

EEEN203 Circuit Analysis Laboratory

Measuring the Frequency Response of an Amplifier

A frequency response is a measurement of the gain and phase of a system as a function of frequency. It allows us to simultaneously assess how different sinusoidal signals will be amplified (or attenuated) by a system, as well as how the output signal will be delayed. Measurement of the frequency response is one of the most common laboratory procedures used by electronic engineers.

In this laboratory we will measure the frequency response of a non-inverting amplifier circuit built using an operational amplifier. While learning how to measure a frequency response is important, the “hidden” purpose of this lab is to develop your skills in using the laboratory equipment. You are encouraged to experiment with the equipment and learn what they can do.

A frequency response is a pair of plots, one of which shows the gain of the system (in decibels) as we change the frequency, and the other which shows the phase vs frequency. We use a logarithmic frequency scale so that we can see the system’s behaviour over a wide frequency range. To perform this experiment we choose a set of measurement frequencies. At each frequency we then use an oscilloscope to measure both the gain and phase of the output relative to the input. Many people find measurement of a gain response (ie just gain vs frequency) to be quite intuitive, but the phase response a little trickier. However, you should always include the phase plot because there are many pieces of information that are better extracted from the phase plot.

Experimental Procedure

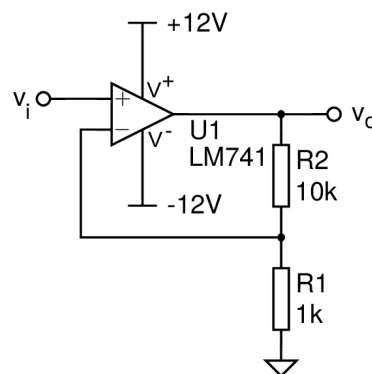


Figure 1: A noninverting operational amplifier circuit.

1. Construct the non-inverting amplifier circuit shown in figure 1 on a breadboard. You should consult a datasheet for a LM741 operational amplifier to find which pins on the package correspond to which electronic function.

You may recall that this circuit has a dc gain of $G = 1 + \frac{R2}{R1}$. Before you insert R1 and R2 into your circuit, you should measure their resistance values with the bench multimeter so that and calculate the expected gain of your amplifier.

2. Configure the power supply to produce two output voltages at ± 12 V and use a multimeter to ensure that you are getting the expected voltages.

Connect these voltages to the amplifier's power rails, as indicated in figure 1 and press the on/off button on the right of the power supply to connect the output voltages to your circuit. Note that you should use this button to turn off the power supply if you want to work on your circuit; there is no need to turn off the power supply with its main switch.

Tool-Tip: Configuring a Dual Power Supply

The GPE-3323 dual power supply can be considered as two adjustable batteries. If you put it into series mode (using the selector buttons between the voltage and current control dials) then the two outputs will have the same voltage, which is controlled by the dials for channel 1. The voltage you set on the dial will appear between the – and + terminals of both channel 1 and channel 2 of the supply. You will want to use the leftmost output (CH1) for +12 V and the CH2 for -12V. The common output is then halfway between the +12 V and -12 V outputs. There are markings below the terminals that show the appropriate connection points, though these are easy to miss.

Importantly, note that the common is **not** connected to ground (the green terminal marked GND on the power supply) by default. You can make this connection with a wire if you wish, which you should for this experiment.

3. Connect the function generator at the circuit's input and configure it to produce a 1.0 kHz sinusoidal signal with a peak-to-peak output voltage of 100 mV.

Tool-Tip: Configuring the Function Generator

To configure the function generator first press the **Sine** button. You can then press **Frequency/Period** to adjust the output to 1 kHz (either using the numerical keypad or the large adjustment dial). Units (kHz in this case) can be entered using the buttons to the right of the screen. Similarly you can then press the **Amplitude/High** button to configure the amplitude of your desired sinusoidal signal.

Tool-Tip: Setting the Function Generator Load Impedance

There is unfortunately a catch with setting the amplitude on these function generators, because by default the generators expect that the load is 50 Ω . This is not true in our case, so we need to tell the function generator that we are connecting it to a high impedance node in our circuit. To do this, start by pressing **More** in the function menu, and then use the buttons by the screen to follow the commands as follows.

More → **Output Menu** → **Load Impedance** → **High Z**

When you are done you will need to again press **Sine** to return to your desired mode. Notice that the amplitude reported by the function generator will now have changed.

4. Connect two oscilloscope probes to the input and output of your amplifier and ensure you get what you expect at the output.

Tool-Tip: Calibrating the Oscilloscope Probes

Before using the oscilloscope you should always calibrate the oscilloscope probes, particularly if you are planning to take measurements at high frequencies (say 100kHz or above as a rule of thumb). First ensure that the slider switches on the probes are in the x10 position and then connect the probe and its ground pigtail to the test points adjacent to the channel one connector (see figure 2). You should be able to see a nominal 1 kHz, 5 V square wave on the corresponding channel, though it may not look very square. Use a small screwdriver to adjust the trimmer in the body of the probe (the part that connects to the scope, see figure 3) until the waveform is as square as possible. Repeat for each scope probe.



Figure 2: Connection of a probe for calibration. Figure 3: Adjustment of probe calibration.

Make sure that the oscilloscope is set for x10 inputs, otherwise its measurements of amplitude will be wrong by a factor of ten. To do this press the yellow button to select the settings for channel 1 (for example). You will hopefully see **Probe x10 Voltage**, but if it says something other than x10, then press the adjacent button and press the **Attenuation** button until it is set to 10x.

Note that this calibration is specific to each input, so once it is calibrated don't swap the probes to another channel, or another scope. If you do you will need to recalibrate.

5. Measure the frequency response of the amplifier, as detailed in the procedure below. Make sure you read through the tips section before you start, as it may save you time.
6. Change R2 to 100 k Ω and repeat the frequency response measurement. You should plot the frequency response on the same graph as the case with R2 = 10 k Ω so that the comparison is clear.

Tool-Tip: Using the Oscilloscope Measurement Functions

The scopes you are using have the facility to measure phase automatically. You can access this through the **Measure** functions, so try using this, along with automatic measurements of voltage, while you draw the second frequency response. You can configure what measurements the oscilloscope displays by pressing **Measure** and then using the buttons to the right of each measurement to adjust what is to be displayed.

Not all oscilloscopes have this facility, so you must know how to measure phase (and gain) without relying on the oscilloscope to tell you directly.

Submission:

You should prepare a brief submission that includes your table of results and your frequency response plots.

Measuring the frequency response

1. Choose a test frequency.
2. Choose an input amplitude that produces a clean, measurable output from the system. You should typically make this as large as is practical to minimise the effects of noise and interference.
3. Use the oscilloscope to measure the amplitudes of both the input and output voltages. You might find it easier if you set the channel to ac coupling, because that should make the sinusoidal signals symmetric around the time axis, which makes everything easier to set up.
4. Calculate the period corresponding to the input frequency (remember $T=1/f$).
5. Measure the time difference where the input crosses zero, and the output crosses zero *in the same direction*. For example, you might measure when each waveform moves from negative to positive voltage. If the output crosses zero after the input, then this is a positive time delay. If it is before then it is a negative delay (which corresponds to a phase lead).
6. Calculate the phase delay in degrees.
We need to convert the time delay into a phase delay. To do this, we can “calibrate” using the period of the input signal. We know that the input signal moves through 360° during one period. So, one degree of phase delay is equivalent to $T/360$. Therefore, the phase delay is given by

$$\phi = 360^\circ \frac{t_{\text{delay}}}{T}$$

7. Plot the gain and phase in your frequency response plots.
8. Return to step 1 at a new test frequency.

You should record your data in a table that look something like the following. Note that the data in the example table is *not* what you should expect from your experiment!

Frequency	$ V_i $	$ V_o $	Gain	Gain	Period, T	t_{delay}	Phase delay
[Hz]	[V]	[V]		[dB]	[s]	[s]	[degrees]
1	1	0.5	0.5	-6.0	1	0.1	36
10	0.5	10×10^{-3}	20×10^{-3}	-34	0.1	0.04	144
20							
50							
100							
:							

Use Matlab to produce your frequency response plots. A basic skeleton to do this as follows, though you will want to add additional features such as axis labels, units and so forth.

```
freq = [1, 10, 20, 50];           % [Hz] Replace this with your data.
period = 1./freq;

Vi = [1, 1, 1, 1];               % [V] Replace this with your data.
Vo = [0.5, 0.5, 0.25, 0.1];      % [V] Replace this with your data.
t_delay = [1, 2, 3, 4];          % [s] Replace this with your data.

gain = Vo ./ Vi;
gain_dB = 20*log10(gain)         % Convert to dB

phase = t_delay ./ period .* 360; % [degrees]

subplot(211)
semilogx(freq, gain_dB)
subplot(212)
semilogx(freq, phase)
```

Tips:

Look before you leap

Begin by quickly sweeping through various frequencies to get a sense of how the system behaves. This will help you choose the range of frequencies that you need to worry about, but also give you a general sense of the response that you are expecting. This is useful as a first step, as it will help you avoid errors when doing the detailed measurements. To do this rapidly just see what happens to the output as you switch the test frequency through successive decades of frequency, for example, 1Hz, 10 Hz, 100 Hz, 1 kHz etc.

As a follow up to this you might as well then collect data at each of these factors of 10. This enables you to quickly sketch an outline of your frequency response curves.

Choose frequencies intelligently

Be intelligent about the frequencies that you use for test signals. We often use a 1-2-5 series for test frequencies, as this gives a reasonably even distribution on the logarithmic frequency axis. That is, we might test at 1 Hz, 2 Hz, 5 Hz, 10 Hz, 20 Hz, 50 Hz, 100 Hz etc. However, in many systems we find large frequency ranges in which both gain and phase are constant, so there is not a lot of reason to take many measurements. You might choose to use a 1-3 series (ie, 1, 3, 10, 30, 100, 300, ...) in these regions to save ourselves some time. Conversely there are frequency ranges when interesting things happen and we will then want to take more measurements. For example, when the gain drops 6 dB from a passband, there is a peak or trough in the gain, or the phase passes through multiples of 45° are all good places to capture additional data.

Plot as you go

Don't wait to collect all of your data before producing your plots. One reason for this is that immediate plotting will catch mistakes when the gain or phase suddenly changes in a way that you didn't expect. A second benefit of this approach is that it will help you choose appropriate frequencies at which to test.

This is easy to do in practice. If you plot your results using a tool like Matlab or a spreadsheet, it is straightforward to and just enter the data as you go and keep your plots up to date.

Alter the input amplitude

When you measure the gain of a linear system, it doesn't matter how large an input signal you use. (That is, if you double the size of the input, then the output amplitude should also double.) There is also nothing that requires that you use the same size input for every frequency. You can always choose an input magnitude that ensures that the output is a clean sinusoid, or at least as clean as you can make it. So, for example, if the output is very small and swamped by noise or 50 Hz interference, you might choose to increase the amplitude of the input to ensure that the output can be seen more easily. Conversely, you might find that the output is too large, so becomes distorted or clipped. In this case you can reduce the amplitude of the input.

Alter input dc offset

Similarly you might find that the output becomes asymmetric or clips at just one extreme (either positive or negative output). In many cases this effect can be reduced by applying a small dc component to the input sinusoidal signal. That is, we apply a test signal $v_i = u + A \cos \omega t$, where u is some constant voltage.

Check linearity

It is always a good idea to check your gain results for a few different input amplitudes. Find a frequency where the measurement is clean and then try increasing the input amplitude by ten, and then decreasing it by ten from the original level. If the system is linear then you should find the same gain in each case. If you don't then this is something you want to know!