

PSpice A/D Manual and Examples, Part 2

In this example we will simulate an inverting operational amplifier using one of the most common commercial operational amplifiers, the μ A741. Begin by using the Parts Browser to place a μ A741 operational amplifier in your schematic. The model of the 741 has been carefully constructed so that its performance in your simulated circuits will be very similar to that of the measured performance of a real 741 operational amplifier circuit. Although you cannot see the inner details of the operational amplifier it is complex enough that you cannot simulate a circuit containing more than two such operational amplifiers.

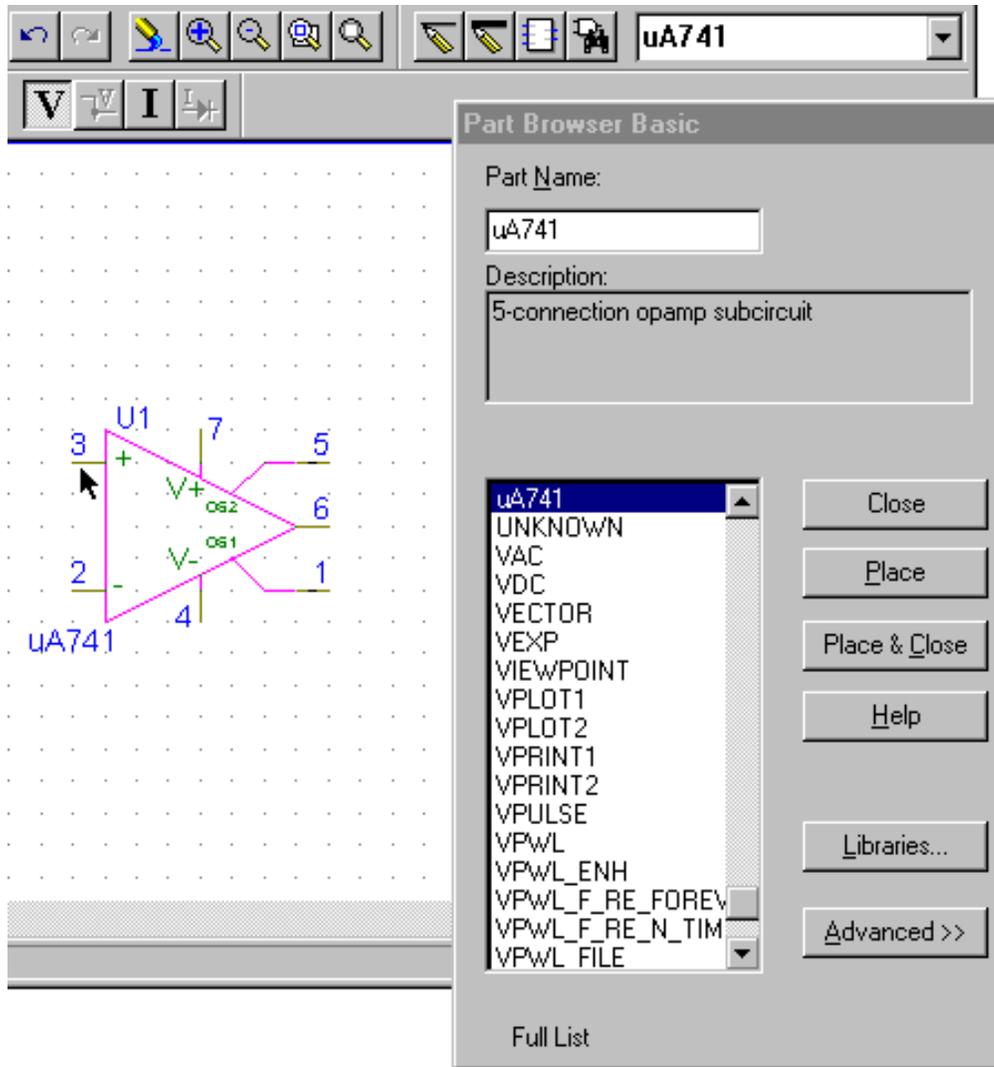


Figure 1. Placing A μ A741 Operational Amplifier In Your Schematic

To put the operational amplifier into the typical orientation seen in electronic schematics (with the inverting terminal up) you need to rotate the symbol twice until it is in the orientation seen in Figure 2.

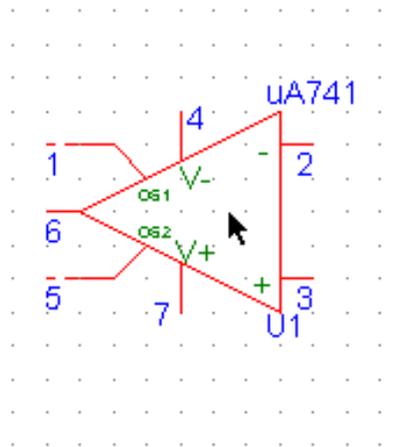


Figure 2. Twice Rotated Operational Amplifier Symbol

This can then be transformed into the conventional configuration by “flipping” the symbol as shown in Figure 3 into the final configuration shown in Figure 4.

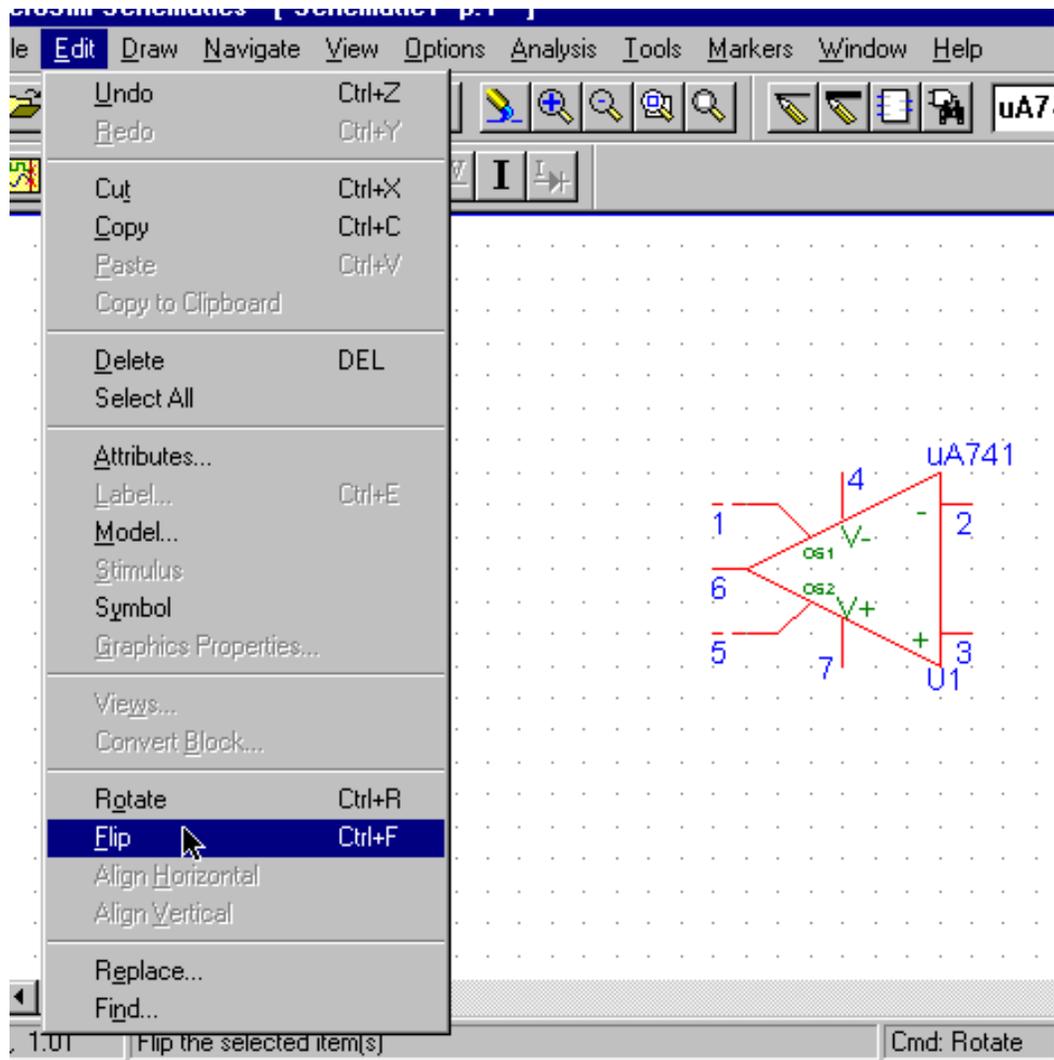


Figure 3. Flip the Operational Amplifier Into Its Final Configuration

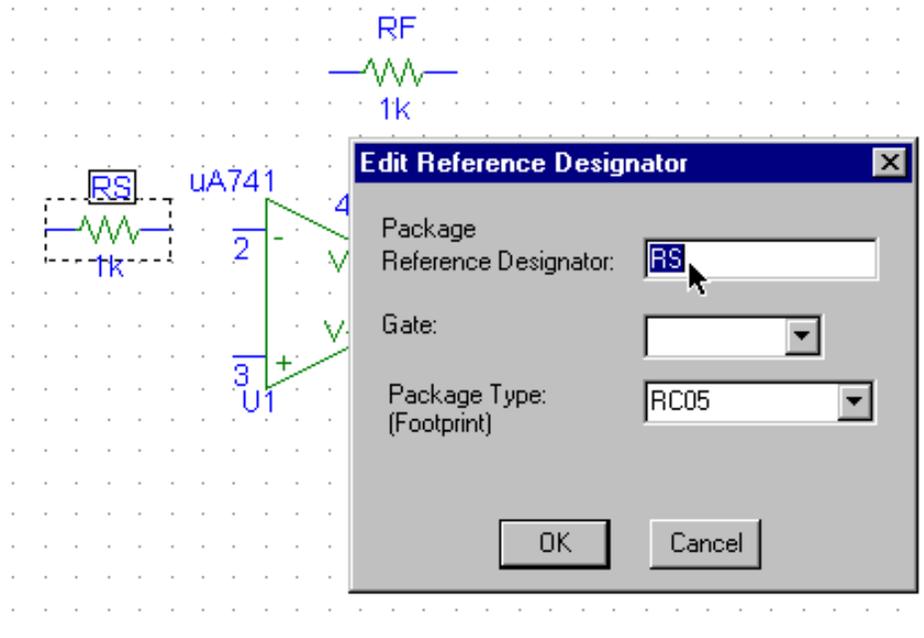


Figure 4. Changing A Component's Reference Name

The labels of resistors placed in the schematic can be changed by simply double left clicking on the symbol's label to get the Reference Designator dialog box shown in Figure 4. You can then change the name to anything you wish such as, in this case, RS.

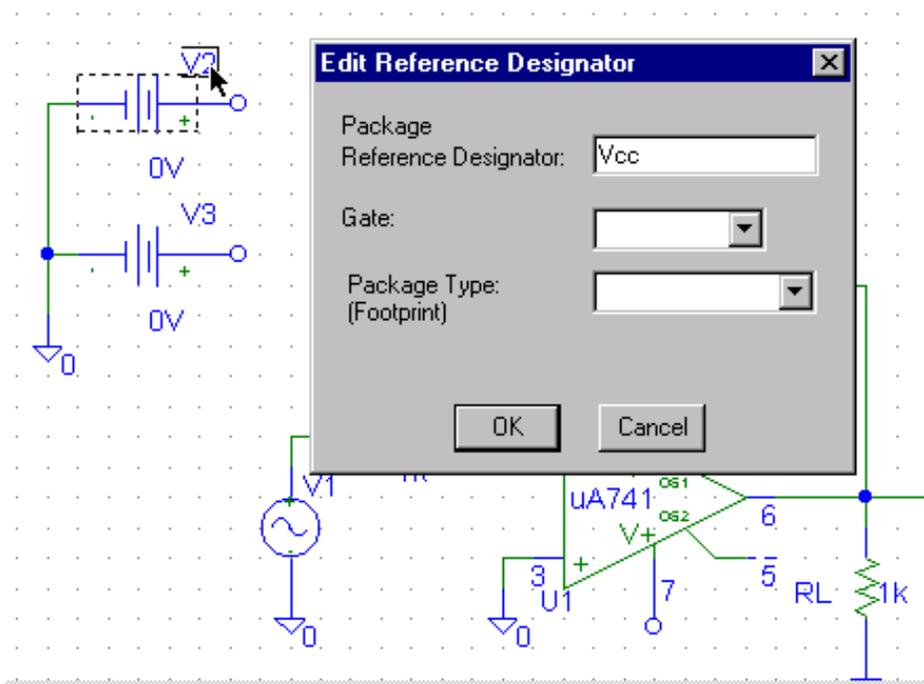


Figure 5. Editing A Component's Label

Drawing a schematic with operational amplifiers in it can get quite complex because of the power supply connections. Fortunately there is a simple technique for connecting components throughout a schematic. This is done by placing a BUBBLE at the end of each wire you want to connect. A bubble is shown at the tip of the cursor in Figure 5. These connections can be placed anywhere in your circuit using the standard Parts Browser dialog box; however, only BUBBLES which have the same name are electrically connected. A BUBBLE's name can be designated by double left clicking on the BUBBLE to get the dialog box shown in Figure 6

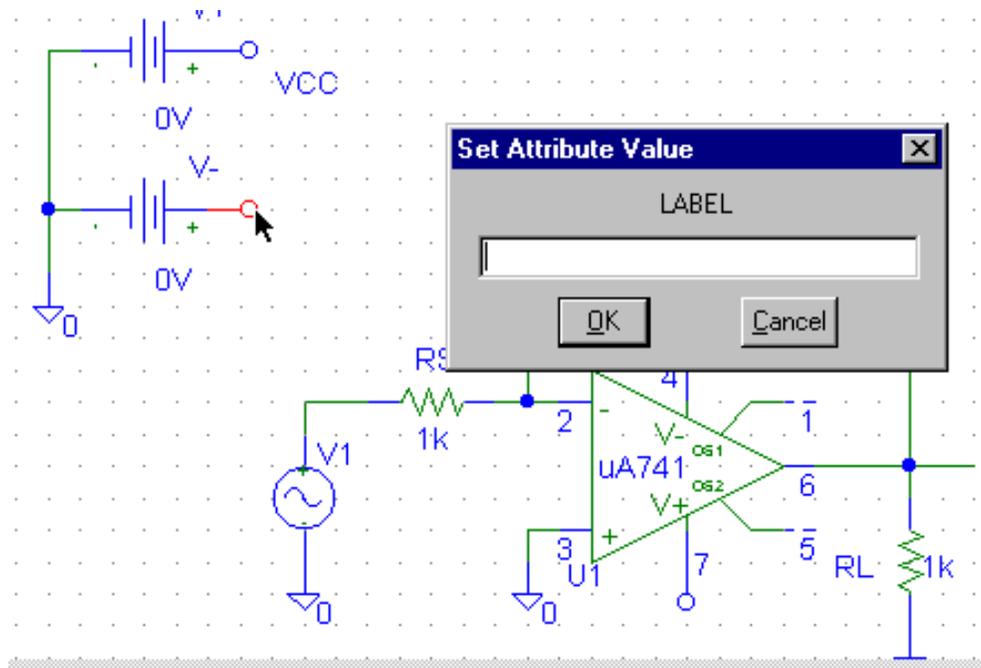


Figure 6. Setting the Name (LABEL) of A BUBBLE

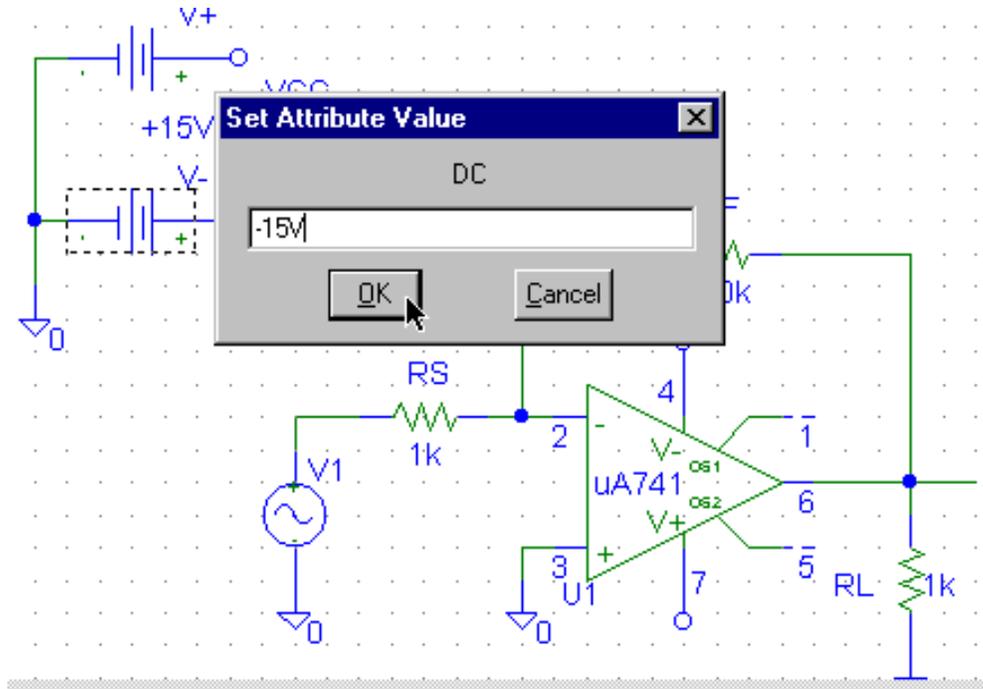


Figure 7. Editing The Value of A DC Voltage Source

Figure 6 and Figure 7 show the specification of a -15 volt voltage source labeled Vee. This is connected to a second DC voltage source labeled Vcc. The two DC voltage sources are connected to a common ground and provide +15 and -15 volts to the operational amplifier just like your Graymark 808 power supply. This can be used to supply the required voltage to the operational amplifier by placing BUBBLES on the operational amplifier's V+ and V- leads as shown in Figure 10.

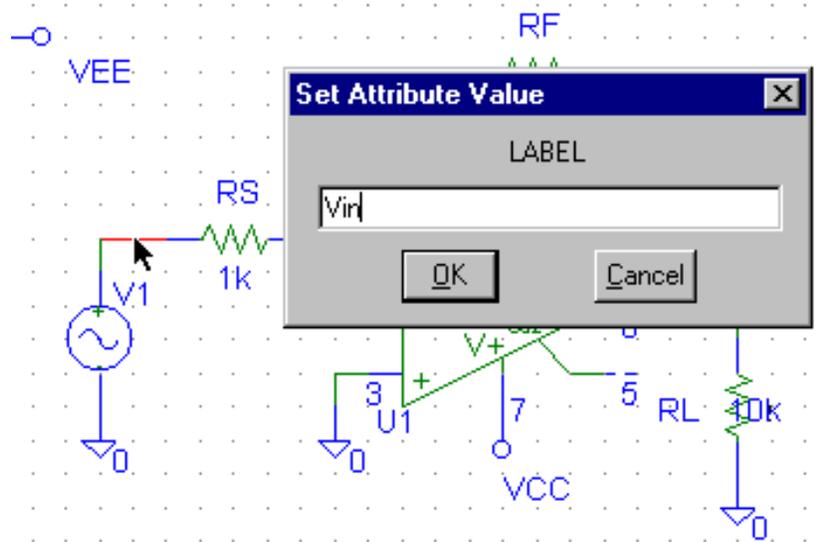


Figure 8. Labeling the Input Voltage

Figure 8 shows a new and useful procedure in PSpice, the labeling of a wire. A wire is a circuit element just like a resistor or voltage source and it can be assigned a unique name to be used in later analysis. In this case, the wire coming from the input voltage source has been selected. By double left clicking the dialog box for assigning a label comes up. Figure 8 shows this wire being assigned the name Vin.

The voltage source connected to the input of the operational amplifier is a new type which you will use extensively in later assignments. It is a voltage source of type VSIN. This source is similar to the function generators you will use in the lab and can supply a voltage of the form

$$VSIN(t) = VDC + VAMPL * SIN(2 * \pi * F * t)$$

where VDC is a constant DC voltage, VAMPL is the peak amplitude of a sine wave voltage, and F is the frequency of the sine wave. These three parameters can all be set by the user. Because there are more than three parameters you have to use the Edit Attribute menu to access all the parameters you can change. To show the parameters you have assigned on the schematic (always a good practice) use the Change Display button to the right of the Edit Attributes dialog box. This brings up the Display Attribute dialog box shown in Figure 9. Click the “Both name and value” selection to show the name and assigned value of the parameter on your schematic.

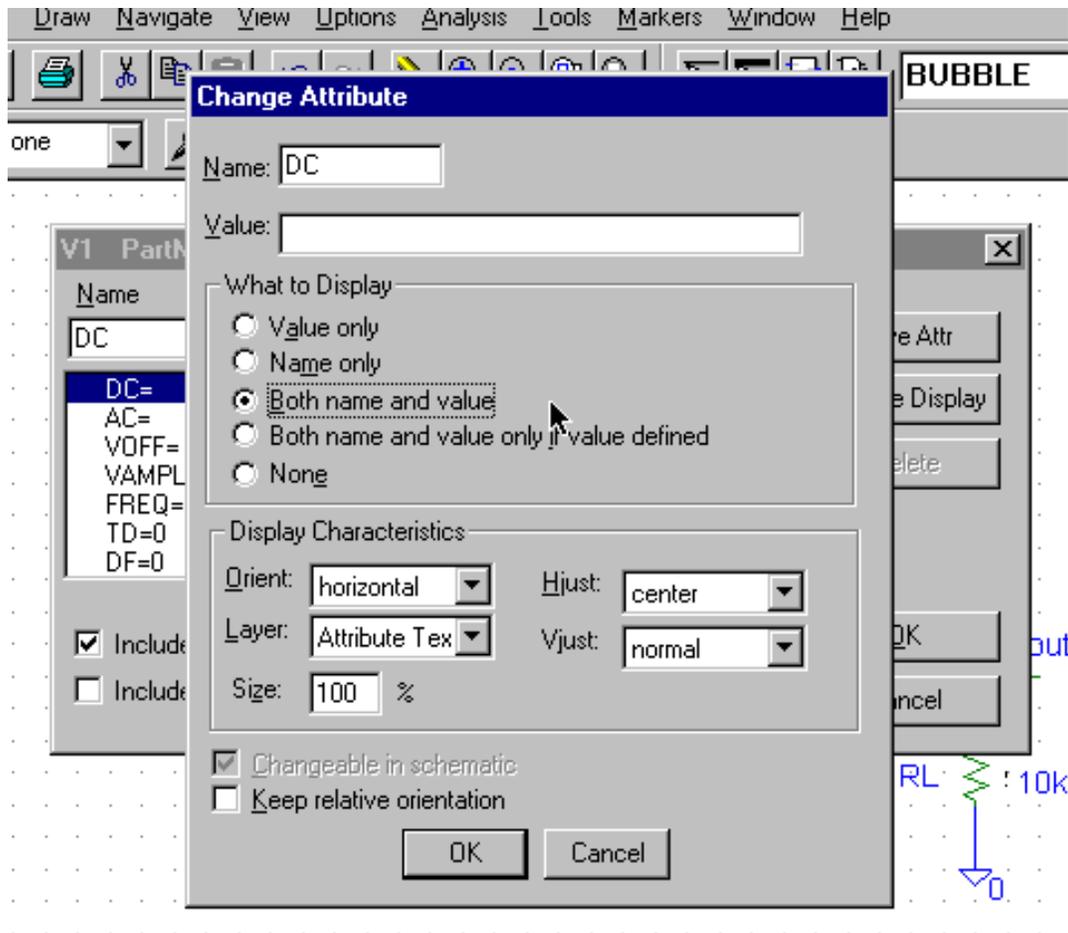


Figure 9. Displaying VSIN's Parameter On A Schematic

The final result of doing all these changes is shown in Figure 10. The power supplies are at the upper left of the drawing and are invisibly connected to the operational amplifier because the labeled BUBBLES have the same name. Note that all the parameters of VSIN (labeled V1) are shown to its left. For convenience in generating plots an input wire is labeled V_{in} and an output wire is labeled V_{out} . The wire labeled V_{out} is only connected at one end.

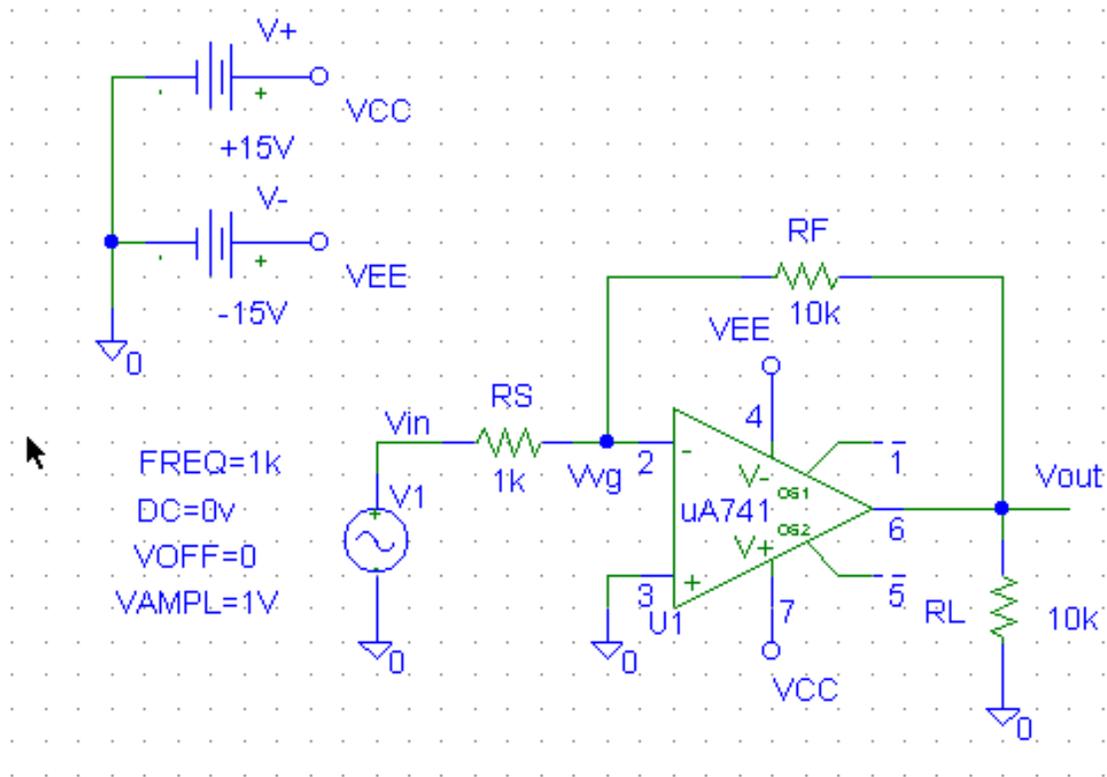


Figure 10. Inverting Operational Amplifier

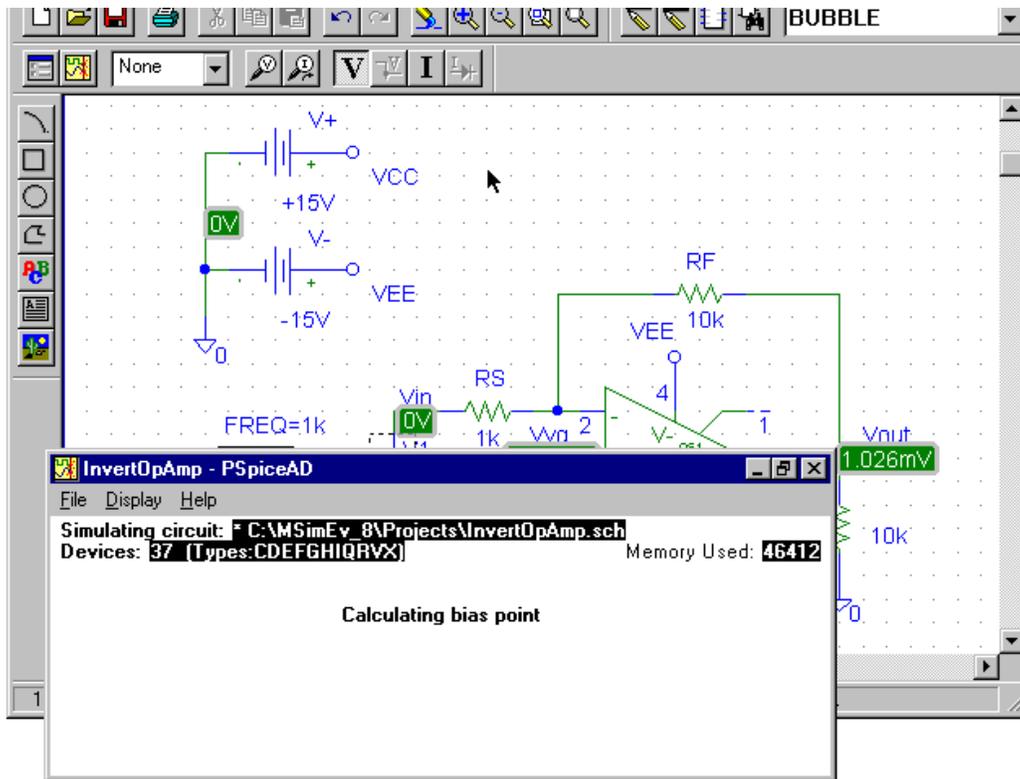


Figure 11. Simulating The Operational Amplifier Circuit

Once you have the circuit shown in Figure 10 you can click on the simulate button to get the analysis window shown in Figure 11. This indicates that PSpice is calculating the circuit values. If everything was properly done you should get the DC voltage values as shown in Figure 12. Remember you have to set the Display Attributes to show the voltages on the schematic. These values look correct. The resistors specify a voltage gain of -10. The input voltage $V_{in}=1$ volt and the output voltage is $V_{out}=-10$ volts in agreement with the prediction. Another interesting observation is that the voltage at the inverting input of the operational amplifier is 72.44 microvolts which is pretty close to the zero (i.e., virtual ground) voltage expected.

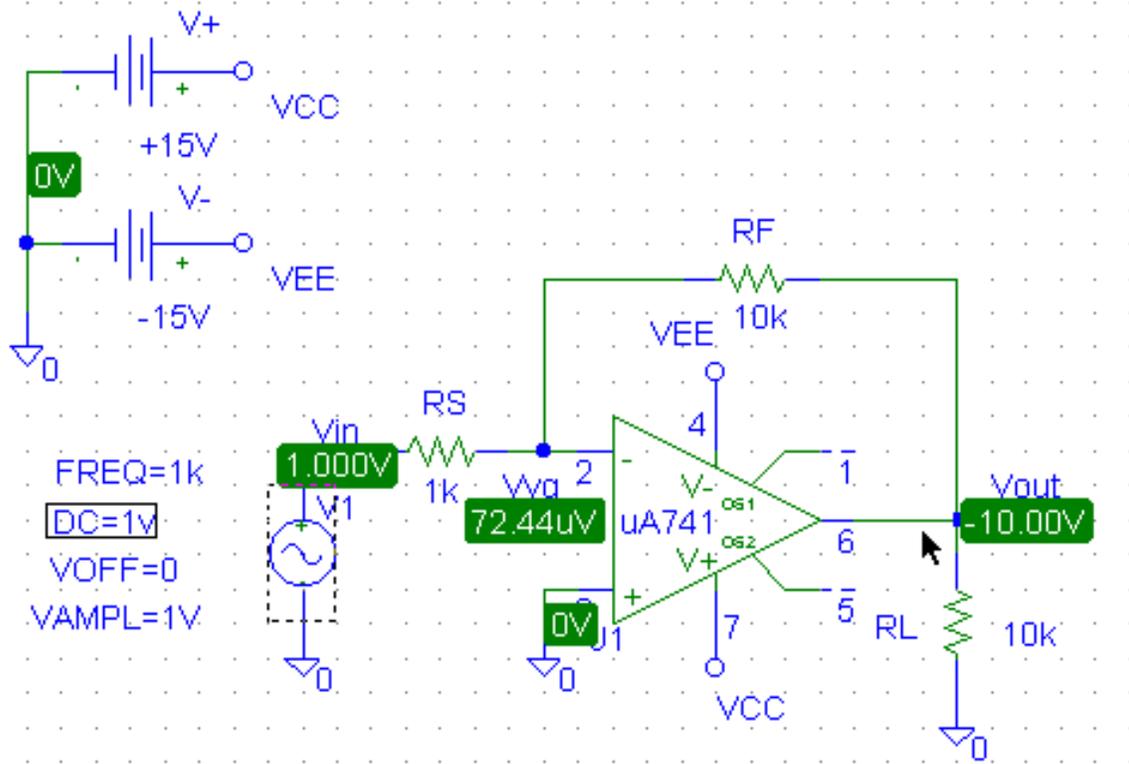


Figure 12. Simulated Operational Amplifier Circuit

So far all you have done is a simple DC analysis. Let's modify the circuit to produce a plot of the output voltage as the input voltage is being systematically varied, i.e. swept. To sweep a voltage source you need to go to the Analysis\Setup menu. In this dialog box you should enable DC Sweep and click the button to get the DC Sweep dialog box shown in Figure 13. A DC sweep means that only the DC component of V1 will be changes; the VAMPL was previously set to zero and does not affect the analysis. In the DC Sweep dialog box you should identify a voltage source as the variable to be swept. The name of the particular source is typed into the Name box. For this sweep, we will linearly sweep the DC component of V1 from -2 to +2 volts in increments of 0.1 volts.

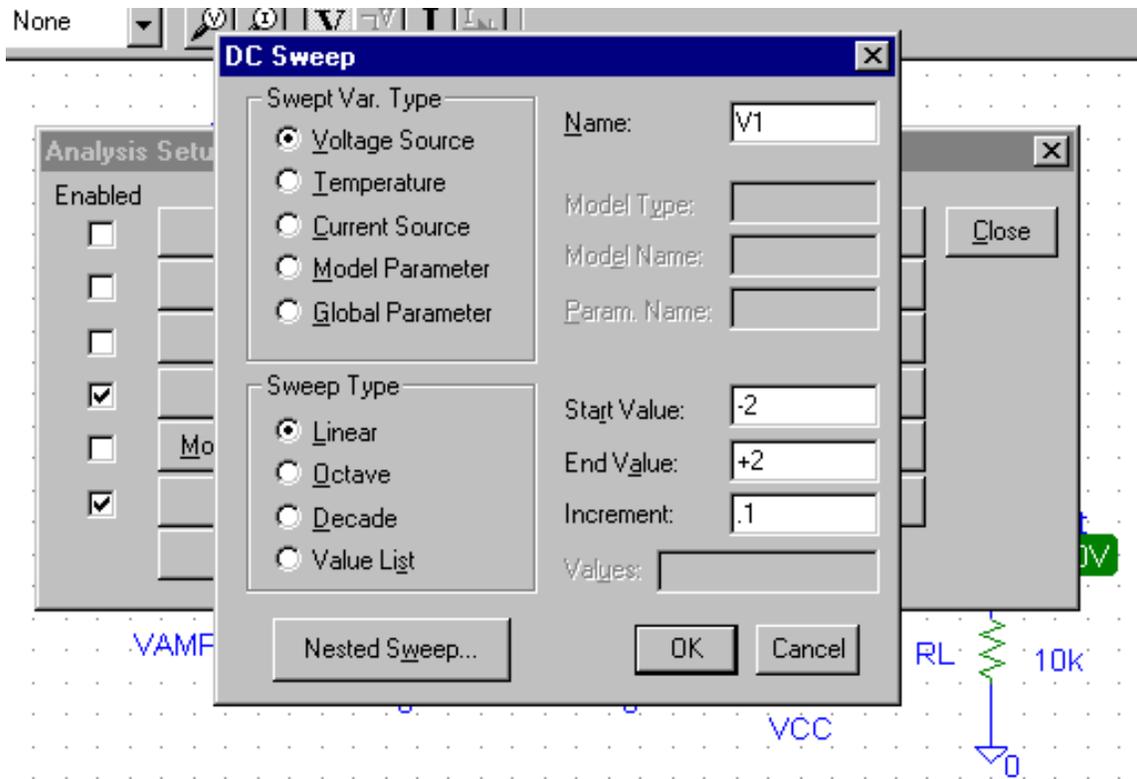


Figure 13. Setting the DC Sweep Parameters for V1

Clocking on the simulation button will start the simulation process and will cause the PSpice window to appear followed by a PROBE window. At this point you need to specify the variables to be plotted. Since the input voltage V_{in} is what is being swept we want to keep V_{in} in the horizontal axis. However, you need to Add a Trace to get a vertical display. Select Add Trace in the menu for the Vertical Axis and choose $V(V_{out})$ as shown in Figure 14.

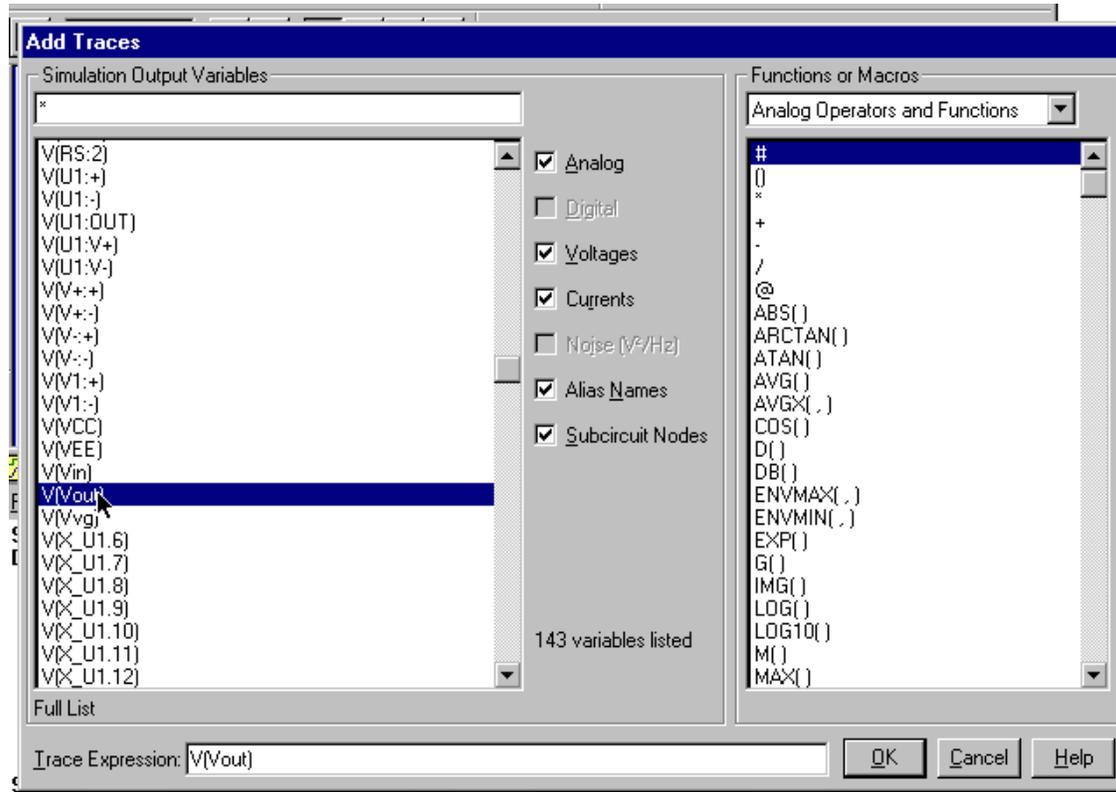


Figure 14. Specify Vout To Be Plotted On Vertical Axis

This will cause PROBE to plot Vout versus Vin as shown in Figure 15.

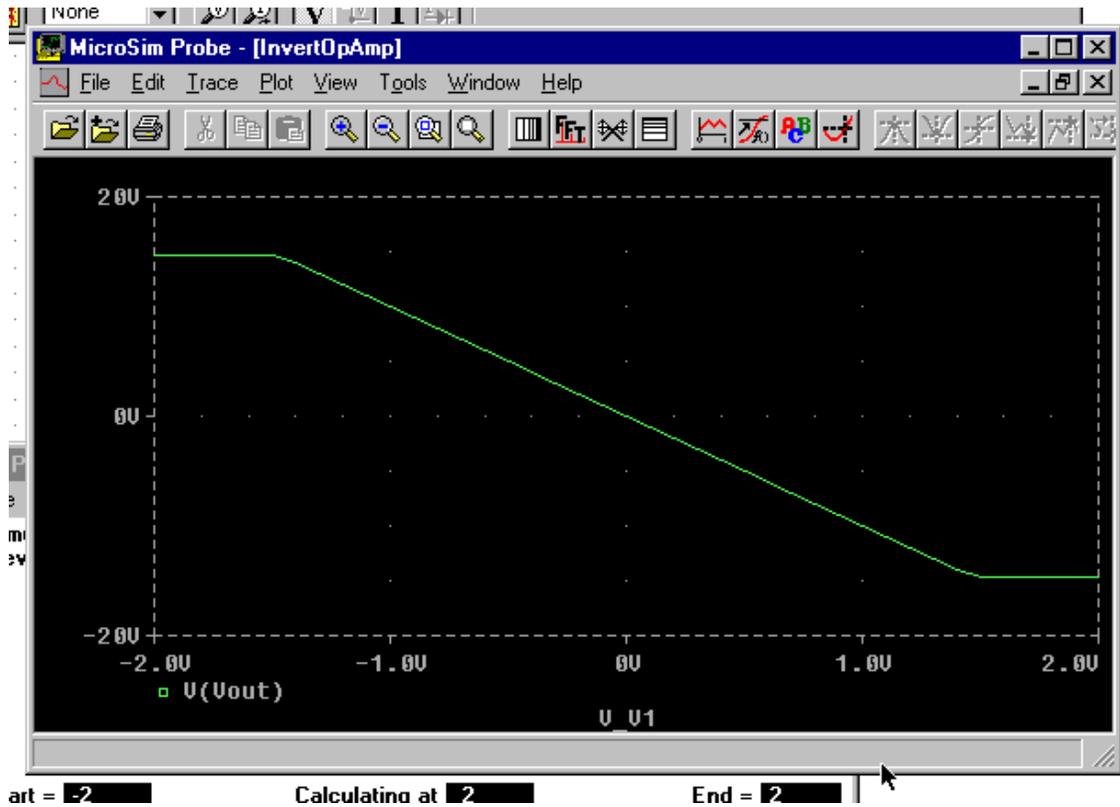


Figure 15. Vout Vs. Vin For The Operational Amplifier Circuit

Note that the output voltage saturates at +15 as -15 as a real operational amplifier should and as you verified in Lab 5.

PROBE is a fairly versatile plotting package. Figure 16 shows the result of adding a second vertical axis. You can select each axis to edit its attributes by clicking on it to select it. Sorry, double clicking doesn't work here. Here, I changed each axis to have a different scale.

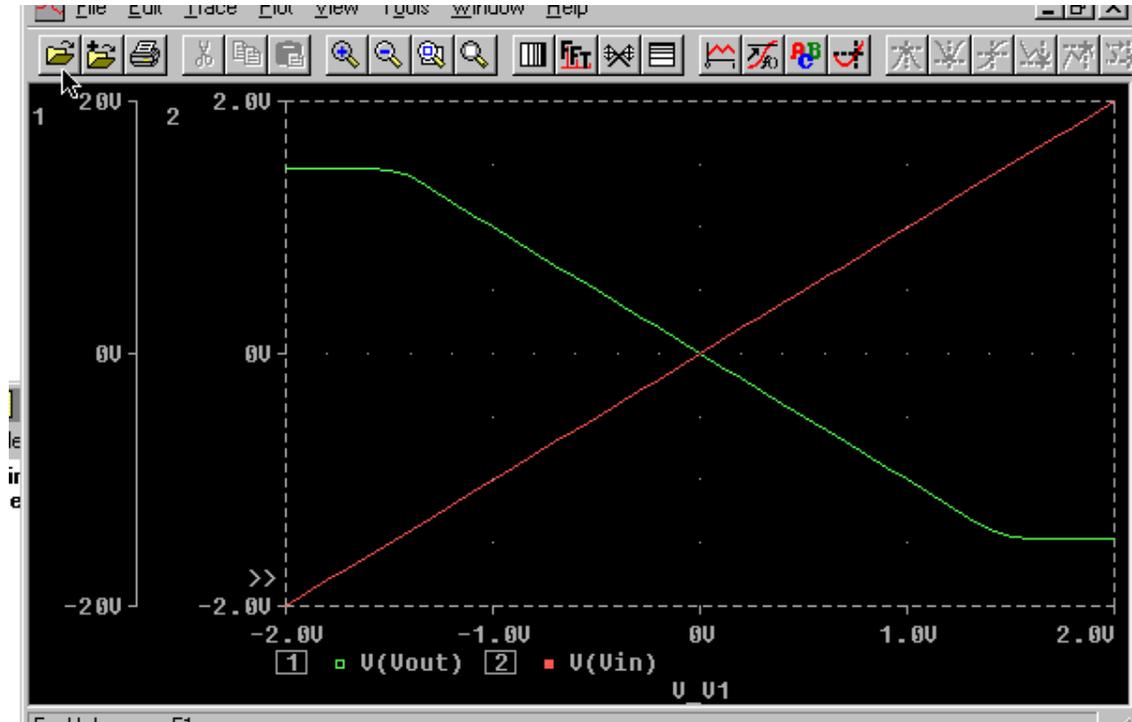


Figure 16. Plot of Vin and Vout Vs. Vin For Amplifier Circuit

As you can see PSpice and Probe provide a very useful tool to test the performance of nearly any circuit you can conceive BEFORE you actually build it. We will be making extensive use of it in this and future classes.