

MULTISIM DEMO 4.1: OPERATIONAL AMPLIFIERS IN MULTISIM

The Operational Amplifier is one of the more exciting components to work with in the study of circuits. The guts of an op amp are rather complicated, but it seals up into a nice lumped component which we can use to build some pretty useful circuits. This demo will introduce the operational amplifier in Multisim.

To begin our study of op amps, we will build a classic op amp circuit...drum roll...the inverting amplifier. For an inverting amplifier, the ideal gain can be expressed as shown below:

$$A = -\frac{R_2}{R_1}$$

To spice (pun definitely intended) things up we will make the feedback resistor R_2 a variable resistor as shown in Fig. 4.1.1 below.

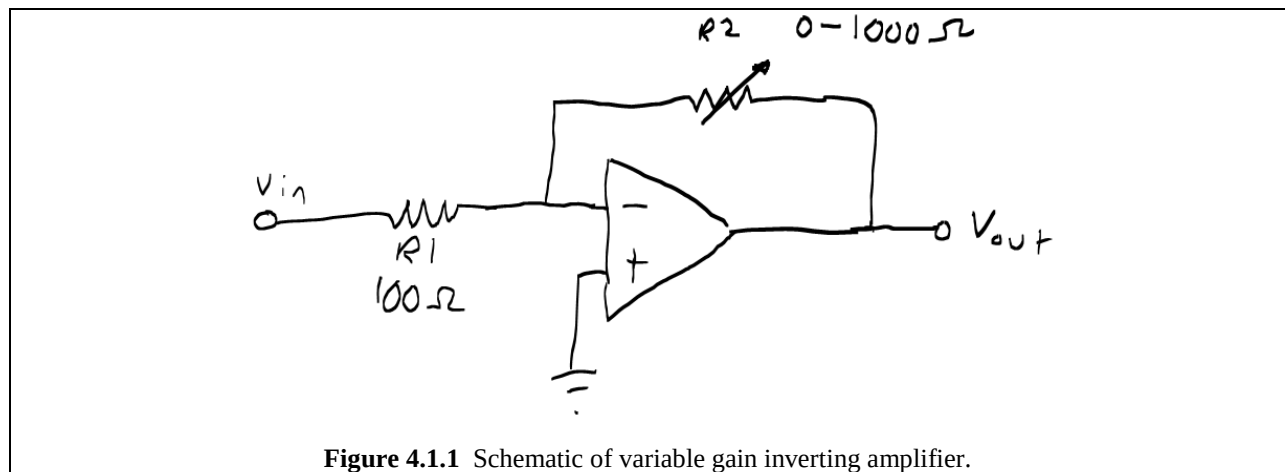


Figure 4.1.1 Schematic of variable gain inverting amplifier.

The inverting amplifier is traditionally drawn as it is shown in Figure 4.1.1 with the – terminal above the + terminal, however, for this Multisim demo, we'll keep the operational amplifier oriented in the manner that has the + terminal above the – terminal.

To obtain an op amp, go to the Select a Component window and set:
(see Fig. 4.1.2 on the next page)

Group: Analog
Family: ANALOG_VIRTUAL
Component: OPAMP_3T_BASIC

To obtain the potentiometer, go to the Select a Component window and set:

Group: Basic
Family: POTENTIOMETER
Component: 1k

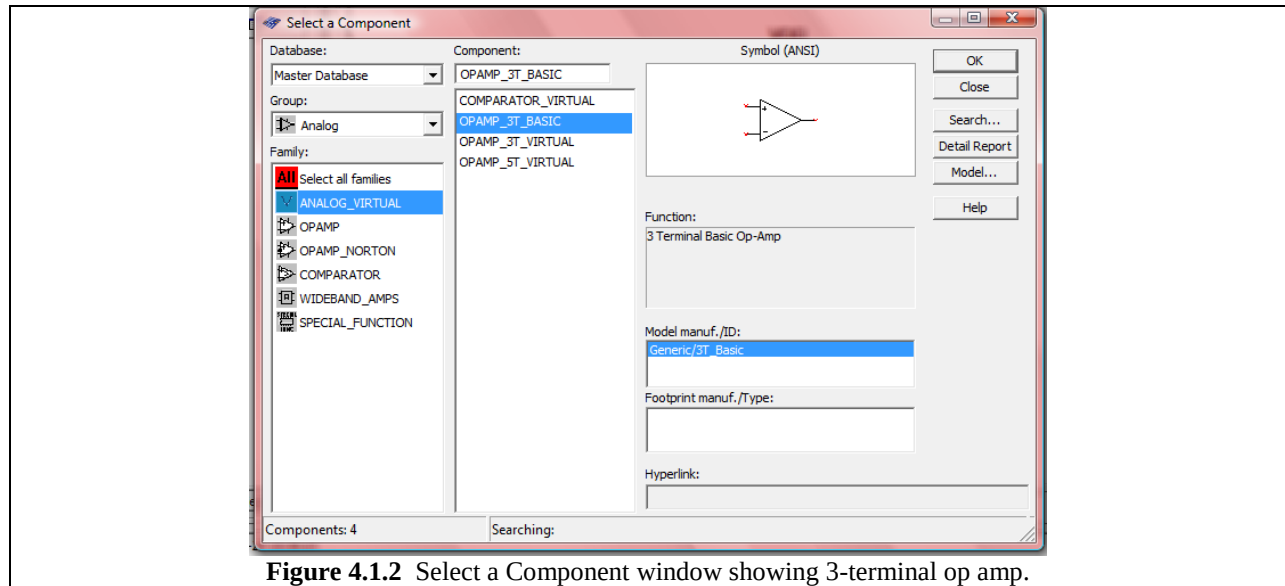


Figure 4.1.2 Select a Component window showing 3-terminal op amp.

Now let's build the circuit. You'll notice that when you place the 1 k Ω potentiometer, the arrow angles to the left. This indicates that the percentage value visible tells us the resistance between the top pin and the left pin of the potentiometer (see Fig. 4.1.3 below). Because of the orientation of the inverting amplifier, we will need to vertically flip the potentiometer so that it resembles the orientation shown in Figure 4.1.4 below. To do this you can right click on the potentiometer and click on Flip Horizontal, or simply highlight the potentiometer and utilize the key combination **Alt-x**. Once everything is put together your circuit should look like Fig. 4.1.4. Make sure your potentiometer is oriented correctly and that it is set to 50%!

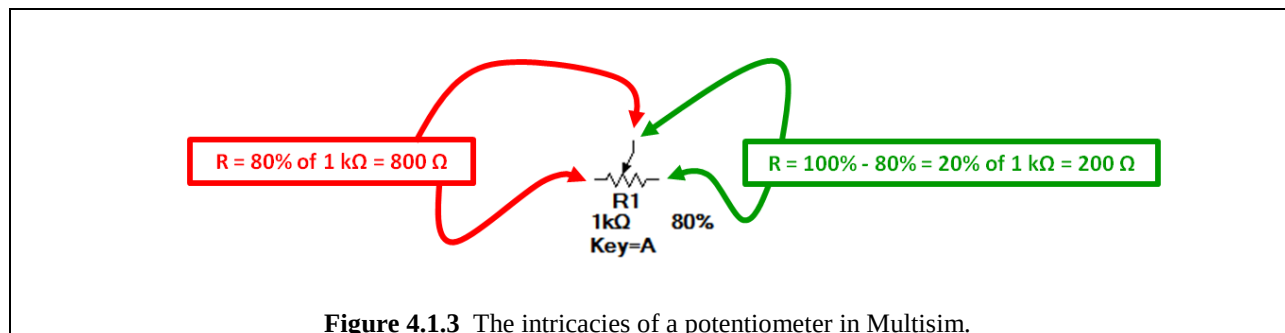
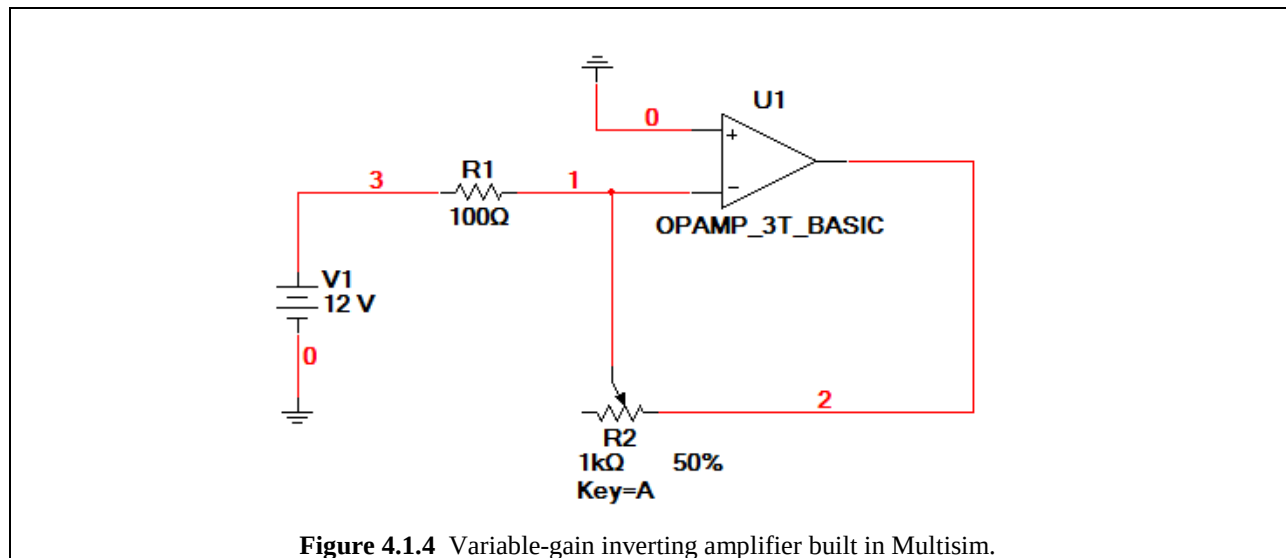


Figure 4.1.3 The intricacies of a potentiometer in Multisim.

*Note: **Alt-x** flips horizontally while **Alt-y** flips vertically. For “non-chiral” components like resistors, flipping the component on either axis does not actually change the component’s symbol...flipping it is the same as rotating it several times in the appropriate direction. However for “chiral” components such as an op amp, flipping will fundamentally change the symbol on the schematic, and no amount of rotating will ever re-orient the symbol back to how it originally was placed. Only flipping in the appropriate counter-direction will properly return it to its original form. For example, when you flipped the potentiometer above, you will never be able to rotate the potentiometer back to how it was originally. You will have to flip it again to cancel the original flip.*



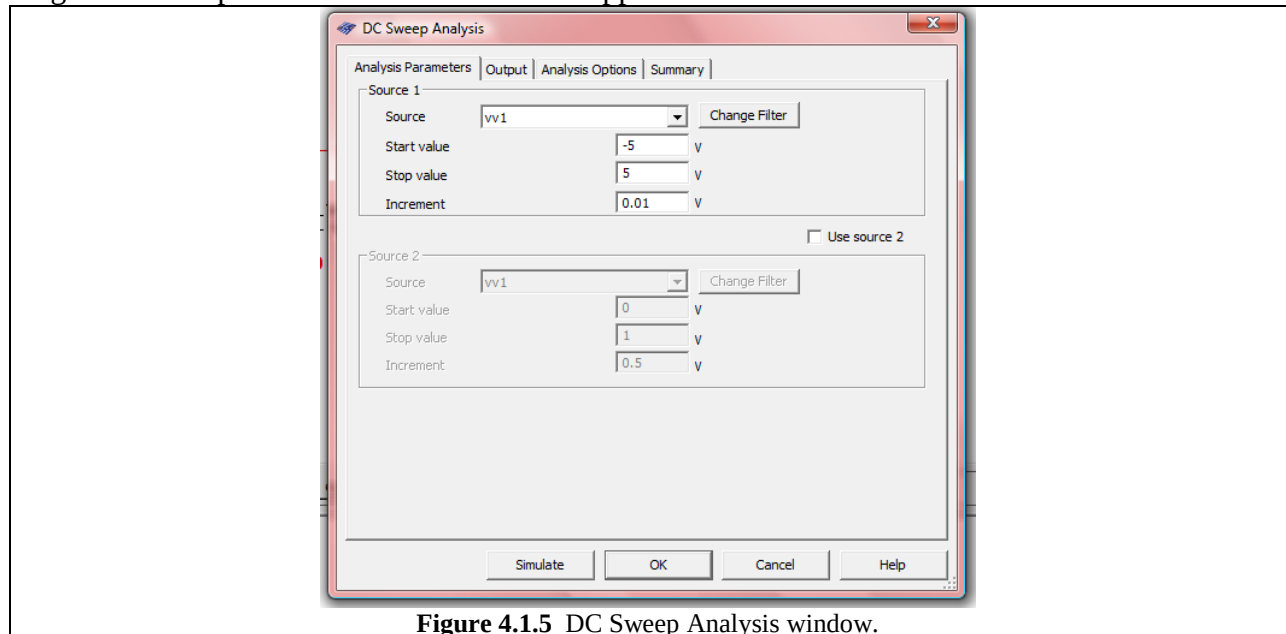
So now we want to see what the output voltage is for a given input voltage. To do this let's run the DC Sweep Analysis.

To access the DC Sweep Analysis, go to Simulate>Analyses>DC Sweep...

or

Click on the arrow portion of the  button and then under the drop-down menu select DC Sweep...

Figure 4.1.5 depicts the window which will appear.



Let's swing the input voltage from -5 V to + 5 V. To do this:

1. Under the Analysis Parameters tab, set Source to vv1.
2. Set Start Value to -5 V.
3. Set Stop Value to 5 V.
4. Set increment to 0.01 V.
5. Click on the Output tab, place V(2) in the Selected variables for analysis field, just like you would do for a DC Operating Point Analysis (see Fig. 4.1.6).
6. Under the Analysis Options tab, set the Title for Analysis to be "A = -5" (as shown in Fig. 4.1.7).
7. Press Simulate.

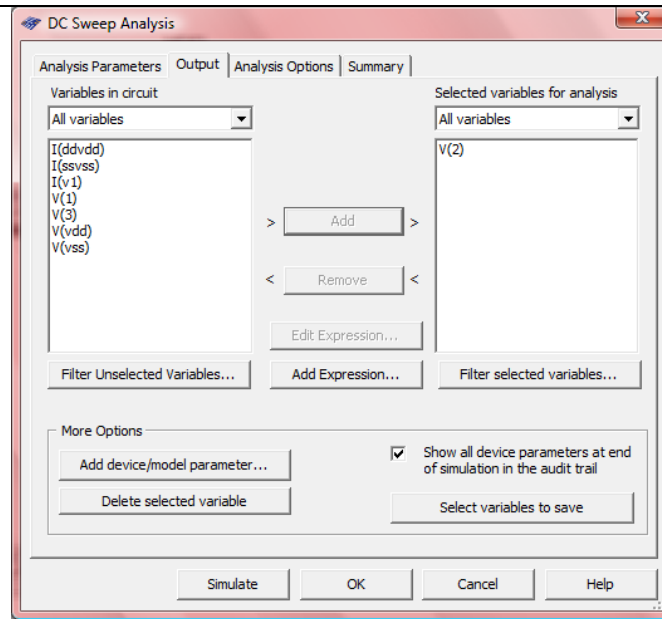


Figure 4.1.6 Output tab of DC Sweep Analysis window.

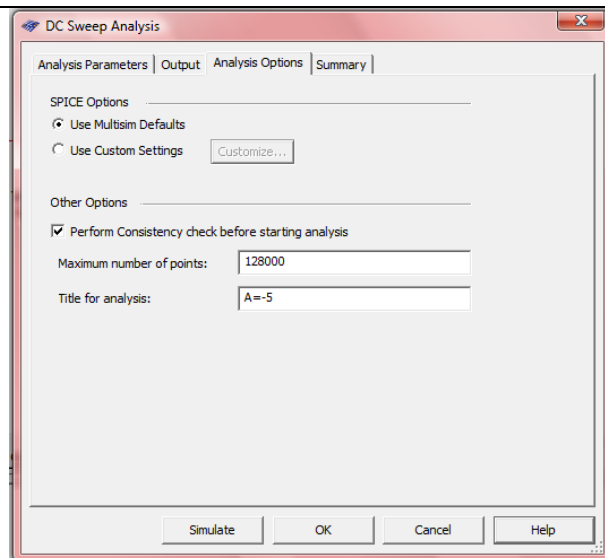


Figure 4.1.7 Titling the Analysis "A = -5".

A window should appear similar to that shown in Fig. 4.1.8 below. This window is commonly called the Grapher window. All Analyses that you will perform in Multisim display their results in the Grapher window.

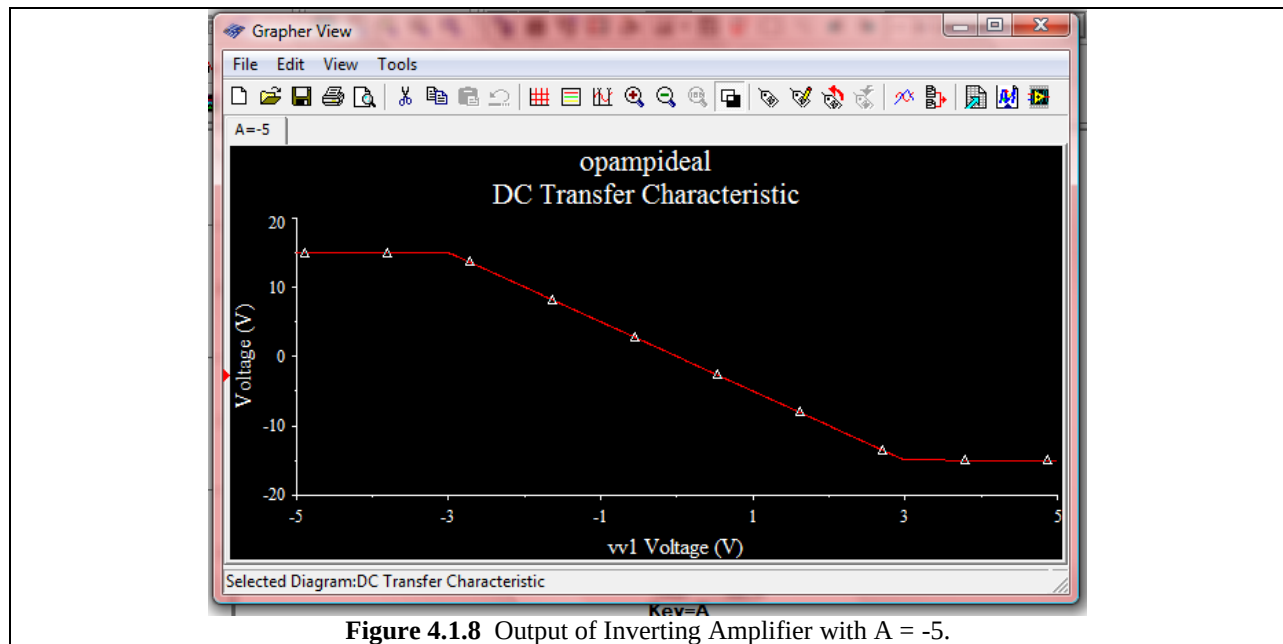


Figure 4.1.8 Output of Inverting Amplifier with $A = -5$.

Exit the Grapher window, and go back to the schematic. Adjust the potentiometer to 80%, meaning the resistance between the arrow and the right side of the potentiometer is $800\ \Omega$. What we've just done is increase the magnitude of the gain of the amplifier...it is now -8. Repeat the DC Sweep Analysis, but this time, set the Title for Analysis to be $A = -8$. Simulate. The result should be similar to that in Fig. 4.1.9 below.

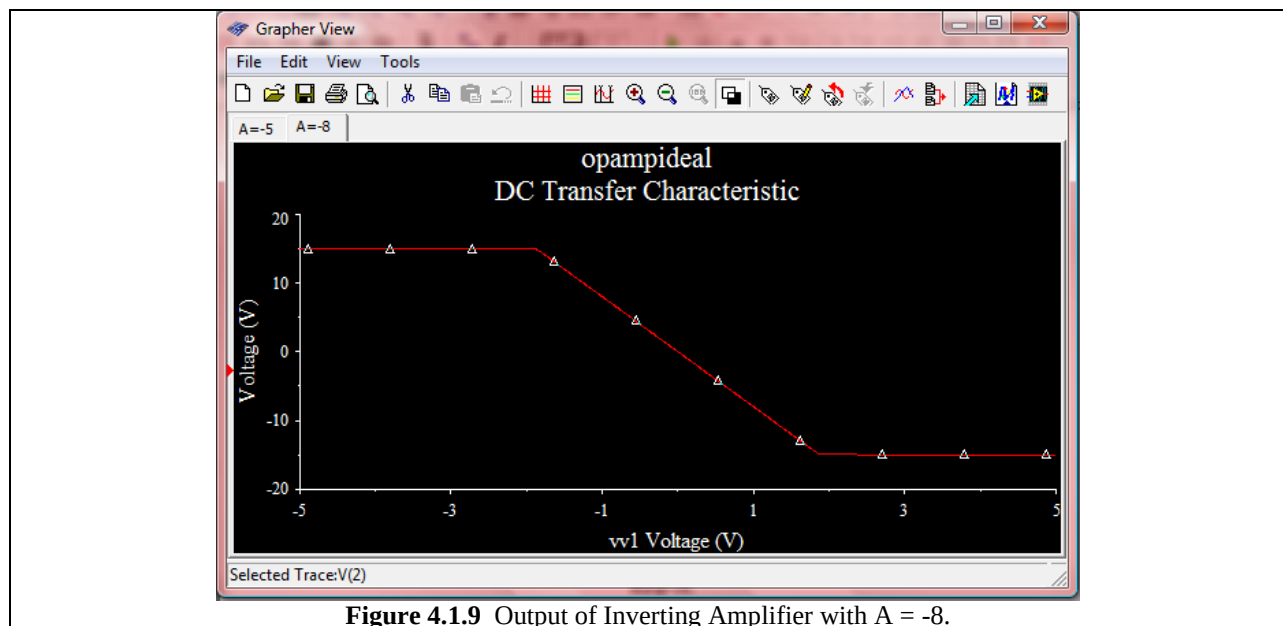


Figure 4.1.9 Output of Inverting Amplifier with $A = -8$.

So we now have in our possession two output-vs-input plots, one for a gain of -5 and one for a gain of -8. It would be really neat if we could put both of those plots on the same graph. Indeed we can. At the top of the Grapher window on the toolbar, click on the Overlay Traces button, . It will bring up the window shown in Fig. 4.1.10.

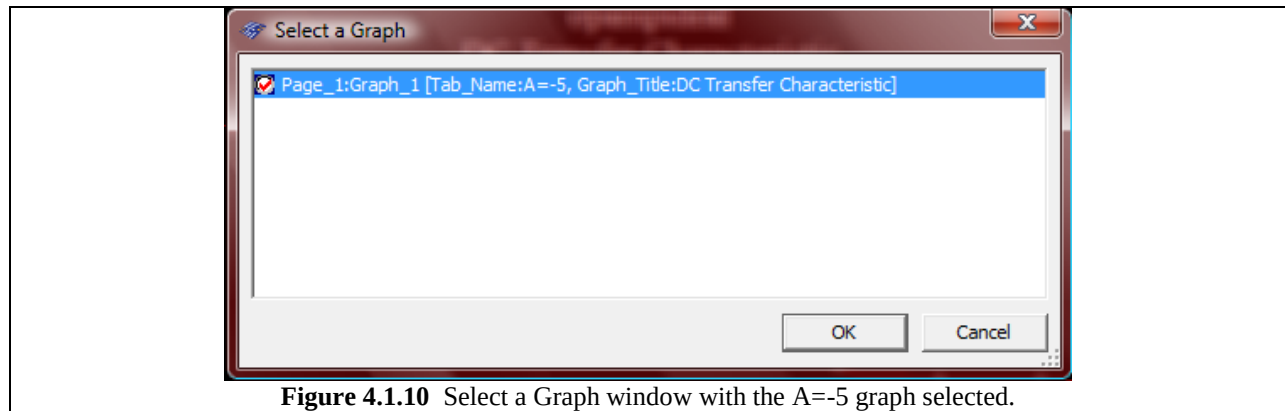


Figure 4.1.10 Select a Graph window with the A=-5 graph selected.

Note: Multisim stores previous Grapher plots, so if you have been doing a lot of simulations before you ran this simulation, your Select a Graph window may be full of a bunch of different selections. To get rid of all of them, on the Grapher window go to Edit>Clear Pages and then in the window which appears check all of the graphs you want throw out and Click OK. Make sure you don't throw out the Graph which has "A = -5" in it, however.

Select the graph (if there is more than one) which has "A = -5" somewhere in its title (see Fig. 4.1.10 for specific title) and then click on OK. The plot which results should resemble Fig. 4.1.11 shown below.

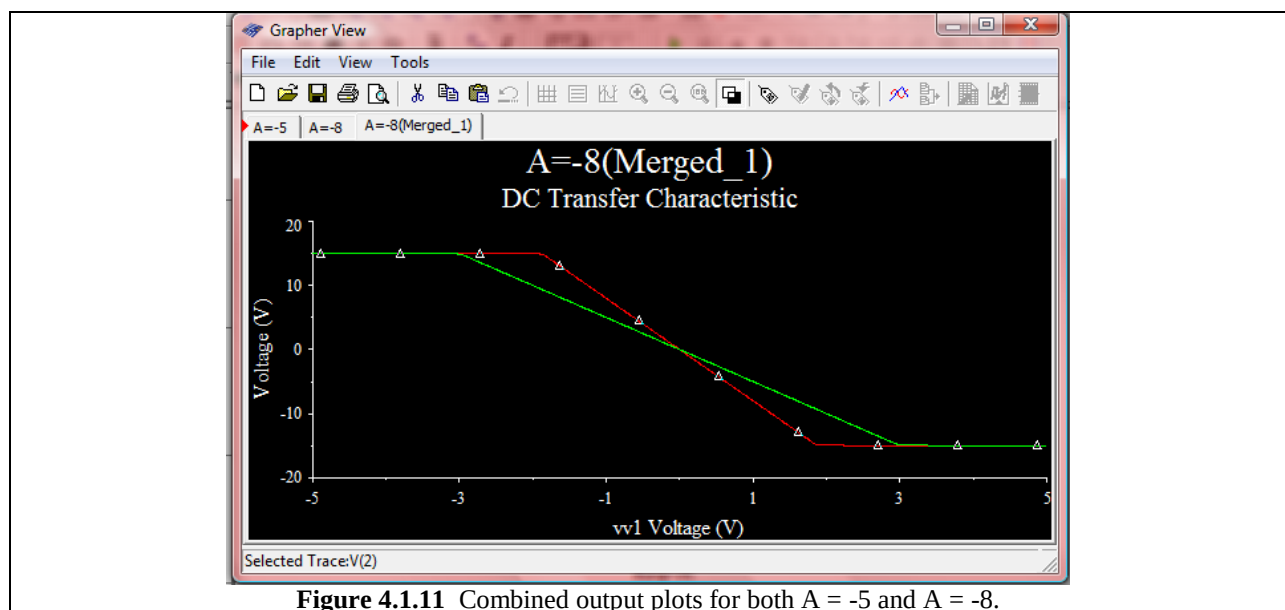


Figure 4.1.11 Combined output plots for both A = -5 and A = -8.

Figure 4.1.11 allows us to compare the input-vs-output plots of the inverting amplifier with two different gains. The steeper trace (red) corresponds to the gain of higher magnitude (-8). The reason the two plots seem to level out above and below a certain input voltage is because the 3-terminal op amp which we used of this circuit has a built-in output limit of ± 15 V. If the op amp had an unlimited output range, then the traces would be simply two diagonal lines but because of this limit, they level out at ± 15 V.

Making Pretty Graphs

Note: The parts below are optional, but helpful hints on how to make you have really nice graphs since by default, the graphs produced by Multisim are readable, but primarily only on the computer. If you have a need to use graphs produced by Multisim in a report, it will most likely be necessary to change the colors and things to make them more aesthetically pleasing.

First off, if we want to present this graph in homework or something, graphs with black backgrounds are totally hard to read and waste a lot of ink. To change this simply click on the Reverse Colors button, , found at the top of the Grapher tool bar. That automatically makes the background white and changes the text and scale to black.

Now let's click on the Page Properties button, , also located at the top of the page. This brings up the window shown in Fig. 4.1.12 below. Change the Tab Name to " $A = -8/A = -5$ " and change the Title to "Inverting Amplifier Gains." Observe the changes to the Grapher window and tabs.

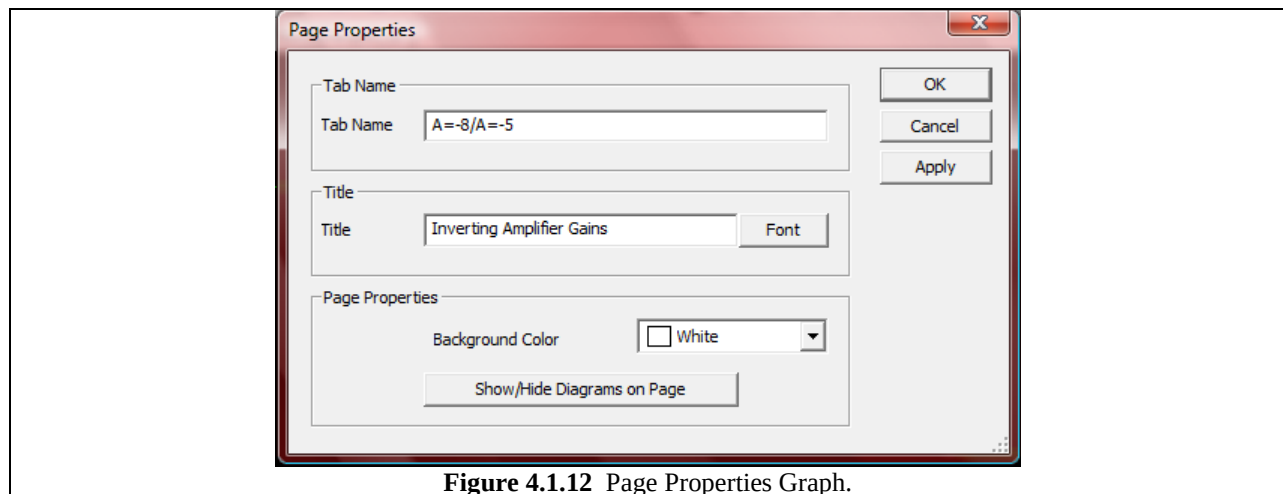


Figure 4.1.12 Page Properties Graph.

Now let's click on the Properties button, , located at the top of the page. This brings up the window shown in Fig. 4.1.13 on the next page. Under the Bottom Axis tab, change the Label to "Input Voltage (V)", and under the Left Axis tab, change the Label to "Output Voltage (V)." Additionally, under the Traces tab, you can alter the properties of the traces. Use the Trace field with the up and down arrow to select a specific trace to alter. For the green trace (Trace 2), change the label to " $A = -5$ " and increase Pen Size to 4 (see Fig. 4.1.14 on the next page). For the red trace (Trace 1), change the label to " $A = -8$ ", and increase the Pen Size to 4.

While still in the Graph Properties window, under the General tab, set Grid to on. Click Apply and then Click OK.

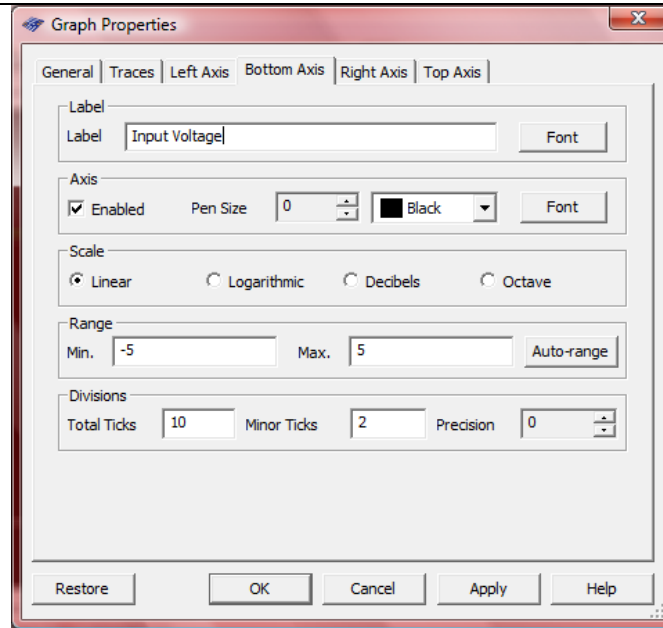


Figure 4.1.13 Renaming the Bottom Axis in the Graph Properties window.

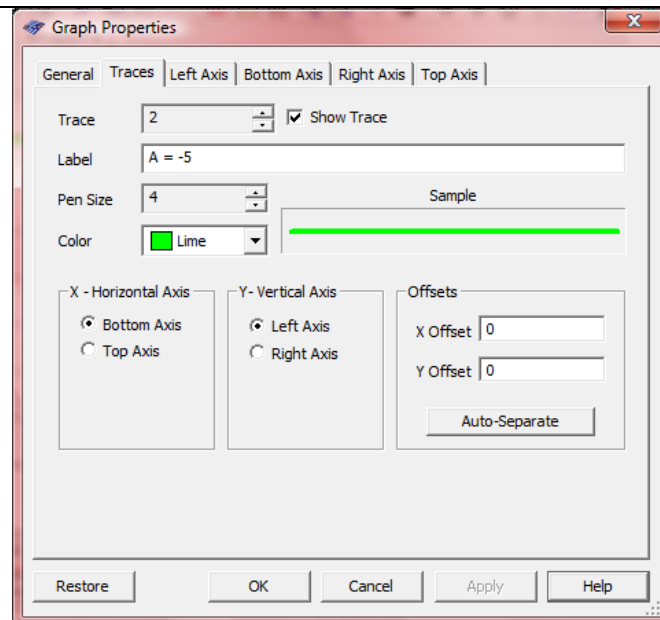


Figure 4.1.14 Adjusting the properties of Trace 2.

Before looking at your beautiful graph, do one more thing: Click on the Show/Hide legend button, , at the top of the toolbar. Now your graph should look like Fig. 4.1.15.

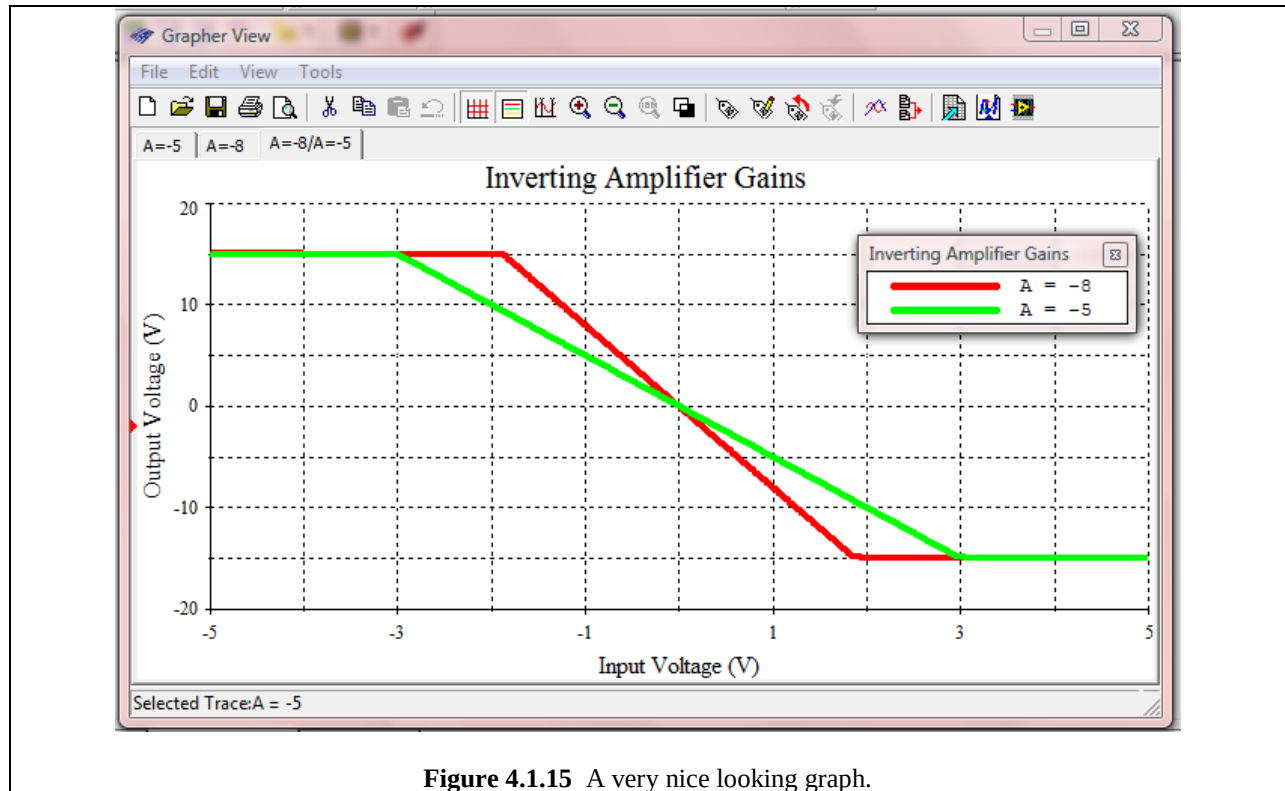


Figure 4.1.15 A very nice looking graph.

There are numerous other modifications you can make to your plots in order to turn them into beautiful, easily readable documents to convey knowledge, and these general rules which we have just gone over will apply to any Analysis which outputs signals into the Grapher View window.

If I were a teacher grading late at night after not having eaten for ten hours and I came across the graph shown in Figure 4.1.11, I would take off points because it is hard to read and poorly labeled. But if I were given the graph shown in Fig. 4.1.15, I would not take off points. In fact, I would probably give bonus points because it is so wonderful.

Nice graphs are one of the cornerstones of good engineering.