

AC Measurements with the Agilent 54622D Oscilloscope

Objectives: At the end of this experiment you will be able to do the following:

1. Correctly configure the 54622D for measurement of voltages.
2. Perform correct AC measurements with the 54622D, and compare these with theoretical values for simple AC circuits.
3. Use the subtract mode to measure voltage drops across components anywhere within a circuit.

Experimental Procedure:

0. Reset the configuration of the 54622D oscilloscope by holding down the AUTOSCALE KEY while turning on the POWER. Keep the AUTOSCALE key down until the “system concerns detected” message appears.

This removes anyone else's settings from the configuration memory of the scope, which could interfere with your measurements.

1. Connect the circuit of Figure 1 below. Pay particular attention to the placement of oscilloscope probes. Use standard 1:1 (BNC-to-alligator clip) connections to the scope.

Connect channel 1 to test point 1, and channel 2 to test point 2.

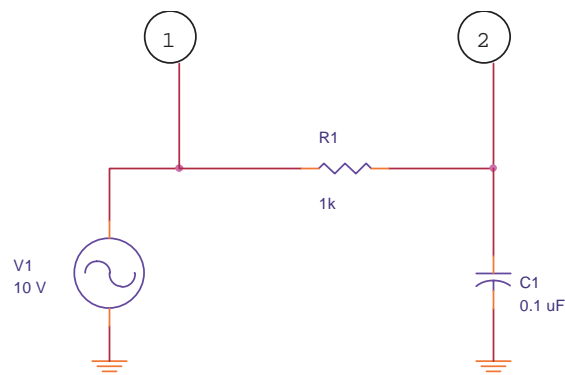




Figure 1: A Test Circuit for Measurement

2. Set the TRIGGER to channel 1. To do this, press EDGE, and then choose “1” from the soft-key menu below the screen. This sets test point 1 as the “frame of reference” for all phase measurements.



Figure 2: Soft Key Menu

3. Set the signal generator to 1.59 kHz, 10 V (RMS). Make sure the generator is set to “High-Z” output mode.

4. Press the  and  buttons to turn on the traces for channels 1 and 2.

Both of the buttons should be lit, and two traces should appear on the screen.

Adjust the oscilloscope horizontal and vertical controls to get one or two cycles of the signal on screen.

5. Record the measurements from channels 1 and 2 here:

Channel 1: $V_{pp} =$ _____ Phase = 0 Degrees (Reference)

$V_{rms} =$ _____

Channel 2: $V_{pp} =$ _____ Phase = _____

$V_{rms} =$ _____

6. Using phasor techniques, calculate the following. Report these in polar form. Use the back of this sheet for your calculations.

$I_T =$ _____ $V_{R1} =$ _____

$V_{C1} =$ _____

7. The measurement V_{C1} is the same as V_2 in the circuit above. This should be the same as the reading from oscilloscope channel 2. Compare the two values (Channel 2 reading, step 5 and V_{C1} , step 6). They should be within 10% of each other.

Differential AC Signal Measurement

It's easy enough to measure AC signals that are ground-referenced, but what about "floating" measurements? We know that the oscilloscope ground can't be connected anywhere but the circuit physical ground. Therefore, when we connect a probe to a test point, we are measuring the voltage between the test point and ground.

What if we want to measure the voltage *between* two test points (such as the voltage V_{12} in Figure 1, which is the same as V_{R1})? We can't just move the scope ground to test point 2, because that will short out C_1 .

To measure across a component that doesn't have one end connected to ground, connect probes to both sides of the component and use subtract mode.

In subtract mode, the oscilloscope calculates the difference between the signals on its channel 1 and channel 2 input connectors and displays this as a third trace on the screen. Since this sort of reading isn't referenced directly to ground, it is sometimes called a "floating" or "differential" measurement. It's quite easy to set up on the 54622D.


Measuring V_{R1} using Subtract Mode

1. Place channels 1 and 2 of the oscilloscope across the component whose voltage is to be measured. The previous setup is correct for measuring V_{R1} .



Caution: If you move channel 1 to a different location in the circuit, your trigger reference will be lost and phase measurements will become invalid. If this is a problem, connect the EXTERNAL TRIGGER input to the trigger reference point (usually the generator, test point 1) and choose EXTERNAL from the EDGE menu.

Since channel 1 is still connected to the input signal source, you will not lose trigger, so you don't need to worry about using the external trigger in this circuit.

2. For the calculated difference voltage to be meaningful, channels 1 and 2 should be set to the same vertical sensitivity. If channel 1 is set to "5 V/division" then channel 2 should also be set to "5 V/division." Make sure that this is so.

3. Press the  button on the oscilloscope. (It should light up.) Note that there are several options available, many of which you'll use in more advanced course work.
4. On the SOFT KEY MENU below the screen, choose 1 - 2. This will cause a new trace to appear on the screen. This trace is the *difference* in the voltage between channels 1 and 2.

NOTE: The 1 - 2 function will work even if channels 1 and 2 are OFF (not displaying.)

5. You will need to set the VERTICAL SCALE FACTOR for the difference trace. (It typically defaults to a small value such as 500 mV/division, which is way too small for this application.) To do this:
 - a) Press the SETTINGS button on the Math SOFT KEY MENU.
 - b) Press the SCALE button on the SOFT KEY MENU and adjust the  knob until the desired scale factor is achieved.
 - c) To move the difference trace up or down on the screen, press the OFFSET button on the SOFT KEY MENU and adjust the  knob until the trace is where you want it.
6. You should now see V_{R1} displayed as the new trace. (You can turn off channel 1 and/or 2 to make it easier to see). Record it below.

V_{R1} measured value: V_{pp} _____ V_{rms} _____ Phase _____

7. Compare the results from step 6 to those from step 6 (theoretical calculations) in the first part of this experiment. Your values should be within 10%.

Questions

1. What should you do with the 54622D oscilloscope each time you begin using it to assure that no settings from a previous user are in effect?

2. Why was it important that the TRIGGER be set to channel 1 in this procedure?

3. Explain what is meant by the term “floating measurement.”

4. Explain how to set up the scope to measure the voltage across a component that has no leads directly connected to circuit ground.

5. What have you learned in this experiment?

Frequency Effects in RC, RL Circuits

Introduction

When circuits are constructed only with ideal resistors, their response is always the same, regardless of the applied AC signal frequency. The addition of either inductance or capacitance to circuits causes them to be *frequency sensitive*.

A frequency sensitive circuit changes its response based on the frequency of the AC signal applied. The most common applications of frequency sensitivity are *filters*. Filters are circuits that selectively block or pass frequencies.

A *low pass* filter passes all frequencies from DC up to its *corner frequency*. At the corner frequency, the filter only passes 70.7% of the original input signal voltage. Above the corner frequency, the filter's voltage output "rolls off," which means that increasing the frequency above the breakpoint (corner frequency) causes the response to rapidly fall. Both of the circuits of Figure 1 are low pass filters. We sometimes call them "first order" filters since they contain only one capacitor or one inductor.

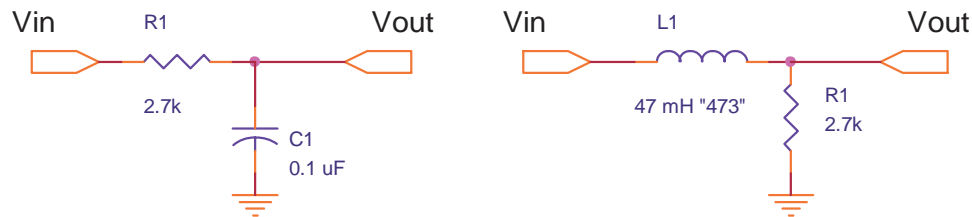


Figure 1: Low Pass Filters

A *high pass* filter passes all frequencies from infinity (very high frequency) down to its corner frequency. Like a low pass filter, a high pass filter passes 70.7% of the input signal voltage at the break frequency. The output signal voltage rapidly rolls off below the break frequency. The circuits of Figure 2 are high pass filters, and they are also "first order" filters like the units in Figure 1. High pass filters block DC.



Figure 2: High Pass Filters

Corner Frequency

The corner frequency of a first-order filter (such as any of the units above) is defined as the frequency where the filter's output voltage is 70.7% of the input voltage. (Later you'll learn that this corresponds to a response of -3 dB, which is also a half-power point).

At the corner frequency (also called the *critical* frequency, or *break* frequency), the reactance (X_L or X_C) exactly equals the resistance in the circuit, and the phase shift at the output is

either +45 degrees (high pass) or -45 degrees (low pass). We can easily calculate this frequency by:

$$(2-1) f_c = \frac{1}{2\pi\tau} \text{ Where } \tau \text{ ("tau")} \text{ is the RC time-constant or the L/R time-constant.}$$

Therefore, for a filter built with a resistor and capacitor:

$$(2-2) f_c = \frac{1}{2\pi RC}$$

And for a filter built with an inductor and resistor:

$$(2-3) f_c = \frac{1}{2\pi(L/R)}$$

You can easily obtain formulas (2-2) and (2-3) if you remember to merely substitute the time-constant of the circuit.

Measuring the Corner Frequency using Test Equipment

To measure the corner frequency of a filter, you need to connect a variable-frequency signal generator (AC source) to the filter's input, and an oscilloscope (or true-RMS AC voltmeter) to the filter's output, as shown below in Figure 3.

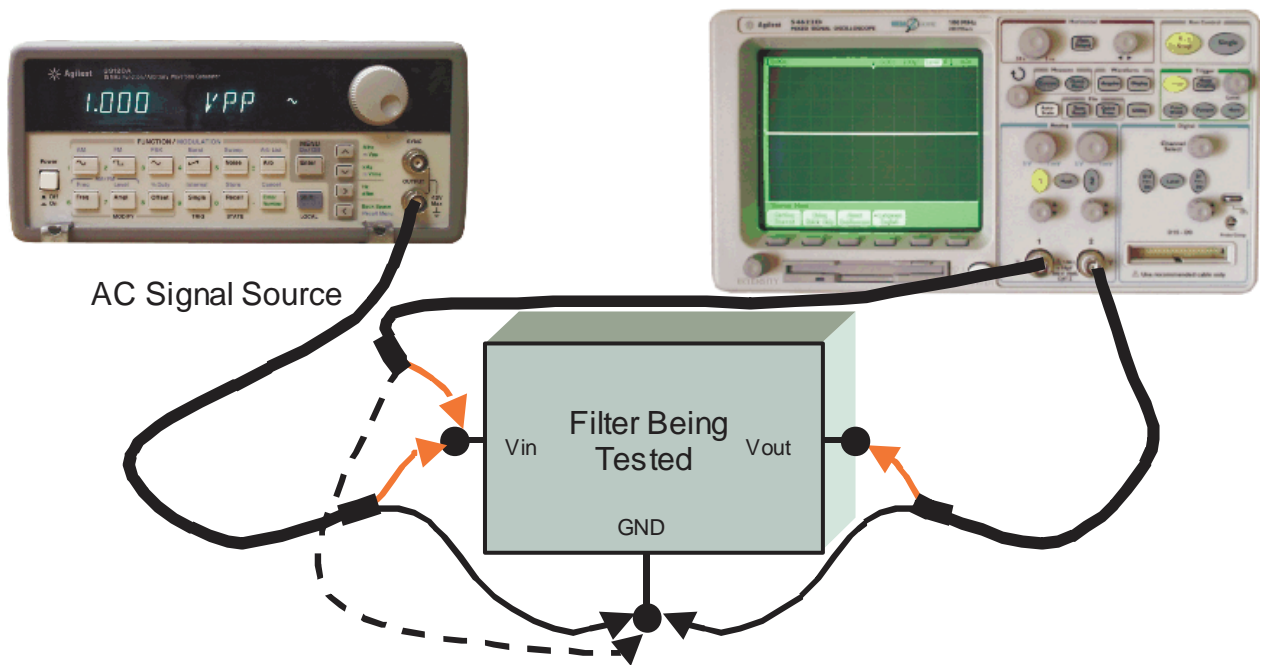


Figure 3: Setup for Measuring Filter Corner Frequency

In the setup of Figure 3 (schematically shown in Figure 4), scope channel 1 monitors the input voltage being applied to the filter, and scope channel 2 displays the filter's output. Compare Figure 4 to Figure 3, and make sure you understand how (and why) things are connected the way they are!

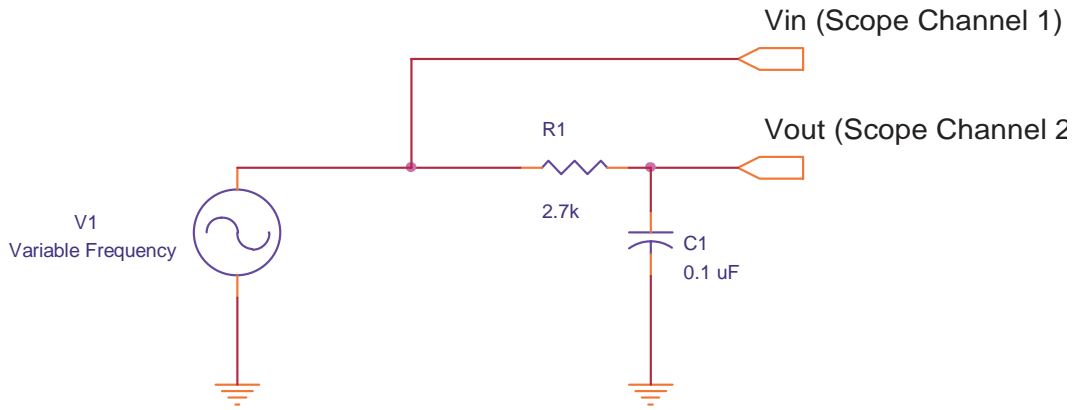


Figure 4: Schematic Diagram of Test Setup

The signal generator provides the stimulus (input signal) for the filter. Most signal generators hold their amplitude constant as the frequency is changed, but it is a good idea to watch the input voltage just in case. This is one of the reasons for applying scope channel 1 to the input of the filter.

To measure the corner frequency, proceed as follows:

1. Connect the test equipment up to the filter and verify that you can get proper input and output signals.

Make sure that channels 1 and 2 of the scope are set to the same vertical sensitivity (volts/division)!

2. Adjust the amplitude of the signal generator until the peak-to-peak amplitude of channel 1 fills the scope screen as shown in Figure 5. (This will be the “100%” level.)

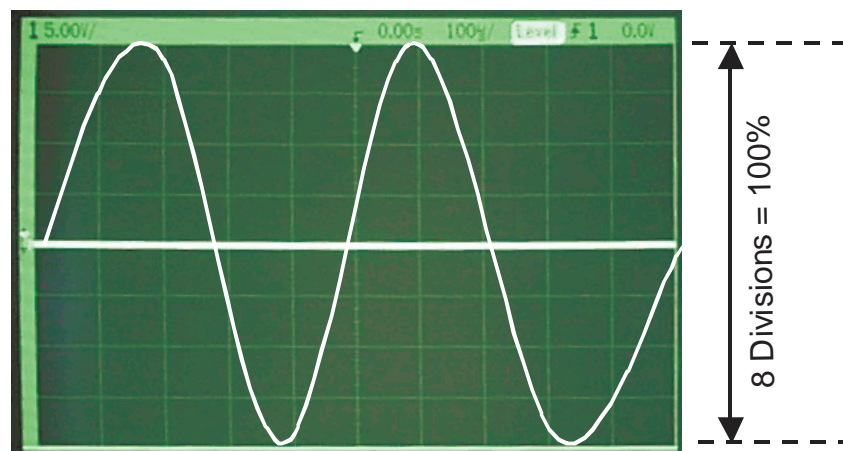


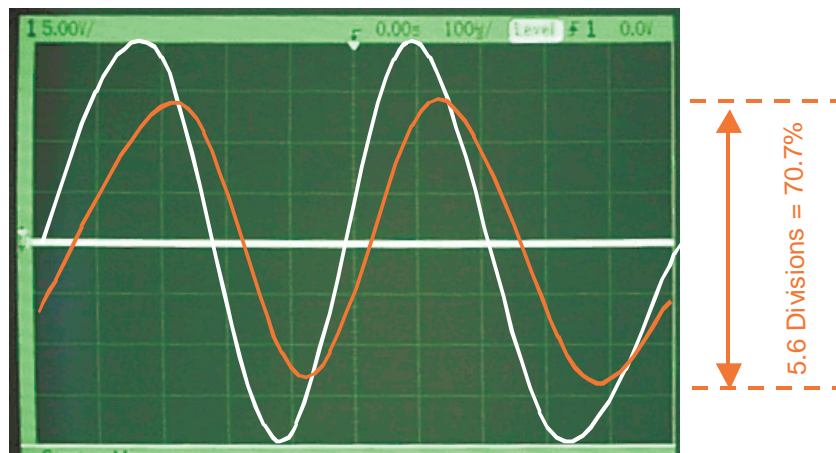
Figure 5: 100% Reference Level (Channel 1 shown alone; Channel 2 off)

- Now remember that at the corner frequency of the filter, the output signal will be 70.7% of the input voltage. Therefore, at the corner frequency, the output voltage displayed on channel 2 will be:

$$V_{\text{OUT}} = (70.7\%)(8 \text{ Divisions p-p}) = 5.6 \text{ Divisions peak-to-peak.}$$

To complete the procedure, enable the signal on channel 2 and adjust the FREQUENCY of the signal generator until the channel 2 trace is about 5.6 divisions peak-to-peak, as shown in Figure 6.

When you reach this frequency, you have found the break frequency of the filter.



*Figure 6: Both Channels Displayed at the Corner Frequency.
Ch 1 = White; Ch 2 = Red.*

- To double-check your result, note that the scope should display +/- 45 degrees of phase shift at this frequency.

Laboratory Procedure

1. Connect the circuit of Figure 4, and calculate the corner frequency of the filter.

$$f_c(\text{calculated}) = \underline{\hspace{2cm}}$$

2. Identify the filter as a high or low pass. (See Figures 1 and 2).

Configuration:

2. Using the procedure outlined in the previous section, sweep the filter to find the corner frequency.

$$f_c(\text{measured}) = \underline{\hspace{2cm}} \text{ (Should be within 10\% of the calculated value)}$$

3. Build the second circuit from Figure 1 (R-L). Calculate its corner frequency. If you don't have a 47 mH inductor (may be marked "47 mH" or "347"), use an inductor in the range of 10 mH to 100 mH and base your calculations on its value.

$$f_c(\text{calculated}) = \underline{\hspace{2cm}} \text{ (Should be within 20\% of the calculated value)}$$

4. Build the RC high-pass filter from Figure 2. Again calculate and measure the corner frequency.

$$f_c(\text{calculated}) = \underline{\hspace{2cm}}$$

$$f_c(\text{measured}) = \underline{\hspace{2cm}}$$

Questions

1. What is the difference between a low and high pass filter?

2. What is meant by the term “corner frequency” of a filter?

3. What is the output voltage percentage and phase angle of a first-order low pass filter at the corner frequency?

4. Explain the experimental setup of Figure 3 in your own words.

5. What have you learned in this experiment?

Series and Parallel Resonant Circuits

Objectives: At the end of this experiment you will be able to do the following:

1. Calculate the resonant frequency of a series or parallel circuit.
2. Calculate the Q and bandwidth of a series or parallel resonant bandpass filter circuit.
3. Use test equipment to measure the bandwidth and Q of a resonant filter by the frequency-sweep method.

Introduction

In many applications a low pass or high pass filter is inadequate. We often need to pass a selected *range* of frequencies. For example, we may need to pass the frequency range 705 kHz to 715 kHz in a radio receiver's tuner section so that a 710 kHz radio signal can be "selected" while rejecting other undesired station frequencies.

A filter that passes a range of frequencies (and rejects others) can be built by combining a low pass and high pass filter. Such a filter is called a *bandpass* filter.

Although we can build a bandpass filter by simply combining an RC high pass and an RC low pass (or RL high pass and RL low pass), this isn't a very efficient way of filtering. The easiest way of building a bandpass filter is to use a *resonant circuit*.

A *resonant circuit* contains both a capacitance and an inductance. We can construct both series and parallel resonant filter circuits, as shown in Figure 1.



Figure 1: Series and Parallel Resonant Filter Circuits

Action of the Series Resonant Bandpass Circuit

To understand how the series resonant filter works, imagine that we first introduce a very low frequency (close to DC) at the V_{in} test point. You know that capacitors block DC. C_1 will provide a very high opposition to the low frequency signal (even though L_1 might "want" to pass it easily.) Very low frequencies can't make it through the filter.

Now place a very high frequency into the V_{in} input. Now C_1 appears as a short (capacitors appear as shorts to high frequencies), but L_1 appears as an open (since X_L increases with frequency). The circuit also blocks very high frequencies!

There is a frequency that can easily pass through this filter. This is the frequency where the reactance of L1 and C1 are equal. This frequency is called the *resonant frequency* of the circuit, and is calculated by:

$$f_0 = \frac{1}{2\pi\sqrt{LC}} \quad (\text{Pronounced "f - not" })$$

At resonance, 100% of the signal passes through the filter because X_L and X_C cancel each other (they have opposite polarities; $Z_L = jX_L$ and $Z_C = -jX_C$).

Above the resonant frequency, X_L is increasing and X_C is decreasing, which means that the inductor blocks signal above the resonant frequency. The opposite happens below the resonant frequency.

Filter Bandwidth and Quality Factor

The frequency range that a filter can pass is called the *bandwidth*. It is the frequency difference between the -3 dB (70.7 % voltage response, half-power) points of the filter, as shown in Figure 2.

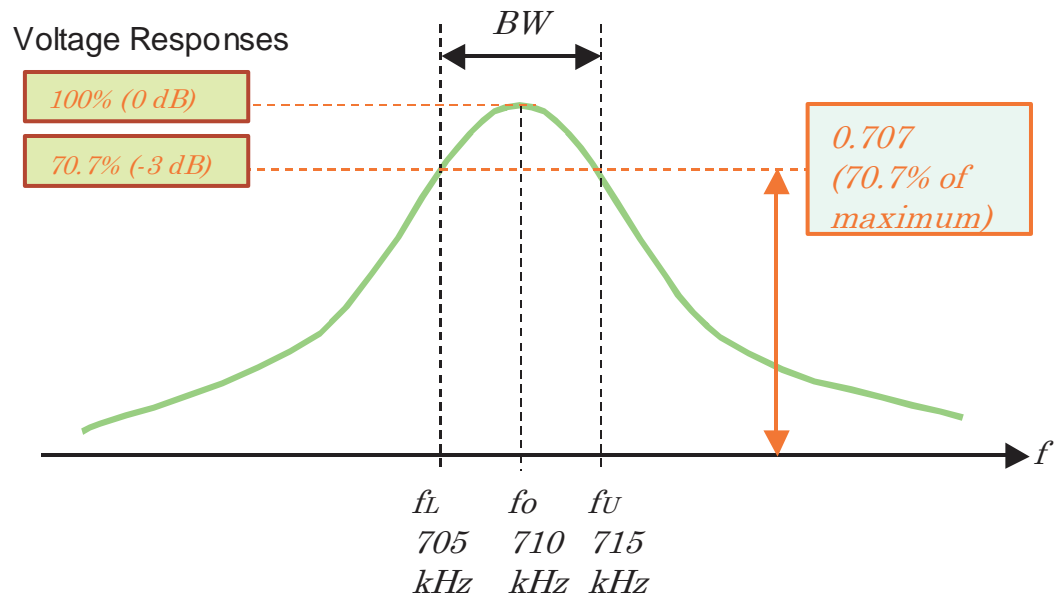


Figure 2: Bandwidth of a Resonant Filter

If we have measured the frequencies f_L , f_o and f_U , we can calculate the filter's bandwidth:

$$BW = f_U - f_L$$

Where f_L and f_U are the lower and upper frequencies at which the voltage response has become 70.7 % of maximum (half-power, -3 dB points). We can easily measure f_L , f_o , and f_U by using a variable frequency (swept) generator and an oscilloscope. Therefore we can easily measure the center frequency and bandwidth of a resonant filter.

In the figure above, the center frequency (f_o) of the filter is 710 kHz, and bandwidth is:

$$BW = f_U - f_L = 715\text{kHz} - 705\text{kHz} = \underline{\underline{10\text{kHz}}}$$

We can also calculate the bandwidth of the filter without doing any measurements at all, if we can find the Q of the circuit. For a series-resonant filter, the Q is calculated by:

$$Q = \frac{X_S}{R_S} \text{ Where } X_S \text{ is the } X_L \text{ or } X_C \text{ at resonance, and } R_S \text{ is the total series resistance.}$$

To find X_S , it is usually easiest to find X_L at resonance ($X_L = 2\pi f_o L$), or you can use the following "shortcut:"

$$X_L = X_C = Z_0 = \sqrt{\frac{L}{C}}$$

The above formula directly calculates the "resonant reactance" or "characteristic impedance" of the circuit without having to find f_o . It's commonly used in radio frequency analysis.

Once Q is known, the bandwidth can be calculated:

$$BW = \frac{f_o}{Q} \text{ (For } Q \geq 10)$$

If the quality factor is less than 10, this formula becomes only an approximation, and more exact methods would need to be used.

I Need a Coffee Break!

You may feel the need for a coffee break at this point. Or perhaps you'll walk a mile or two. That's normal for your first exposure to resonant circuits! Just keep in mind the following key ideas about resonant bandpass filter circuits:

- The circuit rejects frequencies above and below the center frequency, f_0 .
- The bandwidth is the range of frequencies the circuit can pass. It is the frequency distance between the two 70.7% response points.
- The bandwidth of a resonant filter is controlled by the Q of the circuit. The higher the Q , the smaller the bandwidth.
- The Q is controlled by the ratio of reactance to resistance in the circuit.

Action of the Parallel Resonant Circuit

Let's examine the parallel resonant circuit of Figure 1. Notice that the inductor and capacitor are now essentially in "shunt" (parallel) with the signal, and they now contact ground. The resistor R_p limits the current and defines the Q of the circuit.

At low frequencies (near DC), the X_L of $L1$ is very small. Since $L1$ is hooked to ground, it "shunts" (shorts) all of the signal to ground at low frequencies.

At very high frequencies, the X_L of $L1$ is very high and it no longer shorts the signal to ground. However, the X_C of $C1$ is nearly zero at very high frequencies, so $C1$ shunts very high frequencies to ground, again preventing them from passing to the output.

At the resonant frequency ($X_L = X_C$), $L1$ and $C1$ have *no* effect since their combined impedance is near infinity. Therefore nearly 100% of the signal passes to the output.

Like a series resonant circuit, a parallel resonant circuit has a center frequency where maximum response occurs:

$$f_0 = \frac{1}{2\pi\sqrt{LC}}$$

The bandwidth of the parallel resonant circuit is measured and defined exactly the same way as well:

$$BW = f_U - f_L = 715kHz - 705kHz = \underline{10kHz} \quad (\text{Where } f_U \text{ and } f_L \text{ are defined in Figure 2})$$

The bandwidth is also calculated the same way if Q is known:

$$BW = \frac{f_0}{Q} \quad (\text{For } Q \geq 10)$$

In fact the only difference (which is critical) is how Q is calculated. For a parallel circuit, Q is calculated as follows:

$$Q = \frac{R_p}{X_p} \text{ Where } R_p \text{ is the parallel lumped resistance, and } X_p \text{ is the resonant reactance.}$$

X_p is calculated by finding X_L or X_C at resonance (or using the Z_0 formula), just as in series resonance.

Measuring the Center Frequency and Bandwidth of a Resonant Filter

The setup for measuring the center frequency and bandwidth of a resonant filter is shown below in Figure 3. It should be familiar to you, as it is exactly the same as the configuration used in the previous experiment. You will be interpreting the readings a little differently, though.

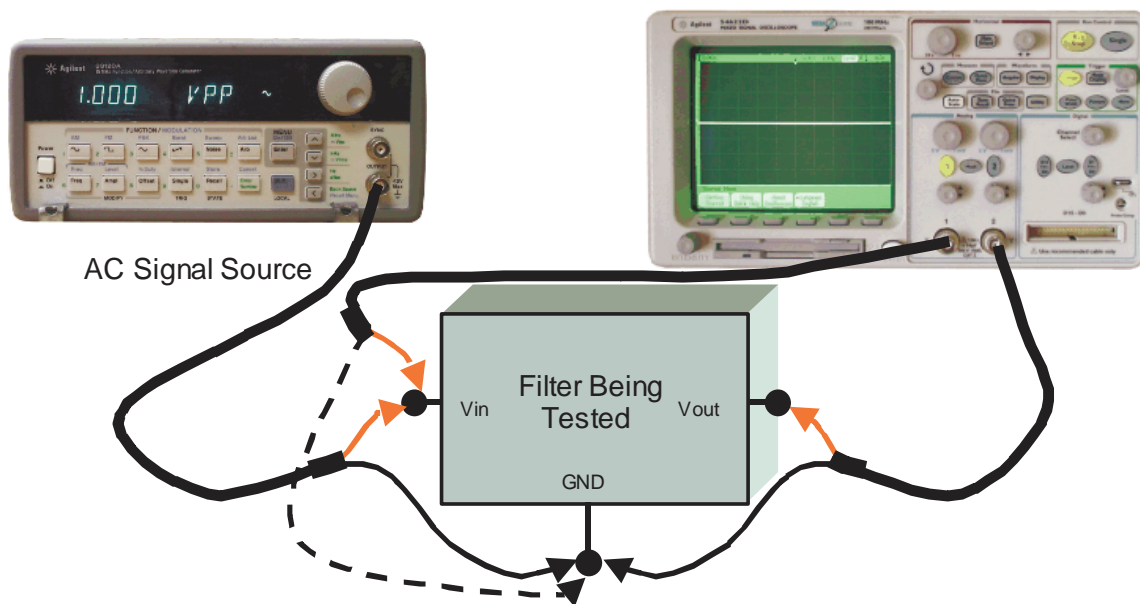


Figure 3: Setup for Measuring Center Frequency and Bandwidth

In this setup, a variable frequency AC signal source is used (this is the “swept” signal source). The filter is connected, and while varying the frequency of the generator, the filter’s output on channel 2 is observed. *Channel 1 is not watched in particular, except to occasionally check that it is not changing in amplitude.*

The frequency at which maximum output occurs on channel 2 is the center or resonant frequency. The generator frequency is then moved slowly up in frequency until the channel 2 response falls to 70.7% of its maximum value; this is the upper limit frequency f_U of the filter. Finally, the lower limit frequency f_L is found by slowly lowering the generator frequency (passing through resonance) until again the response falls to 70.7% of the maximum.

Laboratory Procedure

1. Connect up the series resonant filter of Figure 1 using the test setup of Figure 3.
2. Calculate the resonant frequency, Q , and bandwidth of the filter. Write the formulas below and show your work. You should get a resonant frequency of about 159 kHz.

$$f_0 = \underline{\hspace{2cm}}$$

$$Z_0 (X_L \text{ or } X_C \text{ at the resonant frequency}) = \underline{\hspace{2cm}}$$

$$Q = \underline{\hspace{2cm}}$$

$$BW = \underline{\hspace{2cm}}$$

3. Using the frequency sweep procedure described previously, find these three frequencies:

$$f_0 = \underline{\hspace{2cm}}$$

$$f_L = \underline{\hspace{2cm}}$$

$$f_U = \underline{\hspace{2cm}}$$

4. Calculate the bandwidth based on your answers from Question 3.

$$BW = \underline{\hspace{2cm}}$$

Note: You will see a wider bandwidth than the theoretical value because the inductor you're using is not ideal. It has an internal equivalent series resistance R_s which degrades the Q of the filter and widens the bandwidth.

5. How do the *calculated* and *measured* versions of these values compare?

6. Connect up the parallel resonant filter of Figure 1 using the test setup of Figure 3.
7. Calculate the resonant frequency, Q , and bandwidth of the filter. Write the formulas below and show your work. You should get a resonant frequency of about 50.3 kHz.

$$f_0 = \underline{\hspace{2cm}}$$

$$Z_0 \text{ (} X_L \text{ or } X_C \text{ at the resonant frequency)} = \underline{\hspace{2cm}}$$

$$Q = \underline{\hspace{2cm}} \text{ (Be sure to the PARALLEL } Q \text{ formula!)}$$

$$BW = \underline{\hspace{2cm}}$$

8. Using the frequency sweep procedure described previously, find these three frequencies:

$$f_0 = \underline{\hspace{2cm}}$$

$$f_L = \underline{\hspace{2cm}}$$

$$f_U = \underline{\hspace{2cm}}$$

9. Calculate the bandwidth based on your answers from Question 8.

$$BW = \underline{\hspace{2cm}}$$

10. How do the *calculated* and *measured* versions of these values compare?

Questions

1. What is the difference between a series and parallel circuit?

2. How is resonance defined (in terms of X_L and X_C) for a resonant circuit?

3. What is the bandwidth of a resonant circuit?

4. Explain how to measure bandwidth of a resonant filter circuit.

5. What have you learned in this experiment?
