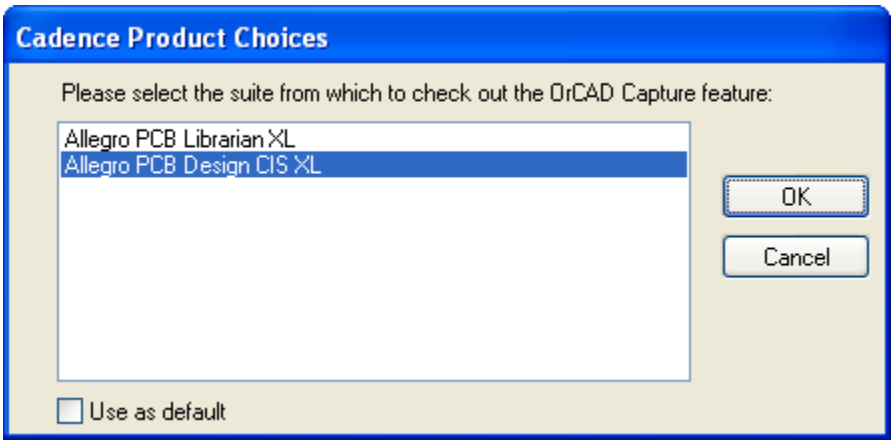
PSpice with Cadence

1. Creating Circuits
2. AC Analysis
3. Step Response
4. Dependent Sources
5. Variable Phase VSin Source

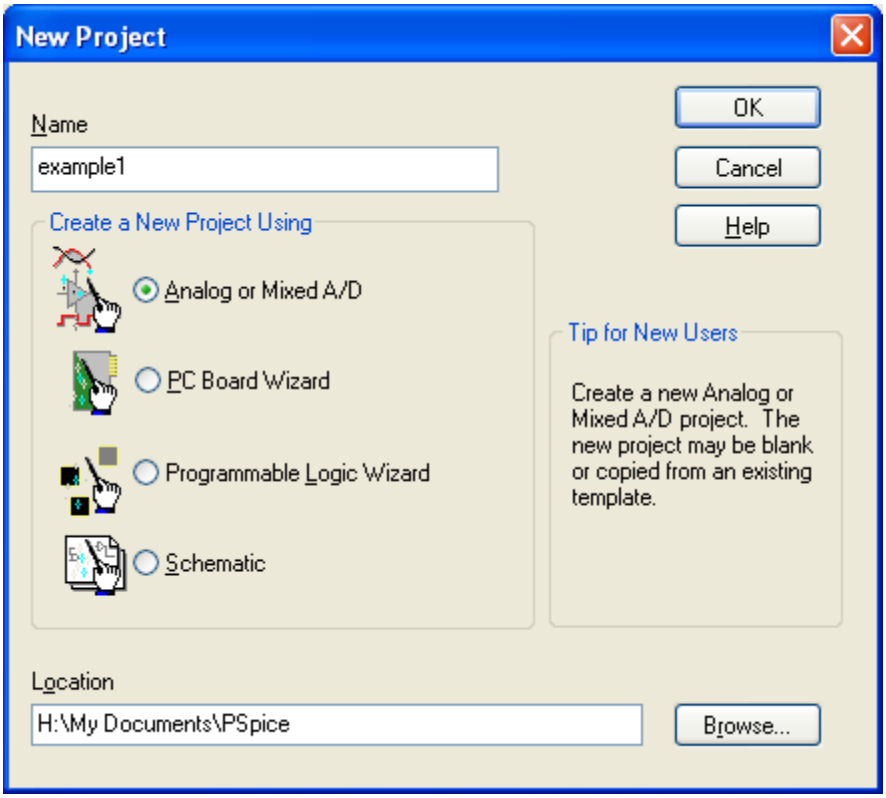
Creating Circuits

Select '**Start → Engineering → Cadence → Capture**' from the start menu.

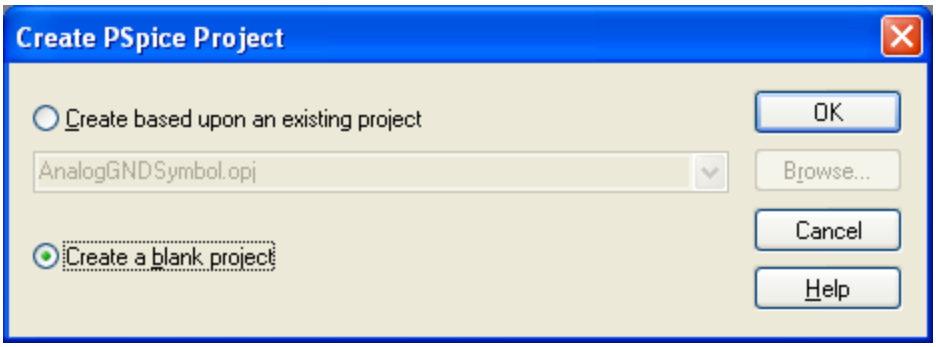


When this dialog box appears, select **Allegro PCB Design CIS XL**

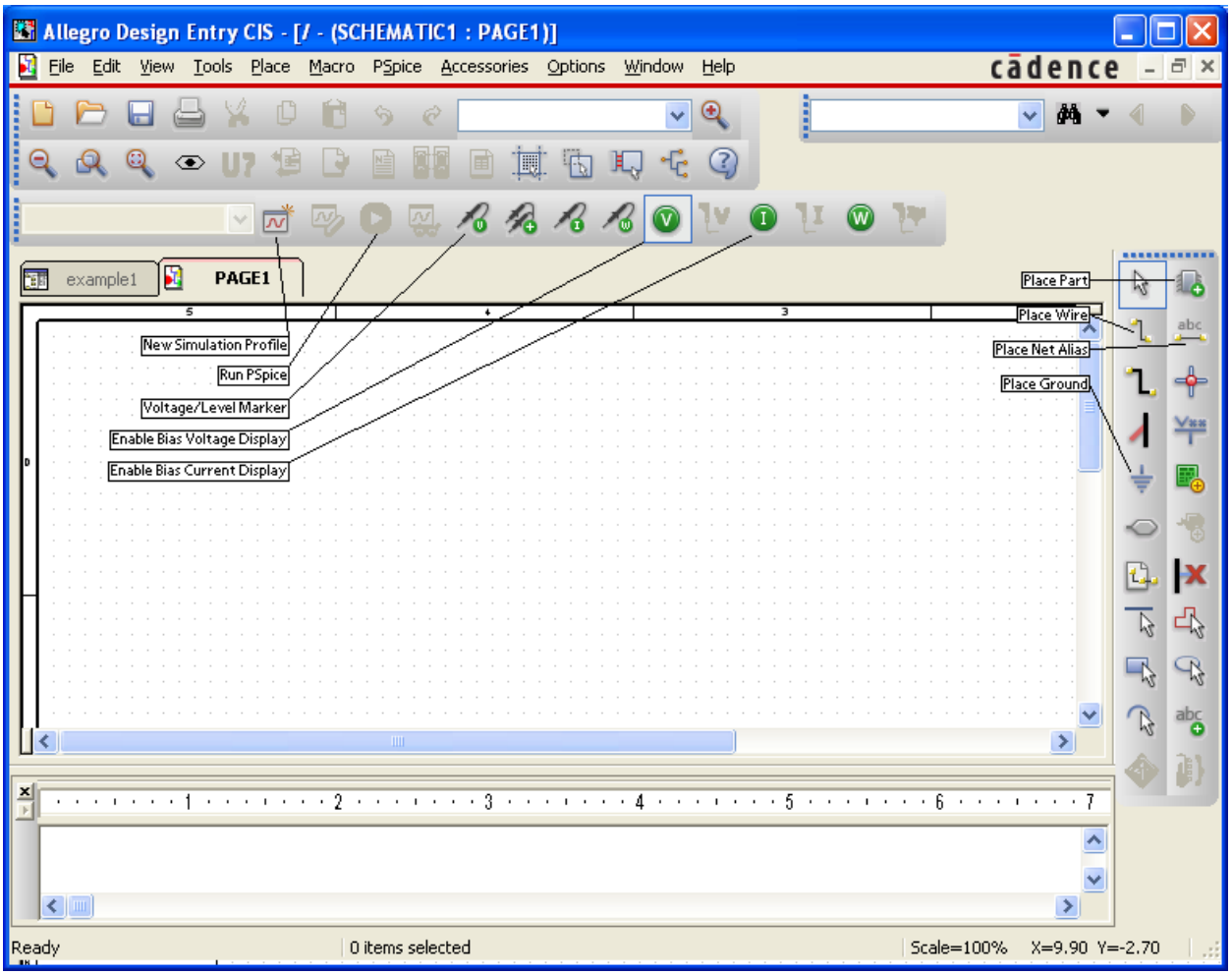
Select '**File → New → Project**' in the menu bar.



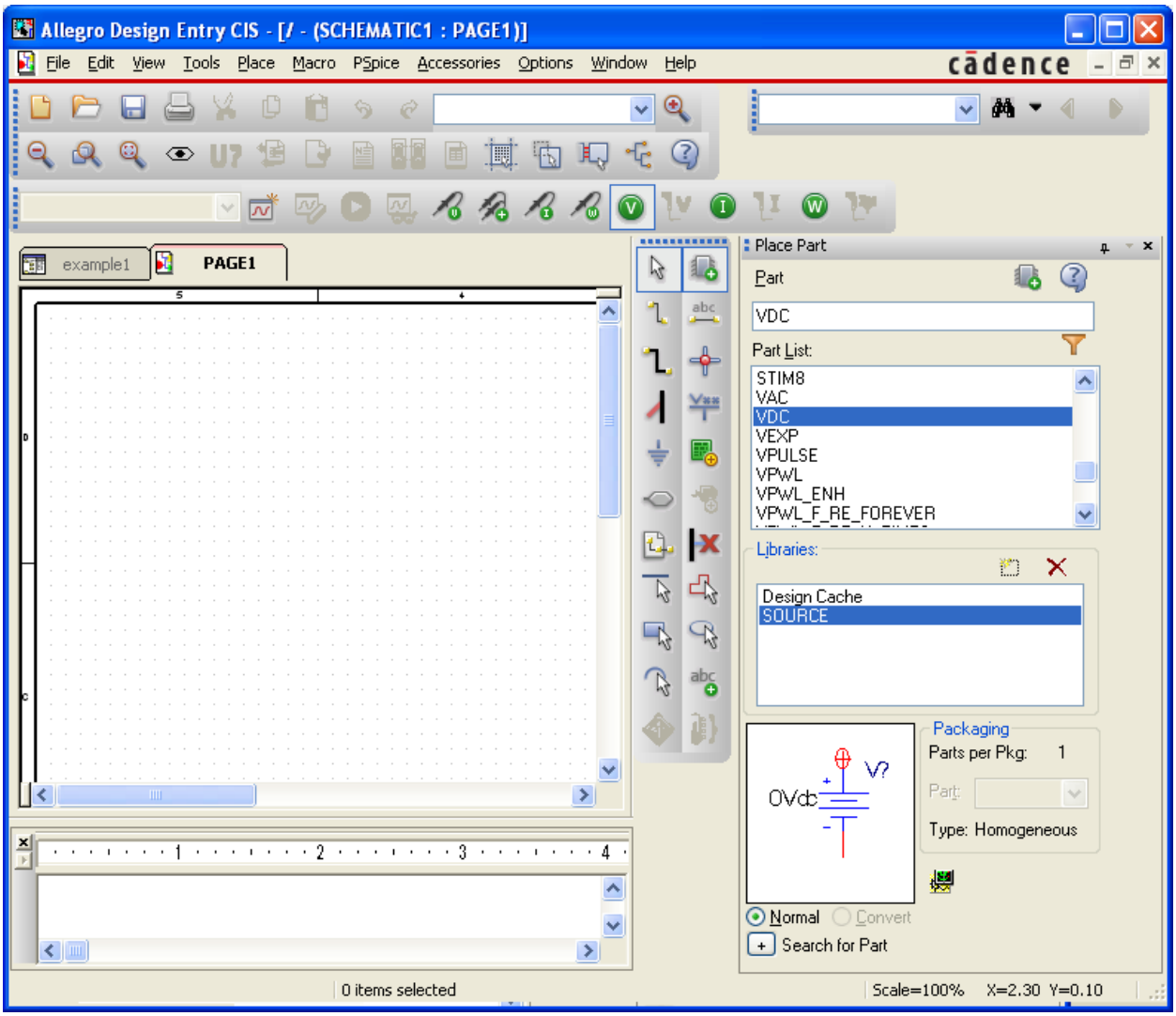
Type **example1** in the **Name** field, select the **Analog or Mixed A/D** project type, set the location to **H:\My Documents\PSpice**, and click **Ok**.



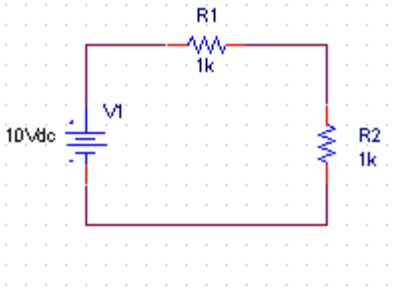
Select **Create a blank project** and click **Ok**. The **Schematic Editor** will open



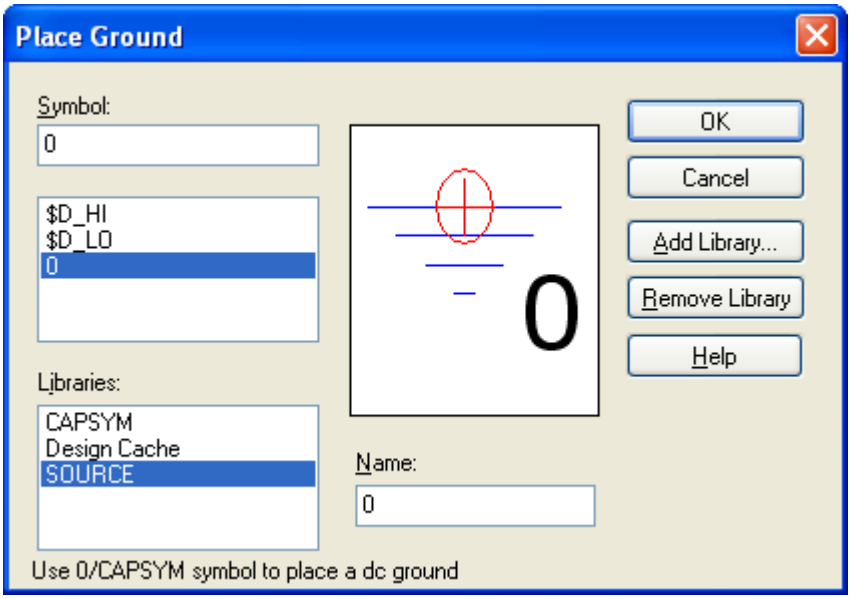
Once the Schematic Editor opens, select the **Place Part** tool on the right sidebar, and click the **Add Library** button. In the file dialog that appears, select the **source** library.



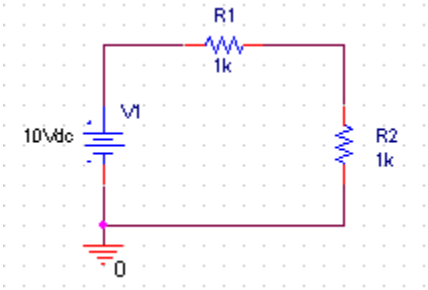
In the **Part List**, select the part 'VDC' and then click the **Place Part** button or hit **enter**. Place the part by clicking in the schematic, and then press escape to stop placing DC sources. Double click on the text **0Vdc** to change the voltage, set it to **10Vdc**



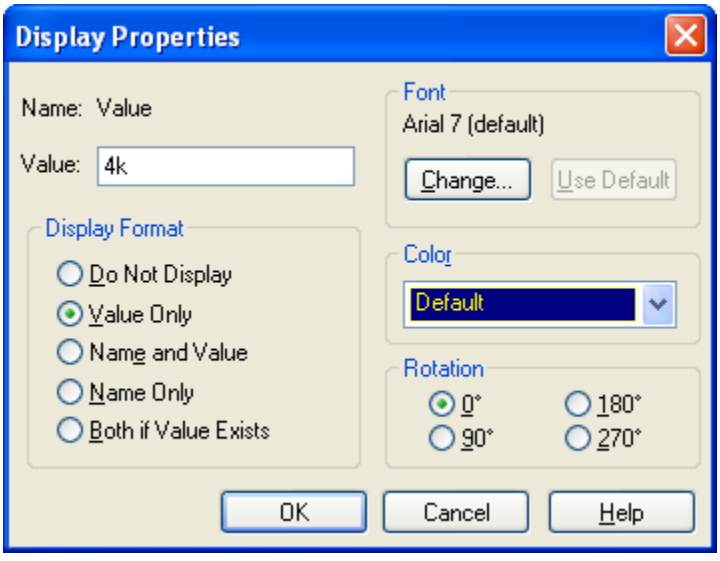
Construct this circuit by selecting the **analog** library and placing two resistors, which appear in the part list as **R**. Rotate one of them by pressing the '**r**' key while placing it. Connect it all together by selecting the **Place Wire** tool and clicking on the points to be connected.



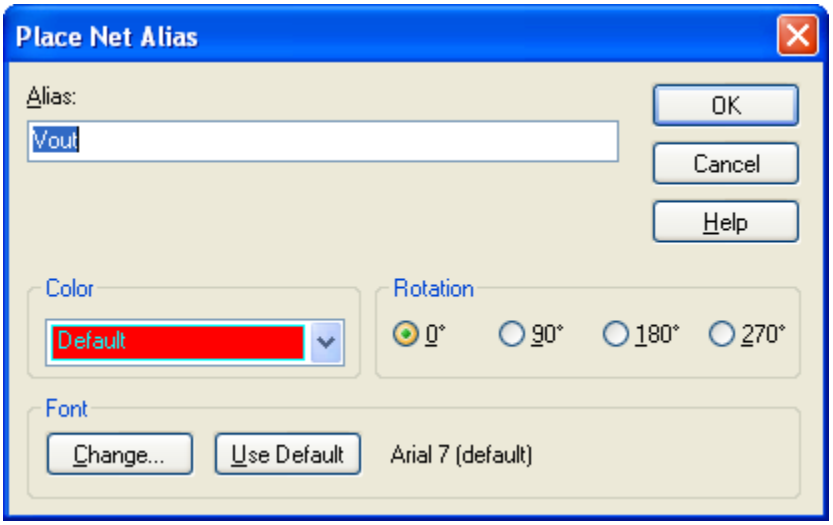
In order to simulate a circuit, PSpice needs a ground node attached to it. Select the **Place Ground** tool in the toolbar. Be warned that there are multiple libraries providing the '**0**' symbol. Select the '**Source**' library before selecting the '**0**' symbol, then click **Ok**.



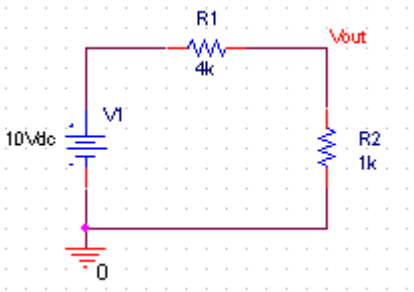
Place the ground symbol in the schematic and connect it to the circuit



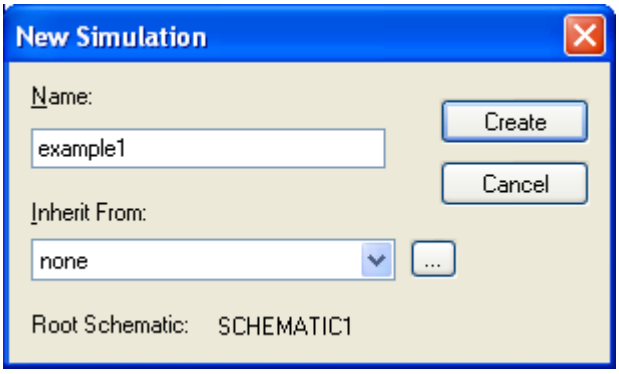
Double click the resistance value of the horizontal resistor, and set it to '4k'.



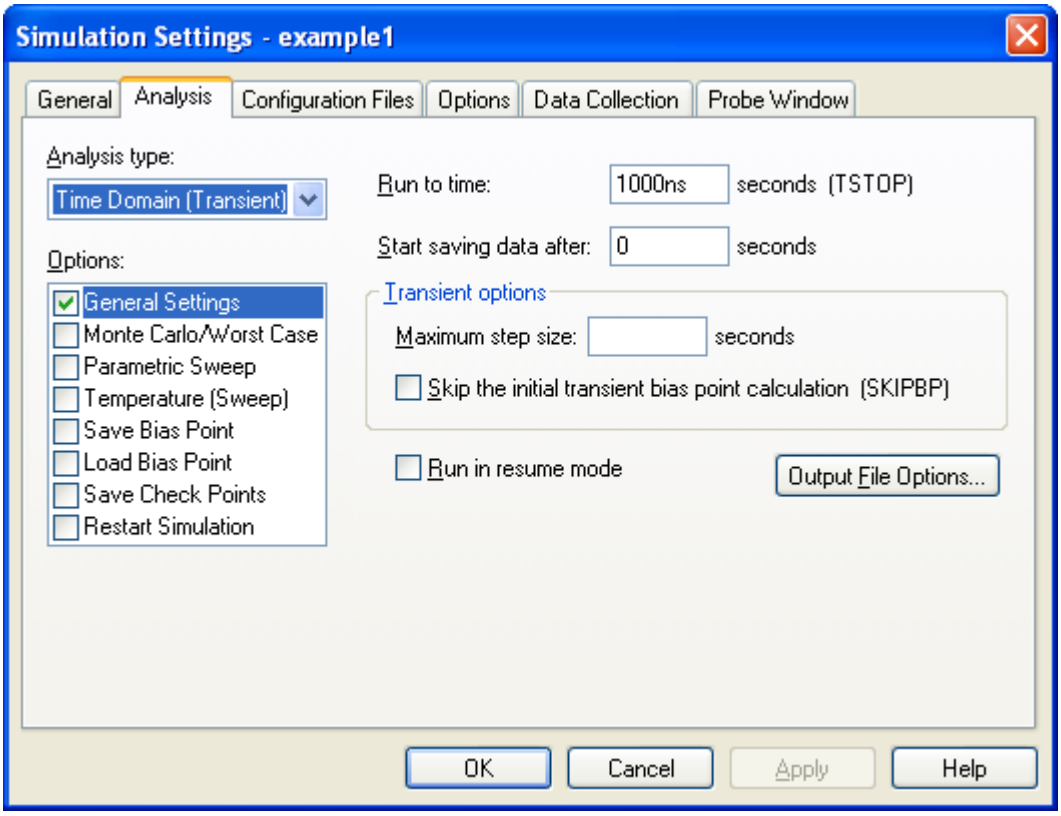
It is important to name the nodes you want to plot so that you can find them easily. Select the **Place Net Alias** tool, enter the name **Vout** and click **Ok.**



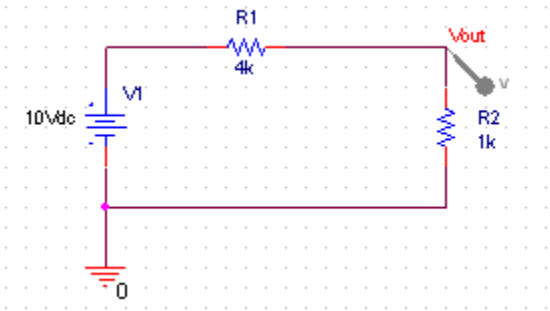
Place the alias between the two resistors



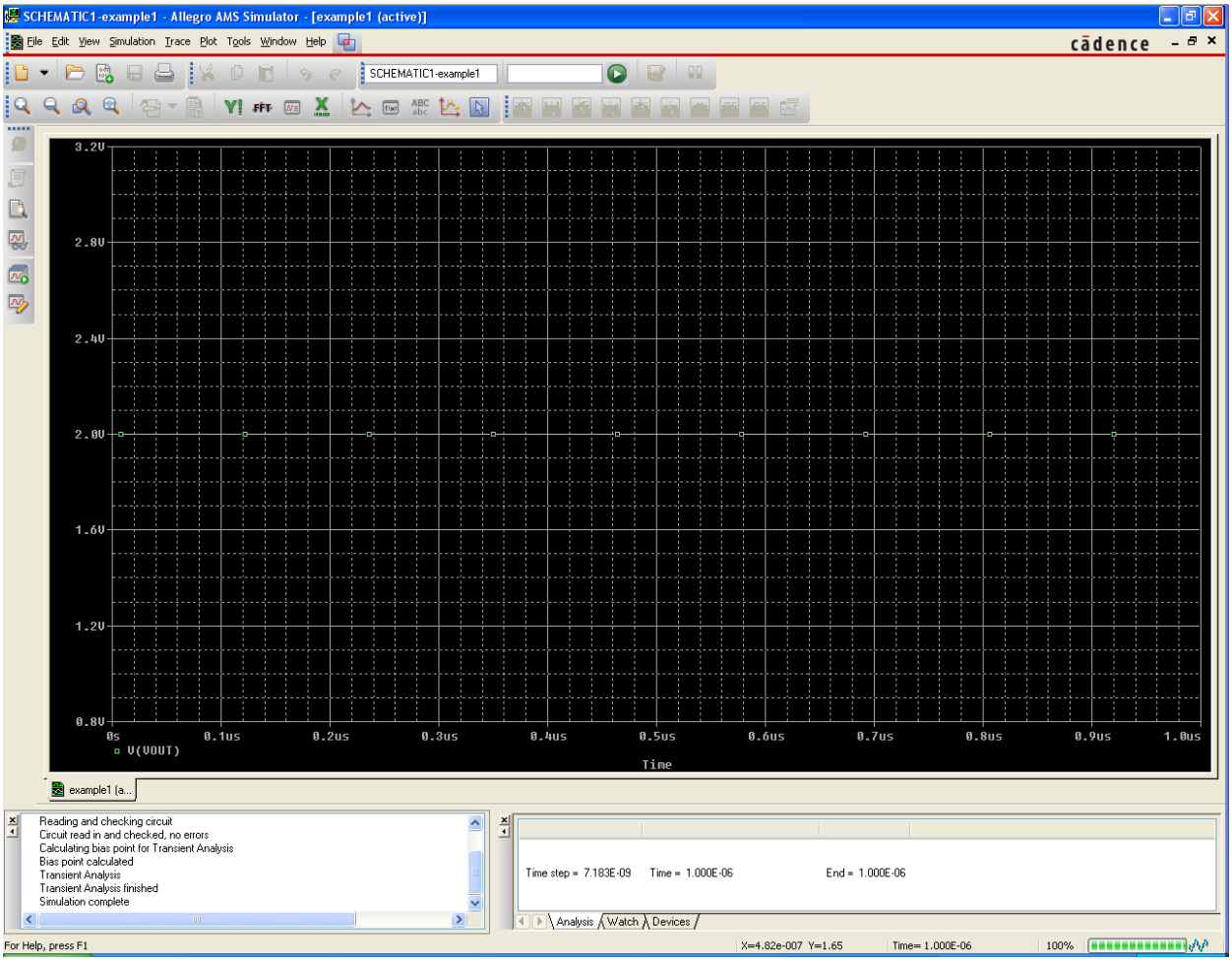
Click the **New Simulation Profile** button to configure the simulation. Enter the name ‘example1’ in the dialog that appears and click **Create.**



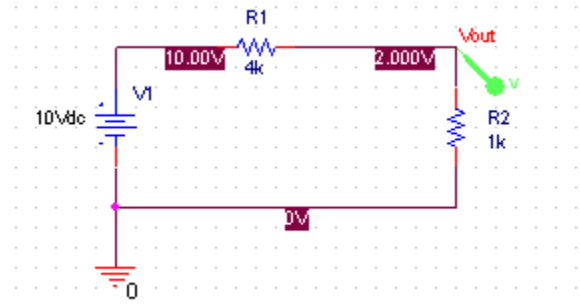
This will bring up the **Simulation Settings** dialog. Click **Ok** to accept the default settings.



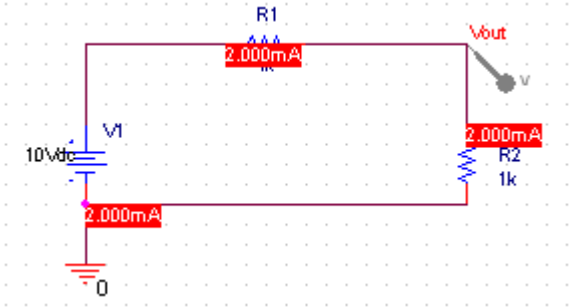
Click on the **Voltage/Level Marker** button and place the marker on **Vout** as shown. Click the **Run PSpice** button.



The simulation results will appear as shown, with the voltages at all probes plotted. Close this window now, and return to the schematic.



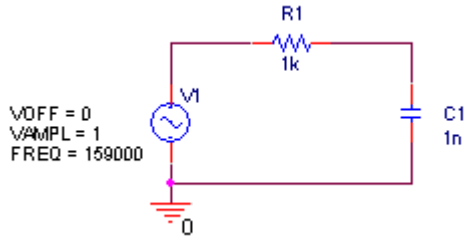
If the **Enable Bias Voltage Display** button is not already selected, click it. Voltage markers should appear at every node of the schematic as shown.



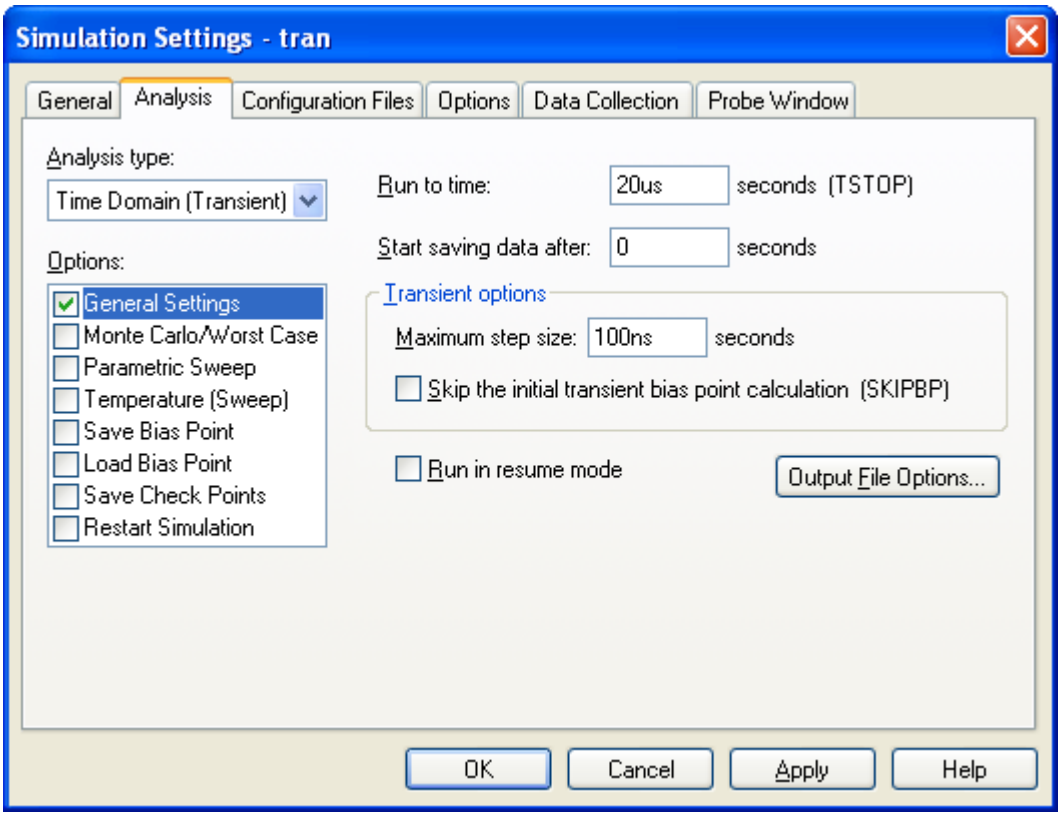
Now turn off the **Bias Voltage Display** and enable the **Bias Current Display**. Cadence will display the DC current through the circuit as shown.

AC Circuit Analysis

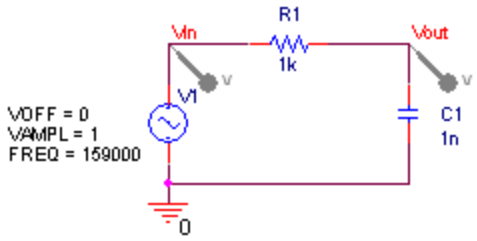
Transient Analysis



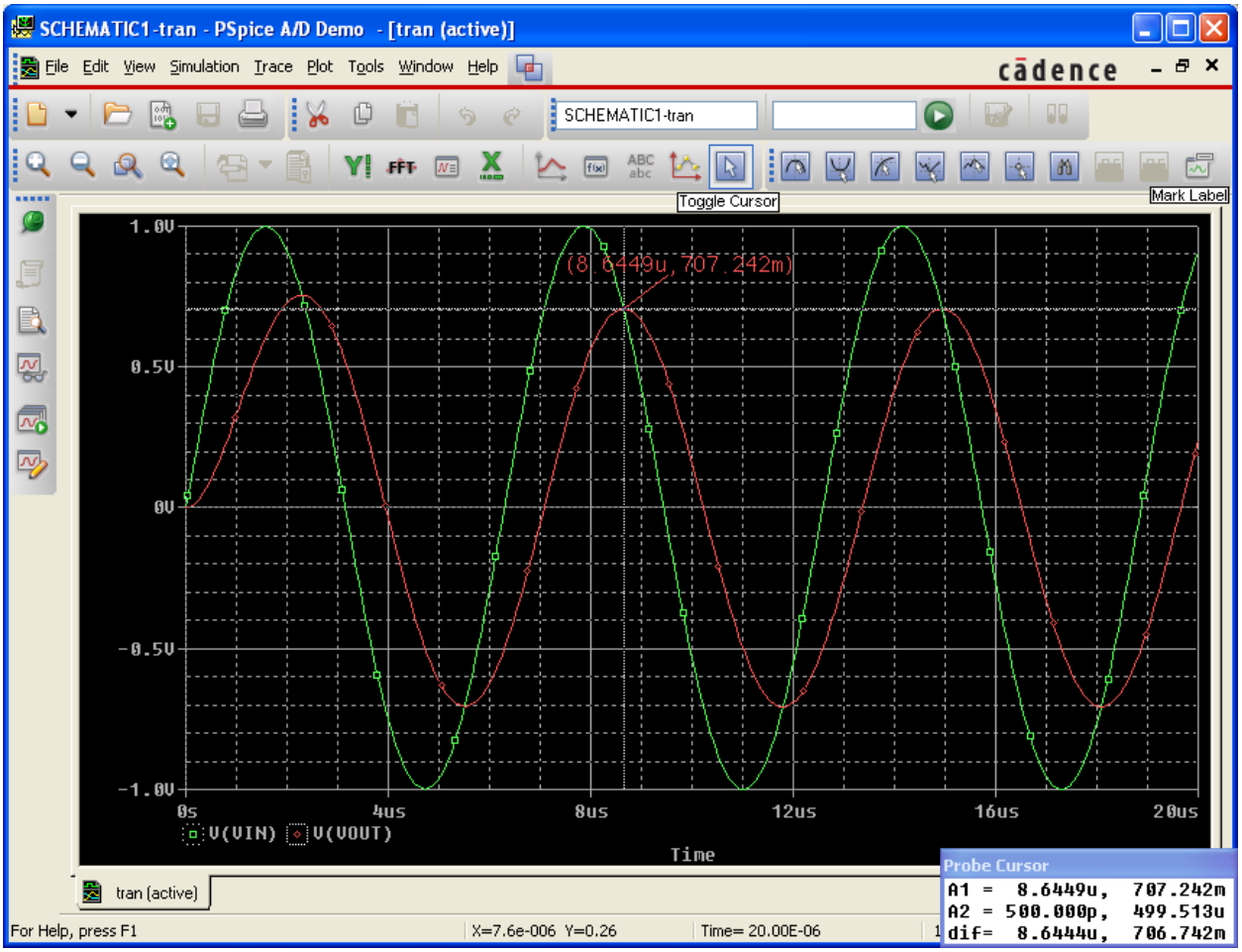
Create a new project and assemble the circuit shown. The voltage source is part '**VSIN**' in the '**SOURCE**' library.



Configure the simulation with the **New Simulation Profile** button and enter the name '**tran**'. Set the **Analysis type** to '**Time Domain (Transient)**', the run time to **20us** and the maximum step size to **100ns.**



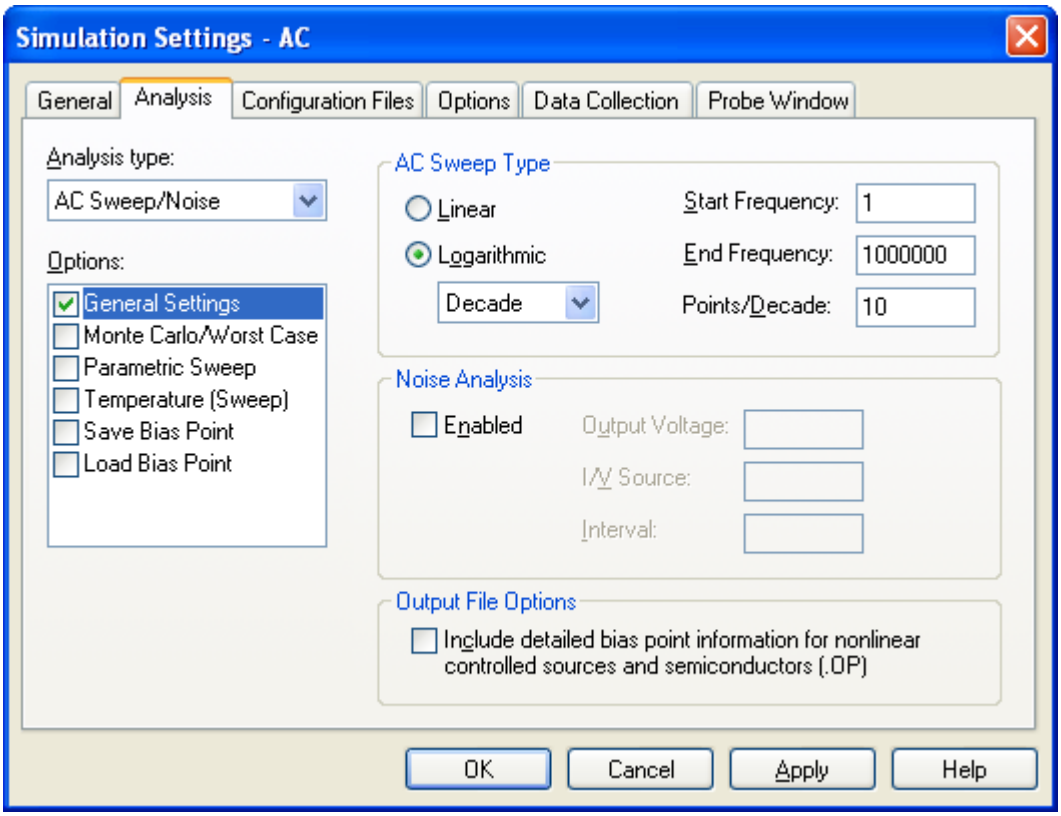
Place voltage markers on each side of the resistor. Name the nodes **Vin** and **Vout**. Run the simulation and the results window should appear. Click the **Toggle Cursor** button and left click the colored dot for **Vout** in the legend. Use the mouse to drag the cursor over to the second peak of **Vout** and note the amplitude. It should be 3dB smaller than the peak voltage of **Vin** (0.707V). Click on the **Mark Label** button to label the point.



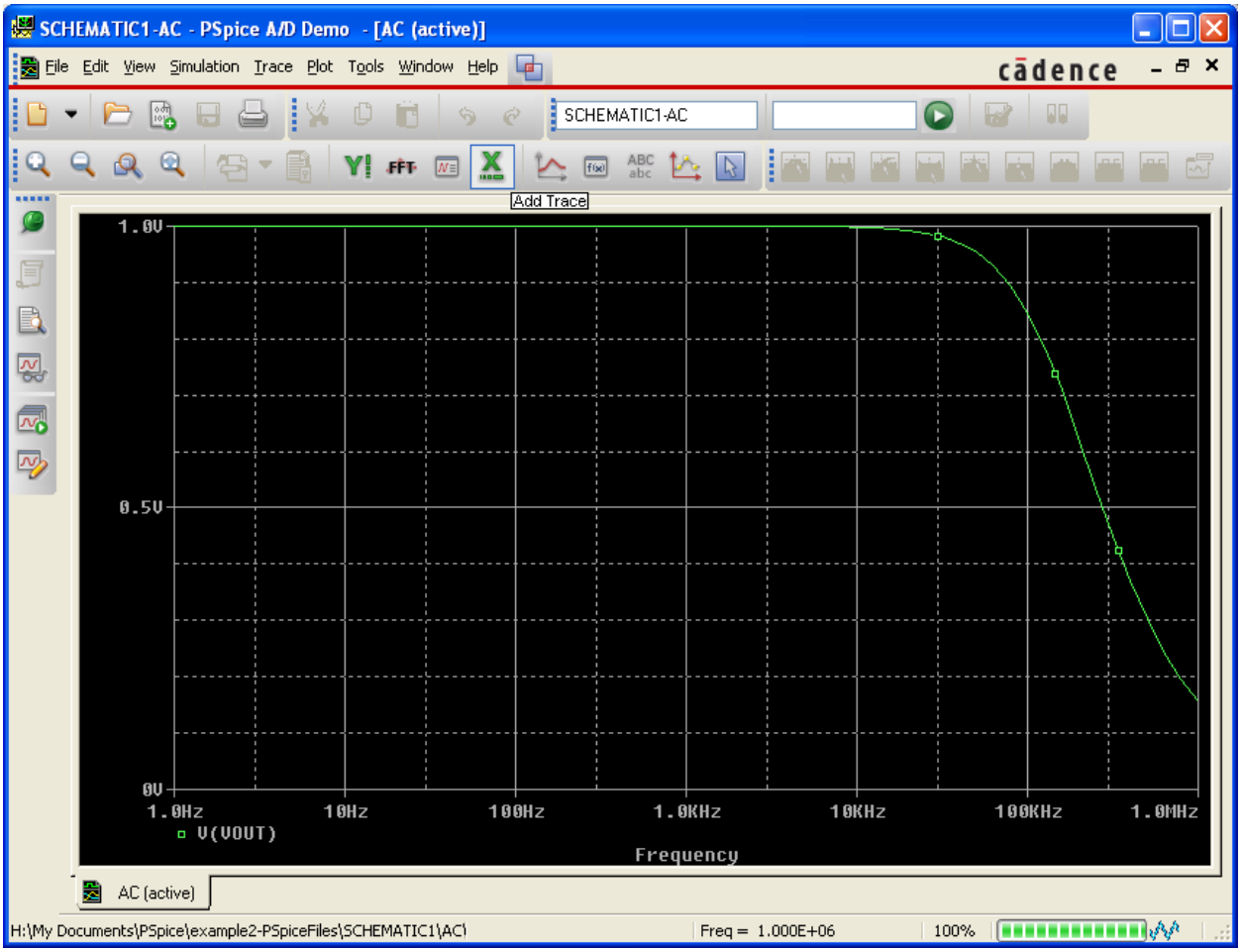
AC Analysis

Close the simulation and modify the circuit, replacing the **VSIN** source with a **VAC** source. Leave the new source with the default attributes. Configure a new simulation profile with name **AC**. Set the analysis

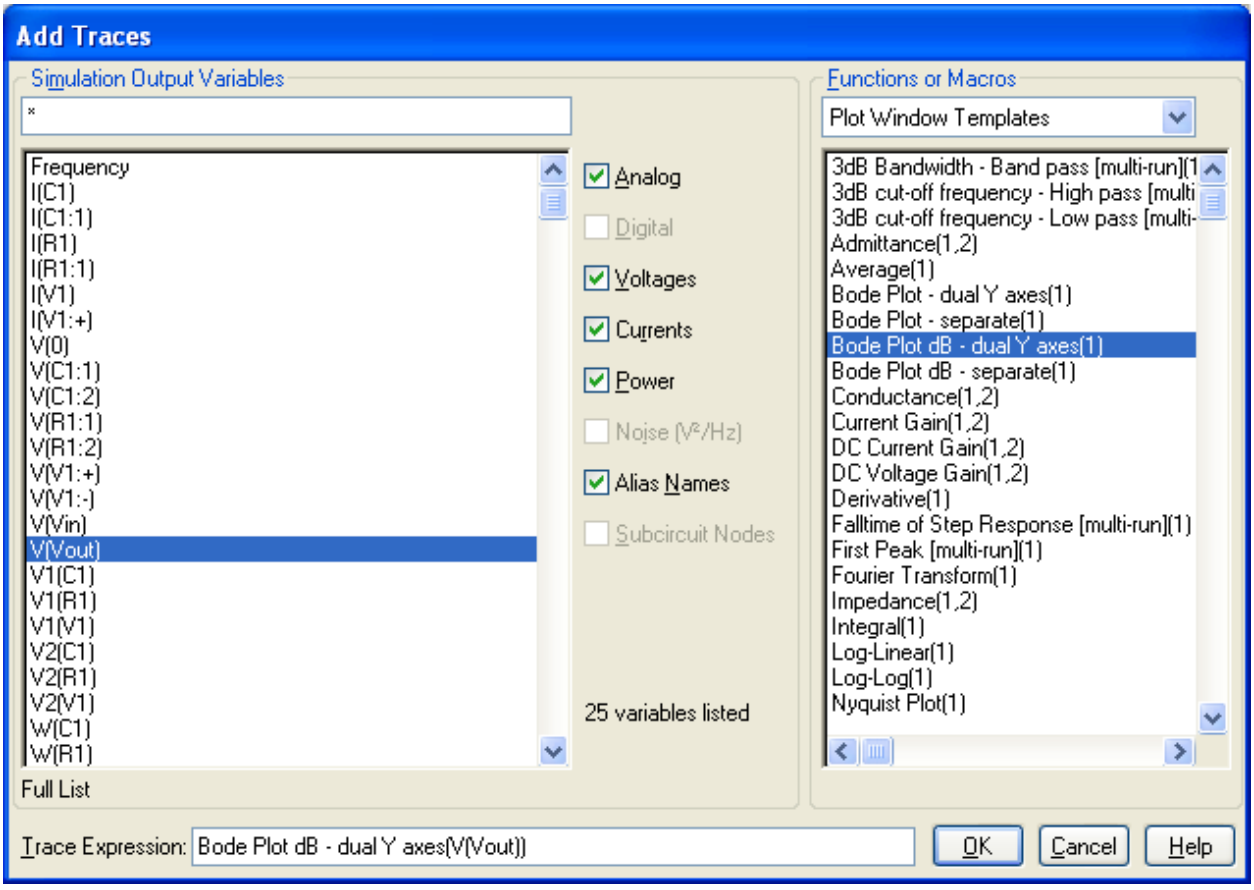
type to **AC Sweep/Noise**, the sweep type to **Logarithmic**, the frequency range from **1-1000000**, and the points/decade to **10**.

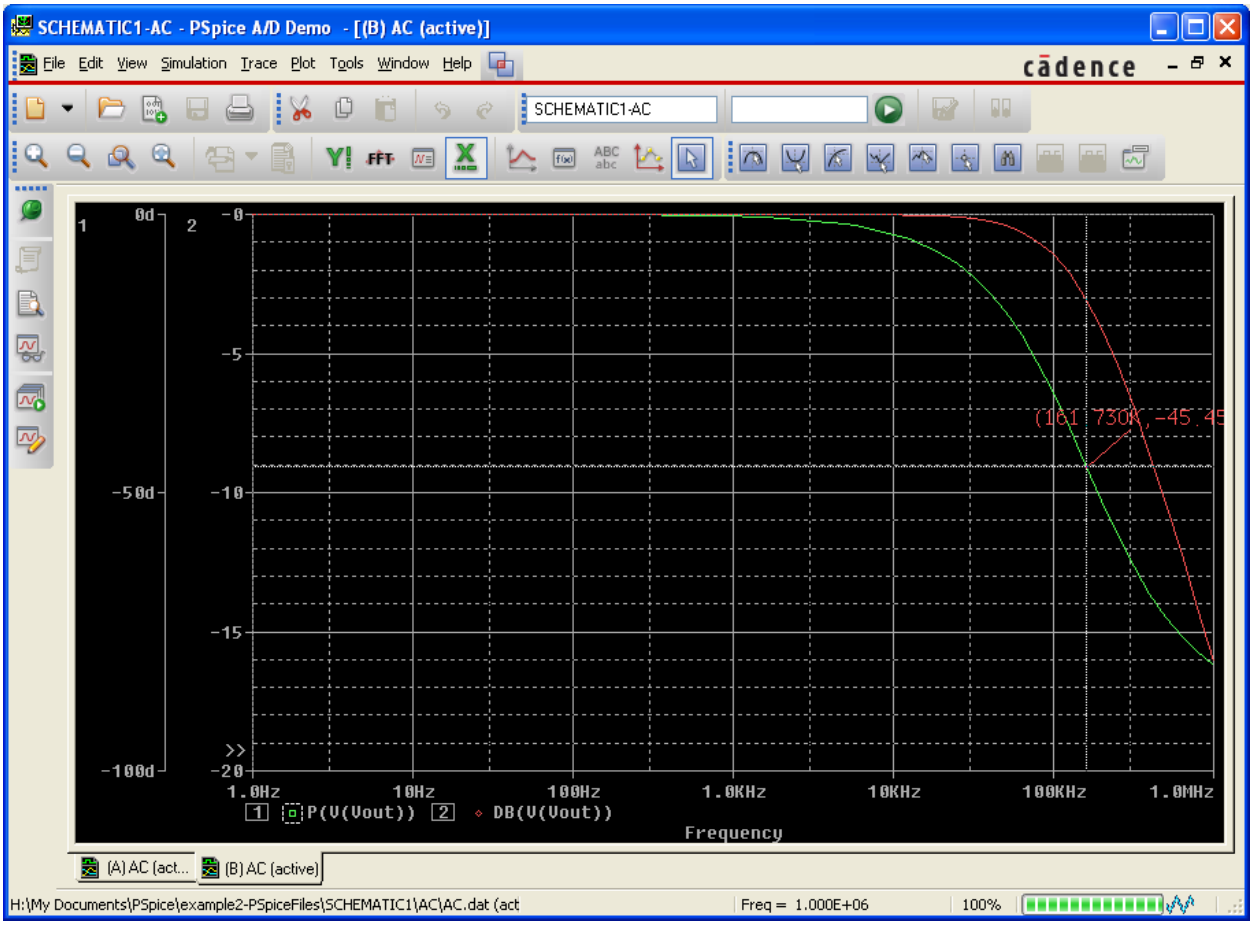


Creating a new simulation profile deletes the preexisting voltage markers, so re-add one on **Vout** and run the simulation.

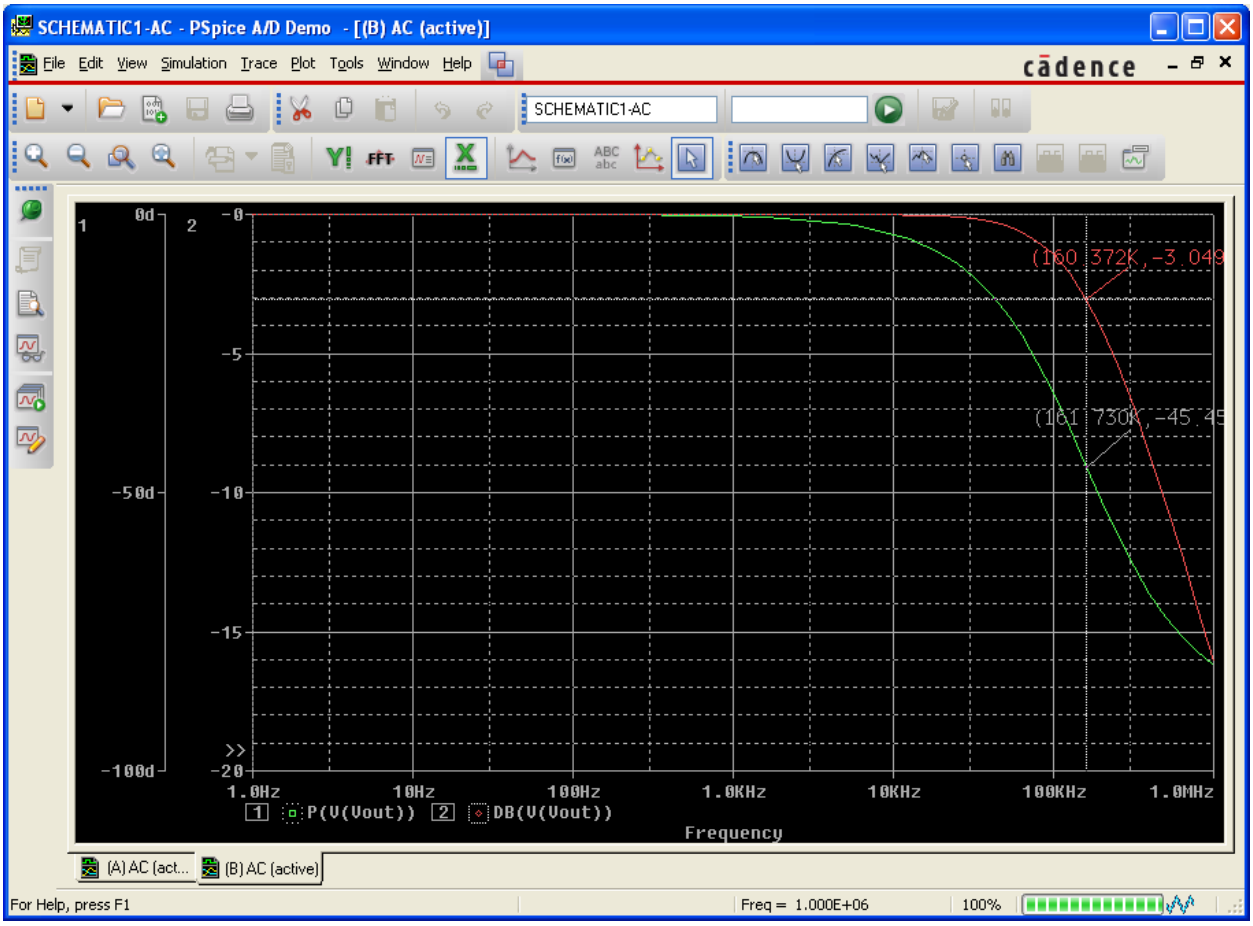


Click the **Add Trace** button and select **Plot Window Templates** from the **Functions or Macros** drop-down menu. Select **V(Vout)** on the left and **Bode Plot dB – dual Y axes(1)** on the right, then click **Ok**.


Add Traces
Simulation Output Variables
*
Frequency
I(C1)
I(C1:1)
I(R1)
I(R1:1)
I(V1)
I(V1:+)
V(0)
V(C1:1)
V(C1:2)
V(R1:1)
V(R1:2)
V(V1:+)
V(V1:-)
V(Vin)
V(Vout)
V1(C1)
V1(R1)
V1(V1)
V2(C1)
V2(R1)
V2(V1)
W(C1)
W(R1)
Analog
Digital
Voltages
Currents
Power
Noise (V²/Hz)
Alias Names
Subcircuit Nodes
25 variables listed
Full List
Functions or Macros
Plot Window Templates
3dB Bandwidth - Band pass [multi-run](1
3dB cut-off frequency - High pass [multi
3dB cut-off frequency - Low pass [multi-
Admittance(1,2)
Average(1)
Bode Plot - dual Y axes(1)
Bode Plot - separate(1)
Bode Plot dB - dual Y axes(1)
Bode Plot dB - separate(1)
Conductance(1,2)
Current Gain(1,2)
DC Current Gain(1,2)
DC Voltage Gain(1,2)
Derivative(1)
Falltime of Step Response [multi-run](1)
First Peak [multi-run](1)
Fourier Transform(1)
Impedance(1,2)
Integral(1)
Log-Linear(1)
Log-Log(1)
Nyquist Plot(1)
Trace Expression:
Bode Plot dB - dual Y axes(V(Vout))
OK
Cancel
Help



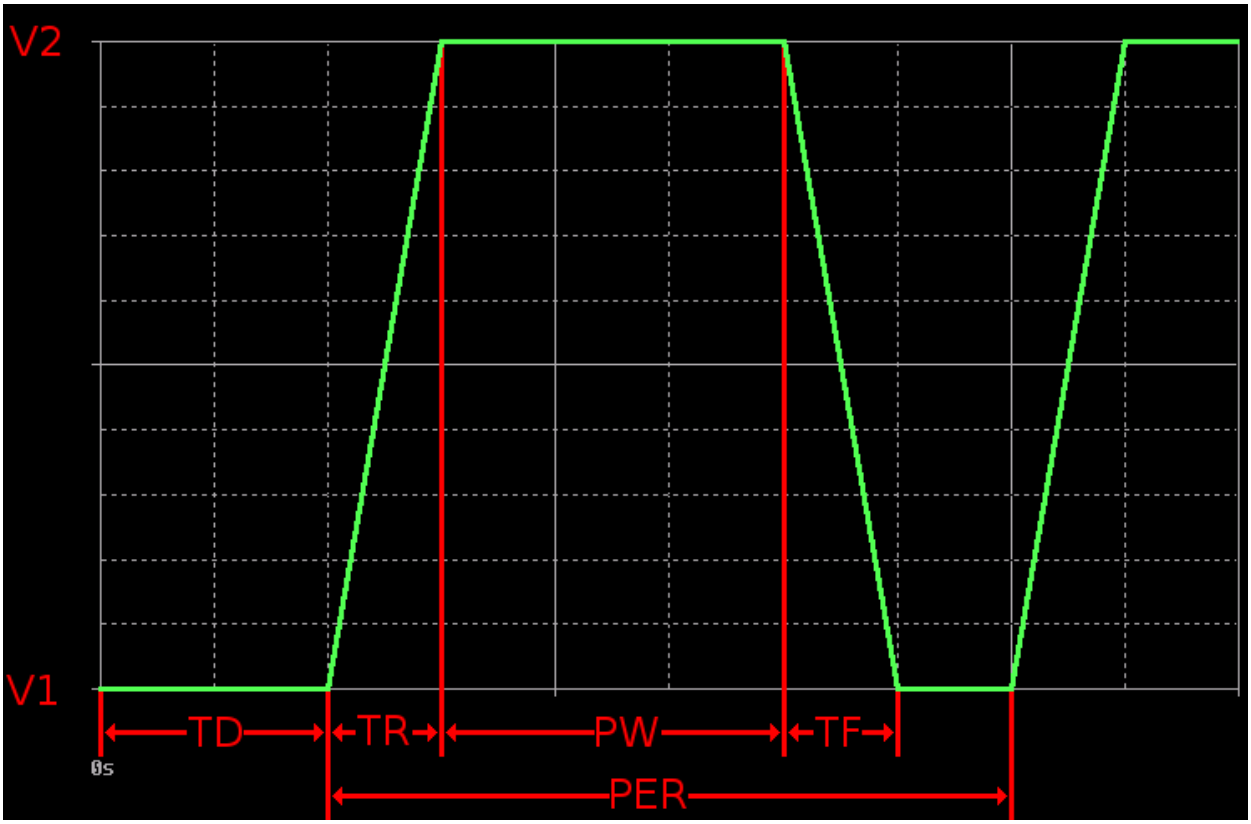
You should see a plot similar to this one. The green trace shows phase and the red trace shows magnitude. Use the cursor to find the point where the phase hits -45, and label it.



Find and mark the point where the magnitude is -3dB.

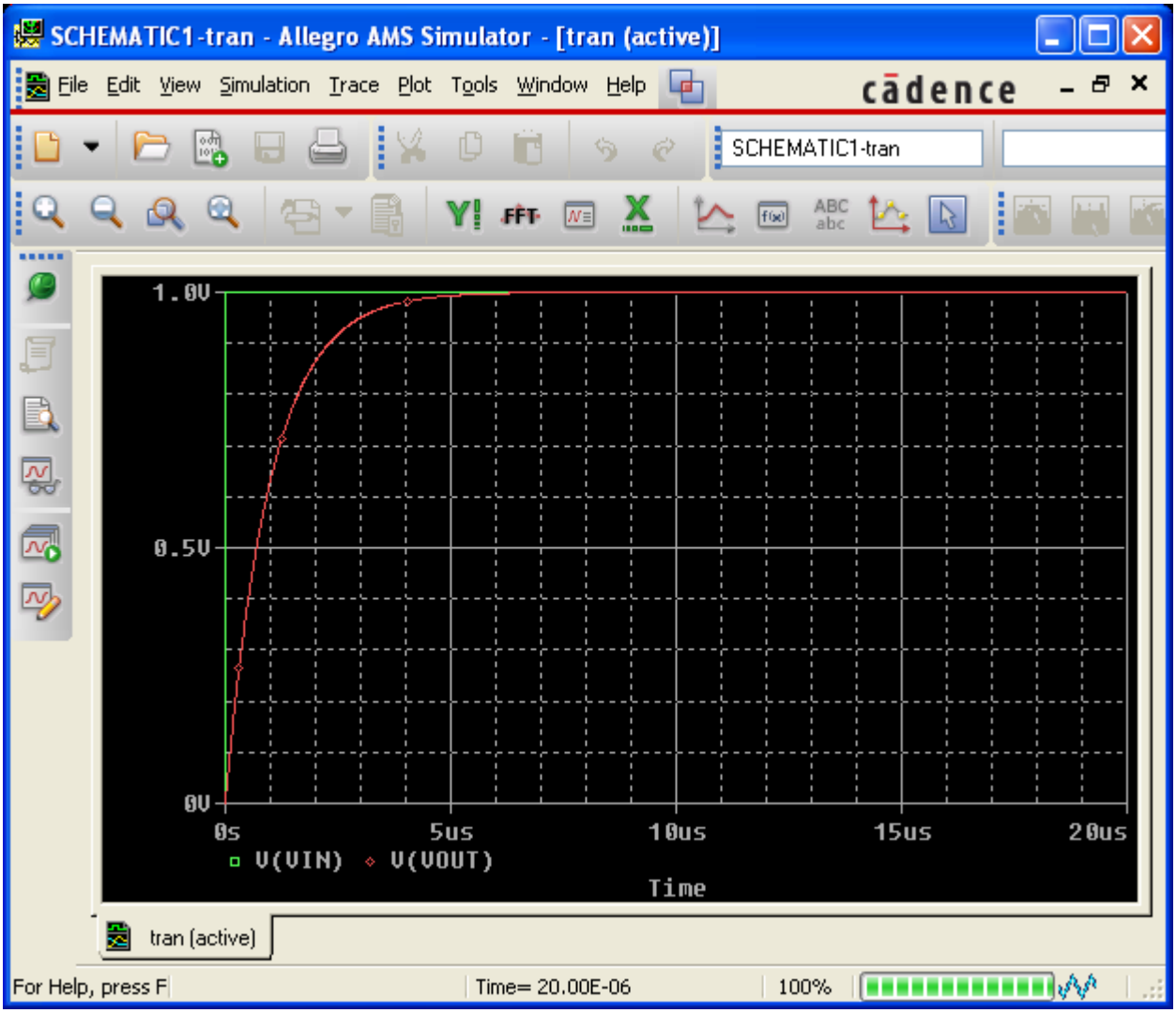
Step Response

To simulate a step response, create a new project and perform a transient analysis using the same steps as above, but with a **VPULSE** voltage source. The **VPULSE** source has 7 parameters affecting the waveform.



V1 = First Voltage
V2 = Second Voltage
TD = Initial Delay
TR = Absolute Rise Time
TF = Absolute Fall Time
PW = Pulse Width
PER = Period

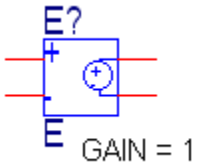
To simulate a step response, we use a **VPULSE** source set to V1=0V, V2=1V, TD=0, TR=1ps, TF=1ps, PW=1s, and PER=2s. A 1 picosecond rise/fall time is extremely small with regard to the simulation time, so it closely approximates the ideal step function.


SCHEMATIC1-tran - Allegro AMS Simulator - [tran (active)]
File Edit View Simulation Trace Plot Tools Window Help
cadence
SCHEMATIC1-tran
1.0V
0.5V
0V
0s
5us
10us
15us
20us
V(VIN) V(VOUT)
Time
tran (active)
For Help, press F
Time= 20.00E-06
100%

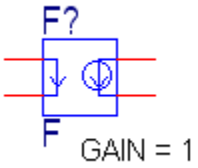
Dependent Sources

The **ANALOG** library provides four dependent sources:

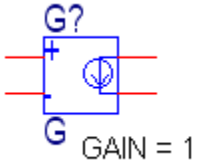
- E – Voltage Controlled Voltage Source



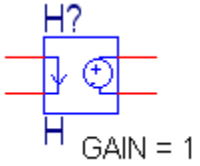
- F – Current Controlled Current Source



- G – Voltage Controlled Current Source



- H – Current Controlled Voltage Source



Variable Phase VSin Source

The phase isn't displayed on the schematic for the **VSIN** source, but it can be added. Double-click on it to bring up the property editor. Find the '**PHASE**' field and select it, then click the '**Display…**' button. Select **Name and Value** from the **Display Format** list, then click **Ok**.

The '**PHASE'** property is now displayed on the schematic, and can be moved and edited the same as the default properties.