

11.2.1: Signals with Multiple Frequency Components

Overview:

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
- Digilent 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:

This lab assignment concerns the circuit shown in Figure 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:

$$\omega = 0 \Rightarrow \left| \frac{V_{OUT}}{V_{IN}} \right| = \frac{1}{2} \quad (1)$$

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:

$$\omega = \infty \Rightarrow \left| \frac{V_{OUT}}{V_{IN}} \right| = 0 \quad (2)$$

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 frequencies¹.

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).

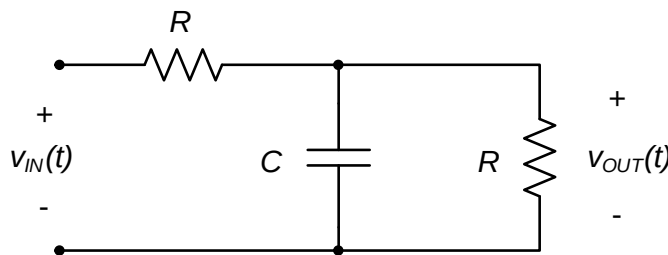


Figure 1. RC circuit.

Pre-lab:

ANALYSIS

Determine the magnitude response of the circuit shown in Figure 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)?

¹ The circuit is said to be a *low pass filter*.

Lab Procedures:

- a. Construct the circuit of Figure 1, using $R = 1k\Omega$ and $C = 100nF$.

DATA

- 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:

$$v_{IN}(t) = 20[\sin(1000\pi t) + \sin(2000\pi t) + \sin(20,000\pi t)] \quad (3)$$

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.

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 Figure A1.
 - Type the mathematical expression for the signal being created. Multiplication is denoted by *, $\text{PI} = \pi$, and **SIN** = sine. Operator precedence is as in most calculators. Thus, the mathematical expression in equation (3) can be implemented as:

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

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 Figure 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.

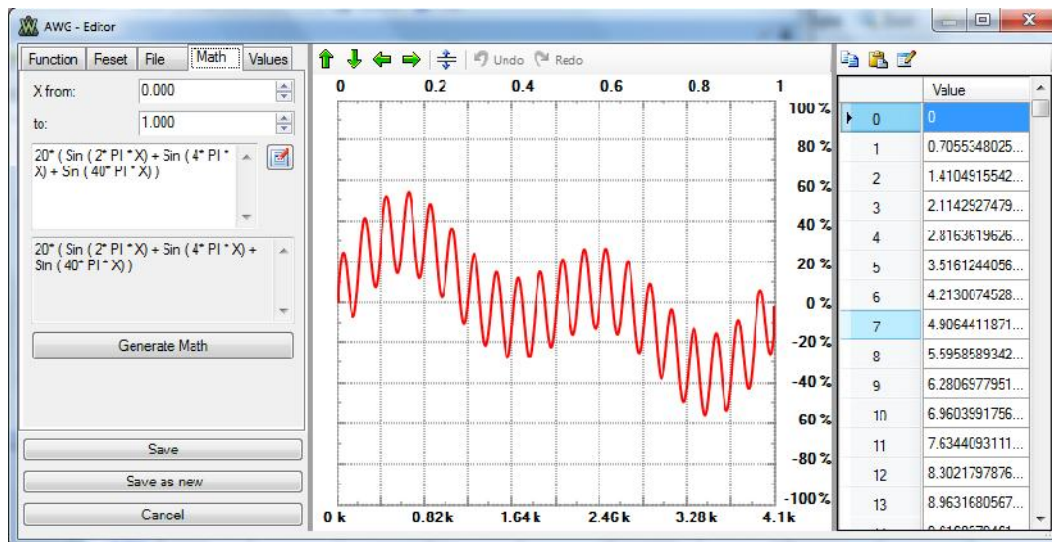
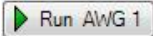
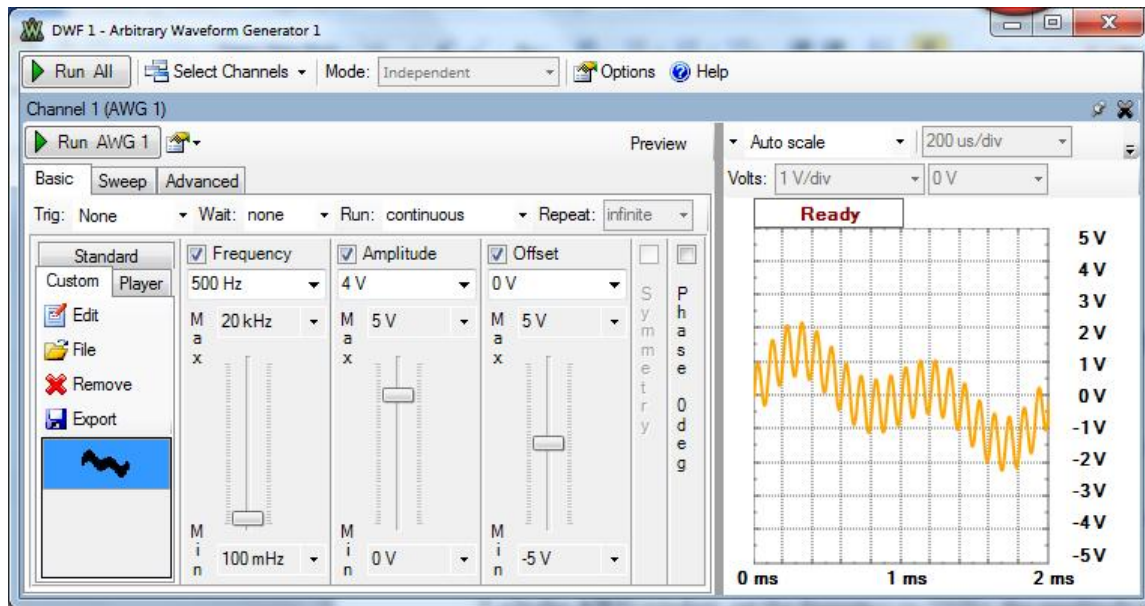


Figure A1. Sample AWG Editor window.

Real Analog – Circuits 1
Lab Project 11.2.1: Signals with Multiple Frequency Components

2. 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 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 Figure B1; your window should look.

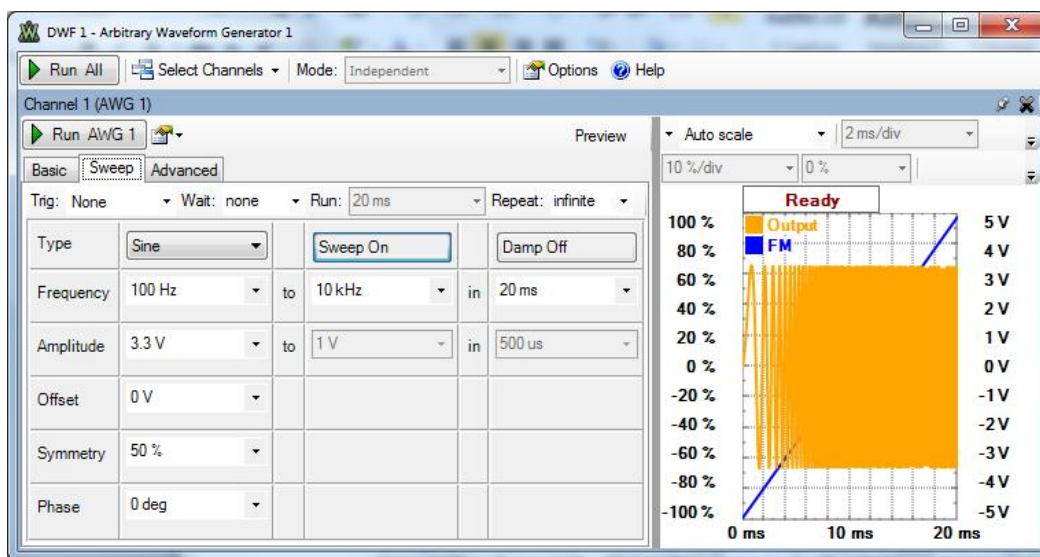


Figure B1. Example arbitrary waveform generator window.

- Click on **Run AWG 1** to play back the signal.