Oscilloscopes, Signal generators and RF impedance analysis

In today's lab, you'll use a frequency generator together with the storage oscilloscope. I recommend only that you make only slight adjustments to the equipment until you learn a bit about how it works. For the students that have the 2001 generators, the frequency will need to be read either from the oscilloscope or from the computer screen. You'll find a step-by-step procedure at the end of this writeup that will help with data measurement and acquisition.

About RMS values:

Suppose a sinusoidally varying current is applied across a resistor (We're initially assuming the circuit is purely resistive):

$$I = I_m \sin(\omega t)$$

$$V = V_m \sin(\omega t)$$

The instantaneous power radiated is given by:

$$P = IV = I_m V_m \sin^2(\omega t)$$

It is not the instantaneous power which is so interesting: it is the time average power. This is given by:

$$\langle P \rangle = I_m V_m \langle \sin^2(\omega t) \rangle$$

From last semester, we know the time average has the value of ½. Thus:

$$\langle P \rangle = \tfrac{1}{2} I_m V_m$$

If we wrote, instead of the peak voltages and currents, these peaks scaled, we can make the form look the same as for DC. Thus, we define:

$$I_{rms} \equiv \tfrac{I_m}{\sqrt{2}} : V_{rms} \equiv \tfrac{V_m}{\sqrt{2}}$$

Then for a purely resistive circuit, we would have:

$$\langle P \rangle = I_{rms} V_{rms}$$

You need to know in a particular context if something is talking about peak values or rms values.

RC – LC – RLC – RL series circuits

This analysis applies strictly to a series RLC circuit.

Impedance: We need to define some "resistive-like" quantities

(a) Inductive reactance: $X_L = \omega L$

(b) Capacitive reactance: $X_C = \frac{1}{\omega C}$

You can verify that these have units of Ohms.

Let's apply a sinusoidally varying current to the series circuit. At any time across the current is varying throughout the circuit as:

$$I = I_m \sin(\omega t)$$

Impedance for a ***series*** RLC circuit is then given by:

$$Z = \sqrt{R^2 + (X_L - X_c)^2}$$

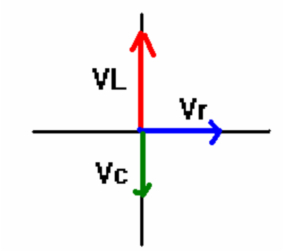
Ohm's law for Impedance:

$$V = IZ$$

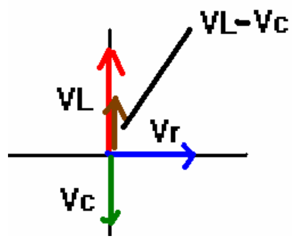
The potential drop across the entire circuit will in general not be in phase with the applied current. In fact, it will vary as:

$$V = V_{max} \sin(\omega t + \delta)$$

For a purely capacitive circuit, the voltage across the capacitor lags behind the current by 90 degrees and the voltage drop across the capacitor is given by $V_c=IX_c$. For a RL circuit, the voltage across the inductor leads applied current by 90 degrees and the voltage drop across the inductor is given by $V_L=IX_L$. Across a purely resistive circuit, the voltage drop is in phase with the applied current and is given by $V_R=IR$.



We can write this in terms of two vectors now using the difference between VL and Vc.



The magnitude of the instantaneous voltage is then given by :

$$|V| = \sqrt{V_R^2 + (V_L - V_C)^2} = I\sqrt{R^2 + (X_L - X_C)^2}$$

The angle between V_r and this instantaneous voltage is given by:

$$\tan\varphi = \frac{V_L - V_C}{V_R} = \frac{X_L - X_C}{R} = \frac{\omega L - \frac{1}{\omega C}}{R}$$

The "power factor" is the cosine of this angle, and the average power radiated by the circuit is related to the power factor by:

$$\langle P \rangle = IV = I_{rms} V_{rms} \cos\varphi$$

Here is another way to get the power factor:

Assuming the current is the same in all parts of the circuit, we have:

$$\text{PowerFactor} = \frac{\langle \text{True power} \rangle}{\langle \text{Apparent Power} \rangle} = \frac{I_{rms}^2 R}{I_{rms}^2 Z} = \frac{R}{Z} = \frac{R}{\sqrt{R^2 + (X_L - X_C)^2}} = \cos\varphi$$

Your work today involves analysis of various aspects of the RC and RLC circuit but not actually the power factor. It's worth realizing that the power factor exists, however, and since mostly power is radiated by resistance, it can effectively reduce Joule heating and power losses over DC circuits.

The RC filter circuits

If the inductance is not present, the impedance is given by:

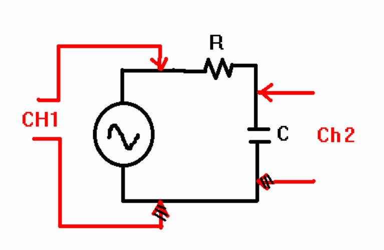
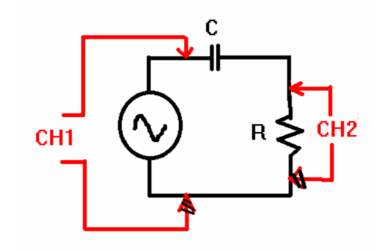
$$Z = \sqrt{R^2 + \left(\frac{1}{\omega C}\right)^2}$$

(1) voltage output from the capacitor: $V_c = IX_c \Rightarrow \frac{V_c}{V_{in}} = \frac{V_{out}}{V_{in}} = \frac{IX_c}{IZ} = \frac{\left(\frac{1}{\omega C}\right)}{\sqrt{R^2 + \left(\frac{1}{\omega C}\right)^2}} = \frac{1}{\sqrt{1 + (\omega RC)^2}}$

(2) Voltage output from the resistor: $\frac{V_R}{Vin} = \frac{IR}{IZ} = \frac{R}{\sqrt{R^2 + \left(\frac{1}{\omega C}\right)^2}} = \frac{\omega RC}{\sqrt{1 + (\omega RC)^2}}$

(1) goes to zero as frequency gets very large. For this reason, it is called a “low pass filter”. (2) goes to zero as frequency gets very low. For this reason, it is called a “high pass filter”. The amazing thing about this is they both are basically the same circuit ... it just depends what you consider your output to be. In the lab today, you’ll measure both. You have to be somewhat careful in practice with how you arrange things since the presence of the ground removes all signals after such a connection.

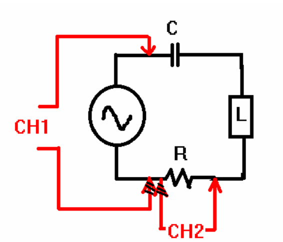
In the table below, you can obtain limiting behavior by looking at the capacitor: at low frequencies, it is essentially replaced by an open switch while at high frequencies, it is essentially replaced by a closed switch.

Low Pass Filter connections	**High pass filter connections**
$\Re \equiv \frac{V_{out}}{V_{in}} = \frac{1}{\sqrt{1+(\omega RC)^2}}$	$\Re \equiv \frac{V_{out}}{V_{in}} = \frac{\omega RC}{\sqrt{1+(\omega RC)^2}}$
Limit: LF: $\Re = 1$	Limit: LF: $\Re = 0$
Limit: HF: $\Re = 0$	Limit: HF: $\Re = 1$
	

For the RC circuit, the phase between I_r and V_r is given by:

$$\tan\delta = \frac{X_c}{R} = \frac{1}{\omega RC}$$

Today I’ll only ask you to measure the phase between an element and the input voltage. You’ll see that the phase does indeed shift and as a result of fitting the response function, the phase shift will automatically be provided for the given fit (and it works pretty well). However, it can not really distinguish in this analysis between the contribution from R and the contribution from C so therefore the fit is to the product RC. Also, if you look closely enough, you’ll see that I fit log10(τ) rather than the time constant itself. You should do the same.



For the RLC circuit, I want you to monitor the voltage drop across the resistor as a function of input frequency and input peak-to-peak voltage and also to take phase measurements. The circuit connections are shown below. The response function is given by:

$$\Re \equiv \frac{V_{out}}{Vin} = \frac{IR}{IZ} = \frac{R}{\sqrt{R^2+\left(\omega L-\frac{1}{\omega C}\right)^2}} = \frac{1}{\sqrt{1+\left(\frac{\omega L}{R}-\frac{1}{\omega RC}\right)^2}}$$

Resonance will occur in this circuit when the inductive reactance is equal to the capacitative reactance:

$$\text{resonance} \Rightarrow X_L = X_c \Rightarrow \omega_r L = \frac{1}{\omega_r C} \Rightarrow \omega_r = \sqrt{\frac{1}{LC}}$$

In the experiment today, the inductance is about 33 mh while the capacitance is about 1μf. This would give a resonant frequency of :

$$\omega_r = \sqrt{\frac{1}{LC}} = \sqrt{\frac{1}{33\times10^{-3}}} \approx 5.5\,\text{kHz}$$

At resonance the response function will reach a maximum value and the power factor will be 1 meaning that the voltage and the current are in phase across the resistor. This circuit is also called the "tank circuit" forms the basis for an enormous number of scientific and real world applications (NMR, Radio, etc.). I want you to observe the response function.

Data analysis:

For the LP filter, we have:

$$\Re \equiv \frac{V_{out}}{V_{in}} = \frac{1}{\sqrt{1+(\omega RC)^2}} \Rightarrow \log(\Re) = -\frac{1}{2}\log\left(1+(\omega RC)^2\right)$$

For low frequencies, the response is essentially flat (1 is the predominate term) while at larger frequencies, the log(response) is nearly linear in log(frequency). The phase is given by

$$\tan\delta = \frac{R}{X_c} = \omega RC$$

The phase fit will be automatically given as a byproduct of the fit to the frequency response.

For the HP filter, we have:

$$\Re = \frac{V_{out}}{V_{in}} = \frac{\omega RC}{\sqrt{1+(\omega RC)^2}} \Rightarrow \log(\Re) = \log(\omega RC) - \frac{1}{2}\mathrm{Log}\left(1+(\omega RC)^2\right)$$

$$\tan\delta = \frac{X_c}{R} = \frac{1}{\omega RC}$$

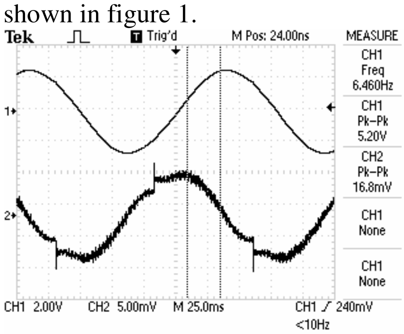
The phase fit will be automatically given as a byproduct of the fit to the frequency response.

The predominate term here is the first: at zero frequency it would diverge, meaning the response function is zero. From there, it increases up to the point at which it levels off at high frequencies and the value there is about given by 1, the log of which is zero. We can fit these outputs to the theoretical response curves and I recommend that you do, but in the logarithmic plane. However, verifying the response curves is secondary to today's lab: it is merely one example application of oscilloscopes to scientific research. For the RLC circuit, I really want you to verify the nature of the response function and also to see how the phase varies up to resonance. In particular, I want you to observe the enormous peak which occurs at resonance in your data. This particular effect is of tremendous importance in many applications. For this reason, I suggest that in the region between 1KHz and 100 KHz, you may want to take 3-4 times as much data as I have recommended as the bare minimum. You'll need to modify the graphing parameters and pull and drag a bit if you do on the spreadsheet.

Frequency Analysis Procedure

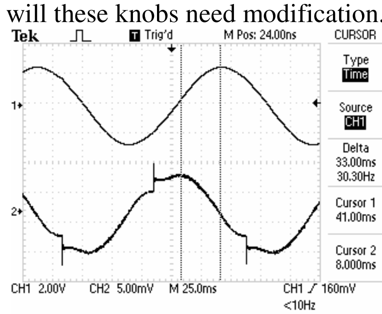
Run TeK Setup 5 from the website and then start the TeK Analyzer program.

Set the frequency on the ith frequency shown on the frequencies chart.
Press the scope "measure" buttons to adjust the <position> knob so that the scan looks somewhat like that shown in figure 1.



You may need to make additional adjustments with the <Volts/Div> knob for CH2 and also, perhaps for CH1. Additionally, you may need to modify the Horizontal <Sec/Div> knob to insure you obtain a complete enough scan for the frequency measurement to be valid (the program will say invalid measurement if this is not correctly set).
You will also notice that by changing the Trigger <Level> knob, the arrow indicting where the trigger will occur can be modified. I suggest, however, not changing this setting after you set it for your experiment.

Next: press the <Cursor> button: this will allow the two vertical position knobs to control the cursor (vertical lines are the cursor) position. You will want to probably associate knob 2 with channel 2 and knob 1 with channel 1. You should confirm that the Type indicator says Time with source Ch1. By placing the cursors on the curve peaks as shown in figure 2, you are able to make a measurement of the phase. You'll want to do this for each measurement. You will not always, however, need to adjust the curve <position>, <Volts/division> and <sec/div> knobs …. Only about once per decade will these knobs need modification.



Now that you have your adjustments made, you'll want to press "R" in the analysis program. This resets the acquisition system. At the lower frequencies it is a bit of a wait until 128 acquisitions are complete. After the acquisition is complete, press the "space bar" to record the measurement. In all of this, however, make sure the program does not say "Invalid Measurement." If it does, what is needed is to modify the <Sec/Div> knob or occasionally the Ch2 <Volts/Div> knob.

The initial settings for the TeK are contained in setup 5. Just to make sure about what these setting are, let me review them with you and you can confirm. To access them in the easiest way possible, run the program from the website: "Tek Setup 5" (I recommend this strongly).

Press <Ch1 Menu> button
You should see the following:
Ch1: Coupling AC : BW Limit Off 60 MHz: Volts/Div: Coarse: Probe 1X: Invert Off.
You can change the setting by pressing the button immediately to the right of the menu item on the TeK.

Press <Ch2 Menu> button
Ch2: Coupling AC: BW Limit Off 60 MHz: Volts/Div Coarse: Probe 1X Invert Off

Press the <Trig Menu> button
Trigger: Type Edge : Source CH1 : Slope Rising: Mode Normal: Coupling AC

Press the <display> button
Display: Type Vectors : Persist 1 sec : format yt
Press the <measure> button to return to the normal display, then the <cursors> button to show cursors.
Note that you could have chosen xy in the display format to show ch1 on the x axis and ch2 on the y axis.