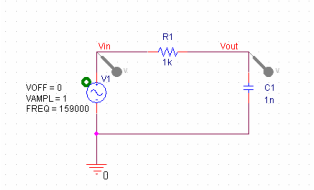
Change the value from <VOFF> to 0. Click OK. Similarly, set the VAMPL attribute to 1 and the FREQ attribute to 159000. (1/(2*PI*R*C)). This circuit is shown below:



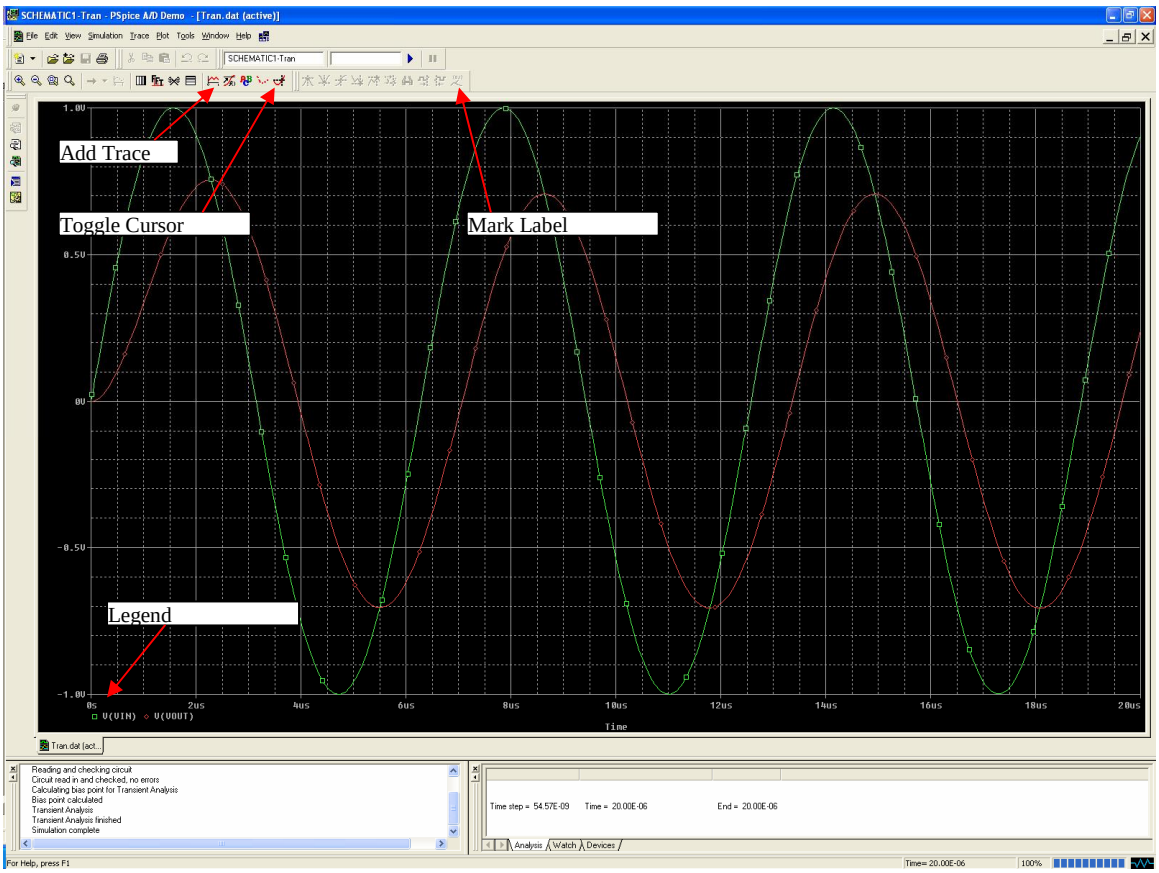
Next, configure the simulation by clicking the **New Simulation Profile** button on the top toolbar. Enter the **Name** Tran and the following dialog box will appear. Set Analysis type: **Time Domain (transient)**, **Run To Time:** 20us, and **Maximum Step Size** to 100ns. Click **OK**.



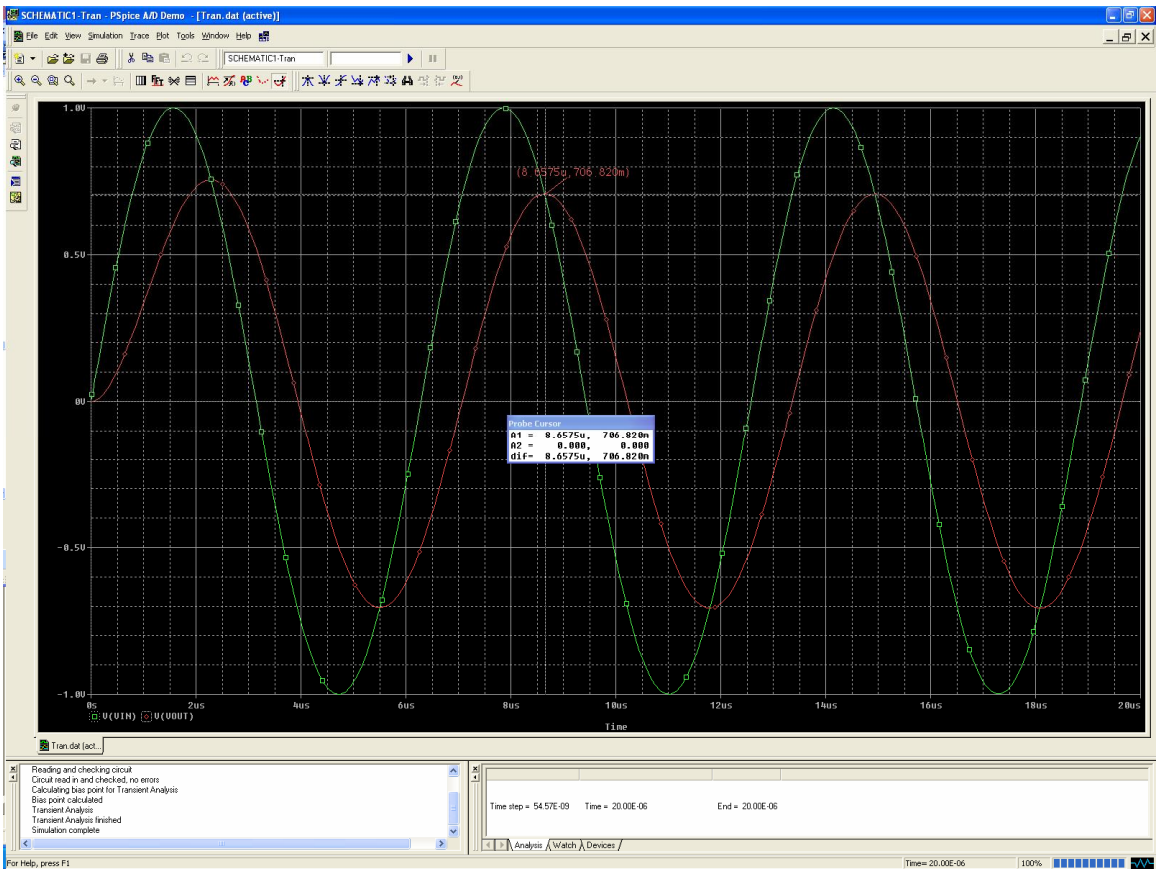
Click on the **Voltage/Level Marker** button and place a marker at the junction of the R and C and at the junction between the **Source** and the **R**. If you are unable to place the marker, you need to create the netlist first (**PSpice → Create Netlist)**. Give the junction between the **R** and **C** the Alias of **Vout** using the **Net Alias** tool as shown below. Give the junction between the **Source** and the **R** the alias **Vin**. This circuit is shown below:



Click the **Run PSpice** button and the PSpice Analysis results will appear as shown below.

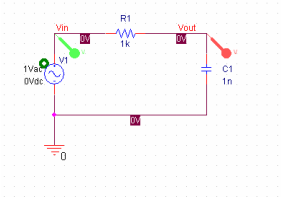


Click the **Toggle Cursor** button to display the cursors and the **Probe Cursor** window. Left mouse click on the red dot in the legend next to **V(Vin)**. This will assign the left mouse button to the Vout trace. Drag the mouse using the left mouse button to 2nd peak of Vout and note the amplitude. It should be 3dB smaller than Vin (0.707V). Click on the Mark Label button to label that point as shown below.

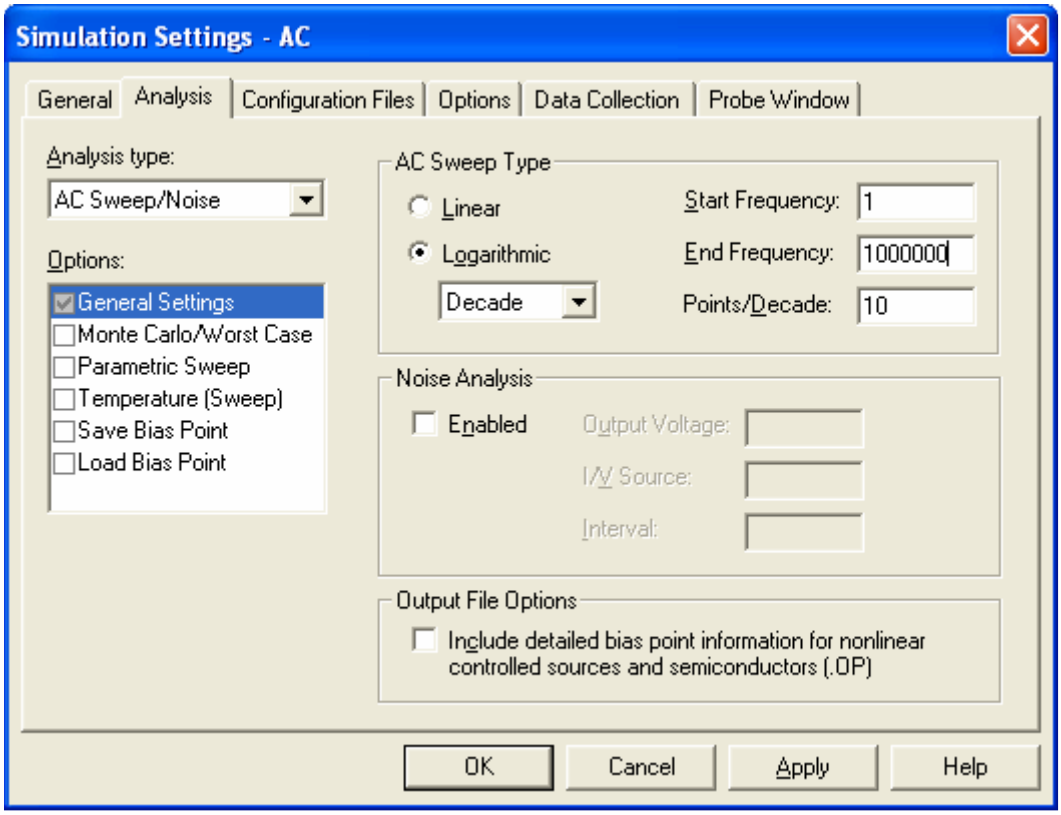

SCHEMATIC1-Tran - PSpice A/D Demo - [Tran.dat (active)]
File Edit View Simulation Trace Plot Tools Window Help
SCHEMATIC1-Tran
1.0V
0.5V
0V
-0.5V
-1.0V
(8.6575u, 706.820m)
Probe Cursor
A1 = 8.6575u, 706.820m
A2 = 0.000, 0.000
dif= 8.6575u, 706.820m
0s 2us 4us 6us 8us 10us 12us 14us 16us 18us 20us
V(VIN) V(VOUT)
Time
Tran.dat (act...
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point for Transient Analysis
Bias point calculated
Transient Analysis
Transient Analysis finished
Simulation complete
Time step = 54.57E-09 Time = 20.00E-06 End = 20.00E-06
Analysis Watch Devices
For Help, press F1
Time= 20.00E-06
100%

AC Analysis:

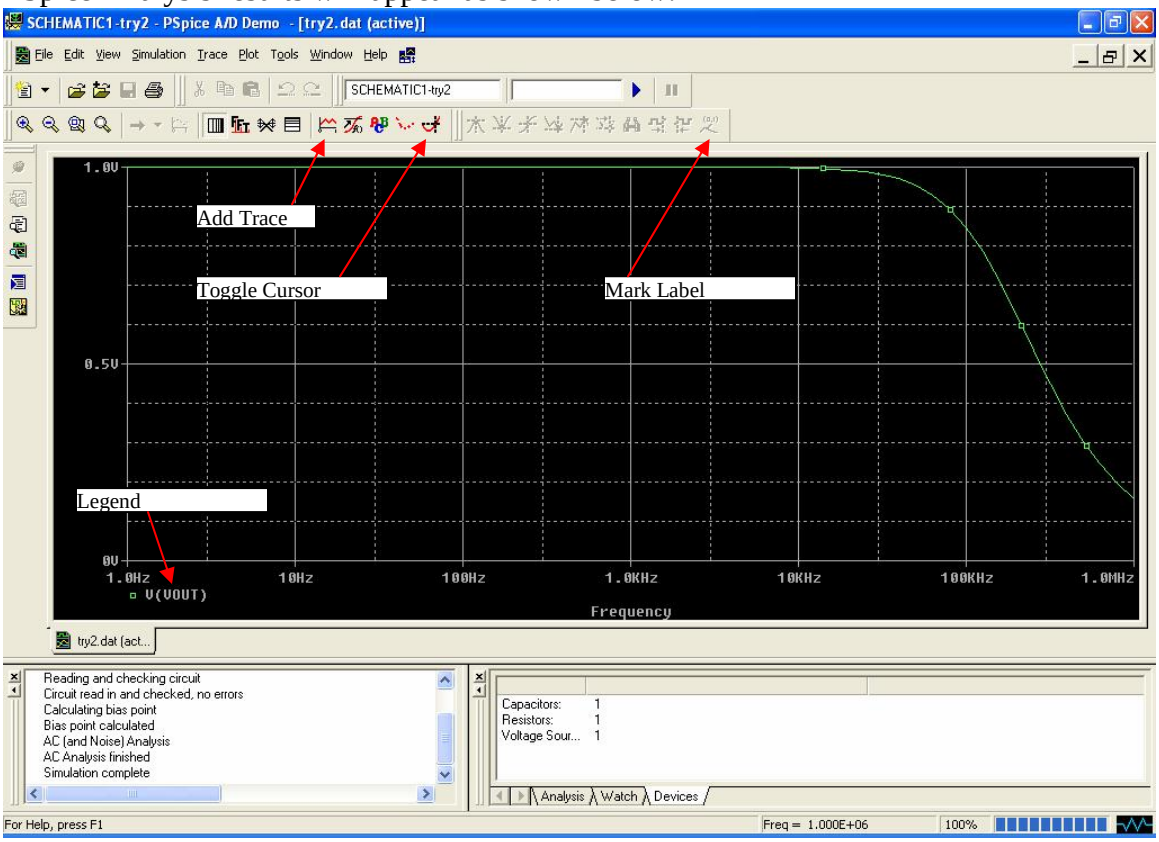
Close the PSpice simulation window. Then, modify the above circuit by deleting the **VSIN** component. Replace this component with the AC source (Place Part **VAC** in **SOURCE** library). The default attributes are correct. This circuit is shown below:



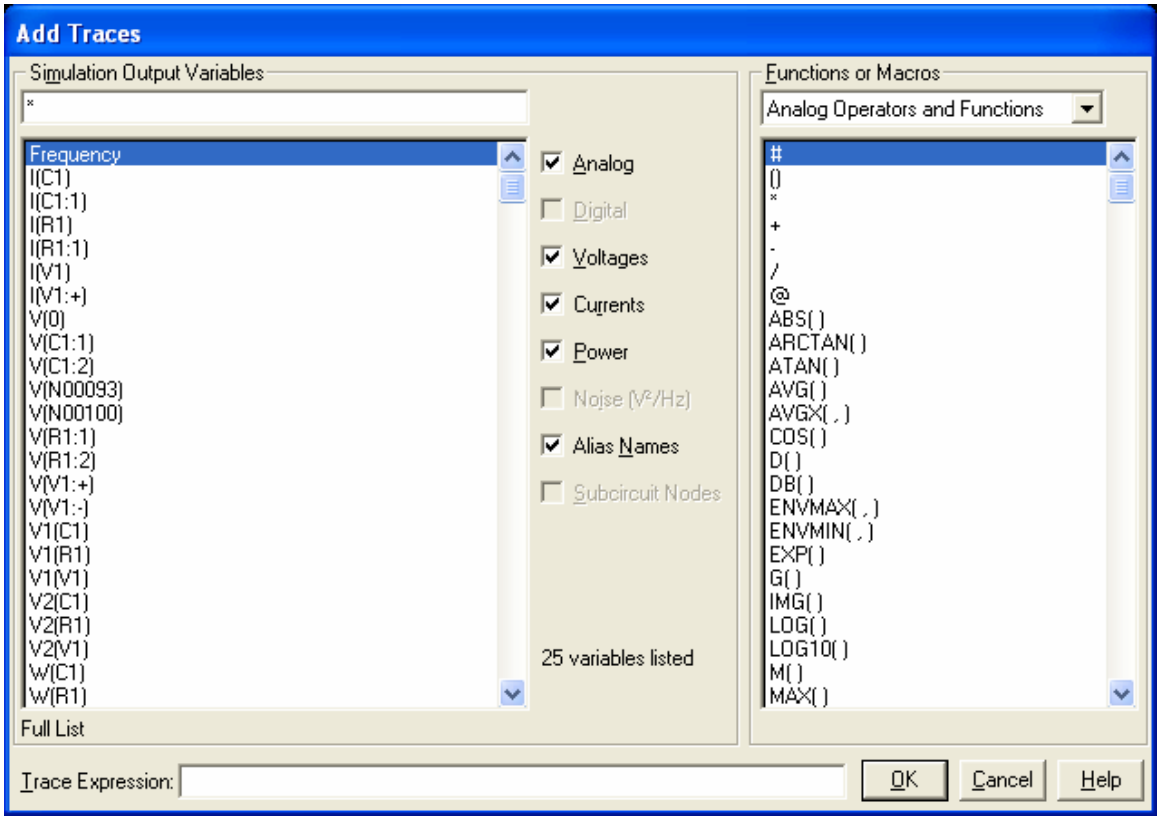
Next, configure the simulation by clicking the **New Simulation Profile** button on the top toolbar. Enter the **Name** AC and the following dialog box will appear. Select Analysis type: **AC Sweep/Noise**, AC Sweep Type: **Logarithmic** and enter the displayed values for **Start Frequency**, **End Frequency** and **Points/Decade**. Click **OK**.



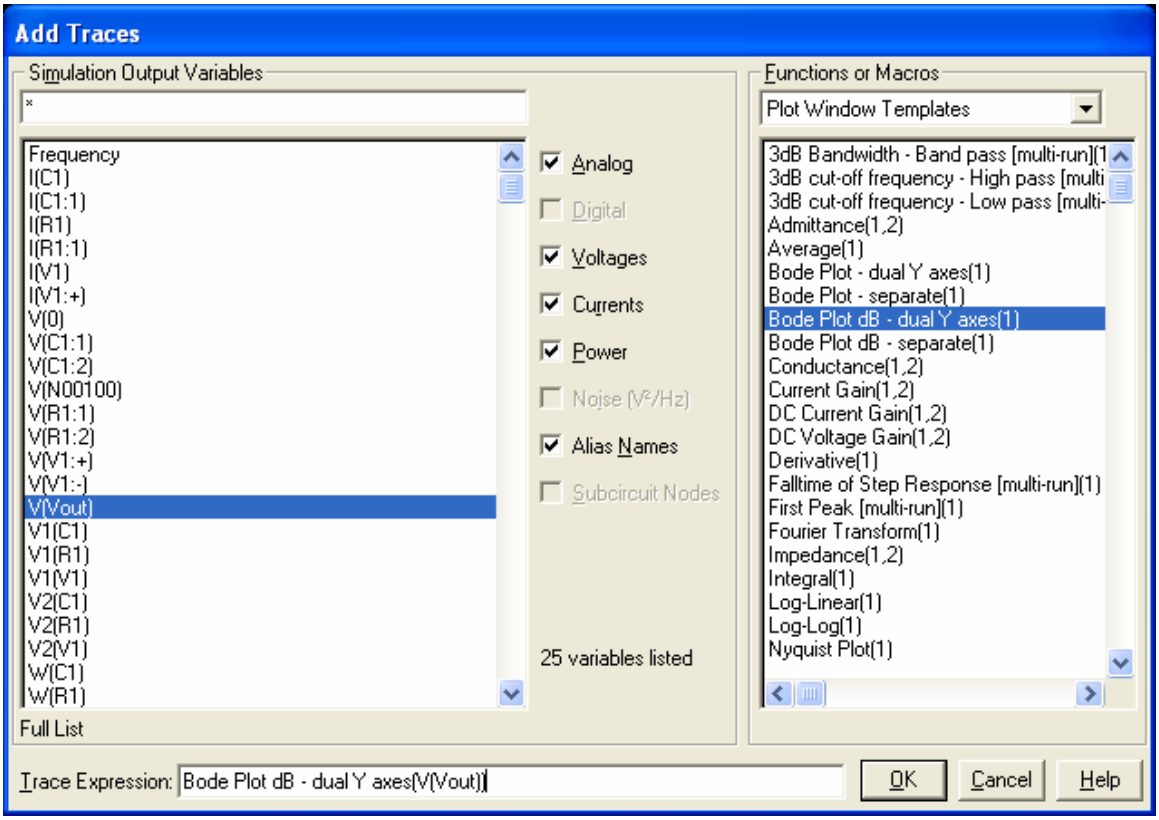
Unfortunately, when you create a new simulation profile, the program deletes the voltage markers. Add a voltage marker the Vout node. Then, click the **Run PSpice** button and the PSpice Analysis results will appear as shown below.



Click on the Add Trace button shown above and the following diaglog box will appear.

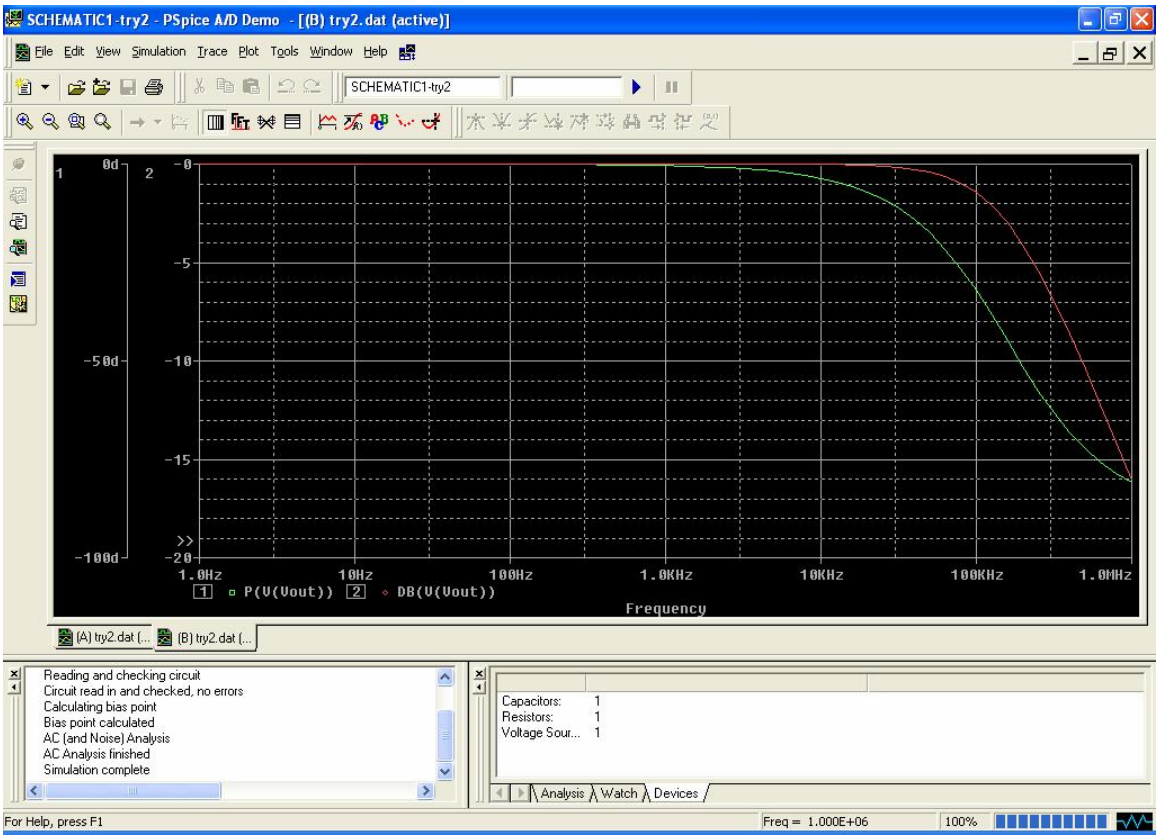


Now select **Plot Window Templates** from the **Functions or Macros** drop down menu and the following dialog box will appear.

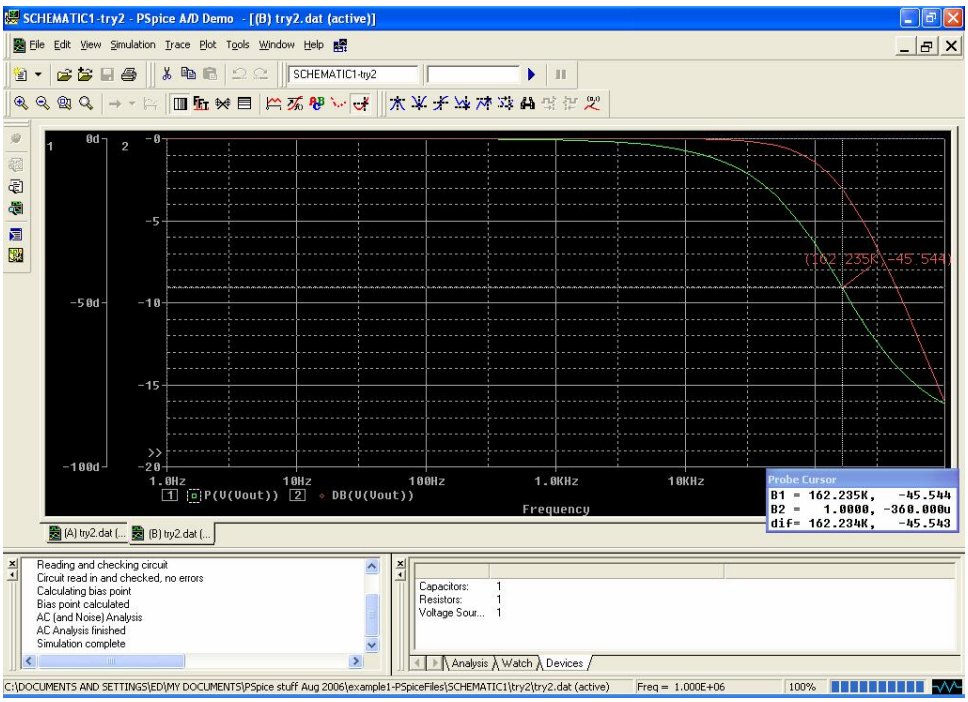


Select Bode Plot dB – dual Y axes(1) on the right side. The cursor in the **Trace Expression** field at the bottom indicates the place to insert the signal name to plot. Select **V(R1:2)** (or **V(C1:2)** which is the same node) from the list of nodes on the left side and click **OK**.

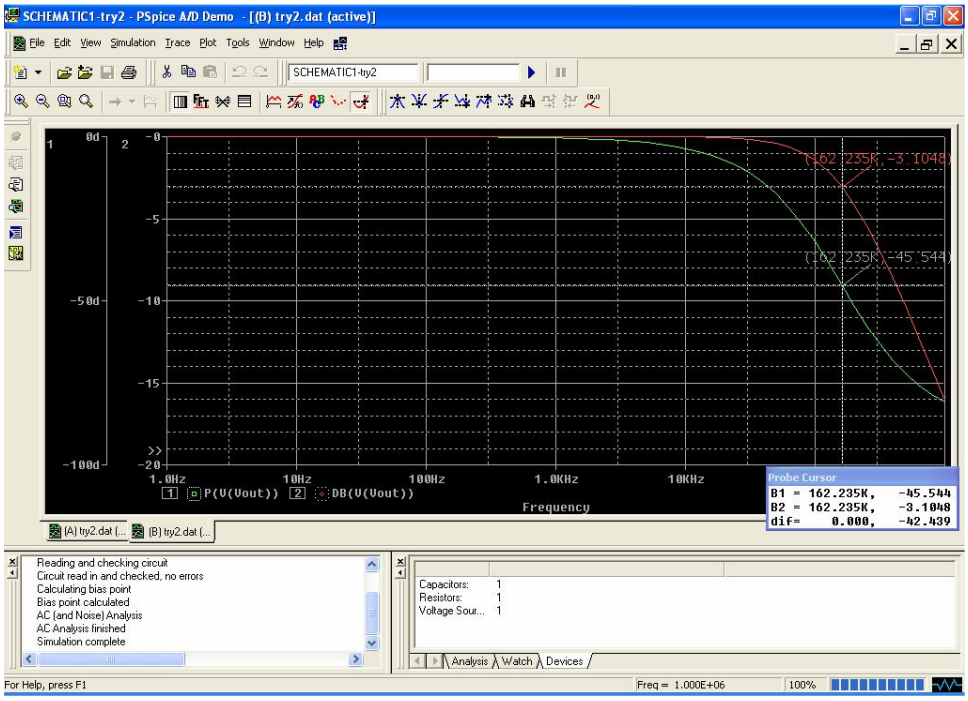
This should be your result. The Phase is indicated by P(V(Vout)) in green (left Y-axis in degrees) and the Magnitude is indicated by DB(V(Vout)) in red (right Y-axis in dB).



Click the Toggle Cursor button to display the cursors and the Probe Cursor window. Left mouse click on the green dot in the legend next to P(V(Vout)). This will assign the left mouse button to the phase trace. Drag the mouse using the left mouse button to find the frequency where the phase is -45 degrees. Click on the Mark Label button to label that point as shown below.

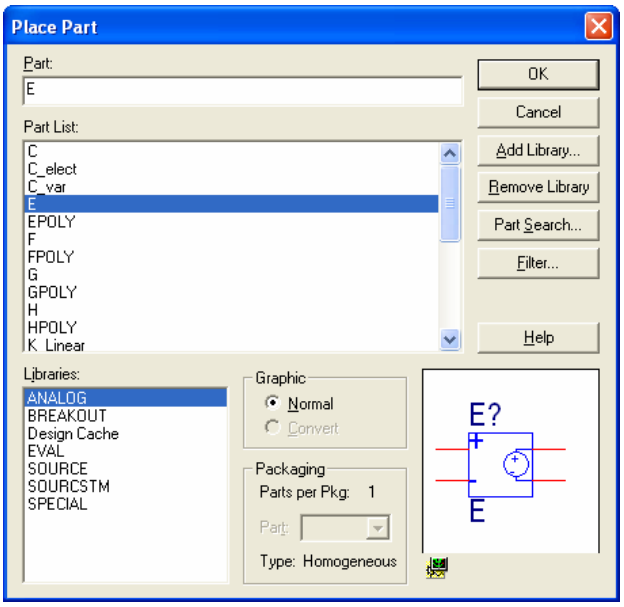


Right mouse click on the red dot in the legend next to DB(V(Vout)). This will assign the right mouse button to the magnitude trace. Drag the mouse using the right mouse button to find the frequency where the magnitude is -3 dB. Click on the Mark Label button to label that point as shown below.

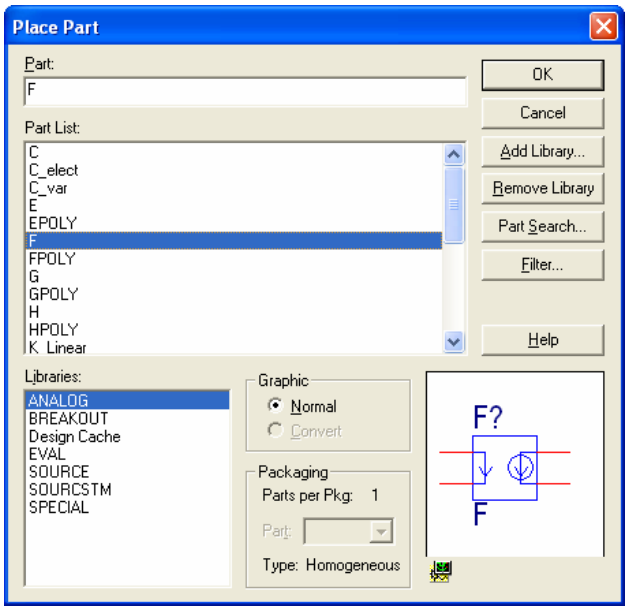


Dependent Sources

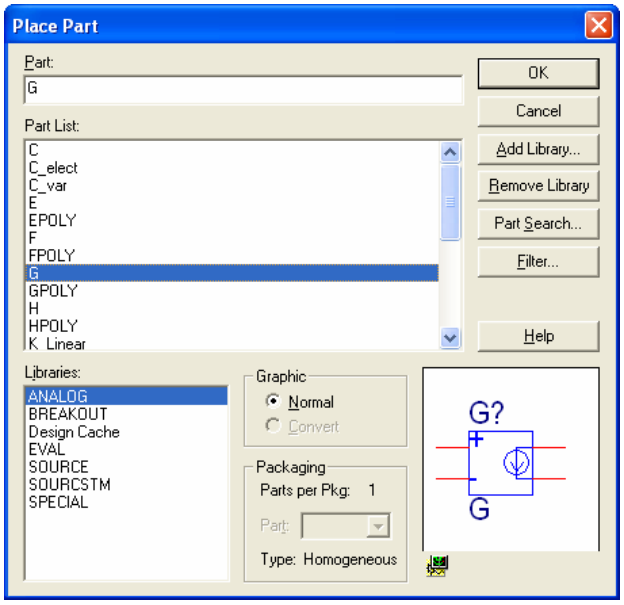
The 4 dependents sources available in the Analog library are shown below:



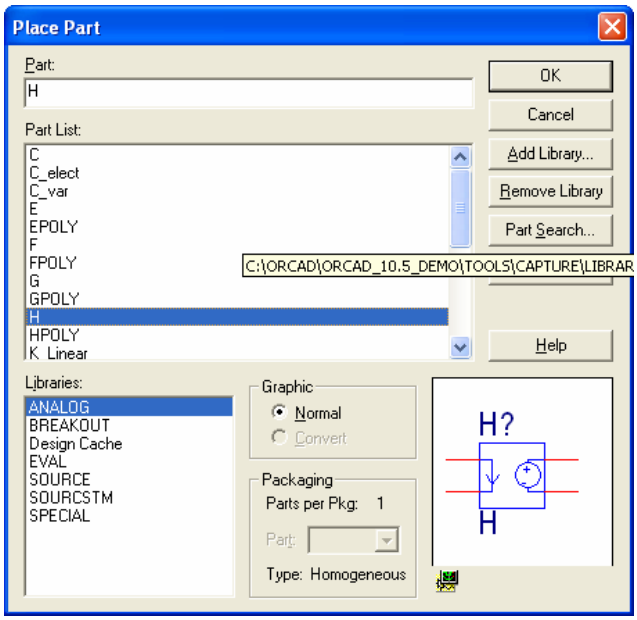
Voltage Controlled Voltage Source



Current Controlled Current Source

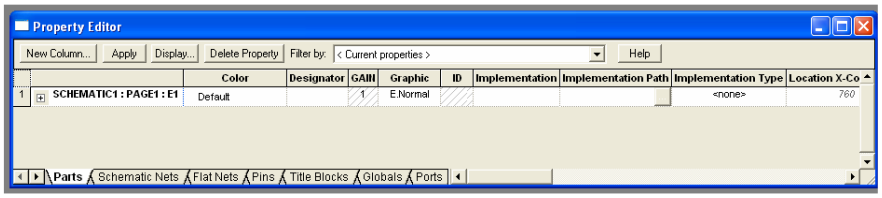


Voltage Controlled Current Source

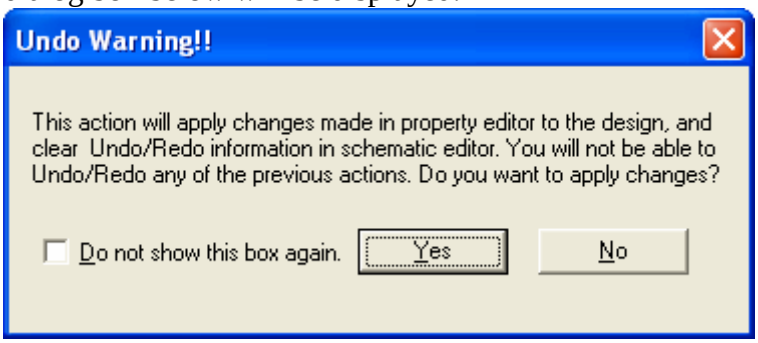


Current Controlled Voltage Source

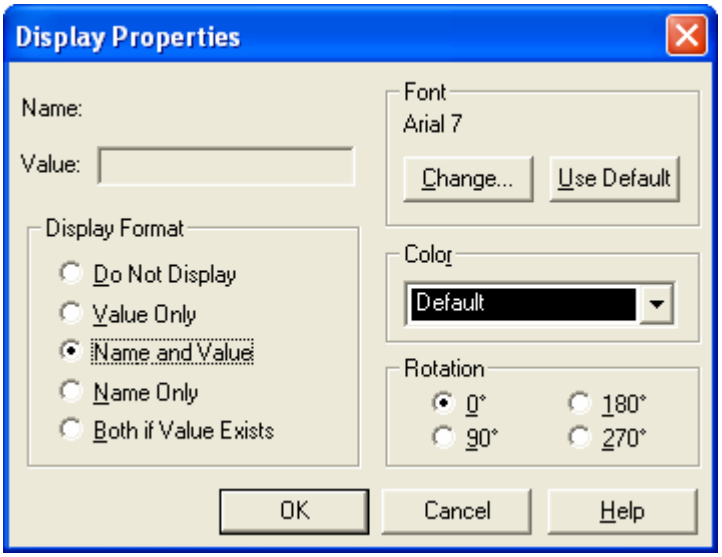
After placing the part, the Gain needs to be set. Double click on the part to bring up the Property Editor shown below:



Click on the 1 in the **Gain** field and change it to the appropriate value. Then click on **Display…** The dialog box below will be displayed.



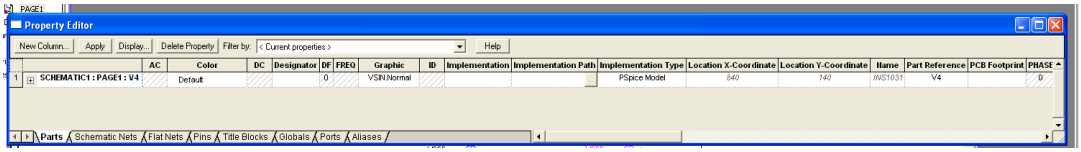
Click **Yes**. The dialog box below will be displayed:



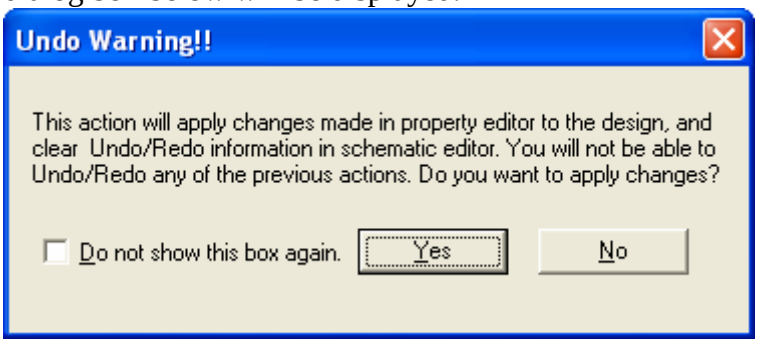
Select **Name and Value** and press **OK**. Click the X in the upper right hand corner of the Property Editor to close it. You can select the Phase property on the schematic and move it to a more convenient spot.

Variable Phase VSin Source

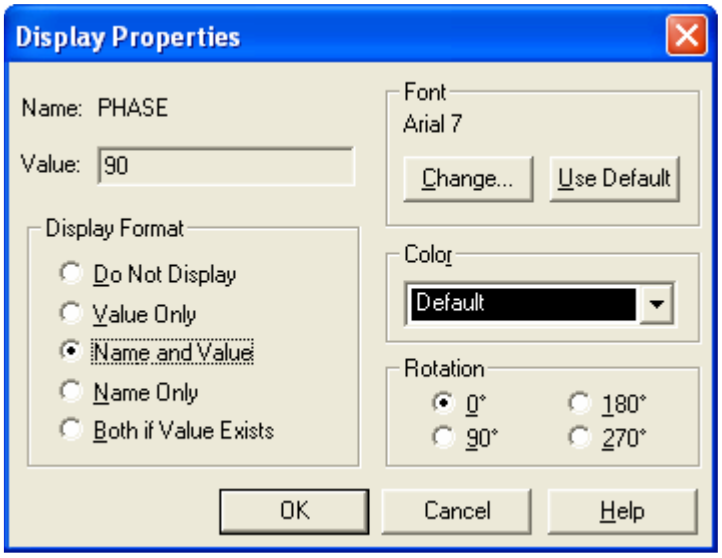
The Phase isn't displayed as a property on the VSin component in the Source library. However, you can set it and display it. To do this, place the VSin component on your schematic. Double click on it to bring up the Property Editor as shown below:



Click on the 0 in the **Phase** field and change it to the appropriate value. Then click on **Display…** The dialog box below will be displayed.



Click **Yes**. The dialog box below will be displayed:



Select **Name and Value** and press **OK**. Click the X in the upper right hand corner of the Property Editor to close it. You can select the Gain property on the schematic and move it to a more convenient spot.