



## Electric Circuit Theory, Experiment 2: Feedback Amplifiers Using OP AMPS

By: Moe Wasserman  
College of Engineering  
Boston University  
Boston, Massachusetts

### Purpose:

To plot the voltage transfer function of simple feedback amplifiers, calculate the gain, and investigate the limits to linear operation.

### Equipment:

- Agilent 3631A Triple Power Supply
- Agilent 34401A Digital Multimeter
- Agilent 33120A Function Generator

### Introduction:

In each of the following experiments, you will "write" a program to instruct the instruments on the lab bench to perform prescribed measurements on your circuits. In some cases your program will analyze the data received by the instruments, and will present the results on the computer screen in graphical and/or tabular form.

The reason why the word "write" is in quotation marks is that the programming language you will use, Agilent VEE, has a purely graphical, rather than text, interface. Conventional program statements will be replaced by objects, or icons on the computer screen, and the order of execution of the objects will be determined by "wires" that connect the icons.

**Before performing any of these experiments, you should read the introductory document, entitled Using Agilent VEE, then work through the four exercises that you will find there.**

The first step in programming any Agilent VEE-controlled experiment is to place on the screen an **Panel Driver** object for each of the instruments to be used. (Recall that there is no instrument panel object for the triple power supply; you must use a **Direct I/O** object.) These objects will be used to set the default configurations for the instruments; they will not be connected by wires to anything else on the screen. Instructions for setting the defaults will be given in each experiment. The settings will be made on the Main Panel of each object unless another panel is designated. Only the necessary settings will be specified; all others are immaterial.

As is often the case, a program may not perform as expected on the first try. If this occurs, two useful debugging tools are available. They can be activated with the menu commands.

***Debug --> Show Data Flow*** and  
***Debug --> Show Execution Flow***.

To use these to best advantage, set the slowest speed for these processes with **File → Default Preferences** on the **General** tab, in the **Debug** group, click the **Data Flow Rate** arrow to show the drop-down list. Set the rate to **1(Slowest)**.

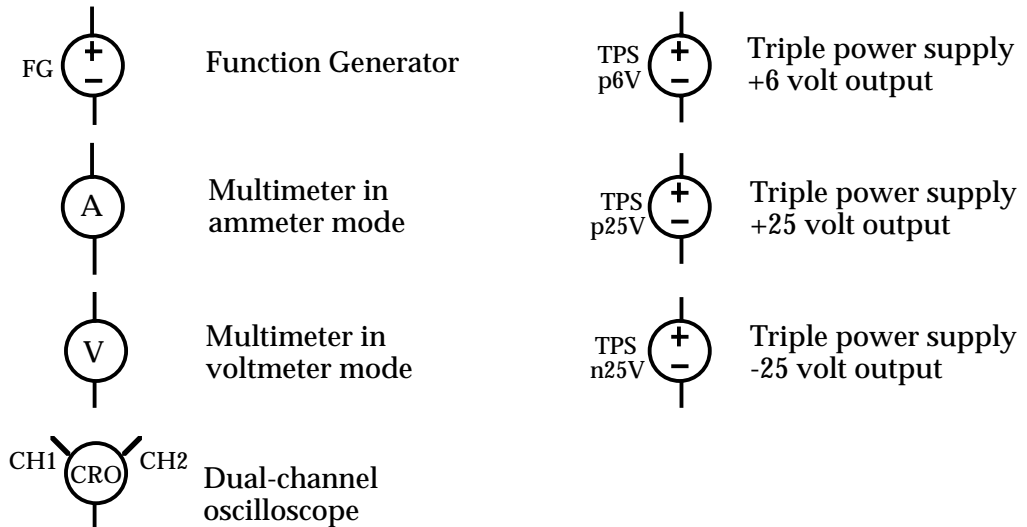
When the program is run, you will see symbols representing data and commands moving along the wires between objects, and each object will be highlighted when it is active.



Operating in this mode is also useful to assist you in understanding how Agilent VEE works. But don't use these options all the time, because they slow down execution considerably.

Always begin an experiment setup by clicking the **reset** button on each instrument panel object. If you don't, the instrument may not be in the proper operating mode when you run the experiment.

The following symbols will be used in the circuit schematics:



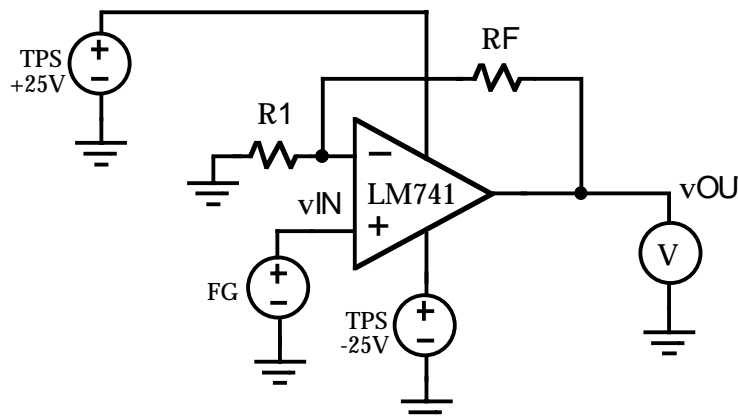
Instructions will be given in considerable detail for setting up and running the program for the first experiment only. It is assumed that you will then be familiar with the basic operations of Agilent VEE.

**Method:**

A stepped sequence of input voltages is applied to the amplifier input, and the corresponding output voltages are measured. A plot of  $v_{OUT}$  vs  $v_{IN}$  is generated. By varying the range of input voltages, the point at which saturation is reached can be identified. The voltage gain can be calculated and displayed.

**Hardware Setup:**

Construct the following circuit for a non-inverting feedback amplifier:





Select resistors whose values you determined in Experiment 1 for  $R_F$  and  $R_1$ , and calculate the expected voltage gain  $v_{OUT}/v_{IN}$  for these values from the equation

$$\frac{V_{OUT}}{V_{IN}} = 1 + \frac{R_F}{R_1}.$$

It is preferable that the two resistors differ by at least a factor of 10. Values in the range between 100  $\Omega$  and 100 k $\Omega$  are best.

### Software Setup:

Place Panel Drivers and Component Drivers for the function generator and multimeter on the screen, and a Direct I/O object for the triple-output power supply. Use the following default settings after resetting the instruments:

Function generator: Function = dc; Units = V peak-peak; Load = infinity  
Multimeter: Function = DC volts; Auto range = on

The transactions to be entered in the Direct I/O object are:

```
WRITE TEXT "output on"  
WRITE TEXT "appl p25v,15,0.1"  
WRITE TEXT "output:track on"
```

These commands set the positive 25-V supply to 15 volts, with a current limit of 100 mA, and turn on tracking mode so that the output of the negative 25-V supply has the same magnitude. All valid commands for this instrument, as well as for the other instruments on your bench, will be found in the User's Manual for each instrument. We use a Direct I/O driver here because no driver is available for the triple power supply. But one could be used for any of the instruments on the bench. They respond faster than the other kinds of driver, and they can issue commands that are not accessible from panel drivers. These advantages come at the cost of needing access to the commands for each instrument and entering them in the appropriate sequence.

The component driver for the function generator requires both an input terminal and an output terminal set to **offset**. The multimeter driver requires a **readings** output terminal.

A convenient way to set and control the input voltage range is to use a For Range object with both a **from** input terminal and a **Thru** input terminal added. Connect the **Thru** input terminal to the output pin of a **Real Knob** or a **Real Slider**, whichever you prefer (try both). These will be found under **Data --> Continuous**. The knob's output pin is also connected to a **Formula** object that contains the statement **"-a"** (no quotes) to change the sign of the knob's output. The negated output goes to the **from** input pin of the **For Range** object. When the limits of the knob range (zero to some positive value) are set in the two upper boxes, and the knob is turned to a value in this range, this value becomes the magnitude of the **from** and **Thru** values in the For Range object. The **step** value is to be set by you. Connect the output of the For Range to the input terminal of the function generator.

The two Execution condition for a Knob or Slider is **Wait for Input**. This means that when a new value is selected on one of these objects, the program will not run until a **start** command is given. An alternative is **Auto Execute (Properties Box)** (double click header), in the **Execution** group, select **Auto Execute**). This will start the program as soon as a new value is selected.

The final object is an **X vs Y Display**. Connect its x-input to the function generator output and its y-input to the multimeter output. The final connection is from the sequence output pin of the function generator to the sequence input pin of the multimeter.



As before, you may add an optional **Start** button, connected to the sequence input pin of the knob.

Figure 2-1 shows a screen layout for this experiment. You should sketch your arrangement in your notebook.

**Procedure:**

Before running the program, predict the range of input voltages over which the  $v_{OUT}$  vs  $v_{IN}$  plot will be linear.

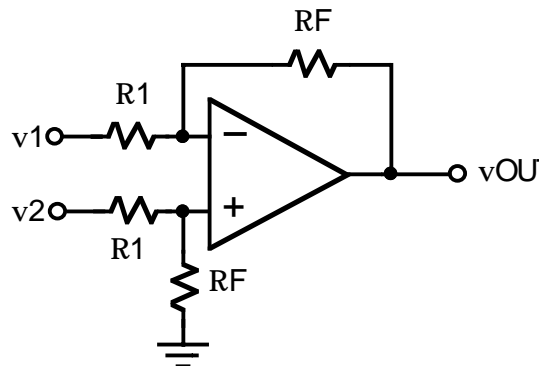
Set the lower limit (left box) of the knob to zero and the upper limit (right box) to a value within your predicted linear range. Use a Step value that will produce a sufficient number of points without requiring too much execution time. Open the XY plot object and run the program. Click **Auto Range** to keep the plot on scale. Did you obtain the expected result?

Now extend the input limit beyond the expected linear range and repeat the experiment. Did you obtain the expected result?

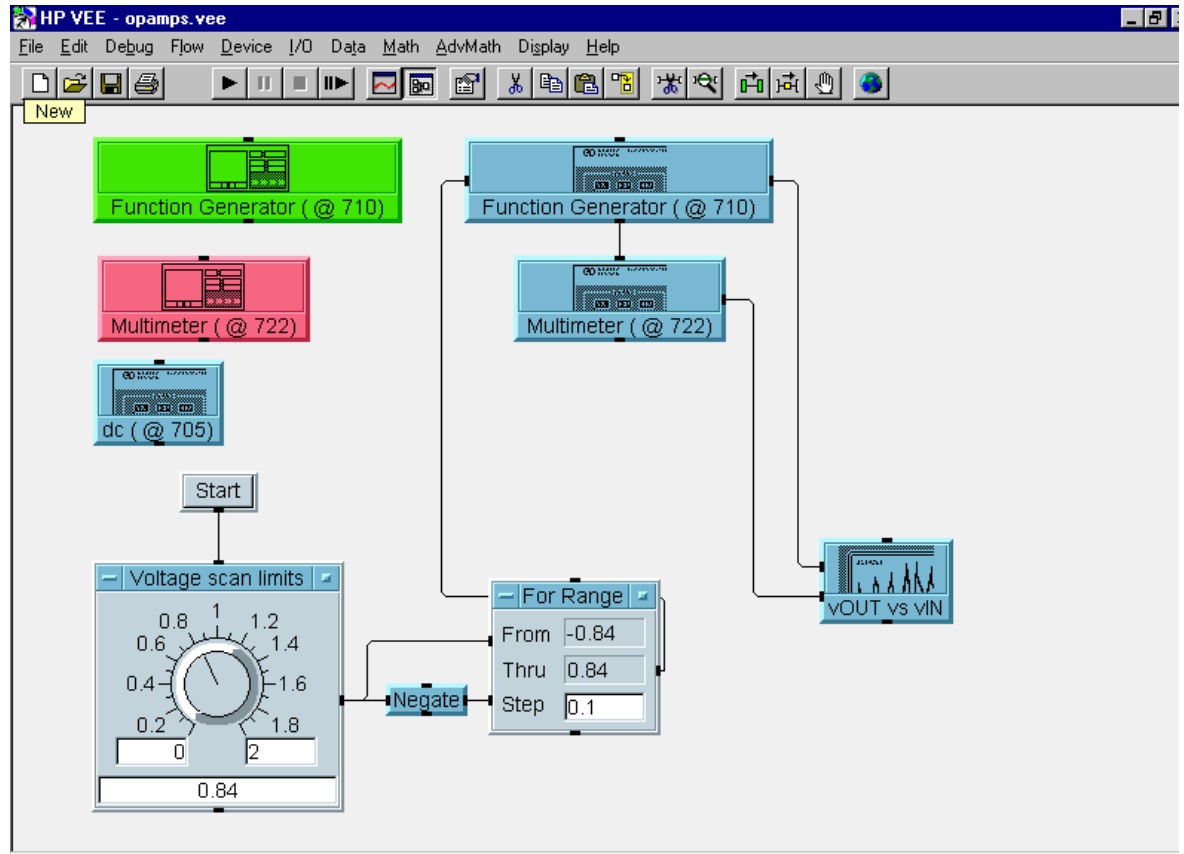
Change your non-inverting amplifier to an inverting configuration, and repeat the experiments. Is it necessary to change your software?

Use your experience from Experiment 1 to add provisions for calculating and displaying the voltage gain of your amplifiers.

If you have time, expand your program's capability so that it can measure the gain of a differential amplifier with this configuration:



$$v_{OUT} = \frac{R_F}{R_1} (v_2 - v_1)$$



**Fig. 2-1 Agilent VEE Layout for the Op Amp Experiment**