

## Real Analog Chapter 11: Frequency Response and Filtering

1300 Henley Court  
Pullman, WA 99163  
509.334.6306  
www.store.digilent.com

### 11 Introduction and Chapter Objectives

In section 10.6, we saw that a system's frequency response provided a steady-state input-output relationship for a system, as a function of frequency. We could apply this frequency response to the phasor representation of the input signal in order to determine the system's steady-state sinusoidal response – we simply evaluated the frequency response at the appropriate frequencies to determine the effect of the system on the input sinusoids. This approach had the potential for simplifying our analysis considerably, particularly for the case in which the input signal contained multiple sinusoids with different frequencies.

In Chapter 10, the signals we considered consisted only of individual sinusoids. It is more useful, however, in some ways to think in terms of the inputs and outputs of the system as functions of frequency, in the same way in which we considered the frequency response of the system to be a function of frequency in section 10.6. We can then perform our analysis of the system entirely in terms of the frequencies involved. This leads to the use of the system's frequency response directly as a design and analysis tool. In many cases, this means that the actual time-domain behavior of the system or signal is of limited interest (or in some cases, not considered at all). Some examples of frequency domain analyses are:

1. *Determining dominant sinusoidal frequency components in a measured signal.* Complex signals can often be represented as a superposition of several sinusoidal components with different frequencies. Identifying sinusoidal components with large amplitudes (the so-called dominant frequencies) can help with many design problems. One application of this is in the area of combustion instability – combustion processes in rocket engines can become unstable due to a variety of reasons, any of which can result in catastrophic failure of the engine. The type of instability which occurs is generally linked to a particular frequency; identification of the frequency of the pressure oscillations associated with the combustion instability is generally the first step in determining the cause of the instability.
2. *Designing systems to provide a desired frequency response.* Audio components in stereo systems are generally designed to produce a desired frequency response. A graphic equalizer, for example, can be used to boost (or amplify) some frequency ranges and attenuate other frequency ranges. When adjusting the settings on an equalizer, you are essentially directly adjusting the system's frequency response to provide a desired system response.

This chapter begins in section 11.1 with a brief review of frequency responses and an overview of the use of the frequency response in system analysis and design. In section 11.2, we discuss representation of signals in terms of their frequency content. At this time, we will also represent the frequency content of the input and output signals and the frequency response of the system in graphical format – this helps us visualize the frequency content of the signals and system. This leads us to think in terms of using a system to create a signal with a desired frequency content – this process is called filtering and is discussed in section 11.3. Using logarithmic scales to represent the signal and system frequency responses can – in many cases – simplify the analysis or design process; this format of

presentation is called a Bode plot, and they are very briefly introduced in section 11.4. We will discuss Bode plots in more depth in later chapters.

It is important to keep in mind that, when we are performing frequency domain analyses, we are restricting our attention to the steady-state sinusoidal response of the system. Frequency domain design and analysis methods are so pervasive that they are often used to infer the system's transient response and/or its response to non-sinusoidal signals, so it is sometimes possible to forget the origins and limitations of the original concepts!

**After completing this chapter, you should be able to:**

- Use the frequency response of a system to determine the frequency domain response of a system to a given input
- State from memory the definition of *signal spectrum*
- Create plots of given signal spectra
- Plot a circuit's magnitude and phase responses
- Check a circuit's amplitude response at low and high frequencies against the expected physical behavior of the circuit
- Graphically represent a system's frequency domain response from provided signal spectra plots and plots of the system's frequency response
- Identify low pass and high pass filters
- Calculate a system's cutoff frequency
- Determine the DC gain of an electrical circuit
- Write, from memory, the equation used to convert gains to decibel form
- Sketch straight-line amplitude approximations to Bode plots
- Sketch straight-line phase approximations to Bode plots

## 11.1 Introduction to Steady-state Sinusoidal Analysis

In section 10.6, we defined the frequency response  $H(j\omega)$  of a system as a complex function of frequency which describes the relationship between the steady state sinusoidal response of a system and the corresponding sinusoidal input. Thus, if a sinusoidal input with some frequency  $\omega_0$  is applied to a system with frequency response  $H(j\omega)$ , the amplitude of the output sinusoid is the input sinusoid's amplitude multiplied by the magnitude response of the system, evaluated at the frequency  $\omega_0$ . The phase angle of the output sinusoid is the sum of the input sinusoid's phase and the phase response of the system, evaluated at the frequency  $\omega_0$ . The overall idea is presented in block diagram form in Fig. 11.1 below.

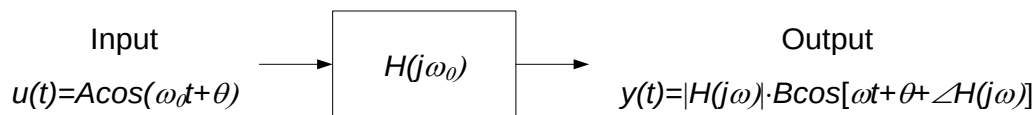


Figure 11.1. Frequency response used to determine steady state sinusoidal system response.

The true power of the frequency response is, however, if we consider both the system's input and output phasors to be complex functions of frequency, in the same way that the frequency response is a complex function of frequency. In this case, the block diagram of Fig. 11.1 can be represented as shown in Fig. 11.2.

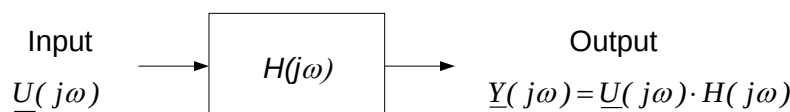


Figure 11.2. Frequency response used to determine system response as a function of frequency.

In Fig. 11.2, the output is determined by multiplying the phasor representation of the input by the system's frequency response. It is important to keep in mind that the arguments of this multiplication are complex functions of frequency – both the input and the frequency response at any frequency are complex numbers, so the output at any frequency is also a complex number. We typically use polar form to represent these complex numbers, so the amplitude of the output signal is the product of the amplitude of the input signal and the magnitude response of the system and the phase of the output signal is the sum of the phase of the input and the phase response of the system. Mathematically, these are expressed as:

$$|Y(j\omega)| = |U(j\omega)| \cdot |H(j\omega)| \quad \text{Eq. 11.1}$$

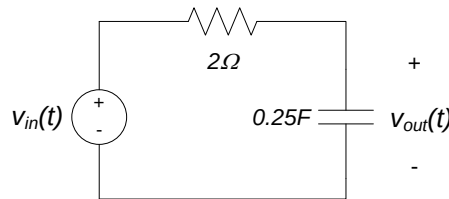
And:

$$\angle Y(j\omega) = \angle U(j\omega) + \angle H(j\omega) \quad \text{Eq. 11.2}$$

We now present two examples of the process defined by equations (11.1) and (11.2) above.

### Example 11.1

Determine the phasor representation for  $v_{out}(t)$  in the circuit shown below as a function of frequency, if the input voltage is  $v_{in}(t) = 3\cos(2t+20^\circ) + 7\cos(4t-60^\circ)$ . (Note: this problem is the same as that of Example 10.19 of chapter 10.6; the difference is primarily philosophical.)



The frequency response of this circuit, for arbitrary resistance and capacitance values, was determined in example 10.18 of chapter 10.6. For our specific resistor and capacitor values, this becomes:

$$H(j\omega) = \frac{1}{1 + j\omega RC} = \frac{1}{1 + j\omega(2\Omega)(0.25F)} = \frac{1}{1 + j\omega(0.5)} = \frac{2}{2 + j\omega}$$

We can represent the input as a piecewise function of frequency:

$$\underline{V}_{in} = \begin{cases} 3\angle 20^\circ, & \omega = 2\text{rad/sec} \\ 7\angle -60^\circ & \omega = 4\text{rad/sec} \\ 0, & \text{otherwise} \end{cases}$$

The input phasor is now considered to be a function of frequency, whose only nonzero components are at frequencies of 2 rad/sec and 4 rad/sec.

The phasor output is simply the product of the input phasor as a function of frequency and the frequency response. For frequencies, other than 2 rad/sec and 4 rad/sec, the input is zero and the frequency response is finite, so the output is zero. We determined the output phasor at frequencies of 2 and 4 rad/sec in example 10.19 in section 10.6; using those results allows us to write the output phasor directly as:

$$\underline{V}_{out} = \begin{cases} \frac{3}{\sqrt{2}} \angle -25^\circ, & \omega = 2 \text{ rad/sec} \\ \frac{7}{\sqrt{5}} \angle -123.4^\circ, & \omega = 4 \text{ rad/sec} \\ 0, & \text{otherwise} \end{cases}$$

### Example 11.2

The frequency response of a system,  $H(j\omega)$ , and the frequency domain input to the system,  $\underline{U}(j\omega)$ , are given below. The frequency response is dimensionless, the input has units of volts, and the units of frequency are rad/sec. Determine the system output  $\underline{Y}(j\omega)$ .

$$\underline{U}_{j\omega} = \begin{cases} j\omega, & 0 < \omega \leq 1 \\ j(2 - \omega), & 1 < \omega < 2 \\ 0, & \text{otherwise} \end{cases}$$

$$\underline{U}_{j\omega} = \begin{cases} 0, & 0 < \omega \leq 5 \\ 1, & 0.5 < \omega \leq 1.5 \\ 0, & \text{otherwise} \end{cases}$$

The system output is determined from a point-by-point multiplication of the input and the frequency response. In this case, both the input to the system and the system frequency response are defined as piecewise functions of frequency, so we must perform a piecewise multiplication to obtain the output. The output, for various ranges of frequency, is obtained below:

$0 < \omega \leq 0.5$ :

- In this range,  $H(j\omega) = 0$  and the input is finite, so  $\underline{Y}(j\omega) = 0$ .

$0.5 < \omega \leq 1.5$ :

- In this range,  $H(j\omega) = 1$  and  $\underline{U}(j\omega) = j\omega$ , so the output is:  $\underline{Y} = (1)(j\omega) = j\omega$ .

$1 < \omega \leq 1.5$ :

- In this range,  $H(j\omega) = 0$  and the input is finite, so  $\underline{Y}(j\omega) = 0$ .

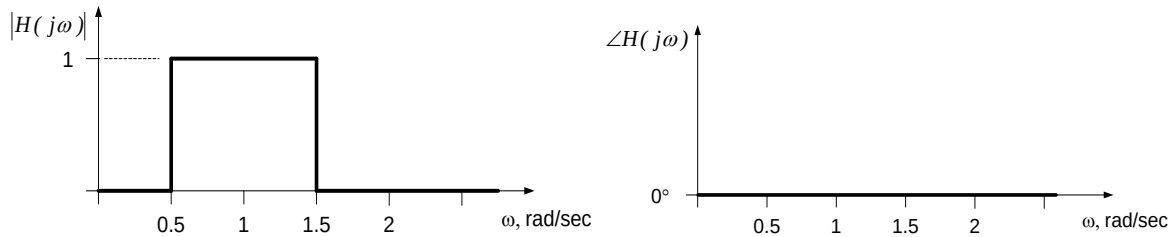
For any other value of frequency, both the input and the frequency response are zero, so the output is obviously zero.

The above results allow us to define the system output in a piecewise fashion as:

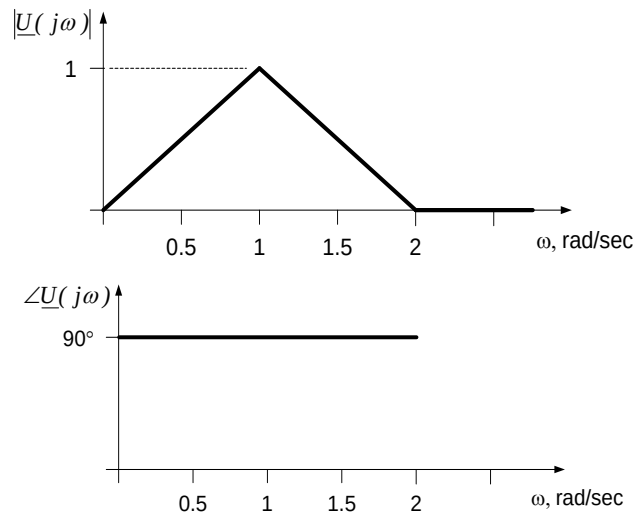
$$\underline{Y}_{j\omega} = \begin{cases} j\omega, & 0.5 < \omega \leq 1 \\ j(2 - \omega), & 1 < \omega \leq 1.5 \\ 0, & \text{otherwise} \end{cases}$$

### Example 11.3

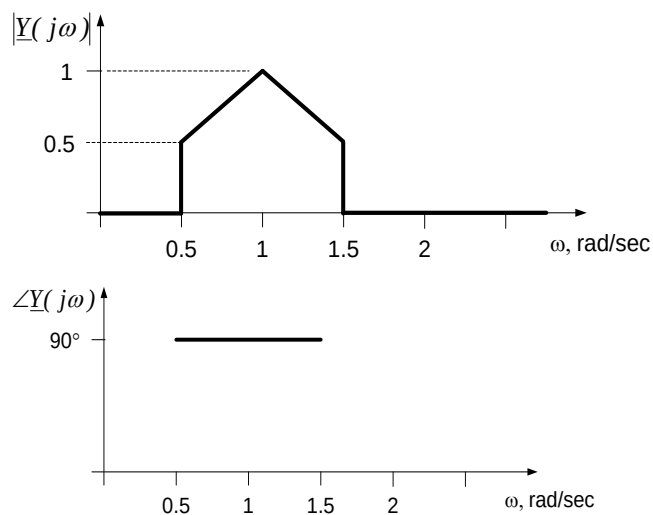
For the signals of example 11.2, plot: the magnitude and phase responses of the system and the magnitude and phase of both the input and output signals. The magnitude and phase response of the system are shown to the left and right below. The phase of the input is indicated as zero for all frequencies, since the input is real-valued for all frequencies. Strictly speaking, however, the phase is not well defined when the magnitude response is zero. (The phase angle provides the direction of a number from the origin of the complex plane, it is difficult to tell what direction “zero” is from itself!)



The magnitude and phase of the frequency domain input are shown below. The  $90^\circ$  phase shift over the range of 0 to 2 rad/sec is due to the factor of “j” in the frequency response. The phase is not indicated where the input amplitude is zero; this is again because the phase angle of “zero” is not well defined.



The magnitude and phase of the output are shown below. It is easy to see from the above figures that the magnitude of the output is the product of the input magnitude and the magnitude response and the phase of the output is the sum of the input phase and the phase response.



The above system is performing a potentially useful operation. Any sinusoidal signals with frequencies between 0.5 and 1.5 rad/sec are passed through the system (they appear at the output, unchanged). Sinusoids with frequencies outside this range are eliminated by the system – they are not present in the system’s output. This system is performing what is called a *band-pass operation*; frequencies within a certain frequency band are passed

through the system, while all other frequencies are stopped. This type of operation can be useful, for example, in communication systems – signals from different radio stations should not overlap or they will interfere with one another.

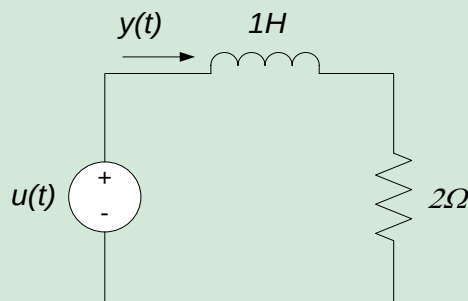
## Section Summary

A very powerful analysis and design techniques for linear systems consists of considering the input and output signals to be phasors which are functions of frequency, so that the amplitude and phase of the input and output signals are both defined at each value of frequency. If this viewpoint is taken,

- The amplitude of the output signal is the frequency-by-frequency product of the amplitude of the input signal and the magnitude response of the system, and
- The phase of the output signal is the frequency-by-frequency sum of the phase of the input and the phase response of the system.
- Calculate the frequency response for the circuit below, if  $u(t)$  is the input and  $y(t)$  is the output.

### 11.1 Exercises

1. Calculate the frequency response for the circuit below, if  $u(t)$  is the input and  $y(t)$  is the output.



2. The input to a system is  $u(t) = 3\cos(2t-60^\circ) + 4\cos(4t+30^\circ) + 7\cos(6t+45^\circ)$ . The system frequency response is given by the piecewise function:

$$H(j\omega) = \begin{cases} 0, & 0 < \omega < 3 \\ 1, & 3 \leq \omega \leq 5 \\ 0, & \omega > 5 \end{cases}$$

What is the system output,  $y(t)$ ?

### 11.2 Signal Spectra and Frequency Response Plots

In previous sections, we used the frequency response of a system to determine the system output, when the input to the system consists of signals comprised of one or more sinusoidal components. This analysis approach is extremely powerful, since it turns out that nearly any signal can be represented as a superposition of sinusoids<sup>1</sup>.

<sup>1</sup> Later in this textbook, we will use *Fourier Series* to represent arbitrary periodic signals and *Fourier Transforms* to represent non-periodic signals. Computer methods, based on these analysis approaches, are commonly used to determine the frequency components present in measured signals. A detailed discussion of these approaches is currently beyond our capabilities; at the moment, we simply ask that you believe that nearly any time domain signal can be represented to some extent in the frequency domain. For the immediate future, we will assume that the frequency domain representation of any signals of interest is directly available.

We will ultimately, therefore, use our frequency domain analysis approaches on a very broad range of input signals, many of which may have very little resemblance to sinusoidal signals.

Often, a graphical representation of the system's frequency response and the frequency content of the signals of interest can facilitate analysis and provide insight into the overall system behavior. We have seen an example of this in example 10.3. In this chapter, we introduce the concept of a signal's *spectrum* – the frequency content of a signal – and we will look more closely at the representation of frequency responses in graphical form. We will conclude this section with an example of the use of signal spectra and frequency response plots to obtain a qualitative representation of a system's frequency domain response to some input.

### 11.2.1 Signal Spectra

The signals currently of interest to us are sinusoidal. Any sinusoidal signal can be written in the form:

$$f(t) = A \cos(\omega_0 t + \theta) \quad \text{Eq. 11.3}$$

The signal is completely defined by its amplitude,  $A$ , its frequency,  $\omega_0$ , and its phase angle,  $\theta$ . Our primary interest in these signals is as inputs and outputs to systems. As indicated in section 10.1, we can characterize systems by their magnitude and phase responses as functions of frequency. To be consistent with this frame of mind, we will consider sinusoidal signals of the type shown in equation (11.3) to be functions of frequency as well. We will thus begin to consider the frequency,  $\omega$ , to be an independent variable, much in the same way that time was treated as an independent variable when we determined time domain responses of first and second order systems.

With frequency treated as an independent variable, the sinusoidal signal of equation (11.3) can be re-written in terms of  $\omega$ :

$$Fj\omega = \begin{cases} A \angle \theta, & \omega = \omega_0 \\ 0, & \text{otherwise} \end{cases} \quad \text{Eq. 11.4}$$

The frequency domain representation of a sinusoidal signal has two dependent variables: the amplitude and the phase. Our immediate goal is to represent the frequency content of signals graphically – we will, therefore, need to use two plots: amplitude as a function of frequency and phase as a function of frequency.

A signal's amplitude and phase as functions of frequency is called the *spectrum* of the signal. If we are provided with the spectrum of the signal, we have all the information necessary to completely define the signal. Signal spectra are often presented graphically in terms of two plots; we will refer to the plot of amplitude as a function of frequency as the *magnitude spectrum*, while the plot of phase as a function of frequency will be called the *phase spectrum*. As an example, the magnitude and phase spectra of the signal provided in equation (11.4) are shown in Fig. 11.3 below.

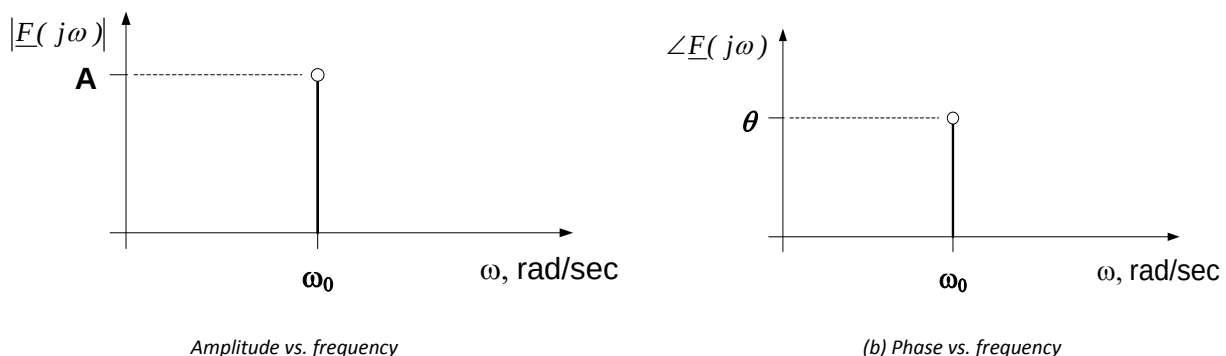
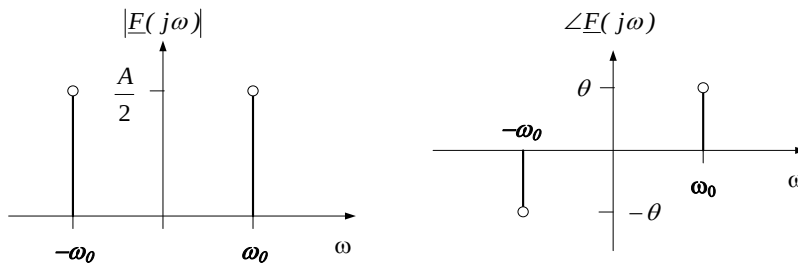


Figure 11.3. Spectrum for the sinusoidal signal of equation (11.4).

**Notes:**

- The term spectrum is most commonly applied to the plot of amplitude and phase vs. frequency of signals in complex exponential form. For example, the signal of equation (11.1) can be written in terms of complex exponentials as:  $f(t) = \frac{Ae^{j(\omega_0 t + \theta)} + Ae^{-j(\omega_0 t + \theta)}}{2} = \frac{A}{2}e^{j\theta}e^{j\omega_0 t} + \frac{A}{2}e^{-j\theta}e^{-j\omega_0 t}$ . The spectral plot of the signal in this form is shown below. Note that spectra in this form have both positive and negative frequencies. In our discussions in this chapter, we will present spectra only in terms of having positive frequencies.



- The time domain representation of a signal has only one dependent variable – the value of the signal at any time. Time domain representations are therefore represented graphically as a single plot with time on the horizontal axis. This difference is, fundamentally, due to the fact that we do not work with complex functions of time; if we could measure a signal with both real and imaginary parts, two plots would be required.)

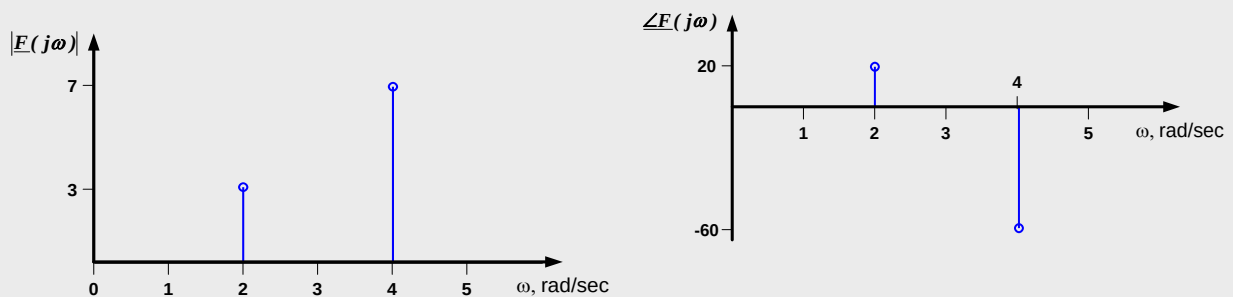
**Example 11.4**

Plot the spectrum of the voltage signal  $v_{in}(t) = 3 \cos(2t + 20^\circ) + 7 \cos(4t - 60^\circ)$ . (Note: this is the same signal as that used in Example 10.20; for a time-domain plot of the signal, see that example.)

The phasor form of this signal can be expressed as a function of frequency as:

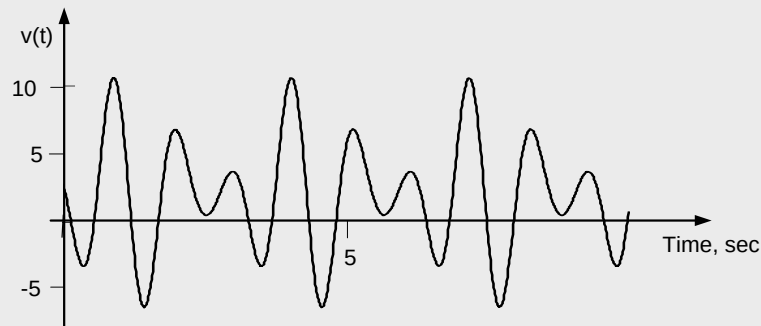
$$V_{in}(j\omega) = \begin{cases} 3\angle 20^\circ, & \omega = 2 \text{ rad/sec} \\ 7\angle -60^\circ, & \omega = 4 \text{ rad/sec} \\ 0, & \text{otherwise} \end{cases}$$

The spectrum is shown below:

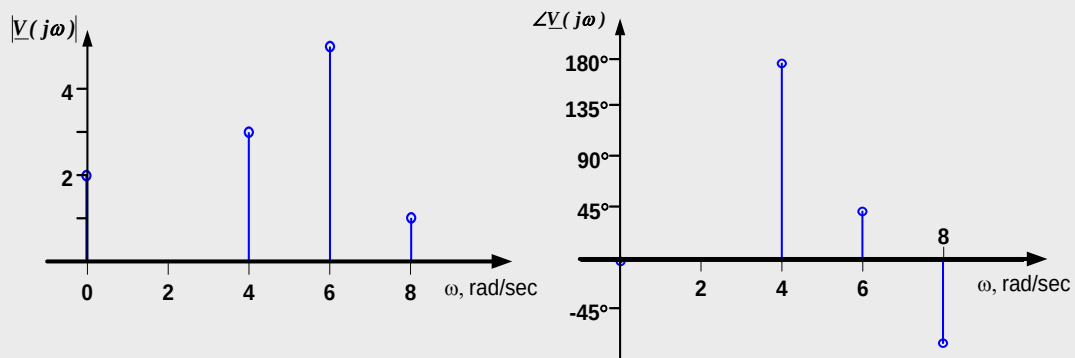
**Example 11.5**

Plot the voltage signal  $v(t) = 2 - 3 \cos(4t) + 5 \cos(6t + 45^\circ) + \cos(8t - 75^\circ)$  and its spectrum.

A plot of the time domain signal is shown below.



A couple of notes should be made about the spectrum of this signal. (1) The constant (DC) value of two corresponds to a frequency of zero rad/sec, since we can write  $2 = 2\cos(0t)$ . (2) Sinusoidal amplitudes are by definition positive, so the negative sign in the  $-3\cos(4t)$  term must be accounted for in the phase. Therefore, we re-write this term as  $+3\cos(4t + 180^\circ)$ . The spectrum of the signal can then be plotted as shown below:

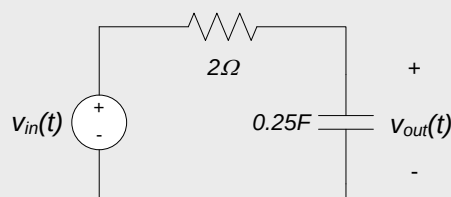


## 11.2.2 Frequency Response Plots

We have previously seen that the frequency response of a system consists of the system's magnitude response and phase response. The magnitude response provides the gain of the system (the ratio between the amplitudes of the output and input sinusoids) as a function of frequency, while the phase response provides the change in phase between the input and output sinusoids, as a function of frequency. In this section, we will emphasize the presentation of this information graphically. We do this by plotting the frequency response of two simple first order circuits in the examples below.

### Example 11.6

Plot the frequency response for the circuit shown below. The voltage  $v_{in}(t)$  is the input and the capacitor voltage  $v_{out}(t)$  is the *desired response*.



A mathematical expression for the frequency response of this circuit was determined in example 11.1 to be:

$$H(j\omega) = \frac{2}{2 + j\omega}$$

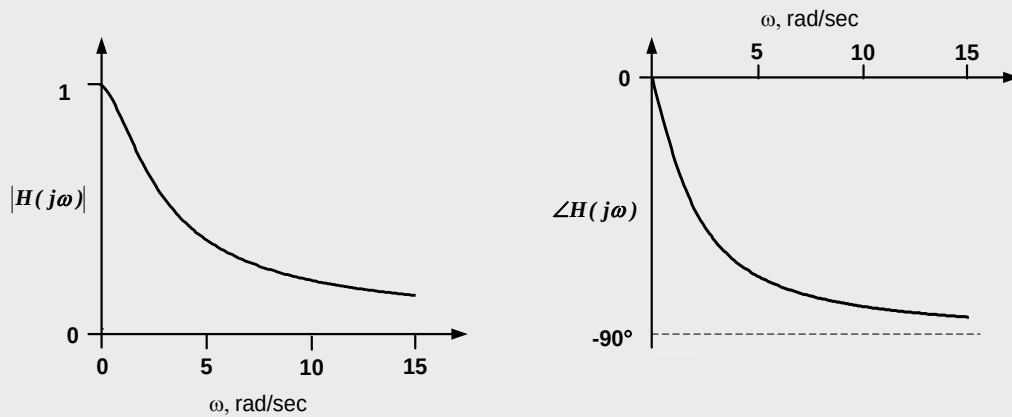
The magnitude and phase responses, as functions of frequency are, therefore:

$$|H(j\omega)| = \frac{2}{\sqrt{4 + \omega^2}}$$

And:

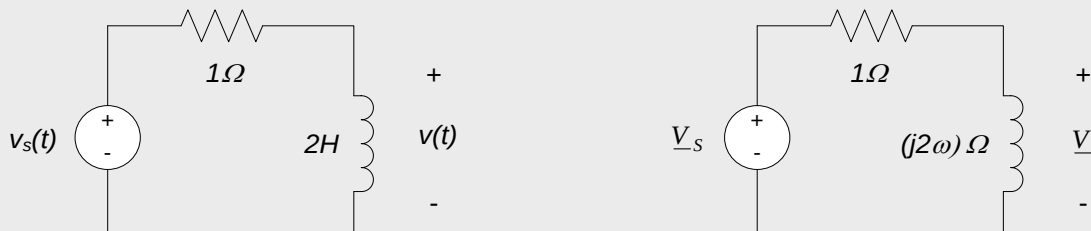
$$\angle H(j\omega) = -\tan^{-1}\left(\frac{\omega}{2}\right)$$

Plotting these functions results in the graphical frequency response shown below:



### Example 11.7

Plot the frequency response for the circuit shown to the left below. The source voltage  $v_s(t)$  is the input and the inductor voltage  $v(t)$  is the output.



We begin by converting the circuit to the frequency domain. Representing the input and output signals as phasors and the circuit elements as impedances results in the circuit to the right above. This circuit suggests that the output voltage can be determined from the input voltage via a voltage divider formula:

$$\underline{V} = \frac{j2\omega}{1 + j2\omega} \cdot \underline{V}_s$$

The circuit frequency response,  $H(j\omega)$ , is the ratio of the output phasor to the input phasor:

$$H(j\omega) = \frac{\underline{V}}{\underline{V}_s} = \frac{j2\omega}{1 + j2\omega}$$

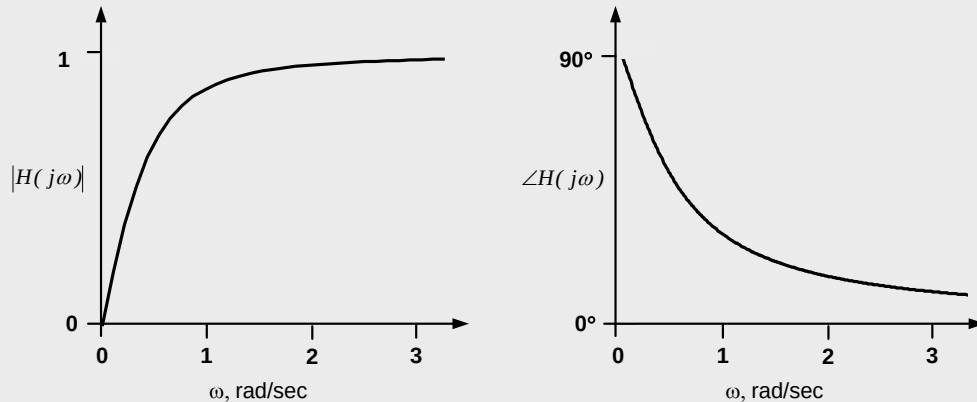
So that the magnitude and phase response of the circuit are:

$$|H(j\omega)| = \frac{2\omega}{\sqrt{1^2 + (2\omega)^2}}$$

And:

$$\angle H(j\omega) = 90^\circ - \tan^{-1}(2\omega)$$

Plots of these functions are shown below.



### 11.2.3 Checking the Frequency Response

A circuit's amplitude response is relatively easy to determine at low and high frequencies. For very low ( $\omega \rightarrow 0$ ) and very high ( $\omega \rightarrow \infty$ ) frequencies, the circuit can be modeled as a purely resistive network. Since resistive networks are relatively easy to analyze (no complex arithmetic is required), this can provide a valuable tool for checking results or predicting expected behavior.

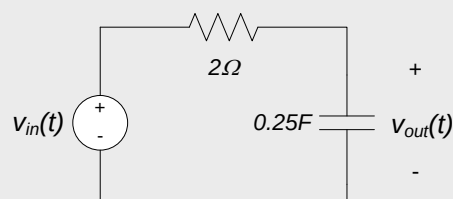
- Capacitors at low and high frequencies: A capacitor's impedance is  $Z_C = \frac{1}{j\omega C}$ . At low frequencies ( $\omega \rightarrow 0$ ), the impedance  $Z_C \rightarrow \infty$ , and the capacitor behaves as an open circuit. At high frequencies ( $\omega \rightarrow \infty$ ) the impedance  $Z_C \rightarrow 0$  and the capacitor behaves like a short circuit.
- Inductors at low and high frequencies: an inductor's impedance is  $Z_L = j\omega L$ . At low frequencies ( $\omega \rightarrow 0$ ), the impedance  $Z_L \rightarrow 0$ , and the inductor behaves as a short circuit. At high frequencies ( $\omega \rightarrow \infty$ ) the impedance  $Z_L \rightarrow \infty$  and the inductor behaves like an open circuit.

Please note that the above statements are relative only to the amplitude response.

### Example 11.8

Use the circuits' low and high frequency behavior to check the amplitude response plots for example 11.6.

The circuit of interest is shown below.

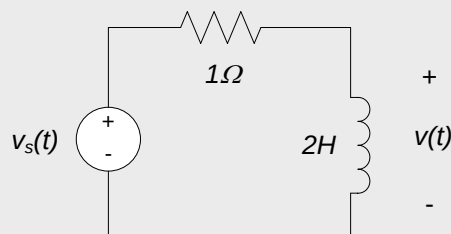


- At low frequencies, the capacitor becomes an open circuit, no current flows through the resistor, and  $v_{out}=v_{in}$ . Thus, the amplitude of the output is the same as the amplitude of the input and the gain as  $\omega \rightarrow 0$  is one.
- At high frequencies, the capacitor becomes a short circuit, so that  $v_{out}=0$ . Since the output amplitude is zero, for a non-zero input, the gain of the circuit as  $\omega \rightarrow \infty$  is zero.

The expected behavior at low and high frequencies agrees with the amplitude response of example 11.6; the amplitude response is one at  $\omega=0$ , and approaches zero as frequency increases.

### Example 11.9

Use the circuits' low and high frequency behavior to check the amplitude response plots for example 11.7. The circuit of interest is shown below:

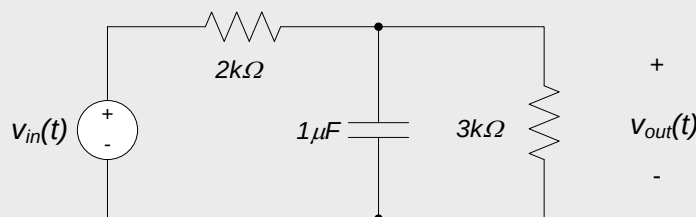


- At low frequencies, the inductor becomes a short circuit, so that  $v=0$ . Since the output amplitude is zero, for a non-zero input, the gain of the circuit as  $\omega \rightarrow 0$  is zero.
- At high frequencies, the inductor becomes an open circuit, no current flows through the resistor, and  $v=v_s$ . Thus, the amplitude of the output is the same as the amplitude of the input and the gain as  $\omega \rightarrow \infty$  is one.

The expected behavior at low and high frequencies agrees with the amplitude response of example 11.7; the amplitude response is zero at  $\omega=0$ , and approaches one as frequency increases.

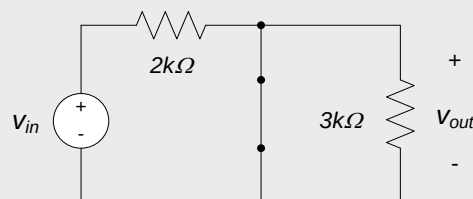
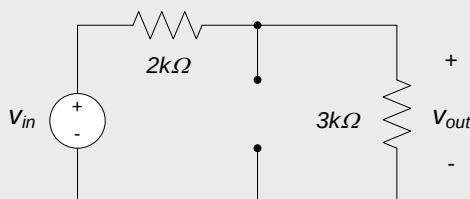
### Example 11.10

What is the gain of the circuit shown below as  $\omega \rightarrow 0$  and  $\omega \rightarrow \infty$ ? The input is  $v_{in}(t)$  and the output is  $v_{out}(t)$ .



As  $\omega \rightarrow 0$ , the capacitor becomes an open circuit, and the overall circuit becomes as shown to the left below.

As  $\omega \rightarrow \infty$ , the capacitor is replaced by a short circuit and the overall circuit becomes as shown to the right below.



As  $\omega \rightarrow 0$ , the circuit becomes a voltage divider, and  $v_{out} = \frac{3k\Omega}{2k\Omega + 3k\Omega} \cdot v_{in}$ , so that the circuit's gain is  $\frac{3}{5}$ . As  $\omega \rightarrow \infty$ ,  $v_{out}$  is measured across a short circuit, and the gain is zero.

### 11.2.4 Graphical Representation of System Response

We can use graphical depictions of the input signal spectrum and the frequency response of the system to obtain a graphical representation of a system's response. This representation can be especially useful in interpreting and understanding the effect of a system on an input signal. This qualitative interpretation of a system's response can be an invaluable aid in system design.

The frequency-domain representation of a system's response is shown in Fig. 11.4. The frequency-domain output is simply a point-by-point multiplication between the input signal and the system's frequency response. Thus, the amplitude spectrum of the output is simply the point-by-point product of the input signal's amplitude spectrum with the system's magnitude response. The phase spectrum of the output is the sum of the input signal's phase spectrum and the system's phase response. An example of this process and its use in interpreting a system's response is provided in the example below.

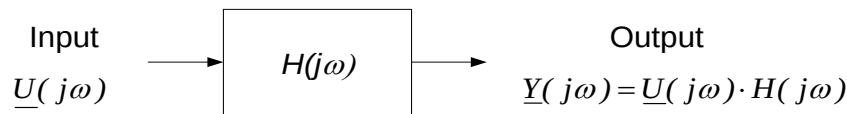
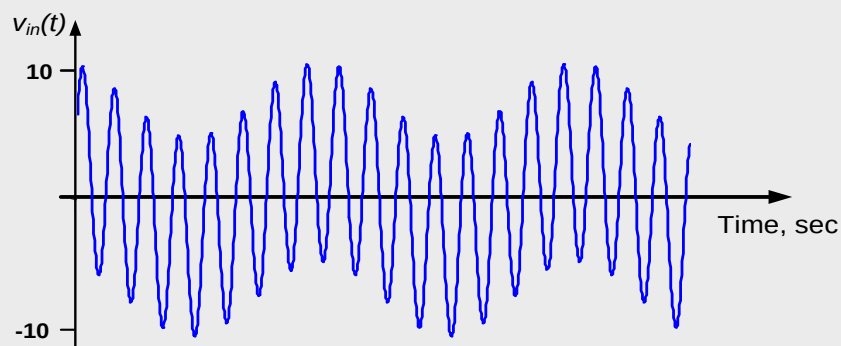


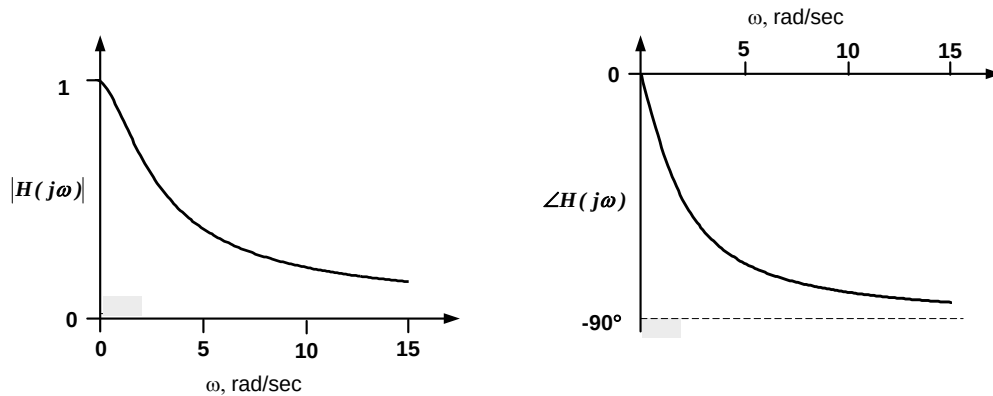
Figure 11.4. Frequency-domain system block diagram.

#### Example 11.11

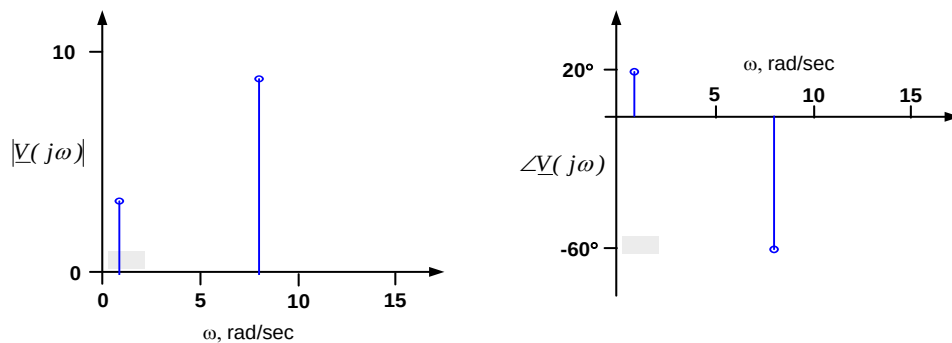
Use graphical methods to interpret the response of the circuit of example 11.6 to an input voltage  $v_{in}(t) = 3 \cos(t + 20^\circ) + 8 \cos(8t - 60^\circ)$ . The input voltage time-domain signal is shown below. The signal consists of a relatively small amplitude, low frequency sinusoid superimposed with a large amplitude, higher frequency signal.



The frequency response of the circuit was determined in example 11.6; the frequency response plots are repeated below:

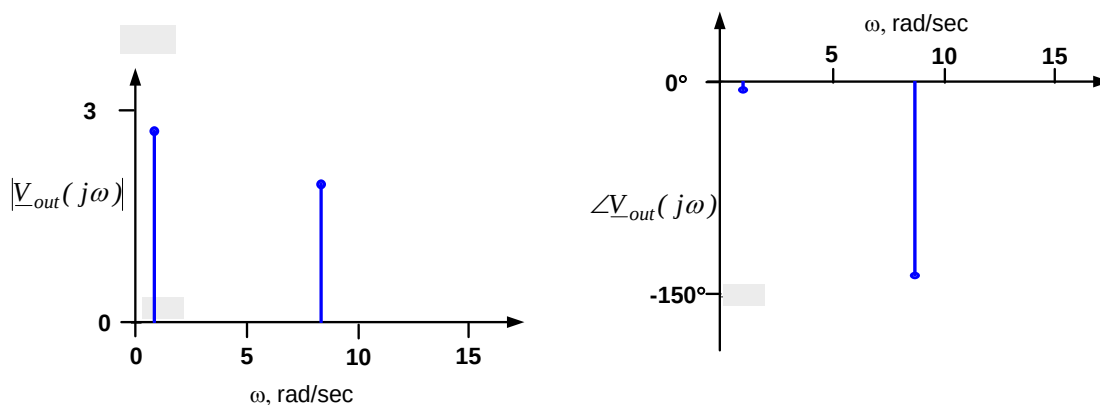


The input signal spectrum,  $\underline{V}(j\omega)$  is shown below.

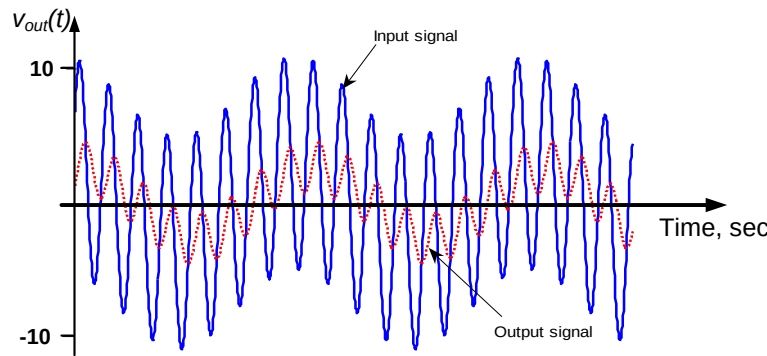


We can obtain the spectrum of the output signal from a frequency-by-frequency product of the input amplitude and the circuit's amplitude response and a frequency-by-frequency sum of the input phase and the circuit's phase response. Keep in mind that when the input amplitude is zero, the output amplitude is also zero and the phase is undetermined.

This process results in the output spectrum shown below. Notice that the circuit attenuates the magnitude of the high frequency component of the signal relative to the low frequency component. The low frequency component is said to be “passed through” the circuit, while the higher frequency component is “stopped” by the circuit.



The time-domain signal of the output signal is shown below, superimposed over the input signal. It is clear that the amplitudes of the higher frequency component of the output signal has been reduced relative to the amplitude of the lower frequency component. This circuit is said to be *frequency selective* – it selects low frequencies to pass and high frequencies to stop.



## Section Summary

- *Signal spectra* are a primarily a graphical representation of the frequency content of a signal. The spectrum consists, in general, of two plots: amplitude as a function of frequency and phase as a function of frequency.
- The signal spectrum is useful, since many signals of interest can be expressed as a superposition of sinusoidal components. This allows signals which do not appear “sinusoidal” to be analyzed using steady-state sinusoidal analysis techniques.
- *Frequency response plots* are a graphical representation of the frequency response of a system. Frequency response plots also consist of two parts: the system’s magnitude response as a function of frequency and the system’s phase response as a function of frequency.
- The combination of the signal spectrum and the frequency response plots can provide valuable insight into a system’s operation. The amplitude of the output signal is the point-by-point product of the input signal’s amplitude and the system’s magnitude response. The phase of the output signal is the point-by-point sum of the input signal’s phase and the system’s phase response.
  - Perhaps even more importantly, the signal spectra and frequency response plots can be used as an effective design tool: if the frequency content of the input signal and the desired frequency content of the output signal are both known, we can determine the system frequency response necessary to provide the desired output signal. This allows us to design our system to perform the desired task
- When calculating a system’s frequency response, it is always important to check your result relative to the expected behavior of the system. This can be done by determining the magnitude response at very low ( $\omega \rightarrow 0$ ) and very high ( $\omega \rightarrow \infty$ ) frequencies. In these extreme conditions the circuit can be approximated as a purely resistive circuit, since energy storage elements can be replaced by either short-circuits or open-circuits as follows:
  - Capacitors at low and high frequencies: A capacitor’s impedance is  $Z_C = \frac{1}{j\omega C}$ . At low frequencies ( $\omega \rightarrow 0$ ), the impedance  $Z_C \rightarrow \infty$ , and the capacitor behaves as an open circuit. At high frequencies ( $\omega \rightarrow \infty$ ) the impedance  $Z_C \rightarrow 0$  and the capacitor behaves like a short circuit.
  - Inductors at low and high frequencies: A inductor’s impedance is  $Z_L = j\omega L$ . At low frequencies ( $\omega \rightarrow 0$ ), the impedance  $Z_L \rightarrow 0$ , and the inductor behaves as a short circuit. At high frequencies ( $\omega \rightarrow \infty$ ) the impedance  $Z_L \rightarrow \infty$  and the inductor behaves like an open circuit.

## 11.2 Exercises

- The input to a system is  $u(t) = 3 \cos(2t - 60^\circ) + 4 \cos(4t + 30^\circ) + 7 \cos(6t + 45^\circ)$ . The system frequency response is given by the piecewise function:

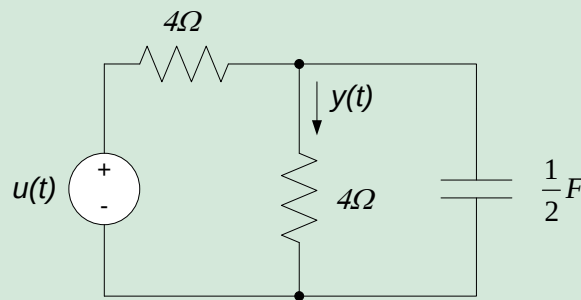
$$H(j\omega) = \begin{cases} 0, & 0 < \omega < 3 \\ 1, & 3 \leq \omega \leq 5 \\ 0, & \omega > 5 \end{cases}$$

Sketch:

- The spectrum of the input signal (magnitude and phase)
- The frequency response of the system (magnitude response and phase response)
- And the output signal spectrum.

Compare your sketches to your results of exercise 2 in section 1.11.

- Sketch the spectrum for the signal  $v(t) = 4 + 3 \cos(2t - 45^\circ) + 2 \cos(4t + 45^\circ)$ .
- Use the circuit behavior at high and low frequencies to check the frequency response you calculated for the circuit of exercise 1 in section 11.1.
- Determine the steady-state response of the output  $y(t)$  for the circuit below at very low and very high frequencies.



## 11.3 Frequency Selective Circuits and Filters

It is common to categorize circuits by the overall “shape” of their magnitude response. As we saw in example 11.11, in some frequency ranges the output amplitude may be high relative to the input amplitude, while in other frequency ranges the output amplitude will be low relative to the input amplitude. If the output amplitude at some frequency is high relative to the input amplitude, the magnitude response at that frequency is “large” and that frequency is said to be *passed* by the circuit. Conversely, if the output amplitude at some frequency is low relative to the input amplitude, the magnitude response at that frequency is “small” and that frequency is said to be *stopped* by the circuit.

Circuits which select certain frequencies to pass and other frequencies to stop are called *frequency selective* circuits or *filters* (since they tend to “filter out” certain frequency ranges of the input signal). The range of frequencies which are passed are called the *passband* of the filter, and the range of frequencies which are stopped are called the *stopband* of the filter. There are four primary categories of filters:

- Low-pass filters* pass low frequencies and stop high frequencies
- High-pass filters* pass high frequencies and stop low frequencies
- Band-pass filters* pass a range of frequencies between two ranges of stopped frequencies
- Band-reject filters* stop a range of frequencies between two ranges of passed frequencies

Filters are also categorized by their *order*. The order of the filter is simply the order of the differential equation governing the filter. Thus a first-order filter is governed by a first-order differential equation, a second-order filter is governed by a second-order differential equation, and so on. Low-pass and high-pass filters can be any order, while band-pass and band-stop filters must be at least second order.

In this chapter, we restrict our attention to first order filters, so we will consider only low pass and high pass filters.

### 11.3.2 Ideal Low-pass and High-pass Filters

We will first introduce the basic concepts relative to first order filters in the context of ideal filters. It must be clearly understood that ideal filters are not *physically realizable* – that is, we cannot construct a physical system which can perform this way<sup>2</sup>. Ideal filters entirely stop all input signals in the stopband and completely pass all signals in the passband. Thus, the magnitude response of an ideal filter is exactly one in the passband and exactly zero in the stop band. First order filters can be either high-pass or low-pass filters.

An ideal low pass filter has a magnitude response as shown in Fig. 11.5. The passband is shown as the shaded area under the magnitude response. The magnitude response is discontinuous – it goes from one to zero instantaneously. The *cutoff frequency*,  $\omega_c$ , defines the boundary between the passband and the stopband. Any signal with a frequency below  $\omega_c$  is passed through the filter without any attenuation; any signal with a frequency above  $\omega_c$  is entirely stopped by the filter – it is not present in the output signal.

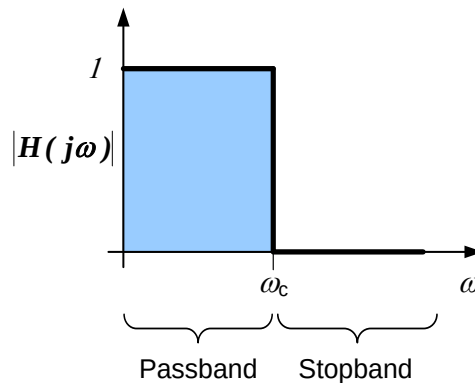


Figure 11.5. Magnitude response of an ideal low-pass filter.

An ideal high-pass filter has a magnitude response as shown in Fig. 11.6. The passband is again shown as the shaded area under the magnitude response. The magnitude response is discontinuous – it goes from zero to one instantaneously. The *cutoff frequency*,  $\omega_c$ , again defines the boundary between the passband and the stopband. Any signal with a frequency below  $\omega_c$  is entirely stopped by the filter while any signal with a frequency above  $\omega_c$  is passed through the filter with no amplitude change.

As previously noted, it is impossible to physically implement an ideal filter. Thus, all electrical circuits implement non-ideal filters. Non-ideal filters do not provide an instantaneous transition between the pass band and the stop band. Non-ideal first order filters are discussed in the following subsections.

<sup>2</sup> In order to implement an ideal filter, the circuit would need to know values of the input signal before the values are applied – that is, the circuit would need to be able to see into the future. The technical term is that the filter would need to be non-causal. It is, of course, impossible to build a device that predicts the future.

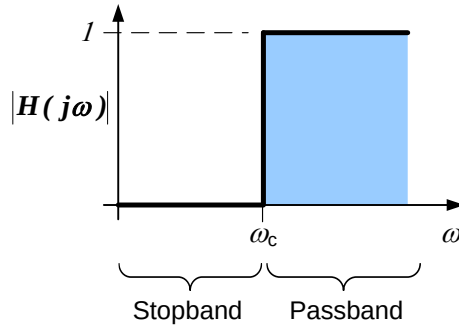


Figure 11.6. Magnitude response of an ideal low-pass filter.

### 11.3.3 First Order Low-pass Filters

The form of the governing differential equation for a first order low pass filter can be written as:

$$\frac{dy(t)}{dt} + \omega_c y(t) = K \cdot u(t) \quad \text{Eq. 11.5}$$

The frequency response of the filter can be determined to be:

$$H(j\omega) = \frac{K}{j\omega + \omega_c} \quad \text{Eq. 11.6}$$

The magnitude response of the filter is thus:

$$|H(j\omega)| = \frac{K}{\sqrt{\omega^2 + \omega_c^2}} \quad \text{Eq. 11.7}$$

The maximum magnitude response is  $\frac{K}{\omega_c}$  when  $\omega=0$  and the magnitude response is zero as  $\omega \rightarrow \infty$ . Thus, the filter is passing low frequencies and stopping high frequencies. The differential equation describes a low-pass filter.

The magnitude response of the filter is shown in Fig. 11.7. The frequency response is a smooth curve, rather than the discontinuous function shown in Fig. 11.5. There is no single frequency that obviously separates the passband from the stopband, so we must choose a relatively arbitrary point to define the boundary between the passband and the stopband. By consensus, the cutoff frequency for a low-pass filter is defined as the frequency at which the magnitude response is  $\frac{1}{\sqrt{2}}$  times the magnitude response at  $\omega=0$ . For the magnitude response given by equation (11.7), the cutoff frequency is  $\omega=\omega_c$ . This point is indicated on Fig. 11.7.

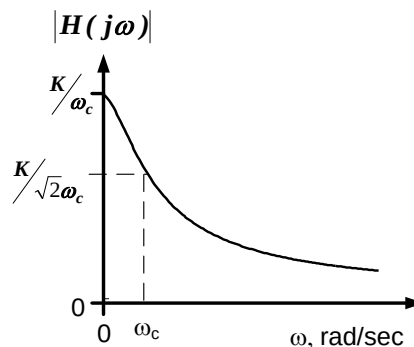


Figure 11.7. Magnitude response of first order low-pass filter.

### 11.3.4 First Order High-pass Filters

The form of the governing differential equation for a first order low pass filter can be written as:

$$\frac{dy(t)}{dt} + \omega_c y(t) = K \cdot \frac{du(t)}{dt} \quad \text{Eq. 11.8}$$

The frequency response of the filter can be determined to be:

$$H(j\omega) = \frac{jK\omega}{j\omega + \omega_c} \quad \text{Eq. 11.9}$$

The magnitude response of the filter is thus:

$$|H(j\omega)| = \frac{K\omega}{\sqrt{\omega^2 + \omega_c^2}} \quad \text{Eq. 11.10}$$

The maximum magnitude response is approximately  $K$  when  $\omega \rightarrow \infty$  and the magnitude response is zero at  $\omega=0$ . Thus, the filter is passing high frequencies and stopping low frequencies. The differential equation describes a high-pass filter.

The magnitude response of the filter is shown in Fig. 11.8. As with the non-ideal low-pass filter, the frequency response is a smooth curve, rather than the discontinuous function shown in Fig. 11.6. Again, there is no single frequency that obviously separates the passband from the stopband, so we must choose a relatively arbitrary point to define the boundary between the passband and the stopband. Consistent with our choice of cutoff frequency for the low-pass filter, the cutoff frequency for a high-pass filter is defined as the frequency at which the magnitude response is  $\frac{1}{\sqrt{2}}$  times the magnitude response at  $\omega \rightarrow \infty$ . For the magnitude response given by equation (1.10), the cutoff frequency is  $\omega=\omega_c$ . This point is indicated on Fig. 11.8.

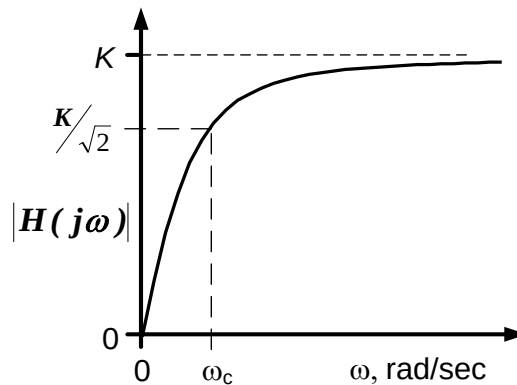


Figure 11.8. Magnitude response of first order high-pass filter.

#### Notes:

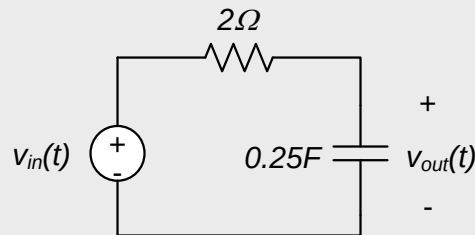
- The cutoff frequency is also called the corner frequency, the 3dB frequency, or the half-power point.
- The cutoff frequency for both low-pass and high-pass filters is defined as the frequency at which the magnitude is  $\frac{1}{\sqrt{2}}$  times the maximum value of the magnitude response.
- It can be seen from examples in section 11.2 that the phase response of a first order low-pass filter is  $0^\circ$  at  $\omega=0$  and decreases to  $-90^\circ$  as  $\omega \rightarrow \infty$ . The phase response is  $-45^\circ$  at the cutoff frequency.
- It can be seen from examples in section 11.2 that the phase response of a first order high-pass filter is  $90^\circ$  at  $\omega=0$  and decreases to  $0^\circ$  as  $\omega \rightarrow \infty$ . The phase response is  $45^\circ$  at the cutoff frequency.

- For both low-pass and high-pass filters, the cutoff frequency is the inverse of the time constant for the circuit, so that  $\omega_c = \frac{1}{\tau}$ .
- The circuit's response at zero frequency is generally an important parameter to consider. This is called the DC gain, and is the ratio of the output amplitude to the input amplitude for a constant input. A constant input corresponds to a cosine with zero frequency. Low pass filters have a relatively high DC gain and a correspondingly large response to a constant input. High pass filters have a low DC gain; they have little or no response to constant inputs.

We conclude this section with examples of circuits from section 10.2 which implement low-pass and high-pass filter operations.

### Example 11.12: First Order Low-pass Filter

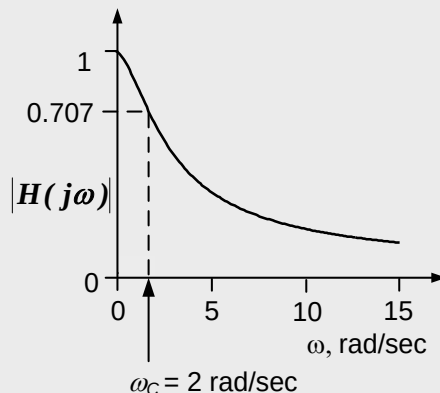
The circuit below is the circuit from example 11.6. The input is  $v_{in}(t)$  and the output is  $v_{out}(t)$ .



In example 11.6, the frequency response was determined to be:

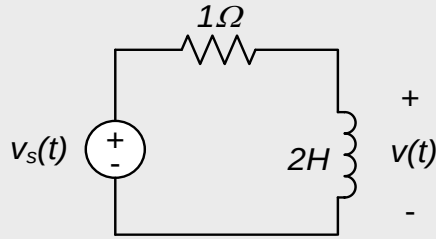
$$H(j\omega) = \frac{2}{2 + j\omega}$$

The maximum value of the magnitude response is one at a frequency of zero radians/second and the magnitude response goes to infinity as  $\omega \rightarrow \infty$ , so the circuit acts as a low-pass filter. Comparing the amplitude response above with equation (2) above, it can be seen that the cutoff frequency is  $\omega_c = 2$  rad/sec. The amplitude response, with the cutoff frequency labeled, is shown below.



### Example 11.13: First Order High-pass Filter

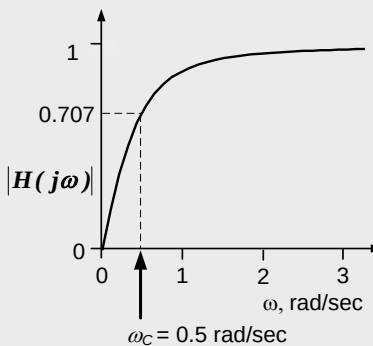
The circuit below is the circuit from example 11.7. The input is  $v_s(t)$  and the output is  $v(t)$ .



The frequency response of the circuit was previously determined to be:

$$H(j\omega) = \frac{V}{V_s} = \frac{j2\omega}{1 + j2\omega}$$

The maximum value of the magnitude response is one as  $\omega \rightarrow \infty$  and goes to zero at a frequency of zero radians/second, so the circuit acts as a low-pass filter. Comparing the amplitude response above with equation (4) above, it can be seen that the cutoff frequency is  $\omega_c = 0.5$  rad/sec. The amplitude response, with the cutoff frequency labeled, is shown below.

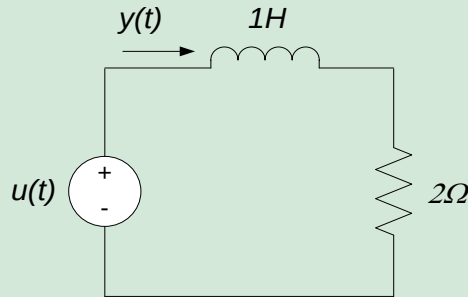


## Section Summary

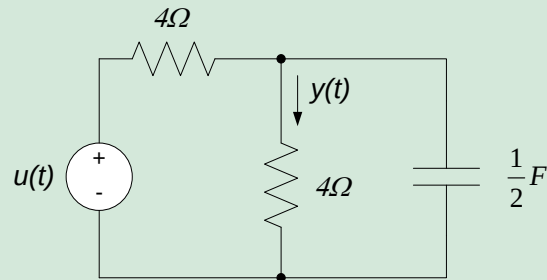
- Filters are frequency-selective systems. These systems provide a desired relationship between the input signal spectrum and the output signal spectrum. The filter does this by *passing* certain frequencies from the input to the output and *stopping* some frequencies from propagating from the input to the output. Nomenclature is as follows:
  - The range of frequencies which are passed is called the *passband* of the filter
  - The range of frequencies which are stopped is called the *stopband* of the filter
- Filters are broadly categorized as follows:
  - *Low-pass filters* pass low frequencies and stop high frequencies
  - *High-pass filters* pass high frequencies and stop low frequencies
  - *Band-pass filters* pass a range of frequencies between two ranges of stopped frequencies
  - *Band-reject filters* stop a range of frequencies between two ranges of passed frequencies
- In this chapter, we were concerned only with first-order low-pass and high-pass filters. These filters are primarily characterized by the following parameters:
  - Cutoff frequency: the cutoff is defined as the frequency at which the magnitude is  $\frac{1}{\sqrt{2}}$  times the maximum value of the magnitude response.
  - *DC gain*: the DC gain is the ratio of the output amplitude to the input amplitude for a constant input (a cosine function with zero frequency). Low pass filters have a relatively high DC gain and a correspondingly large response to a constant input. High pass filters have a low DC gain; they have little or no response to constant inputs.

### 11.3 Exercises

1. What is the cutoff frequency of the circuit below? (You may want to use your results from exercise 1 of section 11.1.)



2. Use the circuit behavior at high and low frequencies and your time constant calculated in exercise 1 above to sketch the frequency response of the circuit of exercise 1. Label your frequency response to include DC gain and cutoff frequency. Is the circuit a high-pass or low-pass filter?
3. Calculate the time constant and the cutoff frequency for the circuit below, if  $u(t)$  is the input and  $y(t)$  is the output. Verify that the cutoff frequency is the inverse of the time constant. Use the circuit behavior at high and low frequencies to determine whether the circuit is a high-pass or low-pass filter.



4. Calculate the frequency response of the circuit of exercise 3. Compare your frequency response to your results of exercise 3. Resolve any differences between the two.

### 11.4 Introduction to Bode Plots

Plotting a systems' frequency response on a linear scale, as done in sections 11.3 and 11.4, has a number of drawbacks, especially for higher-order systems<sup>3</sup>. An alternate format for plotting frequency responses, called a Bode plot, is therefore commonly used<sup>4</sup>. On Bode plots, the amplitude response is essentially presented as a log-log plot, while the phase response is a semi-log plot. Some reasons for this are:

- Use of logarithms converts the operation of multiplication and division to addition and subtraction. This can simplify the creation of frequency response plots for higher order systems.
- Frequencies and amplitudes of interest commonly span many orders of magnitude. Logarithmic scales improve the presentation of this type of data.

<sup>3</sup> Higher order systems are often modeled as a series of lower-order systems (or in the technical parlance, *cascaded* lower-order systems). A combination of cascaded frequency responses is multiplicative. Multiplying several frequency responses together can be tedious

<sup>4</sup> Bode plots use logarithmic scales. Logarithms convert multiplication to addition. Thus, a cascaded set of frequency responses in a logarithmic scale add – adding frequency responses is significantly simpler than multiplying them.

- Human senses are fundamentally logarithmic. The use of logarithmic scales is therefore more “natural”. (This is the reason for use of the Richter scale in measuring earthquake intensity, and the decibel scale in measuring sound levels. It is also the reason that increasing a musical tone by one octave corresponds to doubling its frequency.)

### 11.4.1 Properties of Logarithms

Since Bode plots employ logarithms extensively, we will briefly review some of the basic properties of logarithms before proceeding further. Bode plots rely upon base-10 logarithms ( $\log_{10}$ ), so we will restrict our attention to base-10 logarithms.

A plot of  $\log_{10}(x)$  vs.  $x$  is shown in Fig. 11.9 below. A few important features to note are:

- $\log_{10}(x)$  is a real number only for positive values of  $x$ .
- $\log_{10}(x)$  asymptotically approaches  $-\infty$  as  $x \rightarrow 0$ . The slope of  $\log_{10}(x)$  becomes very large as  $x \rightarrow 0$ .
- The slope of  $\log_{10}(x)$  becomes small as  $x \rightarrow \infty$ .
- From the comments above relative to the slope of  $\log_{10}(x)$ , it can be seen that the sensitivity of  $\log_{10}(x)$  to variations in  $x$  decreases as  $x$  increases (this is the reason why logarithmic scales are used when large variations in  $x$  are encountered – as in Richter scales and musical scales).
- $\log_{10}(1) = 0$

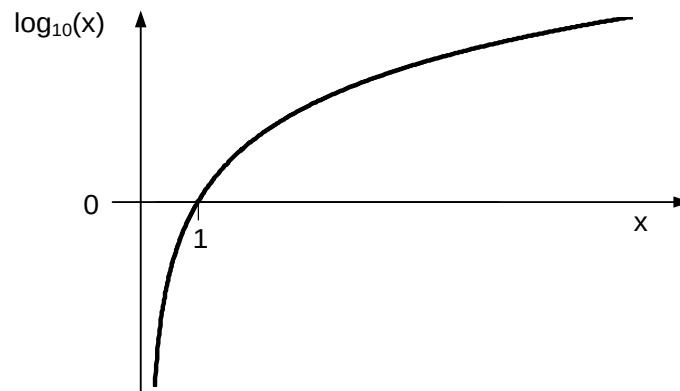


Figure 11.9. Plot of  $\log_{10}(x)$  vs.  $x$

The basic defining property of a base-10 logarithm is that  $x=10^y$ , then  $y=\log_{10}x$ . This property leads to the following rules governing logarithmic operations:

- Logarithms convert multiplication and division to addition and subtraction, respectively. Thus,

$$\log_{10}(xy) = \log_{10} x + \log_{10} y$$

And:

$$\log_{10}\left(\frac{x}{y}\right) = \log_{10} x - \log_{10} y$$

This property is especially useful for us, since determining the spectrum of an output signal results from the product of an input signal's spectrum with the frequency response. Thus, the output spectrum on a logarithmic scale can be obtained from a simple addition.

2. Logarithms convert exponentiation to multiplication by the exponent, so that:

$$\log_{10}(x^n) = n \log_{10} x$$

### 11.4.2 Decibel Scales

Magnitude responses are often presented in terms of decibels (abbreviated dB). Decibels are a logarithmic scale. A magnitude response is presented in units of decibels according to the following conversion:

$$|H(j\omega)|_{dB} = 20 \log_{10}(|H(j\omega)|) \quad \text{Eq. 11.11}$$

Strictly speaking, magnitudes in decibels are only appropriate if the amplitude response is unitless (e.g. the units of the input and output must be the same in order for the logarithm to be a mathematically appropriate operation). However, in practice, magnitude responses are often presented in decibels regardless of the relative units of the input and output – thus, magnitude responses are provided in decibels even if the input is voltage and the output is current or vice-versa.

#### Brief Historical Note

Decibel units are related to the unit “bel”, which are named after Alexander Graham Bell. Units of bels are, strictly speaking, applicable only to power. Power in bels is expressed as  $\log_{10}\left(\frac{P}{P_{ref}}\right)$ , where  $P_{ref}$  is a “reference” power. Bels are an inconveniently large unit, so these were converted to decibels, or tenths of a bel. Thus, power in decibels is  $10 \log_{10}\left(\frac{P}{P_{ref}}\right)$ . Since the units of interest to electrical engineers are generally voltages or currents, which must be squared to obtain power, we obtain  $20 \log_{10}(|H(j\omega)|)$ . The significant aspect of the decibel unit for us is not, however, the multiplicative factor of “20”, but the fact that the unit is logarithmic.

We conclude this subsection with a table of common values for  $|H(j\omega)|$  and their associated decibel values.

| $ H(j\omega) $       | $ H(j\omega) $ in decibels |
|----------------------|----------------------------|
| 10                   | 20                         |
| 1                    | 0                          |
| 0.1                  | -20                        |
| $\frac{1}{\sqrt{2}}$ | -3                         |
| $\frac{1}{2}$        | -6                         |

### 11.4.3 Bode Plots

Bode plots are simply plots of the magnitude and phase response of a system using a particular set of axes. For Bode plots,

- Units of frequency are on a base-10 logarithm scale.
- Amplitudes (or magnitudes) are in decibels (dB)
- Phases are presented on a linear scale

#### Notes

- Since frequencies are on a logarithmic scale, frequencies separated by the same multiplicative factor are evenly separated on a logarithmic scale. Some of these multiplicative factors have special names. For

example, frequencies separated by a factor of two are said to be separated by *octaves* on a logarithmic scale and frequencies separated by a factor of 10 are said to be separated by *decades*.

- Since decibels are intrinsically a logarithmic scale, magnitudes which are separated by the same multiplicative factor are evenly separated on a decibel scale. For example, magnitudes which are separated by a factor of 10 are separated by 20dB on a decibel scale.

One convenient aspect of the presentation of frequency responses in terms of Bode plots is the ability to generate a reasonable sketch of a frequency response very easily. In general, this approach consists of approximating the Bode plot of a system by its asymptotic behavior as a set of straight lines. This is called a “straight line approximation” of the Bode plot; the approach is illustrated for a typical low-pass filter in the following subsection.

### 11.4.4 Bode Plots for First Order Low-pass Filters

The frequency response of a general first order low-pass filter is provided in section 11.3 as:

$$H(j\omega) = \frac{K}{j\omega + \omega_c} \quad \text{Eq. 11.12}$$

Thus, magnitude response of the circuit is:

$$|H(j\omega)| = \frac{K}{\sqrt{\omega^2 + \omega_c^2}} \quad \text{Eq. 11.13}$$

And the phase response of the circuit is:

$$\angle H(j\omega) = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right) \quad \text{Eq. 11.14}$$

To estimate the asymptotic behavior of the frequency response, we consider the behavior of equations (11.13) and (11.14) for the low frequency and high frequency cases. In general, we consider “low” frequencies to be frequencies which are less than a factor of 10 below the cutoff frequency (i.e.  $\omega < \frac{\omega_c}{10}$ , or frequencies more than a decade below the cutoff frequency). High frequencies are typically assumed to be frequencies which are more than a factor of 10 above the cutoff frequency (i.e.  $\omega > 10\omega_c$ , or more than a decade above the cutoff frequency). We consider the high and low frequency cases separately below.

- Low frequencies:
  - The magnitude response given by equation (2) is  $|H(j\omega)| = \frac{K}{\sqrt{\omega^2 + \omega_c^2}}$ . If  $\omega \ll \omega_c$ , the denominator is approximately  $\sqrt{\omega_c^2} = \omega_c$  and the magnitude response  $|H(j\omega)| \approx \frac{K}{\omega_c}$ . If  $\omega \ll \omega_c$ ,  $\frac{\omega}{\omega_c} \approx 0$  and the phase is approximately  $\angle H(j\omega) \approx -\tan^{-1}(0) = 0^\circ$ .
- High Frequencies:
  - If  $\omega \gg \omega_c$ , the denominator of the amplitude response is  $\sqrt{\omega^2 + \omega_c^2} \approx \sqrt{\omega^2} = \omega$ . Therefore, for high frequencies, the magnitude response  $|H(j\omega)| \approx \frac{K}{\omega}$ . If, for high frequencies, we increase the frequency by a factor of 10, we reduce the magnitude response by 20dB (since  $|H(j \cdot 10\omega)| \approx \frac{K}{10\omega} = 0.1 \frac{K}{\omega}$  and the multiplicative factor of 0.1 corresponds to -20dB). Thus, for frequencies well above the cutoff frequency, the magnitude response, presented in Bode plot form, decreases by 20dB/decade. When  $\omega \gg \omega_c$ , the phase response is given by  $\angle H(j\omega) = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right) \approx -\tan^{-1}(\infty) = -90^\circ$ .

**Summary: Low-pass Filter Straight-line Bode Plot Approximations**

The straight line approximation to the magnitude response is constant below the cutoff frequency, with a value (in decibels) of  $20 \log_{10} \left( \frac{K}{\omega_c} \right)$ . Above the cutoff frequency, the Bode plot straight-line approximation has a constant slope of -20 dB/decade.

The straight-line approximation to the phase response is zero degrees up to a frequency of  $\frac{\omega_c}{10}$  and is -90° above a frequency of  $10\omega_c$ . A straight line is used to connect the  $\frac{\omega_c}{10}$  and  $10\omega_c$  frequencies.

A straight-line approximation to the Bode plot for a typical low-pass circuit, with  $K = \omega_c$  (so that the frequency response is  $H(j\omega) = \frac{\omega_c}{j\omega + \omega_c}$  and the DC gain is 1, or 0dB) along with an exact curve is provided below in Fig. 11.10.

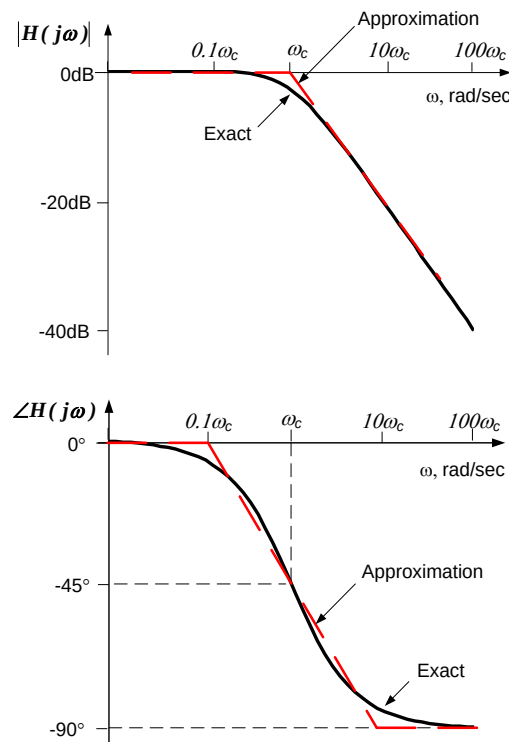
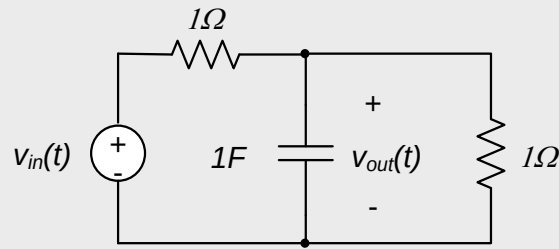


Figure 11.10. Low pass filter straight-line approximation and exact Bode plot.

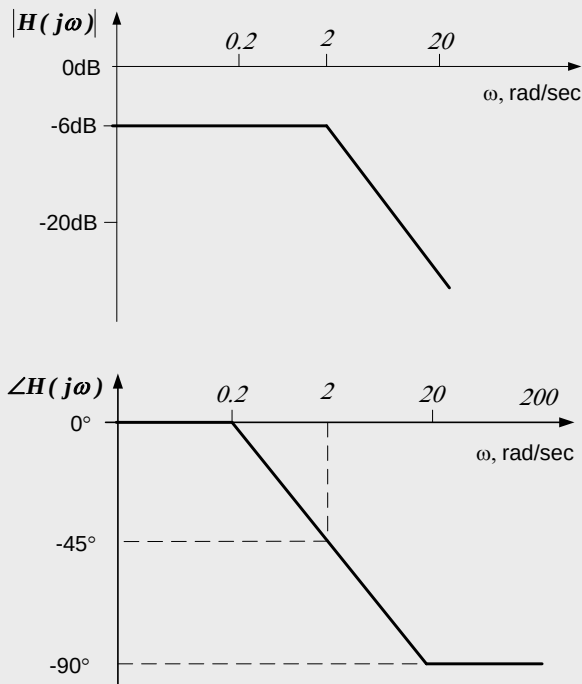
We conclude this section with a numerical example of the straight-line approximation to a Bode plot for a specific circuit.

**Example 11.14**

Sketch a straight-line approximation to the Bode plot for the circuit below. The input is  $v_{in}(t)$  and the output is  $v_{out}(t)$ .



The frequency response for this circuit is  $H(j\omega) = \frac{1}{j\omega + 2}$ . Therefore, the cutoff frequency is  $\omega_c = 2$  rad/sec and the gain in decibels at low frequencies is  $|H(j0)|_{dB} = 20 \log_{10}\left(\frac{1}{2}\right) \approx -6dB$ . Thus, the straight-line magnitude response is -6dB below the cutoff frequency and decreases by 20dB/decade above the cutoff frequency. The straight-line phase response is  $0^\circ$  below 0.2 rad/sec,  $-90^\circ$  above 20 rad/sec and a straight line between these frequencies. The associated plots are shown below.



## Section Summary

- Bode plots are a very useful format for plotting frequency responses. Bode plots provide magnitude responses and phase responses in the following format:
  - Units of frequency are on a base-10 logarithm scale.
  - Amplitudes (or magnitudes) are in decibels (dB)
  - Phases are presented on a linear scale.
  - Magnitude responses in decibels are calculated according to:

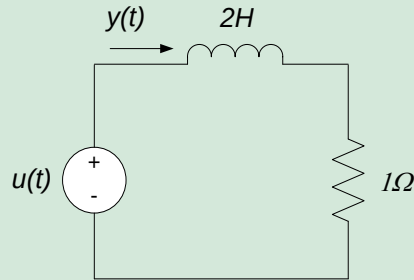
$$|H(j\omega)|_{dB} = 20 \log_{10}(|H(j\omega)|)$$

- The use of logarithmic scales in Bode plots has a number of advantages. Logarithms convert multiplication and division into addition and subtraction, respectively. This provides a significant mathematical simplification in the analysis of higher-order systems. Logarithms are also, in some sense, more “natural” to interpret, since human senses are fundamentally logarithmic.

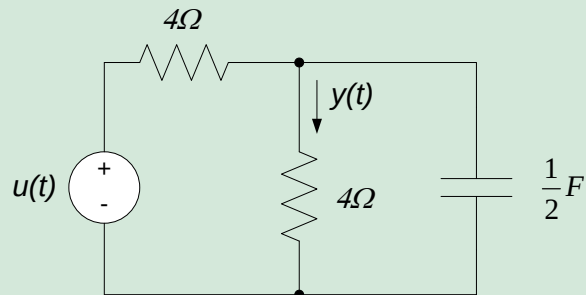
- Bode plots also have the advantage of being approximated fairly well by straight-line approximations. This allows the engineer to sketch a fairly accurate frequency response plot with only a minimal number of calculations.

## 11.4 Exercises

1. Sketch a Bode plot (straight-line approximation) for the circuit below.



2. Sketch a Bode plot (straight-line approximation) for the circuit below.



## Real Analog Chapter 11: Lab Projects

### 11.2.1: Signals with Multiple Frequency Components

In this lab project, we will calculate the magnitude response of an electrical circuit and use this information to infer the effect of the circuit on some relatively complex input signals. In particular, we will apply the following input signal types to the circuit:

- A signal composed of multiple sinusoidal waves of different frequencies
- A sinusoidal signal with a time-varying frequency (a sinusoidal *sweep*)

We will subsequently measure the response of the circuit to these input signals and compare them to our expectations. The goal of this lab project is to see how a circuit's magnitude response affects the "shape" of a signal applied to it. This is a preliminary step to designing a circuit to provide a magnitude response which has a desired effect on an input signal.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of a passive electrical circuit
- Calculate the magnitude and phase responses of a passive electrical circuit
- Use a circuit's magnitude and phase responses to determine the response to a sinusoidal input

#### After completing this lab, you should be able to:

- Measure the response of a circuit to an input signal with multiple sinusoidal components
- Apply a sinusoidal sweep to an electrical circuit

#### This lab exercise requires:

- Analog Discovery module
- Diligent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

##### DEMO

Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.

##### ANALYSIS

Analysis; include principle results of analysis in laboratory report.

##### SIM

Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.

##### DATA

Record data in your lab notebook.

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1. The overall behavior in response to sinusoidal inputs is relatively simple to understand. At very low frequencies, the capacitor has infinite impedance, the circuit becomes a simple voltage divider, and the output voltage phasor amplitude is simply half of the input voltage amplitude:

$$\omega = 0 \Rightarrow \left| \frac{V_{OUT}}{V_{IN}} \right| = \frac{1}{2} \quad \text{Eq. 1}$$

At very high frequencies, the capacitor's impedance is zero, the capacitor behaves like a short circuit, and the output voltage is zero regardless of the input voltage amplitude:

$$\omega = \infty \Rightarrow \left| \frac{V_{OUT}}{V_{IN}} \right| = 0 \quad \text{Eq. 2}$$

Since the circuit's output lets low frequency signals appear in the output (to some extent) and does not let high frequency inputs appear in the output, the circuit is said to "pass" low frequencies and "stop" high frequencies<sup>5</sup>.

In this assignment, we will see how we can infer a circuit's response to relatively complex input voltage waveforms by using simple observations like those of equations (1) and (2).

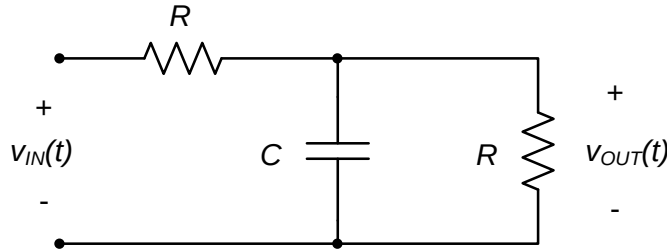


Figure 1. RC circuit.

### Pre-lab:

#### ANALYSIS

Determine the magnitude response of the circuit shown in Fig. 1. If  $R = 1\text{k}\Omega$  and  $C = 100\text{nF}$ , calculate the magnitude response (the ratio of the amplitude of the output sinusoid to the input sinusoid) of the circuit at frequencies of 500Hz, 1000Hz, and 10kHz. Do your magnitude responses at these frequencies agree with the expectations of the circuit response as expressed by equations (1) and (2)?

### Lab Procedures:

#### DATA

- a. Construct the circuit of Fig. 1, using  $R = 1\text{k}\Omega$  and  $C = 100\text{nF}$ .
  - i. Use your oscilloscope to measure both  $v_{IN}(t)$  and  $v_{OUT}(t)$ . Use the waveform generator to apply a custom waveform to the circuit. The waveform will be defined by the expression:

$$v_{IN}(t) = 20[\sin(100\pi t) + \sin(2000\pi t) + \sin(20,000\pi t)] \quad \text{Eq. 3}$$

Notice that the first term in this series has a frequency of 500Hz, the second term a frequency of 1000Hz, and the third term a frequency of 10,000Hz. Instructions for creating and applying a custom signal based on the mathematical expression above are provided in Appendix A of this project.

#### DATA

- ii. Record an image of the oscilloscope window, showing the voltages  $v_{IN}(t)$  and  $v_{OUT}(t)$ .

#### DEMO

- iii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

#### ANALYSIS

- iv. Comment on the overall shape of the input and output signals, relative to your expectations based on the pre-lab analyses and equations (1) and (2).
- b. With the same circuit as in part (a):
  - i. Use the waveform generator to apply a sinusoidal sweep to the circuit. A sinusoidal sweep typically has a frequency which changes linearly with time. The signal we will use starts at a frequency of 100Hz; the frequency increases to 10kHz in 20msec. Detailed instructions for creating and applying this signal are provided in Appendix B of this project.

<sup>5</sup> The circuit is said to be a *low pass filter*.

**DATA**

- ii. Record an image of the oscilloscope window, showing the voltages  $v_{IN}(t)$  and  $v_{OUT}(t)$ .

**DEMO**

- iii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**ANALYSIS**

- iv. Comment on the overall shape of the input and output signals, relative to your expectations based on the pre-lab analyses and equations (1) and (2).

## Appendix A: Creating a Custom Mathematical Waveform

On the **Basic** tab of the waveform generator, select **Custom** and click on **Edit**. An AWG-Editor window should open, providing a number of ways to create custom waveforms.

1. Select the **Math** tab on the **AWG-Editor** window and create the mathematical expression corresponding to the given function. This will require several steps:
  - The horizontal range of values (X, in the waveform generator) are set in the text boxes in the upper right of the window. Your range of X values should be from 0.000 to 1.000, as shown in Fig. A1.
  - Type the mathematical expression for the signal being created. Multiplication is denoted by \*,  $\text{PI} = \pi$ , and  $\text{SIN} = \text{sine}$ . Operator precedence is as in most calculators. Thus, the mathematical expression in equation (3) can be implemented as:

$$20 * (\text{Sin}(2 * \text{PI} * X) + \text{Sin}(4 * \text{PI} * X) + \text{Sin}(40 * \text{PI} * X))$$

Note that the arguments of the sinusoid do not agree with the original mathematical expression.

Actually, the only thing that really matters about these arguments is that the second is twice the first, and the third is twenty times the first. The arguments will all be scaled once we have the waveform generator “play” the signal.

- The final result should look something like that shown in Fig. A1. Don’t worry about the axes in the plot window; the horizontal axis is in “samples” and the vertical axis is “percent of full scale”. Both of these will be scaled when we play back the signal.
- Click **Save**. The **AWG-Editor** window should close.

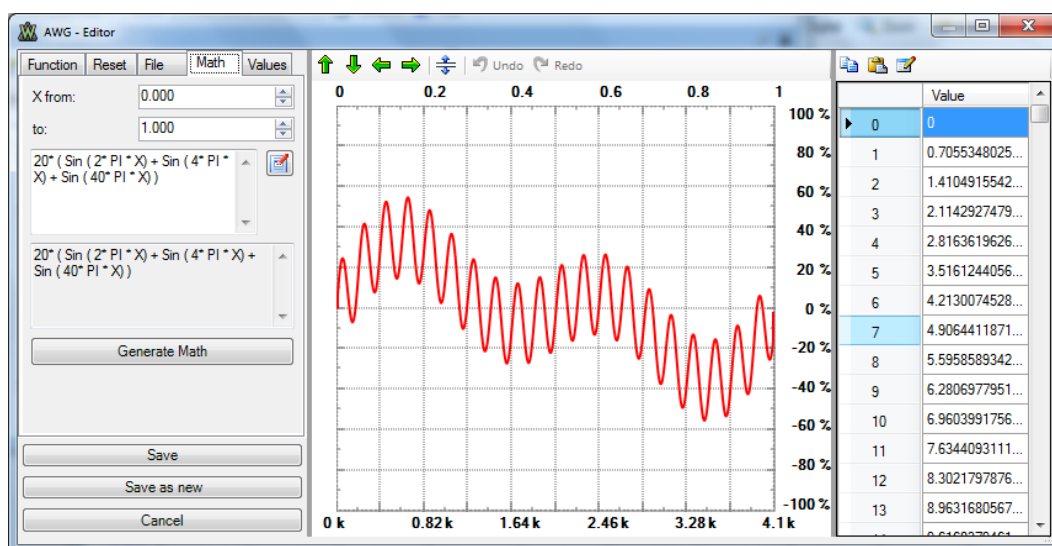
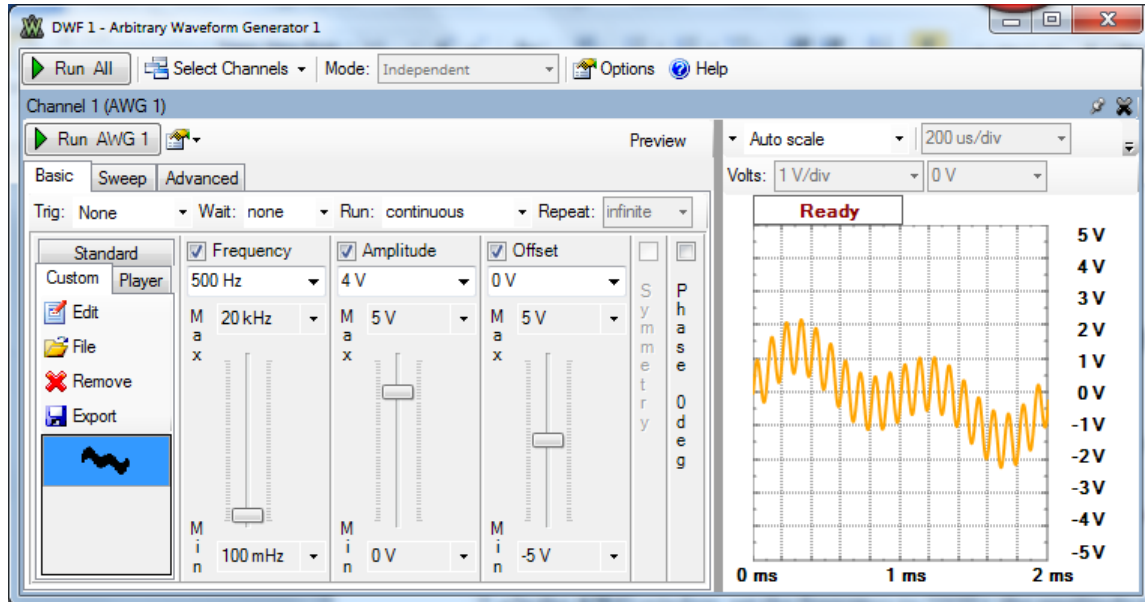


Figure A1. Sample AWG Editor window.

- In the **AWG** window, set the frequency to 500Hz, the amplitude to 4V, and the offset to 0V. Setting the frequency to 500Hz causes the waveform we created to be played back 500 times per second. This causes the frequencies of the sinusoids to be 500Hz, 1000Hz, and 10,000Hz, as desired. The AWG window should look approximately as shown in Figure A2; notice that the plot window indicates that the period of the signal is 2msec, consistent with the 500Hz frequency of the lowest frequency component in the signal. Click on “Run AWG” to play back the signal.



## Appendix B: Creating a Sinusoidal Sweep

- To create a sinusoidal sweep, click on the “Sweep” tab in the Arbitrary Waveform Generator Window. The following steps will set up the waveform as desired for this part of the lab project.
  - In the **Type** text box, select “Sine”. If necessary, choose “Sweep On” in the next column, and “Damp Off” in the last column.
  - Next to **Frequency**, set the range of frequencies used in your sweep, and the time duration over which the sweep occurs. In the leftmost text box, choose 100Hz (this is the initial frequency in the sweep). In the next textbox to the right, choose 10kHz (this is the final frequency in the sweep). In the last text box, choose 20ms (the duration of the sweep is ten seconds). Thus, the range of frequencies is from 100Hz to 10kHz, over a time of 20 milli-seconds. This pattern repeats itself until the user interrupts it.
  - Next to **Amplitude**, select 3.3V. (This will result in a maximum value of 3.3V and a minimum value of -3.3V.)
  - All other text boxes can be left at their default values.
  - The sinusoidal sweep we will apply is displayed in the plot window. The sinusoidal sweep we will apply is displayed in the plot window. Since our sweep changes frequency rapidly, the plot will not provide much detail as to the actual wave shape. If desired, you can adjust the plot parameters to get a better feeling as to the actual shape of the waveform. The **Arbitrary Waveform Generator** window, with the plot parameters chosen above, is shown in Fig. B1; your window should look.

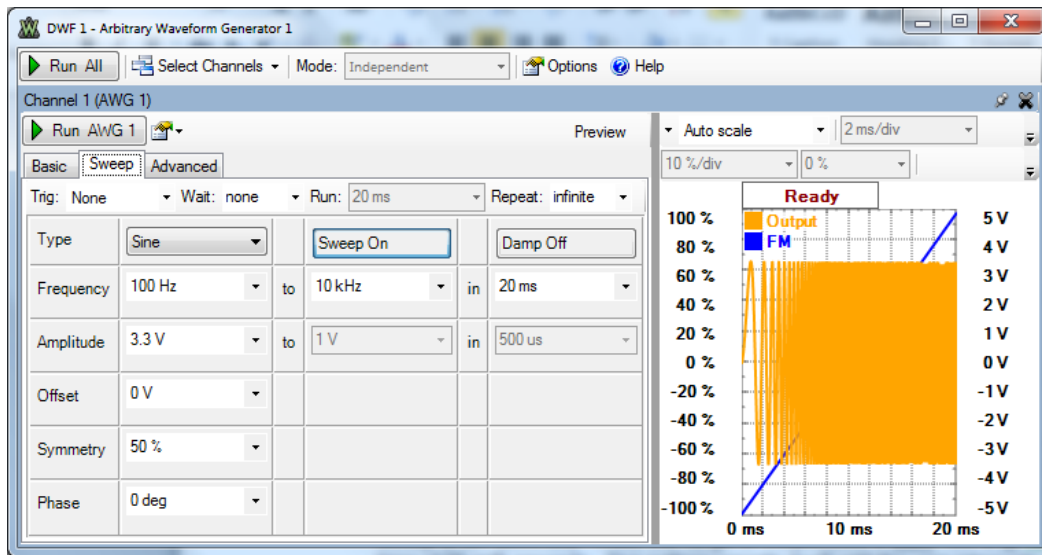


Figure B1. Example arbitrary waveform generator window.

2. Click on “Run AWG 1” to play back the signal.

## Real Analog Chapter 11: Lab Worksheets

### 11.2.1: Signals with Multiple Frequency Components (45 points total)

1. In the space below, provide:
  - a. The magnitude response of the circuit (as a function of  $R$ ,  $C$ , and  $\omega$ ), (3 pts)
  - b. The numerical values of the magnitude response at 500Hz, 1000Hz, and 10kHz, (3 pts) and
  - c. A brief comparison as to whether your magnitude response agrees with the limiting cases of equations (1) and (2). (2 pts)
  
2. Attach to this worksheet an image of the oscilloscope window, showing the voltages  $v_{IN}(t)$  and  $v_{OUT}(t)$ , resulting from the input voltage waveform of equation (3). (8 pts)
  
3. In the space below, briefly discuss the circuit's response to the input of equation (3), relative to you expectations based on the circuit's magnitude response and the input voltage. (6 pts)
  
  
  
  
  
  
  
  
  
  
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's response to the input of equation (3). (4 pts total)

TA Initials: \_\_\_\_\_

5. Attach to this worksheet an image of the oscilloscope window, showing the voltages  $v_{IN}(t)$  and  $v_{OUT}(t)$  resulting from the sinusoidal sweep input voltage waveform of part (b) of the lab procedures. (8 pts)

6. In the space below, briefly discuss the circuit's response to the sinusoidal sweep input voltage, relative to your expectations based on the circuit's magnitude response and the input voltage. (7 pts)
7. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's response to the sinusoidal sweep input. (4 pts total)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 11: Lab Projects

### 11.3.1: Passive RL Filter

This assignment concerns a passive filtering circuit consisting of a series combination of a resistor and an inductor. If the voltage across the inductor is taken as the filter output, the circuit will act as a high-pass filter. However, if we take the voltage across the resistor as the filter output, the circuit acts as a low-pass filter<sup>6</sup>. We will measure both the high-pass and low-pass characteristics of this circuit and compare them to our expectations based on the analytically determined frequency response.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of a passive electrical circuit
- Calculate the magnitude and phase responses of a passive electrical circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a passive first order filter

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of first order filter circuits

#### This lab exercise requires:

- Analog Discovery module
- Diligent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

|                 |                                                                                                                                                                       |
|-----------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>DEMO</b>     | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
| <b>ANALYSIS</b> | Analysis; include principle results of analysis in laboratory report.                                                                                                 |
| <b>SIM</b>      | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
| <b>DATA</b>     | Record data in your lab notebook.                                                                                                                                     |

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1. We will measure the response of both the voltages  $v_R(t)$  and  $v_L(t)$  to the input voltage  $v_{IN}(t)$  and plot the frequency response – both magnitude and phase – of both voltages. Obviously, since we are interested in the frequency response of the circuit, the input voltage will consist of sinusoids.

<sup>6</sup> If the energy in the input does not appear in the inductor voltage, it must be somewhere else (it can't simply disappear). Thus, if the inductor voltage stops low frequencies, the energy in these frequencies must show up somewhere else – in this case, it will be apparent in the resistor voltage (or, equivalently, the inductor current).

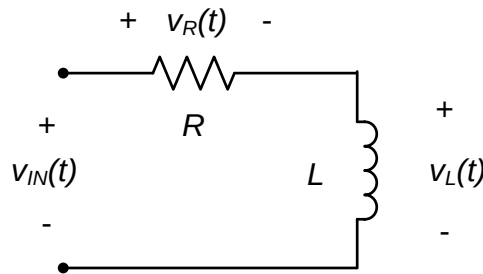


Figure 1. RL circuit.

**Pre-lab:****ANALYSIS**

Determine the frequency responses of both  $v_L(t)$  and  $v_R(t)$  in the circuit shown in Fig. 1, as a function of  $R$  and  $C$ . Use the voltage  $v_{IN}(t)$  as the input in both cases<sup>7</sup>. Calculate the cutoff frequency of the circuit (both outputs share the same cutoff frequency). Also calculate the DC gain and high frequency gain of both outputs.

**ANALYSIS**

Sketch the magnitude and phase responses of both outputs in the circuit of Fig. 1. Label your sketches with the DC gain, high frequency gain, and cutoff frequency.

**Lab Procedures:**

- a. Construct the circuit of Fig. 1, using  $R = 100\Omega$  and  $L = 1\text{mH}$ .

**DATA**

- i. Use your oscilloscope to measure both  $v_{IN}(t)$  and  $v_R(t)$ .

In order to measure the frequency response (amplitude and phase) of your circuit, use the function generator to apply sinusoidal inputs to the circuit. Apply inputs for at least the following frequencies:  $\frac{\omega_c}{10}$ ,  $\frac{\omega_c}{8}$ ,  $\frac{\omega_c}{4}$ ,  $\frac{\omega_c}{2}$ ,  $\omega_c$ ,  $2\omega_c$ ,  $4\omega_c$ ,  $8\omega_c$ , and  $10\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit, as determined in the pre-lab. For each frequency, record the frequency, the input voltage amplitude, the output voltage amplitude, and the time difference between the input and output sinusoids.

**DEMO**

- ii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

- b. Repeat the procedures of part (a):

**DATA**

- i. Except use your oscilloscope to measure  $v_{IN}(t)$  and  $v_L(t)$ . (In this case, the inductor voltage,  $v_L(t)$ , is our output.) Again, for each of the frequencies listed in part (a), record the frequency, the input voltage amplitude, the output voltage amplitude, and the time difference between the input and output sinusoids.

**DEMO**

- ii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**Post-lab Exercises:****SIM**

- a. Use your favorite mathematical analysis software package (MATLAB, Octave, Excel...) to plot the theoretical frequency response determined in the pre-lab, with  $R = 100\Omega$  and  $L = 1\text{mH}$ . Use  $v_R(t)$  as your output, and plot both magnitude and phase vs. frequency. Use a range of frequencies

<sup>7</sup> If the phasor representation of  $v_{IN}(t)$ ,  $v_L(t)$ , and  $v_R(t)$  are  $\underline{V}_{IN}$ ,  $\underline{V}_L$ , and  $\underline{V}_R$ , respectively, we are calculating  $\frac{V_L}{V_{IN}}$  and  $\frac{V_R}{V_{IN}}$ .

from  $\frac{\omega_c}{10}$  to  $10\omega_c$ . Overlay the data you acquired in part (a) of the lab procedures on your plot. Comment on the agreement between the data and the theoretical response. Does the circuit behave as a high-pass or low-pass filter when the output is the resistor voltage?

**SIM**

- b. Use your favorite mathematical analysis software package (MATLAB, Octave, Excel...) to plot the theoretical frequency response determined in the pre-lab, with  $R = 100\Omega$  and  $L = 1\text{mH}$ . Use  $v_L(t)$  as your output, and plot both magnitude and phase vs. frequency. Use a range of frequencies from  $\frac{\omega_c}{10}$  to  $10\omega_c$ . Overlay the data you acquired in part (b) of the lab procedures on your plot. Comment on the agreement between the data and the theoretical response. Does the circuit behave as a high-pass or low-pass filter when the output is the inductor voltage?

## Real Analog Chapter 11: Lab Worksheets

### 11.3.1: Passive RL Filter (55 points total)

1. In the space below, provide the frequency responses of both  $v_L(t)$  and  $v_R(t)$  in the circuit shown in Fig. 1. (5 pts)
2. In the space below, provide the cutoff frequency, the DC gain, and the high frequency gain of both of the frequency responses of part 1. (6 pts)
3. Attach to this worksheet sketches of the magnitude and phase responses of both frequency responses of part 1. (4 pts)
4. In the space below, provide a table showing the input frequencies, the amplitudes of the voltages  $v_{IN}(t)$  and  $v_R(t)$ , and the time difference between  $v_{IN}(t)$  and  $v_R(t)$  for the frequencies specified in part (a) of the lab procedures. (10 pts)
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your signal acquisition from the sensor. (4 pts total)

TA Initials: \_\_\_\_\_

6. In the space below, provide a table showing the input frequencies, the amplitudes of the voltages  $v_{IN}(t)$  and  $v_L(t)$ , and the time difference between  $v_{IN}(t)$  and  $v_R(t)$  for the frequencies specified in part (a) of the lab procedures. (10 pts)

7. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your signal acquisition from the sensor. (4 pts total)

**TA Initials:** \_\_\_\_\_

8. Attach to this worksheet a plot of the theoretical magnitude and phase response with  $v_R(t)$  as the circuit output, overlaid with the data from part 4 above. In the space below, comment on the agreement between the theoretical frequency response and the measured data. Also characterize the circuit as either a high-pass or a low-pass filter. (6 pts)
9. Attach to this worksheet a plot of the theoretical magnitude and phase response with  $v_L(t)$  as the circuit output, overlaid with the data from part 6 above. In the space below, comment on the agreement between the theoretical frequency response and the measured data. Also characterize the circuit as either a high-pass or a low-pass filter. (6 pts)

## Real Analog Chapter 11: Lab Projects

### 11.3.2: Passive RC Filter

This assignment concerns a passive filtering circuit consisting of a series combination of a resistor and a capacitor. If the voltage across the capacitor is taken as the filter output, the circuit will act as a low-pass filter. However, if we take the voltage across the resistor as the filter output, the circuit acts as a high-pass filter<sup>8</sup>. We will measure both the high-pass and low-pass characteristics of this circuit and compare them to our expectations based on the analytically determined frequency response.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of a passive electrical circuit
- Calculate the magnitude and phase responses of a passive electrical circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a passive first order filter

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of first order filter circuits

#### This lab exercise requires:

- Analog Discovery module
- Diligent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

##### DEMO

Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.

##### ANALYSIS

Analysis; include principle results of analysis in laboratory report.

##### SIM

Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.

##### DATA

Record data in your lab notebook.

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1. We will measure the response of both the voltages  $v_R(t)$  and  $v_C(t)$  to the input voltage  $v_{IN}(t)$  and plot the frequency response – both magnitude and phase – of both voltages. Obviously, since we are interested in the frequency response of the circuit, the input voltage will consist of sinusoids.

<sup>8</sup> If the energy in the input does not appear in the capacitor voltage, it must be somewhere else (it can't simply disappear). Thus, if the capacitor voltage stops low frequencies, the energy in these frequencies must show up somewhere else – in this case, it will be apparent in the resistor voltage (or, equivalently, the capacitor current).

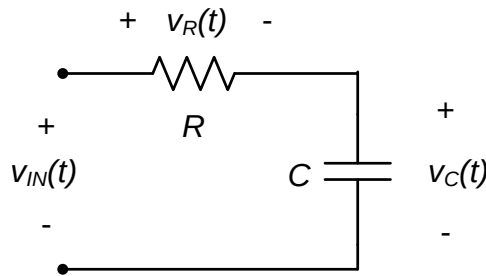


Figure 1. RC circuit.

**Pre-lab:****ANALYSIS**

Determine the frequency responses of both  $v_C(t)$  and  $v_R(t)$  in the circuit shown in Fig. 1, as a function of  $R$  and  $C$ . Use the voltage  $v_{IN}(t)$  as the input in both cases<sup>9</sup>. Calculate the cutoff frequency of the circuit (both outputs share the same cutoff frequency). Also calculate the DC gain and high frequency gain of both outputs.

**ANALYSIS**

Sketch the magnitude and phase responses of both outputs in the circuit of Fig. 1. Label your sketches with the DC gain, high frequency gain, and cutoff frequency.

**Lab Procedures:****DATA**

- a. Construct the circuit of Fig. 1, using  $R=2.2\text{k}\Omega$  and  $C = 4.7\text{nF}$ 
  - i. Use your oscilloscope to measure both  $v_{IN}(t)$  and  $v_R(t)$ .

In order to measure the frequency response (amplitude and phase) of your circuit, use the function generator to apply sinusoidal inputs to the circuit. Apply inputs for at least the following frequencies:  $\frac{\omega_c}{10}$ ,  $\frac{\omega_c}{8}$ ,  $\frac{\omega_c}{4}$ ,  $\frac{\omega_c}{2}$ ,  $\omega_c$ ,  $2\omega_c$ ,  $4\omega_c$ ,  $8\omega_c$ , and  $10\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit, as determined in the pre-lab. For each frequency, record the frequency, the input voltage amplitude, the output voltage amplitude, and the time difference between the input and output sinusoids.

**DEMO**

- ii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**DATA**

- b. Repeat the procedures of part (a):
  - i. Except use your oscilloscope to measure  $v_{IN}(t)$  and  $v_C(t)$ . (In this case, the capacitor voltage,  $v_C(t)$ , is our output.) Again, for each of the frequencies listed in part (a), record the frequency, the input voltage amplitude, the output voltage amplitude, and the time difference between the input and output sinusoids.

**DEMO**

- ii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**Post-lab Exercises:****SIM**

- a. Use your favorite mathematical analysis software package (MATLAB, Octave, Excel...) to plot the theoretical frequency response determined in the pre-lab, with  $R = 2.2\text{k}\Omega$  and  $C = 4.7\text{nF}$ . Use  $v_R(t)$  as your output, and plot both magnitude and phase vs. frequency. Use a range of frequencies

<sup>9</sup> If the phasor representation of  $v_{IN}(t)$ ,  $v_C(t)$ , and  $v_R(t)$  are  $\underline{V}_{IN}$ ,  $\underline{V}_C$ , and  $\underline{V}_R$ , respectively, we are calculating  $\frac{\underline{V}_C}{\underline{V}_{IN}}$  and  $\frac{\underline{V}_R}{\underline{V}_{IN}}$ .

from  $\frac{\omega_c}{10}$  to  $10\omega_c$ . Overlay the data you acquired in part (a) of the lab procedures on your plot. Comment on the agreement between the data and the theoretical response. Does the circuit behave as a high-pass or low-pass filter when the output is the resistor voltage?

**SIM**

- b. Use your favorite mathematical analysis software package (MATLAB, Octave, Excel...) to plot the theoretical frequency response determined in the pre-lab, with  $R = 100\Omega$  and  $L = 1\text{mH}$ . Use  $v_c(t)$  as your output, and plot both magnitude and phase vs. frequency. Use a range of frequencies from  $\frac{\omega_c}{10}$  to  $10\omega_c$ . Overlay the data you acquired in part (b) of the lab procedures on your plot. Comment on the agreement between the data and the theoretical response. Does the circuit behave as a high-pass or low-pass filter when the output is the capacitor voltage?

## Real Analog Chapter 11: Lab Worksheets

### 11.3.2: Passive RC Filter (50 points total)

1. In the space below, provide the frequency responses of both  $v_C(t)$  and  $v_R(t)$  in the circuit shown in Fig. 1. (4 pts)
2. In the space below, provide the cutoff frequency, the DC gain, and the high frequency gain of both of the frequency responses of part 1. (6 pts)
3. Attach to this worksheet sketches of the magnitude and phase responses of both frequency responses of part 1. (4 pts)
4. In the space below, provide a table showing the input frequencies, the amplitudes of the voltages  $v_{IN}(t)$  and  $v_R(t)$ , and the time difference between  $v_{IN}(t)$  and  $v_R(t)$  for the frequencies specified in part (a) of the lab procedures. (10 pts)
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (4 pts total)

TA Initials: \_\_\_\_\_

6. In the space below, provide a table showing the input frequencies, the amplitudes of the voltages  $v_{IN}(t)$  and  $v_C(t)$ , and the time difference between  $v_{IN}(t)$  and  $v_C(t)$  for the frequencies specified in part (a) of the lab procedures. (10 pts)
7. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (4 pts total)
- TA Initials:** \_\_\_\_\_
8. Attach to this worksheet a plot of the theoretical magnitude and phase response with  $v_R(t)$  as the circuit output, overlaid with the data from part 4 above. In the space below, comment on the agreement between the theoretical frequency response and the measured data. Also characterize the circuit as either a high-pass or a low-pass filter. (4 pts)
9. Attach to this worksheet a plot of the theoretical magnitude and phase response with  $v_C(t)$  as the circuit output, overlaid with the data from part 6 above. In the space below, comment on the agreement between the theoretical frequency response and the measured data. Also characterize the circuit as either a high-pass or a low-pass filter. (4 pts)

## Real Analog Chapter 11: Lab Projects

### 11.3.3: Active Low-pass Filter

Active filter circuits provide a number of the same advantages as our previous active circuits:

- As we saw in Lab projects 7.5.1 and 7.5.3, applying a load to an active circuit can have less of effect on the circuit's behavior than if the circuit were passive
- For an active circuit, the output voltage can be larger than the input voltage – the active circuit can amplify a signal.
- The high input impedance of operational amplifiers can help reduce the effect that the passive filter has on the signal source. This can be particularly important in instrumentation applications; since many sensors provide extremely limited power, it is important to process the sensor's output with a circuit which draws little or no power from the sensor.

In this lab project, we will design an active low-pass filter to provide a desired DC gain, cutoff frequency, and input resistance. The circuit used in this project also *inverts* the input – the DC output voltage has the opposite sign as the input voltage.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of an active electrical circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Calculate the magnitude and phase responses of an active electrical circuit
- Design an active filter to provide a desired DC gain, cutoff frequency, and input impedance

#### This lab exercise requires:

- Analog Discovery module
- Diligent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

|                 |                                                                                                                                                                       |
|-----------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>DEMO</b>     | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
| <b>ANALYSIS</b> | Analysis; include principle results of analysis in laboratory report.                                                                                                 |
| <b>SIM</b>      | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
| <b>DATA</b>     | Record data in your lab notebook.                                                                                                                                     |

#### General Discussion:

The circuit of Fig. 1 is an inverting low pass filter. The magnitude response of the circuit is

$$\left| \frac{V_{OUT}}{V_{IN}} \right| = - \frac{\frac{R_2}{R_1}}{\sqrt{1 + (\omega R_2 C)^2}} \quad \text{Eq. 1}$$

In this assignment, we will choose values of  $R_1$ ,  $R_2$ , and  $C$  in the circuit of Fig. 1 to meet design requirements set on:

- **Input resistance:** the input resistance is the ratio of input voltage to input current for a circuit. In the circuit of Fig. 1, the input resistance essentially specifies the resistance  $R_1$ .

- DC gain: the DC gain is, from equation (1), the ratio of  $R_2$  to  $R_1$ . Once  $R_1$  is determined from the input impedance requirement, the DC gain specifies the required value for  $R_2$ .
- Cutoff frequency: once  $R_2$  is known, the cutoff frequency requirement specifies the value of the capacitor  $C$ .

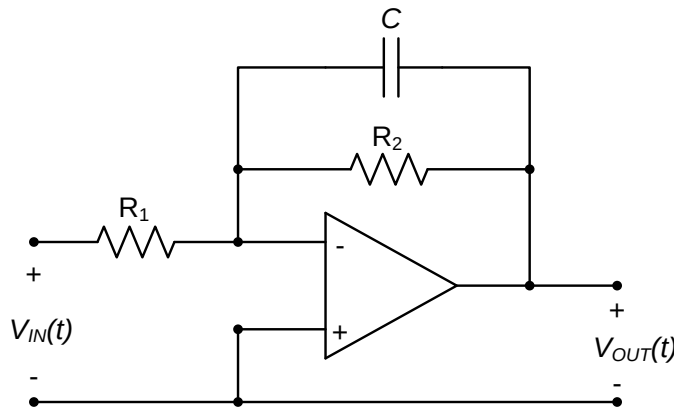


Figure 1. Inverting low-pass filter circuit.

### Pre-lab:

#### ANALYSIS

Design the circuit of Fig. 1 (e.g. choose values for  $R_1$ ,  $R_2$ , and  $C$ ) to meet the following design requirements:

- Input impedance  $\geq 10\text{k}\Omega$
- DC gain = 2
- Cutoff frequency = 10kHz

### Lab Procedures:

#### DATA

Construct the circuit you designed in the pre-lab. Be sure to measure the actual values.

#### DATA

- Measure the magnitude response of the circuit over a range of frequencies  $\frac{\omega_c}{10} < \omega < 10\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>10</sup>. Make sure you measure the response for at least six different frequencies.

#### DEMO

- Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

#### ANALYSIS

- From your measured data, determine the actual cutoff frequency and DC gain of your circuit. Compare your measured values to the design requirements. Comment on the differences between the design requirements and your measured values.

<sup>10</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.

## Real Analog Chapter 11: Lab Worksheets

### 11.3.3: Active Low-pass Filter (40 points total)

1. In the space below, provide the resistance and capacitance values you chose to meet the given design requirements. Also provide the expected values input resistance, cutoff frequency, and DC gain for your circuit. Compare these values to the design specifications. (10 pts)
2. In the space below, tabulate the input frequencies and the magnitude response of your circuit at each of these frequencies. (Note: feel free to include additional data in your table. It may result in partial credit.) (13 pts)
3. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (7 pts total)

**TA Initials:** \_\_\_\_\_

4. In the space below, provide the measured cutoff frequency and DC gain of your circuit (based on your magnitude response of part 2 above). Briefly compare your measured values to the design requirements (a percent difference between the two is always good). Comment on potential reasons for the differences between the design requirements and your measured values. (10 pts)

## Real Analog Chapter 11: Lab Projects

### 11.3.4: Signal Conditioning – Audio Application

When making time-varying measurements, the sensor being used often has at least a few undesirable characteristics. Common shortcomings of sensors are lower than desired sensitivity, excessive noise levels, and DC offsets. *Frequency selective circuits* are often used to *condition* the sensor's output signal to reduce the effects of these shortcomings. *Low-pass filters*, for example, can be used to increase the sensor's low-frequency sensitivity while reducing the high-frequency noise components in the sensor output signal.

In this lab assignment, we will condition the output from a microphone. The microphone is a sensor which converts acoustic waves to electrical signals. The microphone we will be using has fairly low output levels (on the order of 20mV) and a relatively large DC offset. Our goal is to design a filter to amplify the frequencies of interest – those frequencies associated with audio signals – and suppress the DC offset.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of a active electrical circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Categorize frequency-selective circuits as high pass or low pass filters
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Design an active filter high pass filter to provide a desired high frequency gain, cutoff frequency, and input impedance
- Use a frequency selective filter to process a microphone's output signal

#### This lab exercise requires:

- Analog Discovery module
- Diligent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

|                 |                                                                                                                                                                       |
|-----------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>DEMO</b>     | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
| <b>ANALYSIS</b> | Analysis; include principle results of analysis in laboratory report.                                                                                                 |
| <b>SIM</b>      | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
| <b>DATA</b>     | Record data in your lab notebook.                                                                                                                                     |

## 1 Microphone

A microphone converts sound to an electrical signal. We will be using an ADMP504 microphone in this lab project. The ADMP504 microphone is in a surface-mount package; this package does not allow direct implementation in our solderless breadboard. The ADMP504 part in your analog parts kit has been mounted onto a circuit board with pins which can be inserted directly in your breadboard. Figure 1 provides top and bottom views of this circuit board. The ADPM504 microphone is visible on the bottom view of the board, as shown in Fig. 1(a). The pin indicator showing the location of pin 1 can also be seen in the bottom view. Pins are consecutively numbered clockwise from pin 1, also as shown in Fig. 1(a). In the top view of the chip, a small hole is visible – this hole allows

sound to contact the ADMP504 sensor. The hole, and the pin locations seen in the top view are shown in Figure 1(b).

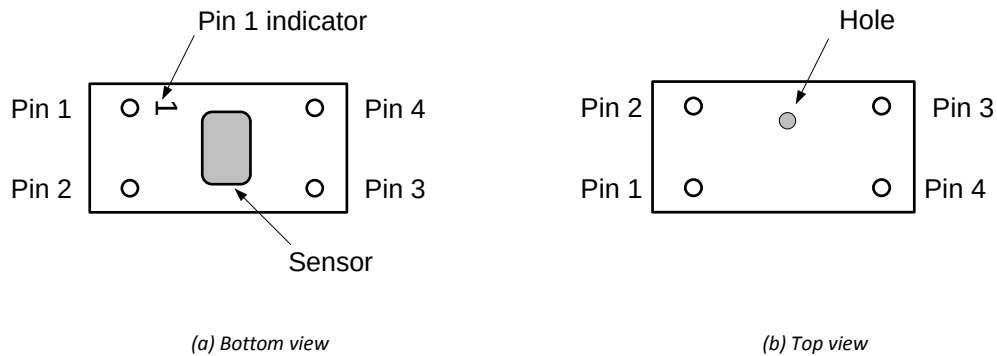


Figure 1. Top and bottom views of the analog parts kit chip on which the microphone is mounted.

The descriptions of the pins shown in Fig. 1 are as follows:

- **Pin 1:** Output voltage. This pin provides a voltage which indicates the audio signal applied to the microphone.
- **Pin 2:** Ground.
- **Pin 3:** Voltage source,  $V_{DD}$ . This power source is necessary for the microphone to work. We will use  $V_{DD} = 3.3V$ .
- **Pin 4:** Not connected.

Additional information about the ADMP504 microphone can be found on the Analog Devices website, <http://www.analog.com>.

### Pre-lab:

None

### Lab Procedures:

#### DATA

- In this portion of the lab project, we will measure the response of the sensor and use this data to determine the amplification necessary to provide us with an output signal of the desired sensitivity.

Use the arbitrary waveform generator to apply 3.3V to pin 3 of the microphone board<sup>11</sup>. Connect pin 2 of the chip to ground. Use one channel of your oscilloscope to measure the voltage difference between pin 1 and ground. Make a sound<sup>12</sup> and verify that you are receiving a signal on the oscilloscope from the sensor. At this stage, you will probably want to use a fairly large time base on your oscilloscope (greater than 100msec/division).

#### DATA

- With a relatively large vertical scale (500 mV/division or so) measure and record the DC offset of the sensor.

#### DATA

- With a relatively large vertical offset (approximately the negative of the DC voltage measured in part i above), set the vertical scale on your oscilloscope channel to be on the order of 10 – 20 msec/division. Measure and record the amplitude of the time-varying signal resulting from your sound source. It may be useful to decrease the time base on your

<sup>11</sup> The waveform generator can be used to apply a constant voltage if the amplitude is set to 0V and the offset is set to 3.3V.

<sup>12</sup> Whistling or snapping your fingers tend to be good sound sources; they are fairly repeatable.

oscilloscope significantly in order to accurately measure this value. Setting a trigger to acquire the waveform may assist you once you have reduced the time base.

**ANALYSIS**

- b. In Part II of this project, we want design a filter which uses the sensor output voltage to produce a minimum 1V peak-to-peak signal with no DC offset. Use the time varying data acquired in part ii above to estimate the amplification necessary to produce the desired 1V amplitude.

## 2 Signal Conditioning Circuit

The circuit of Fig. 2 is an inverting high pass filter. The frequency response of the circuit is:

$$\frac{V_{OUT}}{V_{IN}} = -\frac{R_2}{R_1} \frac{j\omega}{j\omega + \frac{1}{R_1 C}} \quad \text{Eq. 1}$$

So that the magnitude response is:

$$\left| \frac{V_{OUT}}{V_{IN}} \right| = -\frac{R_2}{R_1} \frac{\omega}{\sqrt{\omega^2 + \left(\frac{1}{R_1 C}\right)^2}} \quad \text{Eq. 2}$$

In this assignment, we will choose values of  $R_1$ ,  $R_2$ , and  $C$  in the circuit of Fig. 2 to meet design requirements set on:

- **Input resistance:** the input resistance is the ratio of input voltage to input current for a circuit. In the circuit of Fig. 1, the input resistance is dependent upon both the capacitance and the frequency. However, at high frequencies, the capacitor behaves like a short circuit, and the input resistance is essentially the resistance  $R_1$ .
- **High frequency gain:** the high frequency gain (the gain as  $\omega \rightarrow \infty$ ) is, from equation (2), the ratio of  $R_2$  to  $R_1$ . Once  $R_1$  is determined from the input resistance requirement, the high frequency gain specifies the required value for  $R_2$ .
- **Cutoff frequency:** once  $R_2$  is known, the cutoff frequency requirement specifies the value of the capacitor  $C$ .

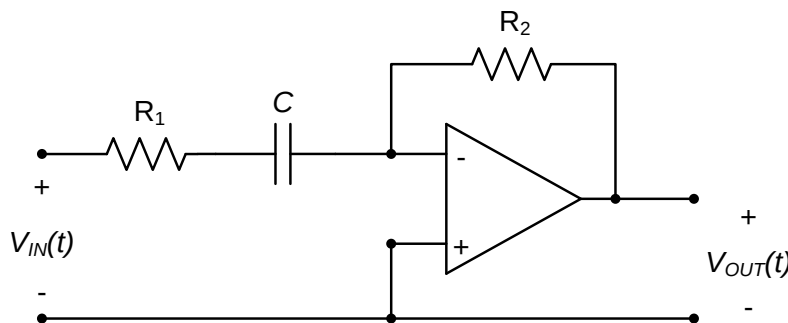


Figure 1. Inverting high-pass filter circuit.

### Pre-lab:

**ANALYSIS**

Design the circuit of Fig. 1 (e.g. choose values for  $R_1$ ,  $R_2$ , and  $C$ ) to meet the following design requirements:

- $R_1 \geq 2k\Omega$ . (This essentially sets the input resistance for the filter.)
- High frequency gain (gain as  $\omega \rightarrow \infty$ ) = Amplification value necessary to produce 1V peak-to-peak output voltage, as determined in the Lab Procedures for Part I.
- Cutoff frequency  $\leq 500\text{Hz}$

**ANALYSIS**

Determine the response of the circuit of Fig. 1 to a DC input voltage. What will the circuit's response be to the DC voltage provided by the sensor?

**Lab Procedures:****DATA**

Construct the circuit you designed in the pre-lab. Be sure to measure the actual values.

**DATA**

- i. Measure the magnitude response of the circuit over a range of frequencies  $\frac{\omega_c}{10} < \omega < 10\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>13</sup>. Make sure you measure the response for at least six different frequencies.

**DEMO**

- ii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**ANALYSIS**

- iii. From your measured data, determine the actual cutoff frequency and high frequency gain of your circuit. Compare your measured values to the design requirements. Comment on the differences between the design requirements and your measured values.

### 3 Overall System Integration

We will now integrate the signal conditioning circuit designed and built in Part II with the sensor of Part I. The goal is to amplify the “important” part of the response of the sensor – the time-varying signal corresponding to the sound – and remove the undesirable DC level in the sensor output. One possibly important drawback to this approach, of course, is that desirable lower-frequency information will also be removed from the data.

**Pre-lab:**

None

**Lab Procedures:****DATA**

- i. Apply the sensor output voltage to the input terminals of the signal conditioning circuit,  $V_{IN}(t)$ . Using the oscilloscope, measure both  $V_{in}(t)$  from the sensor and the signal conditioning unit's output voltage,  $V_{OUT}(t)$  in Fig. 2. Make a sound (clap your hands, whistle,...) and display the resulting waveform on the oscilloscope main window. Record an image of the oscilloscope window, showing the voltages  $V_{IN}(t)$  and  $V_{OUT}(t)$ . Comment on your results relative to your expectations.

**DEMO**

- ii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

<sup>13</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.

## Real Analog Chapter 11: Lab Worksheets

### 11.3.4: Signal Conditioning – Audio Application (100 points total)

## 1. Microphone (20 points total)

1. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your signal acquisition from the microphone. (8 pts)

TA Initials: \_\_\_\_\_

2. In the space below, provide the DC offset and the amplitude of the time-varying portion of the microphone output. (7 pts)

3. In the space below, provide your estimate of the amplification necessary to provide the desired 1V output amplitude. (5 pts)

## 2. Signal Conditioning Circuit (50 points total)

1. Provide below a schematic showing your signal conditioning circuit. Include desired resistance and capacitance values (3 pts)

2. Attach, to this worksheet, analyses which support your circuit design. These analyses should include (at a minimum) your desired high frequency gain, cutoff frequency, and input resistance; chosen resistance and capacitance values, and how these values were chosen. (15 pts)
  
3. In the space below, provide a table containing the following. (15 pts)
  - The frequencies at which the response data was measured
  - The measured data characterizing your signal processing circuit (the input and output voltage amplitudes; time difference)
  - The calculated amplitude and phase response.
  
4. Attach, to this worksheet, a plot of the measured gain and phase vs. frequency for your signal conditioning circuit. In the space below, provide a discussion of measured vs. expected performance. (7 pts)
  
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (10 pts total)

TA Initials: \_\_\_\_\_

### 3. Overall System Integration (30 points total)

1. In the space below sketch your overall system, showing electrical connections between the microphone and the signal conditioning circuit. (3 pts)
2. Attach, to this worksheet, an image of the oscilloscope window, showing the input and output of signal conditioning circuit. In the space below, discuss your results relative to expectations. (10 pts)
3. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (17 pts total)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 11: Lab Projects

### 11.3.5: Signal Conditioning – Vibration Measurement

When making time-varying measurements, the sensor being used often has lower than desired sensitivity and a higher than desired noise level. *Frequency selective circuits* are therefore often used to *condition* the sensor's output signal. *Low-pass filters*, for example, can be used to increase the sensor's low-frequency sensitivity while reducing the high-frequency noise components in the sensor output signal.

In this lab assignment, we will condition the output from a piezoelectric sensor with a low-pass filter. Our goal will be to design a filter to amplify the frequencies of interest – those frequencies below the fundamental vibration frequency of the beam – and suppress higher-frequency components of the signal, which we will interpret as noise.

#### Before beginning this lab, you should be able to:

- Perform frequency-domain analysis of electrical circuits
- Represent sinusoidal signals in phasor form
- Analyze operational amplifier based electrical circuits

#### After completing this lab, you should be able to:

- Use a piezoelectric sensor to measure the vibration of a cantilever beam.
- Estimate the fundamental frequency in a measured signal
- Design and implement a first order low-pass signal conditioning circuit
- Measure the frequency response (gain and phase) of a system

#### This lab exercise requires:

- Analog Discovery module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

**DEMO**

Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.

**ANALYSIS**

Analysis; include principle results of analysis in laboratory report.

**SIM**

Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.

**DATA**

Record data in your lab notebook.

## 1 Piezoelectric Sensor Vibration

The *dynamic response* (or time-varying response) of a mechanical system can be extremely important in the determination of the structural loads in the system. The dynamic response can often be the dominant contributing factor in the stresses in the structure and, as such, can be the primary factor in a structure's failure. The famous Tacoma Narrows bridge failure was caused by the bridge's dynamic response to wind loads. A loading condition is considered to be dynamic if the loading condition changes relatively rapidly compared to how quickly the structure can respond to the load. "Plucking" a guitar string or striking a tuning fork, for example, are dynamic loading conditions, since they set up responses which persist much longer than the actual application of the input.

A system's dynamic response is often interpreted in terms of *vibration*. Vibration is essentially an oscillatory mechanical displacement. In our guitar string and tuning fork examples above, it is apparent that the systems oscillate as a result of the applied input. In general, structural vibrations do not consist of a single, sinusoidal, frequency component. (The tuning fork example is an exception to this rule; tuning forks are designed to vibrate at a single frequency, thus producing a pure auditory tone.) If multiple frequency components are present in a system's dynamic response, these frequency components are generally described in terms of the system's *modes* of vibration. The modes of vibration of the system are the *independent* ways in which the natural response of the system can vibrate. Some modes will be more important than others, from the standpoint of their contribution to the overall dynamic displacement of the system; the most important modes are often called the *dominant* or *fundamental* modes. In the guitar string example above, there are a number of frequencies which contribute to the sound we hear from the guitar (the sound is not a pure tone); these contributions are due to the different vibrational modes of the guitar string.

Vibrations are often measured using piezoelectric sensors. Piezoelectric devices produce a voltage when they are deflected. By bonding the device to a vibrating system, the output voltage from the sensor will indicate the vibration levels the system is experiencing. In this part of the lab assignment, we will induce a vibration in the piezoelectric sensor itself and measure its response. This data will be used to estimate the fundamental mode of vibration of the sensor. The design of the low-pass filter we will use later as our signal conditioning system will be based on this fundamental mode of vibration.

### Pre-lab:

Read the material in Appendix A relative to piezoelectric sensors.

### Lab Procedures:

- a. Connect the leads of one channel of your oscilloscope to the sensor contacts (the tabs protruding from the piezoelectric sensor). "Flick" the sensor and verify that you are receiving a signal on the oscilloscope from the sensor. Adjust the time and amplitude scales on your oscilloscope until the oscilloscope displays what you feel is a reasonable representation of the sensor's deflection as a function of time on the oscilloscope.

#### DEMO

- b. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.
- c. Verify that the piezoelectric sensor provides no steady-state response to a constant input. To do this, use the oscilloscope to monitor the output voltage from the sensor as above. Bend the sensor to a constant deflection (bend it and leave it bent) and monitor the output of the sensor on your oscilloscope. The sensor output voltage should return to zero volts, even though the sensor is still deflected. Comment on your results in your lab notebook. Include a qualitative discussion as to what frequencies the sensor responds to. What type of frequency-selective circuit does the sensor resemble (low-pass, high-pass, band-pass, etc.)?

#### ANALYSIS

- d. Measure the natural response of the sensor. To do this, hold the end of the sensor to which the leads are attached and deflect the other end of the sensor slightly. This corresponds to an initial condition on the sensor deflection. Release the sensor tip suddenly, this allows the sensor to respond to this initial condition. Since no external forces act on the sensor after it is released, this corresponds to the natural response of the sensor.

#### DATA

Acquire the natural response waveform on your oscilloscope. (You may wish to use the **run/stop** button or the **single sequence** capability on your oscilloscope to do this. **Run/stop** will require you to **manually stop the oscilloscopes data acquisition when the signal is acquired**; single sequence will use the oscilloscope trigger to acquire the waveform) Save the waveform to a file.

#### ANALYSIS

Estimate the dominant frequency in the signal and note it in your lab notebook. (Note: this will correspond to the fundamental mode of the sensor; this *mode shape* consists of the sensor

vibrating in a “shape” that we typically associate with the motion of a swimming pool diving board, as shown in Fig. 1.) Also note that the recorded waveform contains frequency components other than the dominant frequency; the signal is not a pure sinusoid with a single frequency.

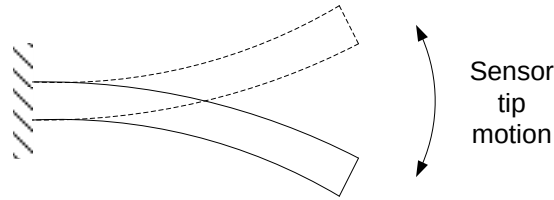


Figure 1. Dominant mode shape for sensor vibration (side view).

## 2 Signal Conditioning Circuit

The voltage output from many measurement systems suffers from two primary shortcomings:

1. The output voltage can be noisy.
2. The sensitivity of the output voltage can be lower than desired.

To overcome the above problems, we will design and implement an electrical circuit to *condition* the output voltage from the sensor before displaying the signal on the oscilloscope. The circuit we will use performs two primary functions, each of which is intended to compensate for one of the above shortcomings. The circuit will:

1. *Low-pass filter* the output signal from the sensor. This will reduce the high-frequency noise in the sensor’s output voltage.
2. *Amplify* the output signal from the sensor. This will increase the sensitivity of the overall measurement.

We will implement the above operations using the circuit shown in Fig. 2. The frequency response of the circuit shown in Fig. 2 is:

$$\frac{V_{OUT}}{V_{IN}} = \frac{R_1 + R_2}{R_1} \frac{\frac{1}{R_3 C}}{j\omega + \frac{1}{R_3 C}} \quad \text{Eq. 1}$$

So the amplitude response of the circuit is:

$$\left| \frac{V_{OUT}}{V_{IN}} \right| = \frac{R_1 + R_2}{R_1} \frac{\frac{1}{R_3 C}}{\sqrt{\omega^2 + \frac{1}{R_3^2 C^2}}} \quad \text{Eq. 2}$$

The amplitude response of the overall signal conditioning circuit of Fig. 2 is shown in Fig. 3. The low frequency gain (as  $\omega \rightarrow 0$ ) of the circuit is  $\frac{R_1 + R_2}{R_1}$ , and the filter’s output goes to zero at high frequencies (as  $\omega \rightarrow \infty$ ). The *cutoff frequency* of the circuit indicates at what frequency the filter’s output begins to decrease rapidly; for our circuit, the cutoff frequency is  $\omega_c = \frac{1}{R_3 C}$ . The DC gain can be used to amplify the output of the piezoelectric sensor in the pass band, while the stop band can be used to eliminate the noise in the signal at high frequencies. (Signals entering the circuit with frequencies below the cutoff frequency – the pass band – are amplified; signals entering the circuit with frequencies above the cutoff frequency – the stop band – are attenuated.)

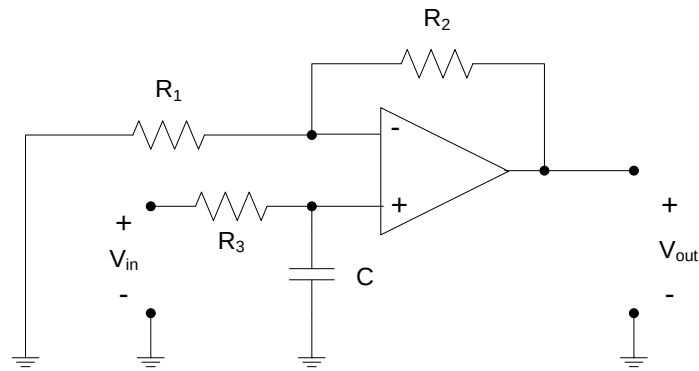


Figure 2. Signal conditioning circuit.

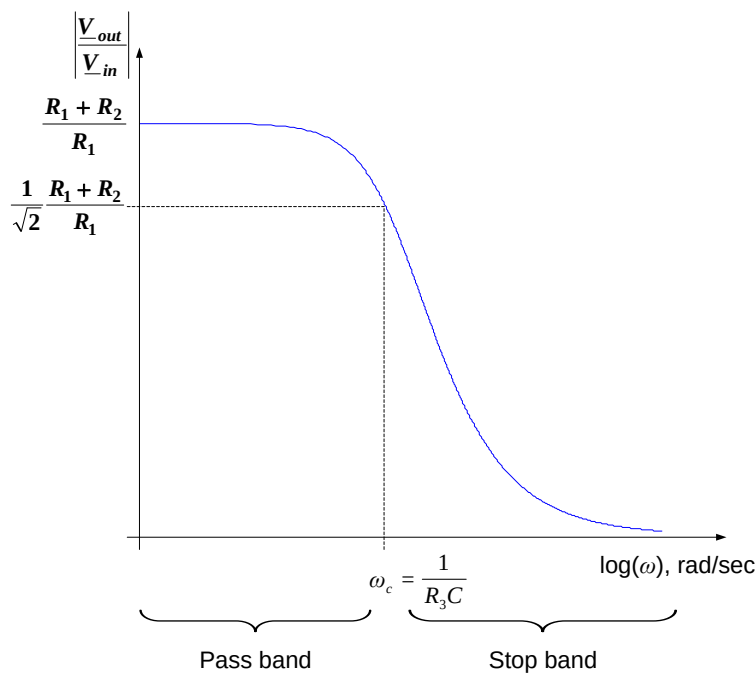


Figure 3. Signal conditioning circuit amplitude response.

**Pre-lab:****ANALYSIS**

- a. Using equation (1) as your starting point, show that the amplitude response of the circuit of Fig. 2 is as provided in equation (2)

**ANALYSIS**

- b. Using equation (1) as your starting point, determine the phase response of the circuit of Fig. 3

**ANALYSIS**

- c. Determine the cutoff frequency of the circuit of Fig. 2. What is the gain and phase of the circuit at the cutoff frequency?

**Lab Procedures:****ANALYSIS**

- a. Design a circuit like that shown in Fig. 2 (e.g. choose  $R_1$ ,  $R_2$ ,  $R_3$  and  $C$ ) to provide a DC gain of approximately two and a cutoff frequency of roughly twice the sensor's dominant vibration frequency as determined in Part I of this lab assignment.

**DATA**

- b. Construct the circuit you designed in part (a). Record actual resistance and capacitance values.

**DATA**

- c. Measure the frequency response (amplitude and phase) of your circuit. To do this, use the function generator to apply sinusoidal inputs to the circuit. Record input voltage amplitude,

output voltage amplitude, the time difference between the two, and frequency for at least 5 or 6 values of frequency; make sure you use a range of frequencies which includes your cutoff frequency. Note: Appendix B of this lab assignment provides tips relative to gain and phase measurement.

**DEMO**

- d. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**ANALYSIS**

- e. Calculate the gain and phase of your circuit for the frequencies measured in part (c). Plot the amplitude response in your lab notebook; use a logarithmic scale on the frequency axes of your plots. Discuss your measured response vs. the expected response from your pre-lab calculations. In particular, compare the actual and expected gain and phase at low frequencies, high frequencies, and the cutoff frequency.

### 3 Overall System Integration

We will now integrate the signal conditioning circuit designed and built in Part II with the sensor of Part I. Since our signal conditioning circuit's cutoff frequency is approximately twice the natural frequency of the sensor, most of the low frequency components in the sensor's output should lie within the pass band of the signal conditioning circuit. The goal is to amplify the "important" part of the response of the sensor and remove the (hopefully) less significant higher frequency content in the sensor output – high frequency noise, for example, will be removed by the low-pass filter. One possibly important drawback to this approach, of course, is that desirable higher-frequency information will also be removed from the data.

#### Pre-lab:

None

#### Lab Procedures:

**DATA**

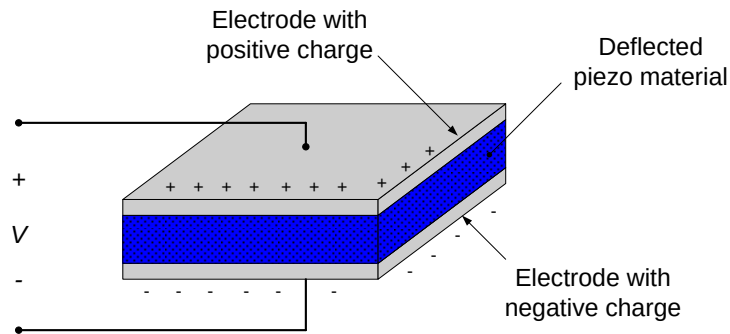
- a. Apply the sensor output voltage to the input terminals of the signal conditioning circuit,  $V_{IN}(t)$ . Using the oscilloscope, measure both  $V_{IN}(t)$  from the sensor and the signal conditioning unit's output voltage,  $V_{OUT}(t)$  in Fig. 2. "Flick" the sensor and record an image of the oscilloscope window, showing the voltages  $V_{IN}(t)$  and  $V_{OUT}(t)$ . Comment on your results relative to your expectations.

**DEMO**

- b. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

### Appendix A: Piezoelectric sensors

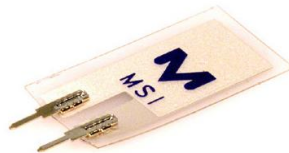
Some materials (certain crystals, for example) produce a charge when they are deflected; this is called a *piezoelectric* effect; materials which exhibit this property are called *piezoelectric materials* or *piezo materials*. If a piezoelectric material is sandwiched between two conductors, or *electrodes*, a voltage difference is produced between the electrodes when the material is deflected. A typical arrangement is shown in the figure below.



A constant (or *static*) deflection of a piezoelectric material will result in a fixed charge at the sensor's electrodes. *Leakage effects*, either within the piezoelectric material or the electronics associated with the sensor, cause this charge to dissipate with time. Thus, piezoelectric sensors cannot generally be used for *static* measurements (measurement of constant values) since the sensor's output voltage will decay to zero if the piezoelectric material's deflection is constant. Piezoelectric devices do, however, make excellent *dynamic* sensors (sensors which record time-varying phenomena) in which the piezoelectric material deflects rapidly relative to the leakage rate.

Piezoelectric sensors are often used in the measurement of time-varying pressures, accelerations, and forces. In these applications, the sensor is set up so that the process to be measured results in deflection of the piezoelectric material; the resulting voltage is used to indicate the desired physical parameter. A force applied to the material, for example, induces a stress in the material with a corresponding deformation of the material.

The piezoelectric sensor provided in the analog parts kit consists of a very thin piezoelectric film sandwiched between two printed electrodes and laminated to a polyester substrate. Contacts are provided to make connections to measure the response voltage. The device is shown below.

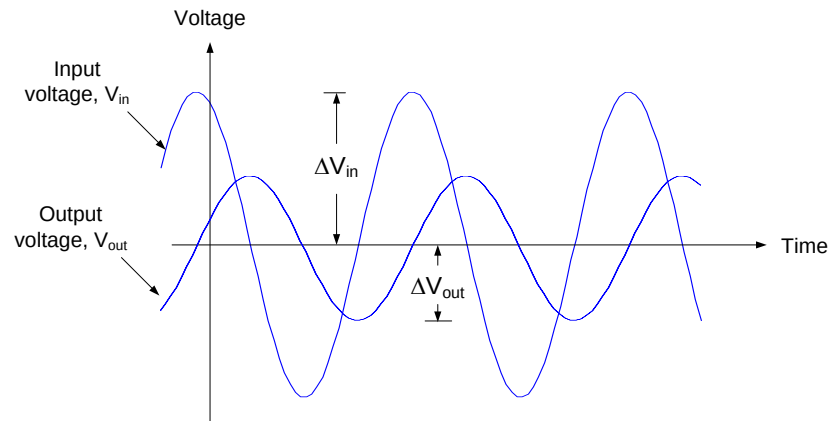


## Appendix B: Measuring Gain and Phase

The gain of a system at a particular frequency is the ratio of the magnitude of the output voltage to the magnitude of the input voltage at that frequency, so that:

$$\text{Gain} = \frac{\Delta V_{out}}{\Delta V_{in}}$$

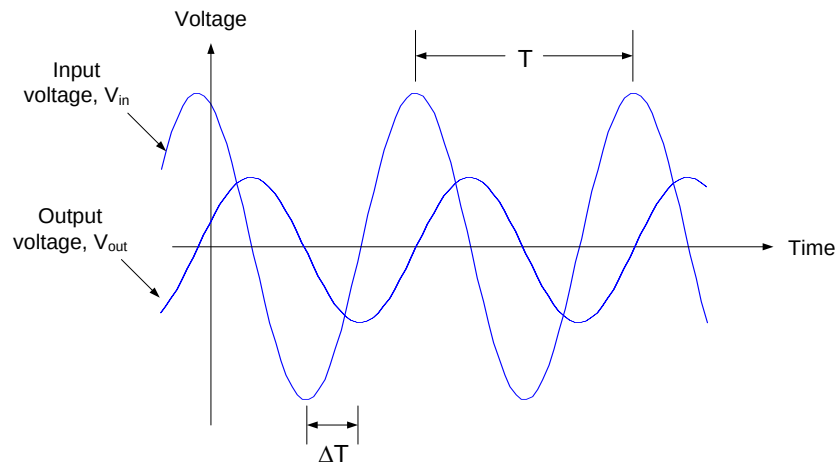
Where  $\Delta V_{out}$  and  $\Delta V_{in}$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



The phase of a system at a particular frequency is a measure of the time shift between the output and input voltage at that frequency, so that:

$$\text{Phase} = \frac{\Delta T}{T} \times 360^\circ$$

Where  $\Delta T$  and  $T$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



## Real Analog Chapter 11: Lab Worksheets

### 11.3.5: Signal Conditioning – Vibration Measurement (100 points total)

#### 1. Piezoelectric Sensor Vibration (20 points total)

1. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your signal acquisition from the sensor. (8 pts)

**TA Initials:** \_\_\_\_\_

2. Attach, to this worksheet, a plot of the sensor's natural response. (7 pts)
3. In the space below, provide your estimate of the dominant frequency of the sensor. Annotate your plot of part 3 to indicate how this dominant frequency was determined. (5 pts)

#### 2. Signal Conditioning Circuit (50 points total)

1. Provide below a schematic showing your signal conditioning circuit. Include desired resistance and capacitance values (3 pts)
2. Attach, to this worksheet, analyses which support your circuit design. These analyses should include (at a minimum) your desired low frequency gain and cutoff frequency; chosen resistance and capacitance values, and how these values were chosen. (15 pts)

3. In the space below, provide a table containing the following. (15 pts)
- The frequencies at which the response data was measured
  - The measured data characterizing your signal processing circuit (the input and output voltage amplitudes; time difference)
  - The calculated amplitude and phase response.
4. Attach, to this worksheet, a plot of the measured gain and phase vs. frequency for your signal conditioning circuit. In the space below, provide a discussion of measured vs. expected performance. (7 pts)
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (10 pts total)

TA Initials: \_\_\_\_\_

### 3. Overall System Integration (30 points total)

1. In the space below sketch your overall system, showing electrical connections between the sensor and the signal conditioning circuit. (3 pts)
2. Attach, to this worksheet, an image of the oscilloscope window showing the input and output of the signal conditioning circuit. In the space below, discuss your results relative to expectations. (10 pts)
3. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (17 pts total)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 11: Lab Projects

### 11.4.1: Passive Low-pass Filter

Bode plots are a convenient way to represent frequency responses. In particular, straight-line approximations to Bode plots are extremely easy to generate. In this lab assignment, we will generate a straight line approximation to the Bode plot for a passive low-pass filter. We will then measure the frequency response of the circuit at several discrete frequencies and compare these to our straight-line approximation. Finally, we will use the Analog Discovery's Bode Transfer Function instrument to automatically generate a Bode plot for the circuit.

#### Before beginning this lab, you should be able to:

- Generate straight line approximations to the Bode plot for a first order circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of an electrical circuit at discrete frequencies
- Use the Bode Transfer Function instrument to automatically measure the Bode plot for a circuit

#### This lab exercise requires:

- Analog Discovery module
- Diligent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:



Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.



Analysis; include principle results of analysis in laboratory report.



Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.



Record data in your lab notebook.

#### General Discussion:

The circuit of interest in this lab project is shown in Fig. 1.

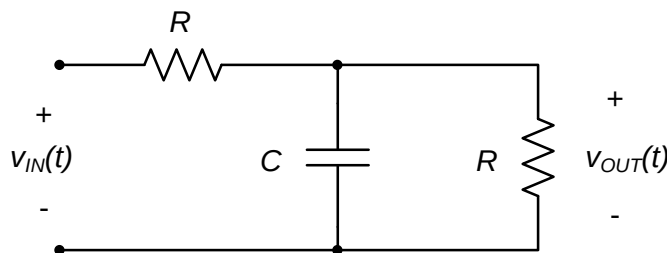


Figure 1. Passive low-pass filter circuit.

**Pre-lab:****ANALYSIS**

Sketch a straight-line approximation to the Bode plot for the circuit of Fig. 1 if  $R = 2.2\text{k}\Omega$  and  $C = 100\text{nF}$ .

**Lab Procedures:****DATA**

- a. Construct the circuit of Figure 1 if  $R = 2.2\text{k}\Omega$  and  $C = 100\text{nF}$ .
- Be sure to measure and record the actual values for the two resistors. If your DMM has a capacitance measurement, record the actual value for  $C$ ; if you do not have the ability to measure capacitance, assume that the nominal value of your capacitor is correct.

**DATA**

- Measure the magnitude response of the circuit at frequencies approximately equal to  $\frac{\omega_c}{8}, \frac{\omega_c}{4}, \frac{\omega_c}{2}, \omega_c, 2\omega_c, 4\omega_c$ , and  $8\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>14</sup>.

**DEMO**

- Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.
- b. Use the Bode Transfer Function instrument to automatically generate a Bode plot for the circuit. To do this, follow the steps below:
- Click on the “More Instruments” icon on the WaveForms main window. Select **Network Analyzer** from the resulting drop-down menu. A window will open, which will allow you to automatically create a Bode plot for the system.
  - Bode plot parameters are set using the options shown in Fig. 2 below. The available options are:
    - Start and Stop:** provide the range of frequencies which will be plotted. Generally, a range of frequencies from about an order of magnitude below the cutoff frequency to about an order of magnitude above the cutoff frequency is a good starting point.
    - AWG Offset and Amplitude:** The offset and amplitude of the input voltage. Use the offset and amplitude that you used to measure the response in part (a).
    - Steps:** The number of data points that will be used to generate the plot and the maximum gain that can be measured. More points will result in a smoother curve, but will take longer to generate.
    - Max-Gain:** Make sure the Max-Gain parameter is large enough to display the entire Bode plot.
    - Bode Scale:** The drop-down menus here allows you to choose the scale on both the magnitude and phase responses.
    - Scope Channels:** Choose the channels to be displayed.

**DATA**

Example parameters for this lab project are shown in Fig. 2. Record an image of the Network analyzer window. Also save the data to a .csv file for later processing by selecting the **File** option on the menu bar and clicking on **Export**.

|       |        |            |     |          |     |                |
|-------|--------|------------|-----|----------|-----|----------------|
| Start | 50 Hz  | AWG Offset | 0 V | Steps    | 100 | Bode Scale     |
| Stop  | 20 kHz | Amplitude  | 1 V | Max-Gain | 1 X | Scope Channels |

Figure 2. Example parameters for this lab project.

<sup>14</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.

### Post-lab Exercises:

**SIM**

Use your favorite mathematical software package (MATLAB, Octave, Excel,...) to plot the frequency response data (gain and phase vs. frequency) acquired by the Bode Transfer Function instrument in part (b) of the lab procedures. Overlay on this plot the data you acquired in part (a) of the lab procedures. Compare the responses from parts (a) and (b) and comment on any differences.

**ANALYSIS****ANALYSIS**

Compare the data you acquired in parts (a) and (b) of the lab procedures with your expectations from the pre-lab. Especially provide an assessment as to whether the original design requirements were met by your design.

## Real Analog Chapter 11: Lab Worksheets

### 11.4.1: Passive Low-pass Filter (45 points total)

1. Attach to this worksheet a sketch of the straight-line approximation to the Bode plot for the circuit of Fig. 1 if  $R = 2.2\text{k}\Omega$  and  $C = 100\text{nF}$ . (5 pts)
2. In the space below, sketch the circuit of Fig. 1 with measured values for the resistances and capacitance (if available). (3 pts)
3. In the space below, tabulate the input frequencies and the magnitude response of your circuit (in decibels) at each of these frequencies. (Note: feel free to include additional data in your table. It may result in partial credit.) (10 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (4 pts total)

TA Initials: \_\_\_\_\_

5. Attach to this worksheet an image showing the output of the Network Analyzer window. (8 pts)

6. Attach to this worksheet a plot showing the Network Analyzer data overlaid with the tabulated data from part 3 above. (10 pts)
  
7. In the space below, comment on the agreement between the data acquired in part 3 and the data acquired with the Network Analyzer in part 6. Also comment on the agreement between your expectations based on your pre-lab analysis and both sets of data. (5 pts)

## Real Analog Chapter 11: Lab Projects

### 11.4.2: Non-inverting Low-pass Filter

Bode plots are a convenient way to represent frequency responses. In particular, straight-line approximations to Bode plots are extremely easy to generate. In this lab assignment, we will generate a straight line approximation to the Bode plot for an active low-pass filter. We will then measure the frequency response of the circuit at several discrete frequencies and compare these to our straight-line approximation. Finally, we will use the Analog Discovery's Bode Transfer Function instrument to automatically generate a Bode plot for the circuit.

#### Before beginning this lab, you should be able to:

- Generate straight line approximations to the Bode plot for a first order circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of an active electrical circuit at discrete frequencies
- Use the Bode Transfer Function instrument to automatically measure the Bode plot for a circuit
- Design an active filter to provide a desired DC gain and cutoff frequency

#### This lab exercise requires:

- Analog Discovery module
- Diligent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

|                 |                                                                                                                                                                       |
|-----------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>DEMO</b>     | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
| <b>ANALYSIS</b> | Analysis; include principle results of analysis in laboratory report.                                                                                                 |
| <b>SIM</b>      | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
| <b>DATA</b>     | Record data in your lab notebook.                                                                                                                                     |

#### General Discussion:

The circuit of Fig. 1 is a non-inverting low pass filter. The DC gain of the circuit of Fig. 1 is:

$$\omega_c = \frac{1}{R_3 C} \quad \text{Eq. 1}$$

And the DC gain of the circuit is:

$$DC \text{ Gain} = \frac{R_2}{R_1} \quad \text{Eq. 2}$$

In this assignment, we will choose values of  $R_1$ ,  $R_2$ , and  $C$  in the circuit of Fig. 1 to meet design requirements set on the cutoff frequency and DC gain.

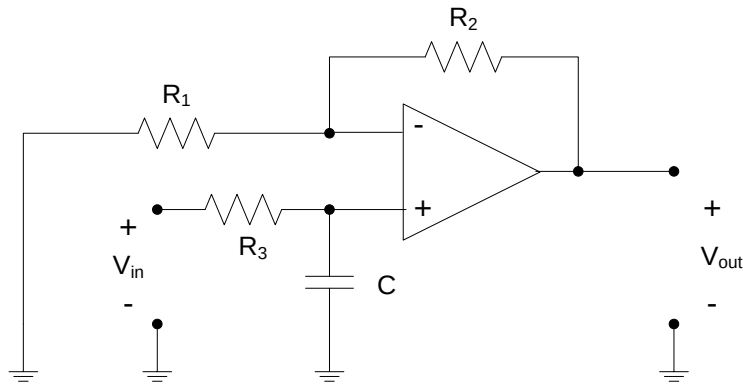


Figure 1. Non-inverting low-pass filter circuit.

**Pre-lab:****ANALYSIS**

Design the circuit of Fig. 1 (e.g. choose values for  $R_1$ ,  $R_2$ ,  $R_3$ , and  $C$ ) to meet the following design requirements:

- DC gain =  $3 \pm 10\%$
- Cutoff frequency =  $1.5\text{kHz} \pm 10\%$

**ANALYSIS**

Sketch a straight-line approximation to the Bode Plot for your design.

**Lab Procedures:****DATA**

a. Construct the circuit you designed in the pre-lab.

**DATA**

- Be sure to measure and record the actual values for  $R_1$ ,  $R_2$ , and  $R_3$ . If your DMM has a capacitance measurement, record the actual value for  $C$ ; if you do not have the ability to measure capacitance, assume that the nominal value of your capacitor is correct.
- Measure the magnitude response of the circuit at frequencies approximately equal to  $\frac{\omega_c}{8}$ ,  $\frac{\omega_c}{4}$ ,  $\frac{\omega_c}{2}$ ,  $\omega_c$ ,  $2\omega_c$ ,  $4\omega_c$ , and  $8\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>15</sup>.

**DEMO**

- Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.
- b. Use the Bode Transfer Function instrument to automatically generate a Bode plot for the circuit. To do this, follow the steps below:
- Click on the “More Instruments” icon on the WaveForms main window. Select **Network Analyzer** from the resulting drop-down menu. A window will open, which will allow you to automatically create a Bode plot for the system.
  - Bode plot parameters are set using the options shown in Fig. 2 below. The available options are:
    - **Start and Stop:** provide the range of frequencies which will be plotted. Generally, a range of frequencies from about an order of magnitude below the cutoff frequency to about an order of magnitude above the cutoff frequency is a good starting point.
    - **AWG Offset and Amplitude:** The offset and amplitude of the input voltage. Use the offset and amplitude that you used to measure the response in part (a).
    - **Steps:** The number of data points that will be used to generate the plot and the maximum gain that can be measured. More points will result in a smoother curve, but will take longer to generate.

<sup>15</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.

- Max-Gain: Make sure the Max-Gain parameter is large enough to display the entire Bode plot.
- Bode Scale: The drop-down menus here allows you to choose the scale on both the magnitude and phase responses.
- Scope Channels: Choose the channels to be displayed.

**DATA**

Example parameters for this lab project are shown in Fig. 2. Record an image of the Network Analyzer window. Also save the data to a .csv file for later processing by selecting the **File** option on the menu bar and clicking on **Export**.

|       |        |   |            |        |   |          |     |   |                                                                                     |                |   |
|-------|--------|---|------------|--------|---|----------|-----|---|-------------------------------------------------------------------------------------|----------------|---|
| Start | 100 Hz | ▼ | AWG Offset | 0 V    | ▼ | Steps    | 200 | ▼ |  | Bode Scale     | ▼ |
| Stop  | 30 kHz | ▼ | Amplitude  | 500 mV | ▼ | Max-Gain | 5 X | ▼ |  | Scope Channels | ▼ |

Figure 2. Example parameters for this lab project.

### Post-lab Exercises:

**SIM**

Use your favorite mathematical software package (MATLAB, Octave, Excel,...) to plot the frequency response data (gain and phase vs. frequency) acquired by the Bode Transfer Function instrument in part (b) of the lab procedures. Overlay on this plot the data you acquired in part (a) of the lab procedures. Compare the responses from parts (a) and (b) and comment on any differences.

**ANALYSIS**
**ANALYSIS**

Compare the data you acquired in parts (a) and (b) of the lab procedures with your expectations from the pre-lab. Especially provide an assessment as to whether the original design requirements were met by your design.

## Real Analog Chapter 11: Lab Worksheets

### 11.4.2: Non-inverting Low-pass Filter (50 points total)

1. In the space below, provide the resistance and capacitance values you chose to meet the given design requirements. Also provide the expected values input resistance, cutoff frequency, and DC gain for your circuit. Compare these values to the design specifications. (8 pts)
2. Attach to this worksheet a sketch of the straight-line approximation to the Bode plot for your design. (5 pts)
3. In the space below, tabulate the input frequencies and the magnitude response of your circuit (in decibels) at each of these frequencies. (Note: feel free to include additional data in your table. It may result in partial credit.) (10 pts)

4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (4 pts total)

**TA Initials:** \_\_\_\_\_

5. Attach to this worksheet an image showing the output of the Network Analyzer window. (8 pts)

6. Attach to this worksheet a plot showing the Network Analyzer data overlaid with the tabulated data from part 3 above. (10 pts)
  
7. In the space below, comment on the agreement between the data acquired in part 3 and the data acquired with the Network Analyzer in part 6. Also comment on the agreement between your expectations based on your pre-lab analysis and both sets of data. (5 pts)

## Real Analog Chapter 11: Lab Projects

### 11.4.3: Non-inverting High-pass Filter

Bode plots are a convenient way to represent frequency responses. In particular, straight-line approximations to Bode plots are extremely easy to generate. In this lab assignment, we will generate a straight line approximation to the Bode plot for an active high-pass filter. We will then measure the frequency response of the circuit at several discrete frequencies and compare these to our straight-line approximation. Finally, we will use the Analog Discovery's Bode Transfer Function instrument to automatically generate a Bode plot for the circuit.

#### Before beginning this lab, you should be able to:

- Generate straight line approximations to the Bode plot for a first order circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of an active electrical circuit at discrete frequencies
- Use the Bode Transfer Function instrument to automatically measure the Bode plot for a circuit
- Design an active filter to provide a desired DC gain and cutoff frequency

#### This lab exercise requires:

- Analog Discovery module
- Diligent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

|                 |                                                                                                                                                                       |
|-----------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>DEMO</b>     | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
| <b>ANALYSIS</b> | Analysis; include principle results of analysis in laboratory report.                                                                                                 |
| <b>SIM</b>      | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
| <b>DATA</b>     | Record data in your lab notebook.                                                                                                                                     |

#### General Discussion:

In this project, we will sketch a straight line approximation to the Bode plot for the circuit. The frequency response of the circuit will then be measured and compared with our expectations.

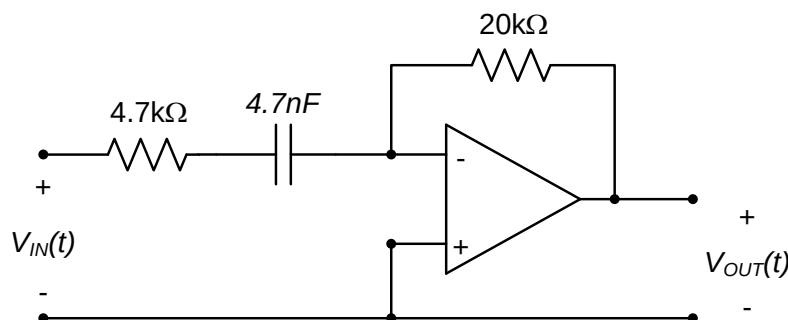


Figure 1. High-pass filter circuit.

**Pre-lab:****ANALYSIS**

Calculate the frequency response for the circuit shown in Fig. 1 and sketch the straight-line approximation to the circuit's Bode plot. Label the plot to show the high frequency gain and the cutoff frequency.

**Lab Procedures:****DATA**

a. Construct the circuit shown in Fig. 1.

i. Measure and record the actual values for  $R_1$ ,  $R_2$ , and  $R_3$ . If your DMM has a capacitance measurement, record the actual value for  $C$ ; if you do not have the ability to measure capacitance, assume that the nominal value of your capacitor is correct.

**DATA**

ii. Measure the magnitude response of the circuit at frequencies approximately equal to  $\frac{\omega_c}{8}$ ,  $\frac{\omega_c}{4}$ ,  $\frac{\omega_c}{2}$ ,  $\omega_c$ ,  $2\omega_c$ ,  $4\omega_c$ , and  $8\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>16</sup>.

**DEMO**

iii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

b. Use the Bode Transfer Function instrument to automatically generate a Bode plot for the circuit. To do this, follow the steps below:

- i. Click on the "More Instruments" icon on the WaveForms main window. Select **Network Analyzer** from the resulting drop-down menu. A window will open, which will allow you to automatically create a Bode plot for the system.
- ii. Bode plot parameters are set using the options shown in Fig. 2 below. The available options are:
  - Start and Stop: provide the range of frequencies which will be plotted. Generally, a range of frequencies from about an order of magnitude below the cutoff frequency to about an order of magnitude above the cutoff frequency is a good starting point.
  - AWG Offset and Amplitude: The offset and amplitude of the input voltage. Use the offset and amplitude that you used to measure the response in part (a).
  - Steps: The number of data points that will be used to generate the plot and the maximum gain that can be measured. More points will result in a smoother curve, but will take longer to generate.
  - Max-Gain: Make sure the Max-Gain parameter is large enough to display the entire Bode plot.
  - Bode Scale: The drop-down menus here allows you to choose the scale on both the magnitude and phase responses.
  - Scope Channels: Choose the channels to be displayed.

**DATA**

Example parameters for this lab project are shown in Fig. 2. Record an image of the Network Analyzer window. Also save the data to a .csv file for later processing by selecting the **File** option on the menu bar and clicking on **Export**.

|       |         |            |        |          |     |                |
|-------|---------|------------|--------|----------|-----|----------------|
| Start | 100 Hz  | AWG Offset | 0 V    | Steps    | 100 | Bode Scale     |
| Stop  | 100 kHz | Amplitude  | 500 mV | Max-Gain | 5 X | Scope Channels |

Figure 2. Example parameters for this lab project.

<sup>16</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.

## Post-lab Exercises:

**SIM**

Use your favorite mathematical software package (MATLAB, Octave, Excel,...) to plot the frequency response data (gain and phase vs. frequency) acquired by the Bode Transfer Function instrument in part (b) of the lab procedures. Overlay on this plot the data you acquired in part (a) of the lab procedures. Compare the responses from parts (a) and (b) and comment on any differences.

**ANALYSIS****ANALYSIS**

Compare the data you acquired in parts (a) and (b) of the lab procedures with your expectations from the pre-lab. Especially provide an assessment as to whether the original design requirements were met by your design.

## Real Analog Chapter 11: Lab Worksheets

### 11.4.3: Non-inverting High-pass Filter (50 points total)

1. In the space below, provide the frequency response you calculated for the circuit of Fig. 1. (5 pts)
2. Attach to this worksheet a sketch of the straight-line approximation to the Bode plot for your design. (5 pts)
3. In the space below, provide your measured values for the resistors and capacitor (if possible). (3 pts).
4. In the space below, tabulate the input frequencies and the magnitude response of your circuit (in decibels) at each of these frequencies. (Note: feel free to include additional data in your table. It may result in partial credit.) (10 pts)
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (4 pts total)

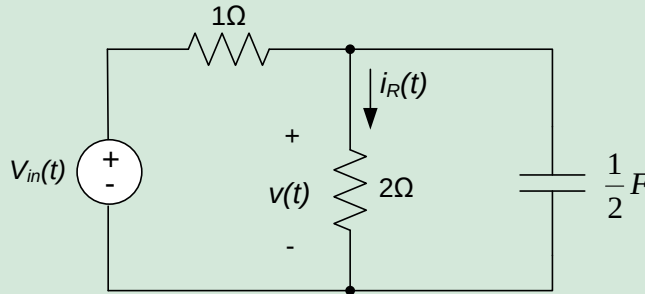
TA Initials: \_\_\_\_\_

6. Attach to this worksheet an image showing the output of the Network Analyzer window. (8 pts)

7. Attach to this worksheet a plot showing the Network Analyzer data overlaid with the tabulated data from part 4 above. (10 pts)
  
8. In the space below, comment on the agreement between the data acquired in part 4 and the data acquired with the Network Analyzer in part 7. Also comment on the agreement between your expectations based on your pre-lab analysis and both sets of data. (5 pts)

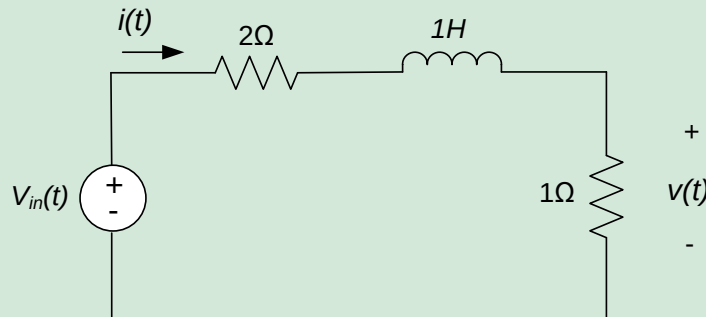
## Real Analog Chapter 11: Homework

- 11.1 For the circuit below, the input voltage  $V_{in}(t) = 3\cos(2t-20^\circ) - 2\cos(3t+30^\circ) + \cos(4t)$ . Determine:
- The steady-state response of the voltage  $v(t)$ .
  - The steady-state response of the current  $i_R(t)$ . (Hint: take advantage of your results from part a)

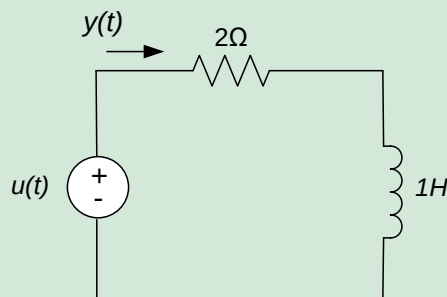


- 11.2 For the circuit of problem 1, sketch the magnitude and phase response of the voltage  $v(t)$ . Label your sketch at  $\omega = 0$ ,  $\omega \rightarrow \infty$  and the circuit's cutoff frequency. Does the circuit behave as a low-pass or high-pass filter? Why?

- 11.3 For the circuit below, the input is the voltage  $V_{in}(t)$ . If the output is the current  $i(t)$ , sketch a straight-line approximation to the Bode plot for the circuit. Label your sketch to include the DC gain and the cutoff frequency of the circuit.

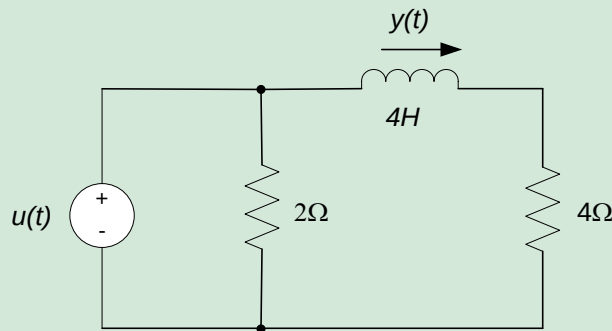


- 11.4 For the circuit below, the input is the voltage  $u(t)$  and the output is the current  $y(t)$ . Sketch a straight-line approximation to the Bode plot. Label your sketch to include the DC gain and the cutoff frequency of the circuit.



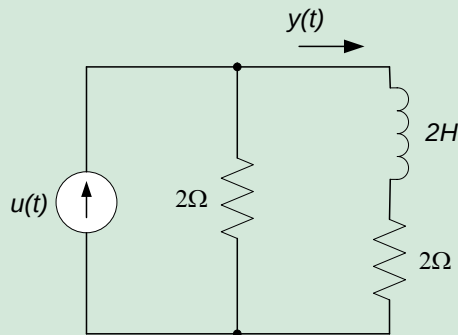
11.5 For the circuit of problem 11.4, sketch a straight-line approximation to the Bode plot for the circuit if the output is the voltage  $v(t)$ . Label your sketch to include the DC gain and the cutoff frequency of the circuit.

- 11.6 For the circuit shown, the input is the voltage source  $u(t)$  and the output is the current through the inductor,  $y(t)$ .
- Determine the frequency response of the circuit.
  - Sketch the magnitude response of the circuit. Label your sketch to show the DC gain and the cutoff frequency of the circuit.
  - Does the circuit behave as a high pass or low pass filter? Explain.



- 11.7 For the circuit of problem 1, the input is the voltage source  $u(t)$ . If the output is the voltage across the inductor,
- Determine the frequency response of the circuit.
  - Sketch the magnitude response of the circuit. Label your sketch to show the DC gain and the cutoff frequency of the circuit.
  - Does the circuit behave as a high pass or low pass filter? Explain.

- 11.8 For the circuit shown, the input is the current source  $u(t)$  and the output is the current through the inductor,  $y(t)$ .
- Determine the frequency response of the circuit.
  - Sketch the magnitude response and phase response of the circuit. Label your sketch to show the DC gain and the cutoff frequency of the circuit.
  - Does the circuit behave as a high pass or low pass filter? Explain.



- 11.9 For the circuit shown, the input is the current source  $u(t)$  and the output is the voltage across the  $6\Omega$  resistor,  $y(t)$ .
- Determine the frequency response of the circuit.
  - Does the frequency response you calculated above agree with your expectations as to the circuit's response at high ( $\omega \rightarrow \infty$ ) and low ( $\omega \rightarrow 0$ ) frequencies? Why or why not?

