1.0 Intro
In this lab, you will build both a low-pass and high-pass filter. First, you will simulate the circuit using the analysis program Multisim. Then, you will build the circuit on the breadboard. For this experiment you will use a 10K resistor and a 0.1 uF capacitor. Using the DMM and the RLC Meter to measure the exact values. Record these values.

2.0 Low-Pass Filter

2.1 Simulated Circuit
Multisim is a simple program which allows the user to create and analyze circuits. We will be building the first order low-pass RC circuit shown in Figure 1.

Figure 1: Low-pass RC Circuit

1) Open Multisim: Start >> Engineering >> NationalInstruments >> Multisim
2) On the far right panel, locate the NI ELVISmx Instruments menu and select the NI ELVISmx Function Generator. Place the Function Generator on the schematic.

3) In the same menu, locate and place the NI ELVISmx Oscilloscope. Double-click on the icon to bring up the scope, which looks exactly like the one we have used for the ELVIS.
4) On the top menu, go to Place >> Component. Select “Basic” from the Group menu. Find the “Resistor” tab. Choose the 10k resistor. Place this component and set the value it to your measured value.

5) In the same menu, locate the “Capacitor” tab, and select a 100 nF capacitor. Place this component and set the value to your measured value.

6) In the same menu, change the Group to “Sources.” Under the “Power_Sources” tab, choose “Ground.” Place this component
7) Wire the diagram to create the low pass RC circuit.

8) Now, we will do some circuit simulation. First, generate a sine wave to stimulate your circuit by double clicking on the Function Generator. Set the frequency to 16 Hz and the amplitude to 1V.

9) Click the Play button (Green Arrow) in the Multisim window.

10) Observe the output waveform amplitude and relative phase. Record these values. Repeat for 160Hz and 1600 Hz.
2.2 Construct an actual circuit

1. You will need 4 wires, a 10k resistor, and a 100nF capacitor. Make the following connections on the breadboard:
   
   a. Wire FGEN on the prototyping board to AI1+
   b. Connect the 10K resistor from FGEN to intermediary node on breadboard
   c. Connect 0.1 uF Capacitor from intermediary node to AI Ground
   d. Wire intermediary node to AI0+
   e. Wire AI0- to AI Ground
   f. Wire AI1- to AI Ground
2. This setup will allow you to use the Bode plotting instrument. Open up this tool from the Start -> Engineering -> National Instruments -> NI ElviSMX Instrument launcher. Change the settings as shown above and click Run.

3. Compare your simulation results (amplitude and phase) to the Bode plot at the 3 frequencies you simulated. The Y-Axis of the Bode plot is in dB where dB is defined as:

   \[ dB = 20 \log_{10}\left(\frac{V_{\text{out}}}{V_{\text{in}}}\right) \]

   For example, -3 dB = \(\frac{1}{\sqrt{2}}\). Often, the -3 dB frequency is referred to as the “Corner Frequency” of the filter. For a simple RC filter, the phase shift should be 45 degrees at that frequency. Note: The X-Axis is Frequency in Hz on a log scale.

4. What is the slope of the Bode Plot beyond the corner frequency in dB/decade?

5. Now run the function generator at 16, 160 and 1600 Hz and measure the output amplitude and phase. Use the Ni-Elvis Scope to do your measurements. Verify that your measurements (amplitude and phase) agree with the Bode plot and the simulation?
6. Change the Function Generator to 16 Hz and Square Wave. Notice how the signal rises and falls exponentially? Determine the rising and falling time constant of the system. The time constant is defined as the time it takes for the signal to reach 63% of its peak voltage.

For an RC circuit, the time constant should be equal to \( R \times C \). Calculate this value and compare it to your experimental time constant. How close were you?
3.0 High Pass Filter

The high pass RC circuit is shown below. Repeat all the above steps (Multisim analysis, building the actual circuit, Bode plot, and Oscilloscope testing) using this new circuit.