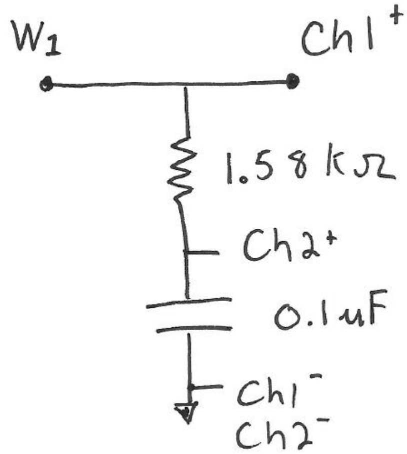
Problem set: RC circuits and the Bode plot

Construct and test the behavior of the RC circuit below.



Connect the W1 (waveform output 1, solid yellow wire) from the Analog Discovery to the top of the resistor. Use the Scope's channel 1 to measure this input to the circuit, and use the channel 2 to measure the output (voltage across the capacitor). Turn on the waveform generator in the Waveforms software. In the WaveGen, make the output signal a **sine wave**, with an offset of 0 volts and amplitude of 1 volt. Open the scope and turn on the two measurement channels so you can see both the filter's input and output signals on top of each other. Adjust the time scale and voltage scale so you can easily observe the output signal relative to the input. Adjust the frequency of the waveform to be lower, higher, and around the characteristic frequency of the filter given by the RC time (don't forget to convert rad/s to Hz). Observe how the filter behaves qualitatively at both low and high frequencies. Watch the amplitude of the output as well as the phase between input and output.

The input to the circuit should therefore be, $V = \sin(\omega t)$. The output is expected to be the form $V_{out} = A\sin(\omega t + \phi)$. From the book, we know that the expected form for amplitude and phase are,

$$A = \frac{1}{\sqrt{1 + (RC\omega)^2}}$$

$$\phi = \text{atan}(-RC\omega)$$

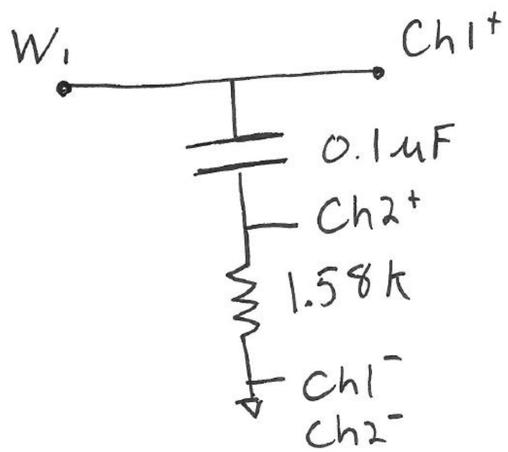
Fill in the table for the experimental and expected values. You might need to think about how you can convert what you can measure on the scope to the measured phase. We can review phase measurement in class.

Freq. (Hz)	Freq. (rad/s)	RCω	A (theory)	A (meas)	Φ (theory)	Φ (meas)
100						
500						
1,000						
5,000						
10,000						

Once you have some intuitive feel for how the circuit is operating, go to the main screen for the Analog Discovery. Click on “Welcome” tab select “network”. The network analyzer will automatically create a Bode plot for you. Adjust the frequency range, 10 Hz to 100 kKHz is a good place to start, though you may want to adjust this range. Click “single” to conduct a single frequency sweep. **Save the Bode plot data through the export feature.**

Now make sure you understand the connection between the table you created and the Bode plot generated by the Network Analyzer. To get some practice in MATLAB, plot the amplitude part of your measured Bode from the Network Analyzer. Superimpose your data from the table on the Bode plot. Remember the experimental output will be in dB while the amplitude, A, is just the ratio of output voltage to input voltage. See the book for converting to (or from) dB.

Switch the placement of the R and C in the same circuit and repeat the exercise. See the circuit below:



In this case the amplitude and phase results change from the previous example. The expected results for amplitude and phase are;

$$A = \frac{RC\omega}{\sqrt{1 + (RC\omega)^2}}$$

$$\phi = \operatorname{atan}\left(\frac{1}{RC\omega}\right)$$

Freq. (Hz)	Freq. (rad/s)	RCω	A (theory)	A (meas)	Φ (theory)	Φ (meas)
100						
500						
1,000						
5,000						

Recreate the Bode plot using the Network Analyzer as you did with the previous circuit. Create the same plot as you did for the other filter.

Deliverable: You can just turn in two plots from this work. The plots should be well-labeled and clear. This is not a lab and no further explanation is needed.