# CE1100 Introduction to Electrical and Computer Engineering

Lab Report #1: RL, RC and RLC Filter Circuits

Laboratory Team Member: Mahir Rahman and Lecture Section # 001

Instructor: James M. Florence

Date: October 6, 2020


## Abstract

The objective is to demonstrate effects when RL, RC, and RLC circuits are supplied different frequencies of voltage for different lengths of time. The results showed no current in steady states, lagging current across high and low pass filter, and sudden changes in transient state. The conclusion was drawn that frequency has an impact on inductors and capacitors.

## Introduction

The experiment is to observe the effects in RL, RC, and RLC circuits at different frequencies of voltage. Multisim is used to records changes in current and voltage, using voltmeters and ammeters, in the DC steady state, AC interactive, AC sweep, and Transient states. Three different circuits are set up, one has DC, other has AC, and the last one has step voltage. This way we create various situations and collect data.

## Procedures and Expected Results

The first circuit that will be observed is an RL circuit. It incorporates a resistor and an inductor and is supplied DC voltage for a long period of time. The circuit diagram is provided below:

**DC Steady State**

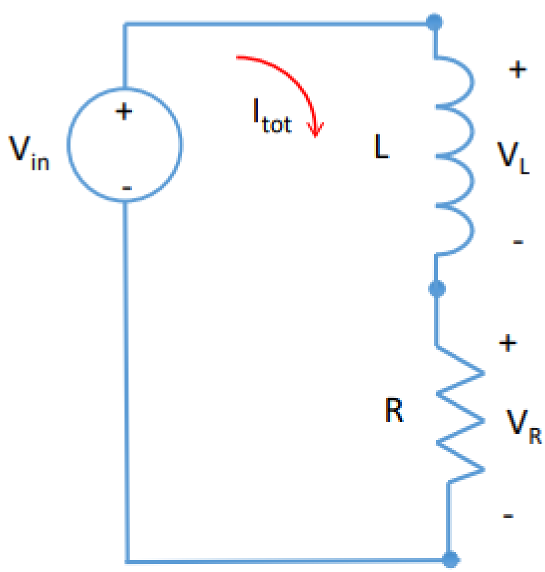



Figure 1.1 A series RL circuit set up as a DC steady

In Figure 1.1, we find an inductor and resistor connected in series. According to KVL:

$$V_{in} = V_L + V_R$$

Where, $V_{in}$ is source voltage, $V_L$ is voltage across inductor and $V_R$ is voltage across resistor.

As circuit is left for a long time in the DC steady state, there will be no voltage drop across the inductor. It will act like any other wire. Therefore, the source voltage will be equal to the resistor voltage ( $V_{in} = V_R$ ). Now, the circuit is set up on Multisim to prove the theory. A DC source of 3.3 V is set up, connecting wires are used to connect inductor of 220 µH and resistor of 560 Ω in series. A voltmeter and ammeter is connected with reference point set to ground. The DC OP is selected from drop down menu, and the simulation is turned on. The results should appear like the table provided below:

| **Quantity** | **Magnitude** |
|---|---|
| Voltage across resistor ($V_R$) / V | 3.3 |
| Voltage across inductor ($V_L$) / V | 0.0 |
| Current through resistor ($I_{tot}$) / mA | 5.9 |
| Voltage across source ($V_{in}$) / V | 3.3 |

Table 1 Expected results for RC circuits in DC Steady State

From Table 1, we observe that voltage across inductor is 0 V. The tabulated values follow KVL. The source voltage and resistor voltage are equal.

The next experiment concerns an RL circuit which demonstrates lagging current when connected to an AC supply in Interactive Mode. The circuit diagram is provided below:

**AC Supply (Lagging Current)**

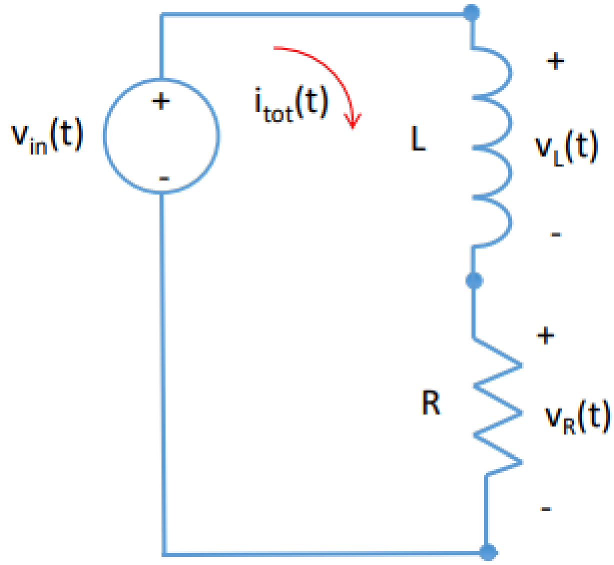



Figure 1.2 A series RL circuit set up as AC interactive

In Figure 1.2, there is the set up like Figure 1, but the source voltage is AC. This means, voltage will go from positive to negative to positive again in a complete cycle. The equations are as follows:

$$v_{in}(t) = L\frac{di_{tot}}{dt} + \mathrm{R}i_{tot}(t) \qquad v_{in}(t) = V_o \sin(\omega t)$$

$$i_{tot}(t) = \frac{V_o}{\sqrt{R^2 + \omega^2 L^2}} \sin(\omega t + \theta_{RL})$$

where:

$$\theta_{RL} = \arctan\left(-\frac{\omega L}{R}\right)$$

Where, $\omega = 2\pi f$, L is inductance, R is resistance, $i_{tot}$ (t) is source current and $V_o$ is highest amplitude of voltage.

According to Faraday's law, a voltage is set up across the inductor which is proportional to the rate of change of current. That is why, in the equation derived using KVL, voltage across inductor is the product of inductance and rate of change of current, but the voltage across resistor is the product of current and resistance. As source voltage is sinusoidal, the sine operator is used. The equation is rearranged to give the third equation, and θ is explained in the fourth equation.

The theory tells us that as the frequency of AC supply increases, the value of source current decreases and a phase difference causes the current to appear as lagging. There will be no impact on source voltage.

On Multisim, the circuit is set up with inductor of inductance 220 μH and resistor of resistance 560 Ω in series, and an AC supply of 3.3 V and 500 kHz. The voltmeter, with reference to ground, and ammeter are connected. The mode is switched to Interactive, and the simulation is started. The expected results are below:

An example in a circuit where $R = \omega L = 1$ and $\theta_L = -\pi/4$

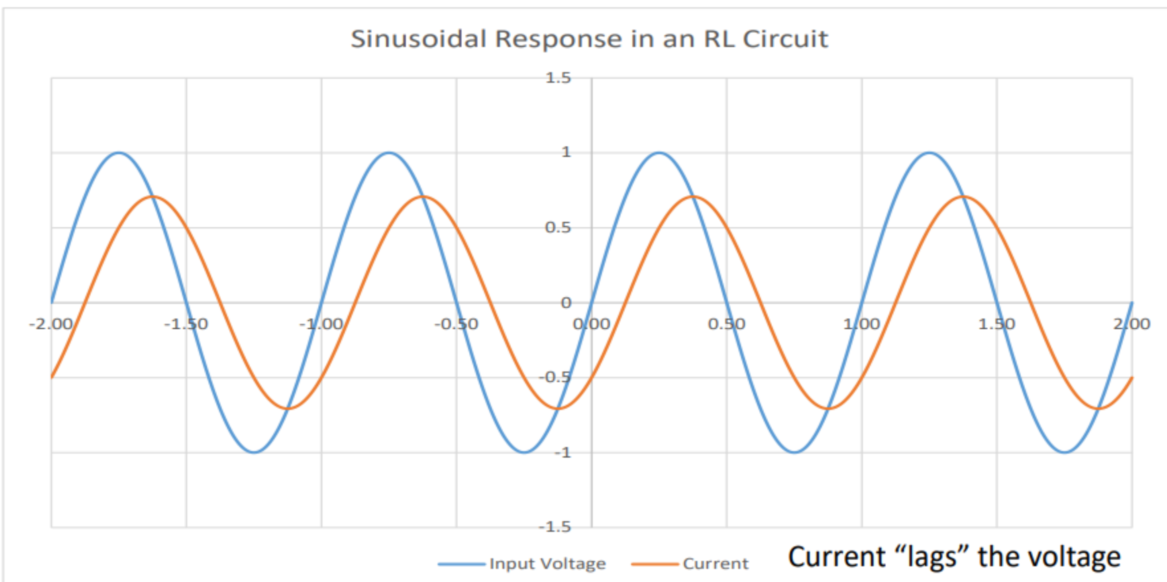



Figure 1.3 Expected lagging current from RL circuit analysis with AC supply

In Figure 1.3, we observe the lagging current we are looking for. The graph should appear like this in our Multisim results.

This time we will be observing an RL circuit's low pass and high pass filter characteristics. First, let us consider the low pass filter. The circuit is set up as shown in Figure 1.4 below:

**Low Pass Filter**

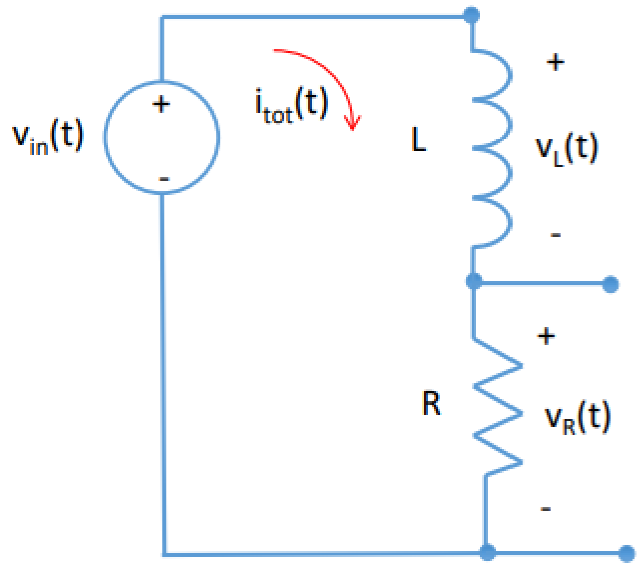



Figure 1.4 An RL circuit set up to demonstrate low pass filter

From Figure 1.4, we see that it is the same set up as Figure 1.2, except that a voltmeter is connected across the resistor. The equations to understand this situation are provided below:

$$v_R(t) = R\, i_{tot}(t)$$

$$v_R(t) = \frac{R\, V_o}{\sqrt{R^2 + \omega^2 L^2}} \sin(\omega t + \theta_{RL})$$

$$v_R(t) = \frac{V_o}{\sqrt{1 + \frac{\omega^2 L^2}{R^2}}} \sin(\omega t + \theta_{RL})$$

$$\theta_{RL} = \arctan\left(-\frac{\omega L}{R}\right)$$

.

In this case, we are observing only resistor voltage. Therefore, it is the subject of second equation. Now, according to the equation, as frequency increases, the denominator becomes large and the voltage across the resistor decreases in third equation.

Now, this is observed in Multisim. A circuit is set up with the same value of inductance and resistance as before, and an AC supply of 3.3 V and frequency of 500k Hz. The AC Sweep option is selected, and simulation is started.

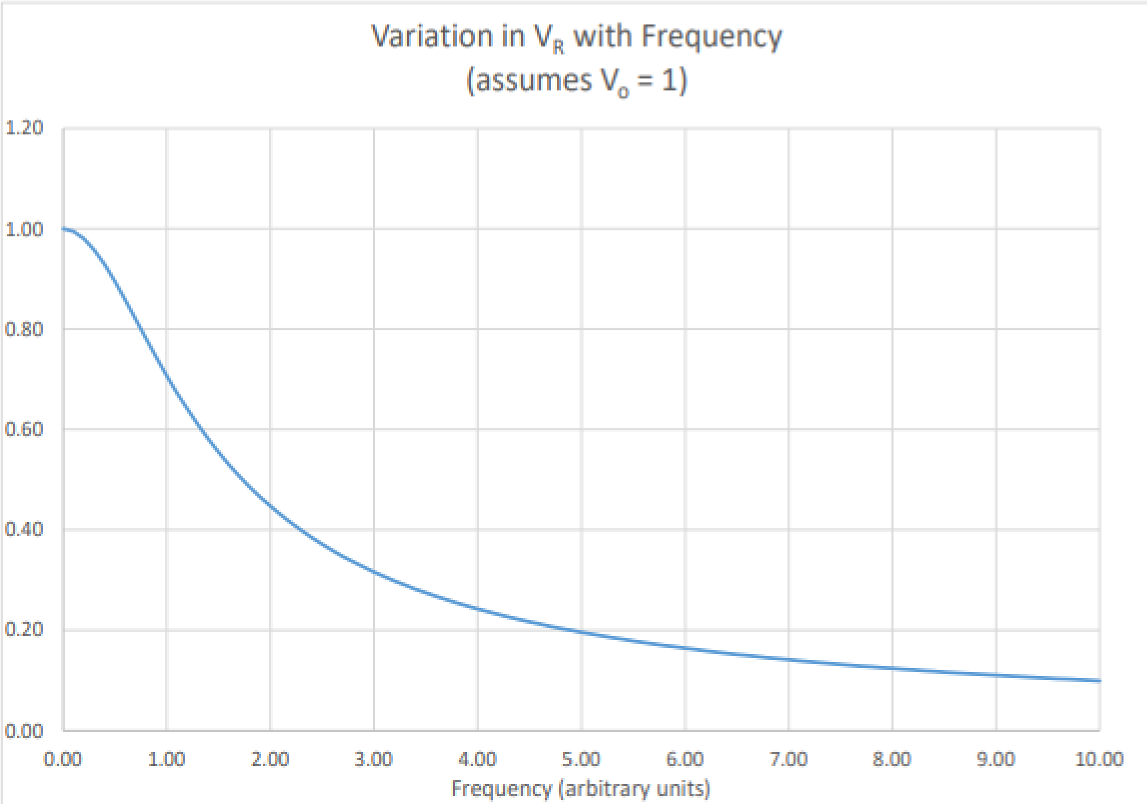



Figure 1.5 Expected results for a low pass filter

From Figure 1.5, we observe the expected shape of the graph. The voltage across resistor falls as frequency increases. It eventually falls to 0 V at very high frequencies.

Now, we observe the high pass filter. The circuit is set up as shown below:

**High Pass Filter**

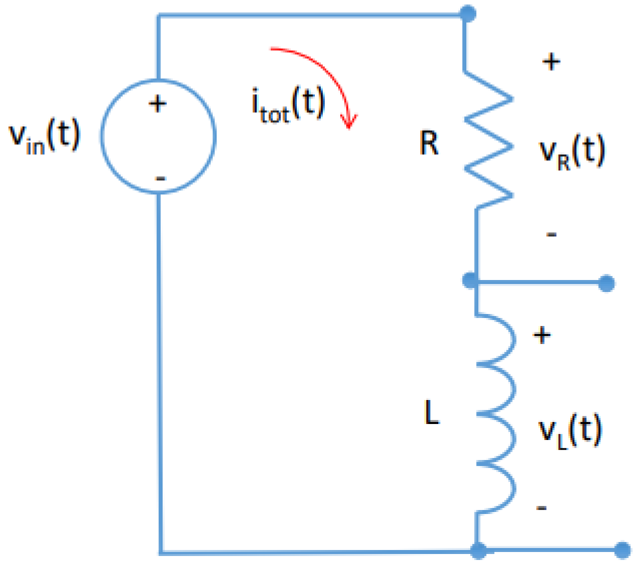



Figure 1.6 RL circuit with voltmeter across inductor, acting as high pass

In Figure 1.6, a voltmeter is connected across inductor instead of resistor. The circuit will now demonstrate high pass filter properties. The equations that demonstrate why this occurs are provided below:

$$v_L(t) = L\frac{di_{tot}(t)}{dt}$$

$$v_L(t) = \frac{V_o}{\sqrt{1 + \frac{R^2}{\omega^2 L^2}}} \cos(\omega t + \theta_{RL})$$

$$\theta_{RL} = \arctan\left(-\frac{\omega L}{R}\right)$$

We are observing the voltage drop across inductor, and so the first equation represents the voltage across the inductor. The $i_{tot}$ is substituted with corresponding value, and rearranged to make $V_L$ (t) the subject of the equation. As ω increases, the denominator of the main fraction decreases, and so $V_L$ (t) increases.

Multisim is turned on with AC sweep and AC voltage of 3.3 V. The resistance and inductance are same as before. Moreover the circuit set up is as in the low pass filter, but the voltmeter is connected to inductor and a reference point is set up between the inductor and resistor.

We can expect the following results:

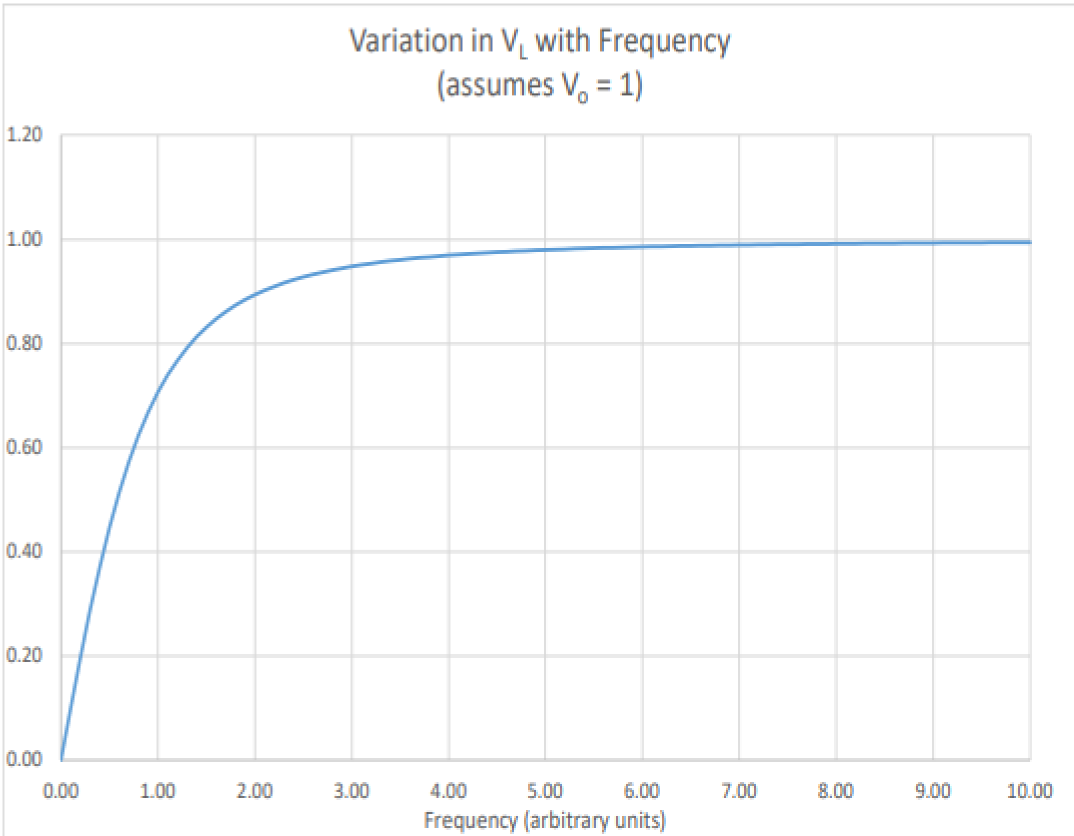



Figure 1.7 RL circuit with expected high pass filter

In Figure 1.7, voltage across inductor is 0 when frequency is 0, but as frequency increases the voltage across it increases. According to Faraday's Law, the rate of change of current is proportional to the voltage induced across the inductor. Therefore, as frequency increases, the rate of change of current increases, and the voltage across inductor increases. We can expect this kind of graph from the analysis.

The last example from RL circuit is the transient analysis.

**Step Up and Down Voltage Circuit**

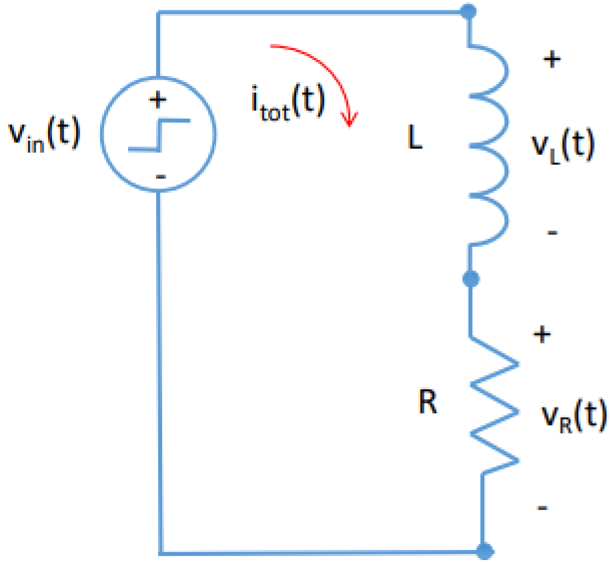



Figure 1.8 RL circuit set up to demonstrate transient effect

From Figure 1.8, we see that a step up or down voltage is connected to the circuit. It shows what happens when voltage is suddenly switched on and off respectively. The source voltage, voltage across resistor and inductor, and source current is observed after circuit is set up.

Let us consider the step up voltage first. The equations are as follows:

$$i_{tot}(t) = I_{tot}(1 - e^{-\alpha t})$$

$$i_{tot}(t) = \frac{V_{in}}{R}(1 - e^{-\alpha t}) \quad with \quad \alpha = \frac{R}{L}$$

The first equation shows current as the function of time. The current through the circuit is source voltage divided by resistance. The RL time constant shows us the time taken for transient effect to pass.

The circuit is now set up on Multisim. The source is set up to be a step up voltage from 0 to 3.3 V. The inductor has a value of 220 µH and resistor has a value of 560 Ω. Voltmeters across source, resistor, and inductor are set up. The source current is recorded using ammeter. The transient function is selected from drop down menu, and the results should come up as below:

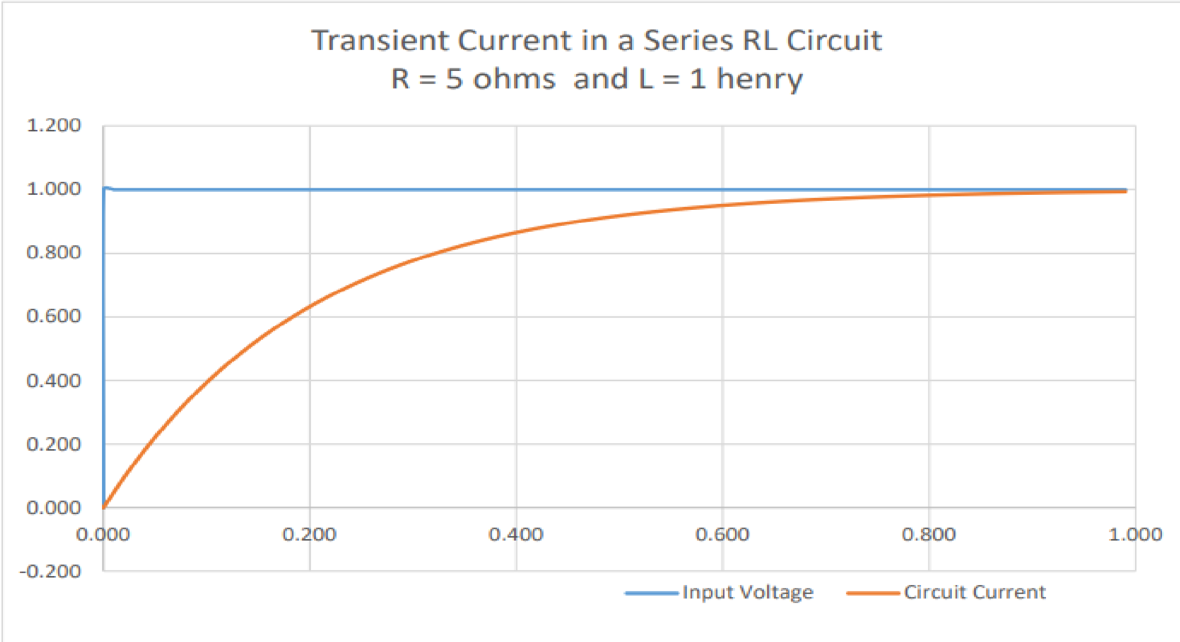



Figure 1.9 Transient effects when voltage is stepped up

From Figure 1.9, we see that the circuit current increases while source voltage immediately jumps to 3.3 V. Additionally, the voltage across inductor will also climb to 3.3 V instantly, and then slowly continue to fall to zero. The voltage across resistor should remain zero initially, but slowly increases as current is supplied for longer time.

The analysis in the results section will demonstrate what the Transient looks like when voltage is stepped down.

For the next part, we will be demonstrating RC circuits. The first circuit will be in the DC Steady State.

**DC Steady State (RC)**

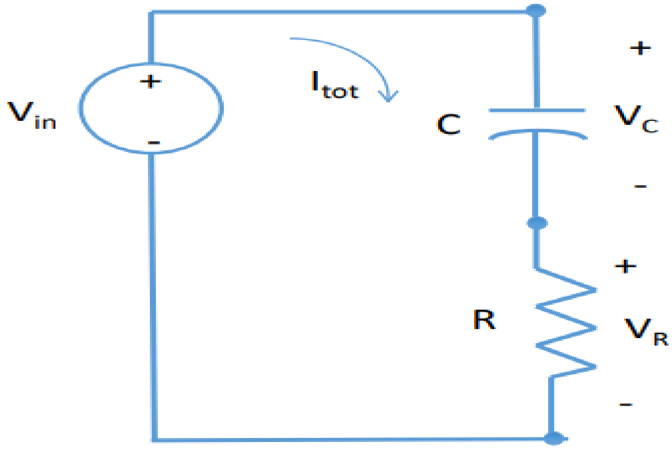



Figure 2.1 RC circuit set up as DC Steady State

From Figure 2.1, we see that a capacitor and resistor are in series, and a DC voltage source has been provided. The source voltage, source current, $V_R$ and $V_c$ is to be measured, and recorded. The expected results and equations are explained below:

$$V_{in} = V_C + V_R$$

$$V_C = V_{in}$$

Where, $V_{in}$ is supply voltage
$V_R$ is resistor voltage
$V_c$ is capacitor voltage

According to KVL, voltage across components will be equal to source voltage. However, when connected across a capacitor, voltage across resistor falls to zero. This is because, the capacitor stores charge and with a long time in the DC steady state, charge builds up and current flow stops. $V_{in}$ is equal to Vc.

The circuit is now set up in Multisim, the DC supply is connected to a capacitor of capacitance 5700 pF and resistor of resistance 560 Ω. Ammeter and voltmeter are connected to source to measure source current and voltage. Two more voltmeters are added across the two components. The DC OP is selected from the drop-down menu, and simulation is started.

| Quantity | Magnitude |
|---|---|
| Voltage across resistor ($V_R$) / V | 0.0 |
| Voltage across capacitor ($V_C$) / V | 3.3 |
| Current across circuit ($I_{tot}$) / A | 0.0 |
| Voltage across source ($V_{in}$) / V | 3.3 |

Table 2 Expected results for RC in DC Steady State

From Table 2, we see that $V_R$ is 0 V while $V_c$ is equal to source voltage. The current flowing is also 0 A. This shows the capacitor stores charge, and eventually stops current flow across resistor. These should be the expected results.

The next RC circuit is going to show leading current when capacitor and resistor are connected in series to an AC supply.

**RC Circuit (AC Supply)**

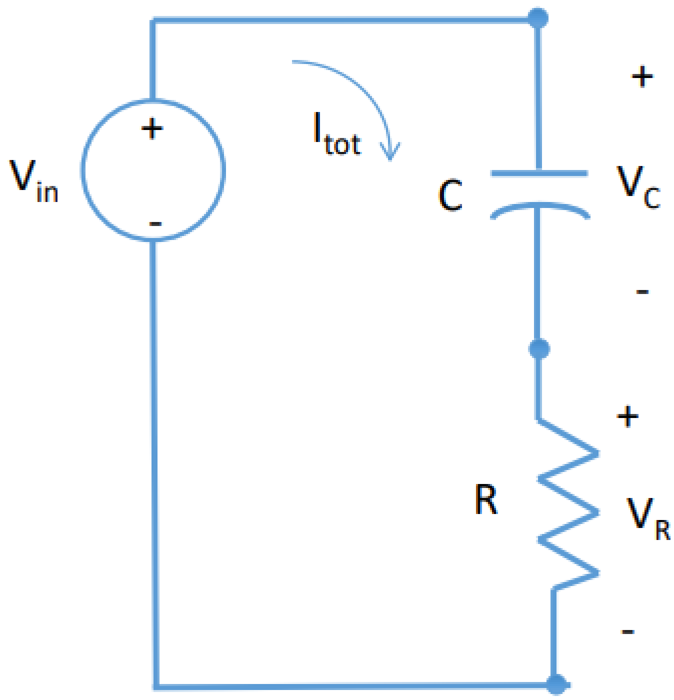



Figure 2.2 RC circuit set up as AC Interactive to demonstrate leading current

From Figure 2.2, we observe that the resistor and capacitor are in series to an AC supply. The current is seen to be leading in this case. The equations help to explain the concept.

$$v_{in}(t) = \frac{1}{C}\int i_c(t)\,dt + \mathrm{R}\,i_{tot}(t)$$

Where, $i_C$ is current across capacitor, $i_{tot}$ (t) is source current, Vin (t) is source voltage and $V_o$ is maximum amplitude of voltage, C is capacitance and $\omega = 2\pi f$.

If we let $v_{in}(t) = V_o \sin(\omega t)$

Then the solution is:

$$i_{tot}(t) = \frac{V_o}{\sqrt{R^2 + \frac{1}{\omega^2 C^2}}} \sin(\omega t + \theta_{RC})$$

where: $$\theta_{RC} = \arctan\left(\frac{1}{\omega RC}\right)$$

As mentioned before, KVL means component voltages add up to source voltage. In terms of capacitance and resistance, voltage across different components is as illustrated in equation 1. The equation is rearranged to make $i_{tot}$ (t) the subject. Moreover, $\theta_{RC}$ decreases as $\omega$ increases, but phase difference becomes more positive. Therefore, current is leading.

Now, the circuit is built in Multisim. An AC voltage of 3.3 V and 30 kHz is connected to a capacitor of 5700 pF and resistor of 560 Ω. $V_R$ is such that the source voltage is half its maximum value. Source voltage and current are measured, and $V_R$ is also recorded. All voltmeters are referenced to ground. The AC Interactive mode is selected and simulation started.

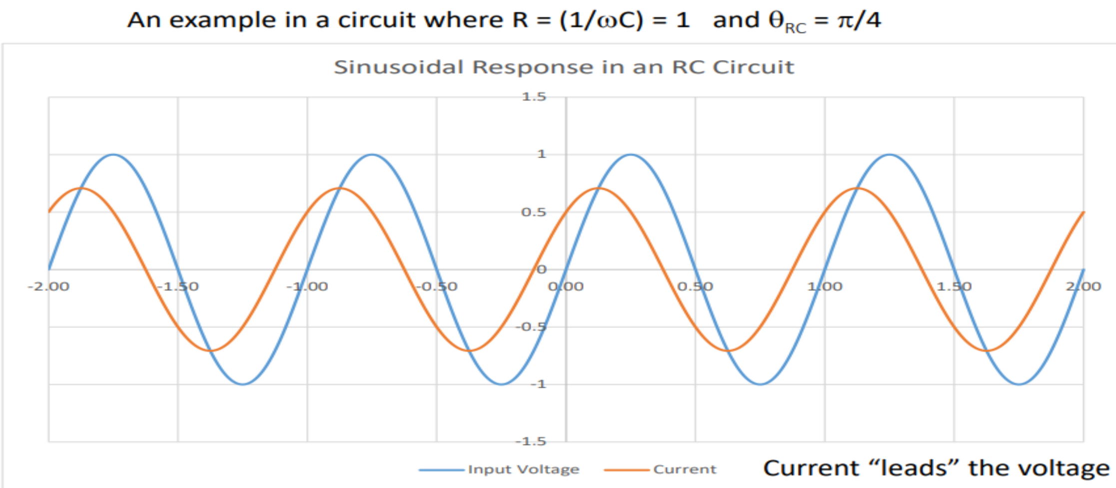



Figure 2.3 RC circuit set up as AC Interactive to demonstrate expected leading current

As shown in Figure 2.3, the current is leading. It is represented by the brown line and it reaches its maximum amplitude faster than the blue line (input voltage). Additionally, $V_R$ is proportional to current, and so should be approximately half of source voltage.

In the next stage, we will be observing the low and high pass filter. Let us begin with the high pass filter.

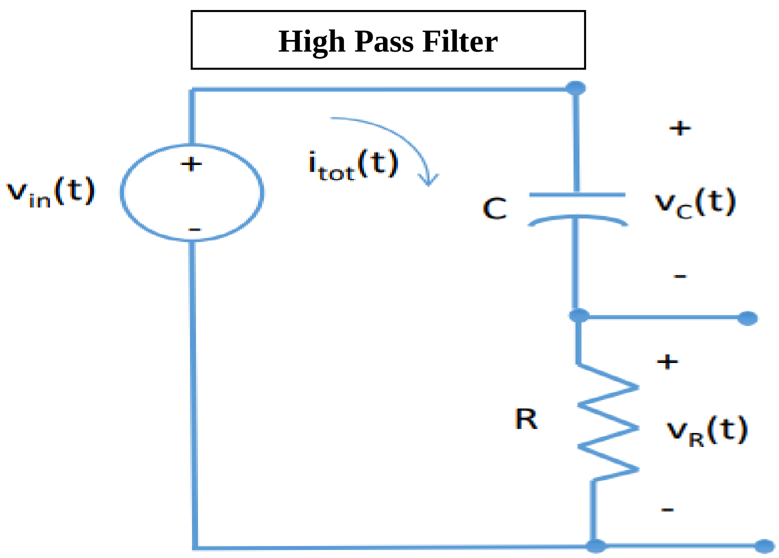



Figure 2.4 RC circuit set up to demonstrate high pass filter

From Figure 2.4, we observe that it is the same as Figure 2.2, with the exception of a voltmeter across resistor. The equations below will help to explain why this circuit is a high pass filter.

$$v_R(t) = R\, i_{tot}(t)$$

giving:

$$v_R(t) = \frac{R\, V_o}{\sqrt{R^2 + \frac{1}{\omega^2 C^2}}} \sin(\omega t + \theta_{RC})$$

$$v_R(t) = \frac{V_o}{\sqrt{1 + \frac{1}{\omega^2 R^2 C^2}}} \sin(\omega t + \theta_{RC})$$

$$\theta_{RC} = \arctan\left(\frac{1}{\omega RC}\right)$$

We will be observing $V_R$, and so it is shown as a product of R and $i_{tot}$ (t). As the frequency increases, ω increases. The denominator of the equation in third line decreases, and so $V_R$ (t) increases. So it can be deduced that resistor voltage increases with increase in frequency, and so it is a high pass filter.

Now, the circuit is observed in Multisim. The AC supply is 3.3 V, connected in series to capacitor of 5700 pF and resistor of 560 Ω. A voltmeter, the ref point being ground, is set up across resistor. The AC Sweep mode is selected, and simulation started.

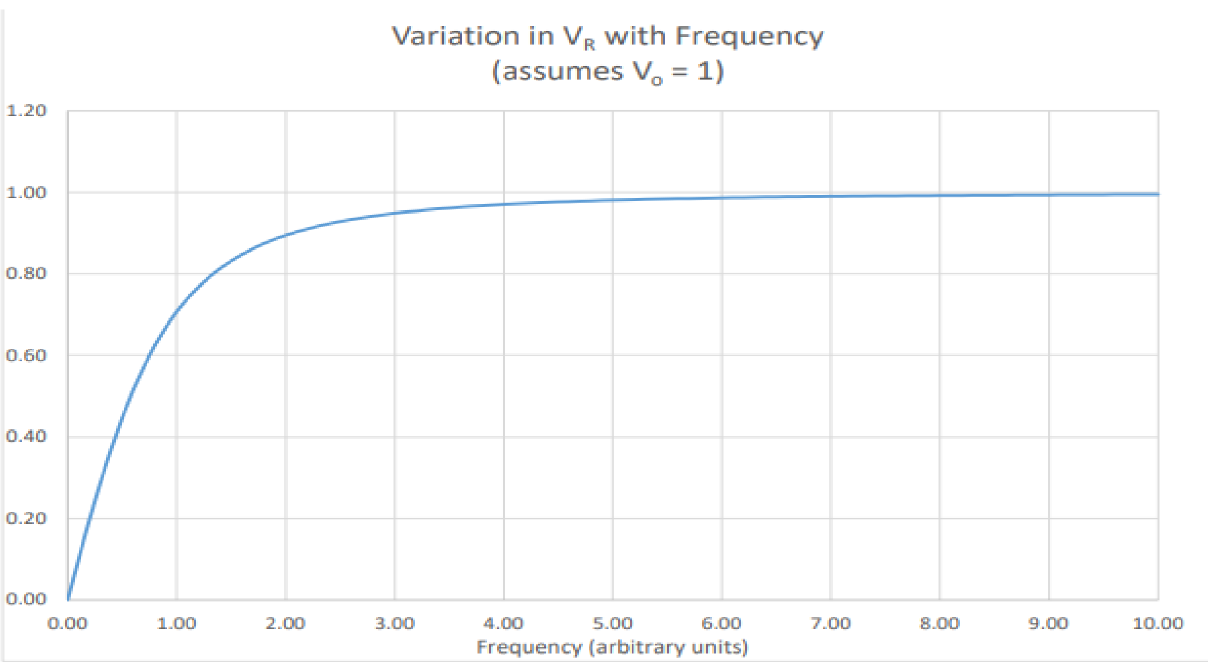



Figure 2.5 RC circuit set up as AC Sweep to demonstrate expected high pass filter

From Figure 2.5, we observe that the light blue solid line, representing $V_R$ is present. The voltage across resistor increases as frequency increases from 0 Hz to 1 MHz. It shows the expected high pass characteristic.

If we look at low pass filter now, there are a few distinct properties that would be observed.

**Low Pass Filter**

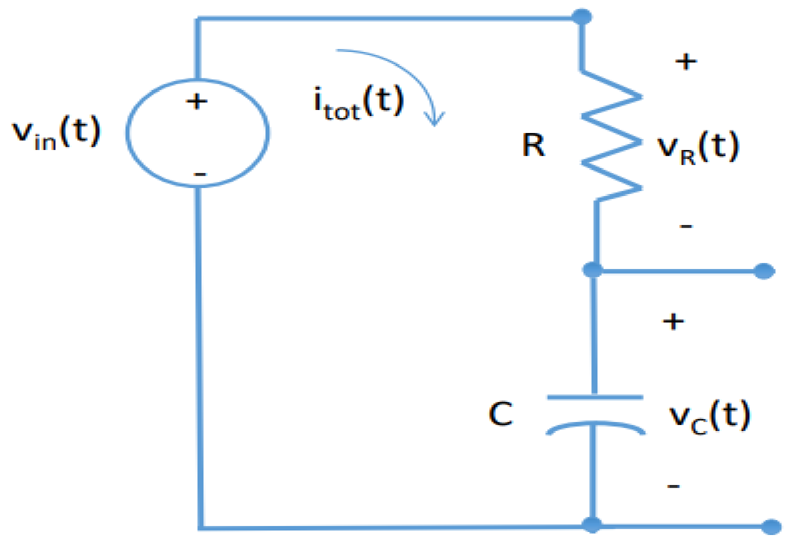



Figure 2.6 RC circuit set up to demonstrate low pass filter

From Figure 2.6, we see it is almost identical to 2.4, but the voltmeter is now across the capacitor. The equations below help us understand the low pass features.

giving: $$v_C(t) = \frac{1}{C}\int i_{tot}(t)\,dt$$

$$v_C(t) = \frac{V_o}{C\sqrt{R^2 + \frac{1}{\omega^2 C^2}}}\int \sin(\omega t + \theta_{RC})\,dt$$

$$v_C(t) = \frac{-V_o}{\sqrt{1 + \omega^2 R^2 C^2}}\cos(\omega t + \theta_{RC})$$

$$\theta_{RC} = \arctan\left(\frac{1}{\omega RC}\right)$$

This time we are observing $V_c$ so it is rearranged in that way. As ω increases, the denominator of the equation in the third line decreases. The overall value of fraction increases, and so $V_C$ (t) increases.

Now, the circuit is demonstrated in Multisim. It is the same set up as the high pass filter, except that this time, the voltmeter is to be connected across capacitor. It is referenced to a ref point between capacitor and resistor. AC sweep mode is turned on and simulation started. The graph should show up like this.

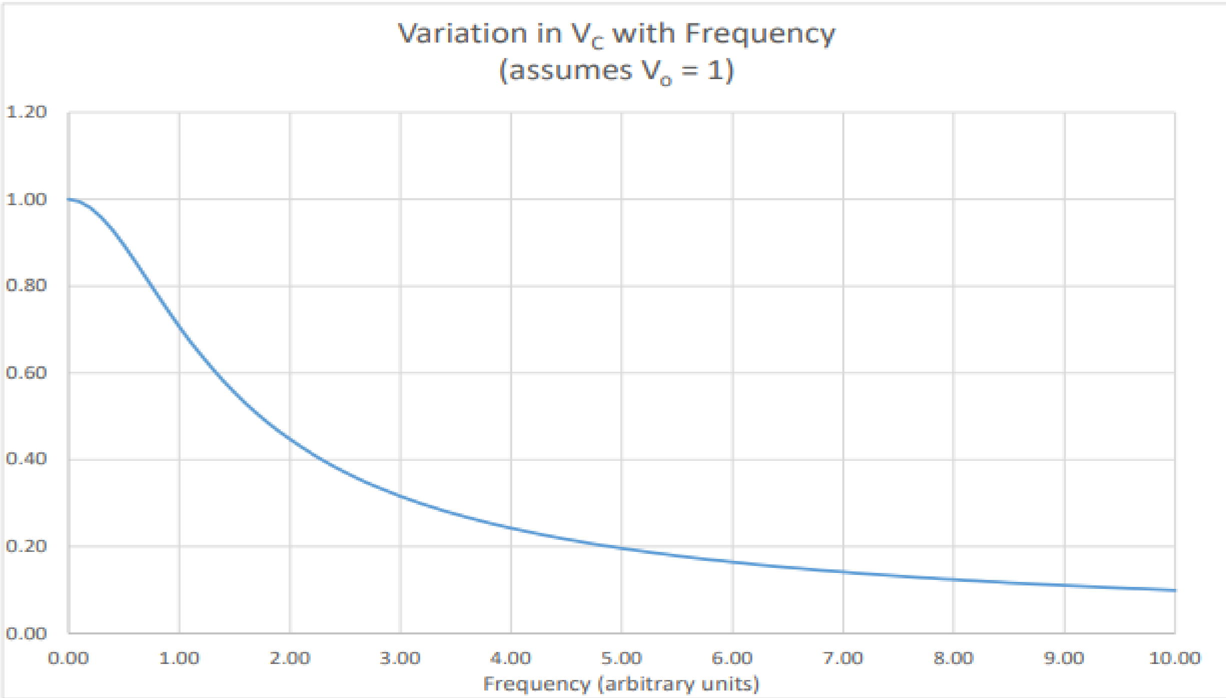



Figure 2.7 RC circuit in AC Sweep to demonstrate expected high pass filter

From Figure 2.7, we observe that an increase in frequency leads to a decrease in voltage across capacitor. The graph ultimately falls to magnitude of 0 as frequency continues to increase. However, it is very high when frequency is 0 Hz. It gives a sense of how it permits high voltage drop at low frequencies.

For the last part, we will be observing Transient Analysis of step up and step down voltage. Let us begin with the circuit diagram (circuit diagram for step and down are same).

**Transient Analysis (RC)**

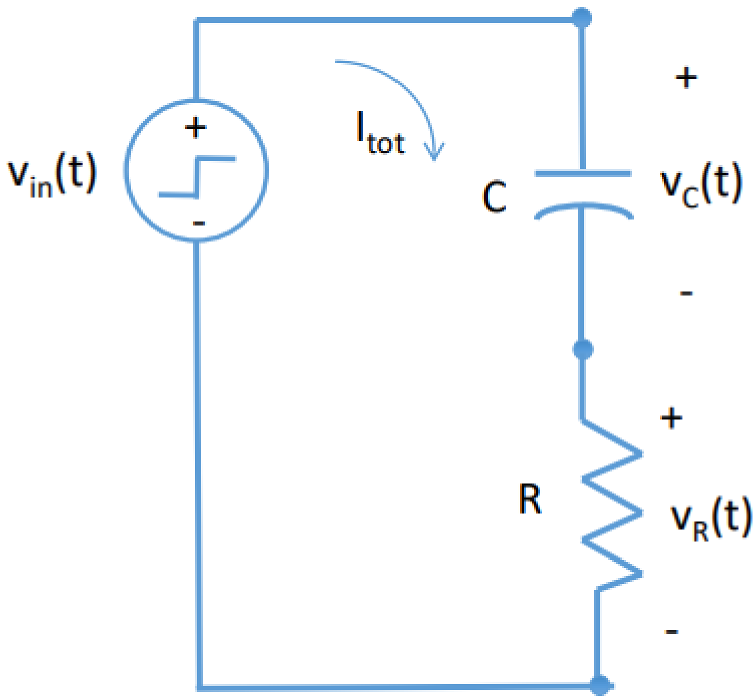



Figure 2.8 RC circuit set up to demonstrate Transient effect for step up and down voltage

From Figure 2.8, we see that a step voltage source is connected in series to a capacitor and resistor. The sudden changes for both are explained below with equations:

$$i_{tot}(t) = \frac{V_{in}}{R} e^{-\alpha t} \quad with \quad \alpha = \frac{1}{RC}$$

The equation shows us that as time increases, $i_{tot}$ (t) decreases when voltage is stepped up from 0 to 3.3 V, and the $i_{tot}$ decreases when voltage is dropped from 3.3 to 0 V. The RC represents the time taken for effects of transient analysis to wear off.

Now, let us observe the circuits in Multisim. First, let us consider step-up voltage. The step voltage source is assigned to change $V_{in}$ from 0 to 3.3 V. A capacitor of 5700 pF and resistor of 560 Ω is connected in series. Voltmeters are placed across source and the two other components, and ammeter is used to record source current. The Transient mode is turned on and the simulation is started.

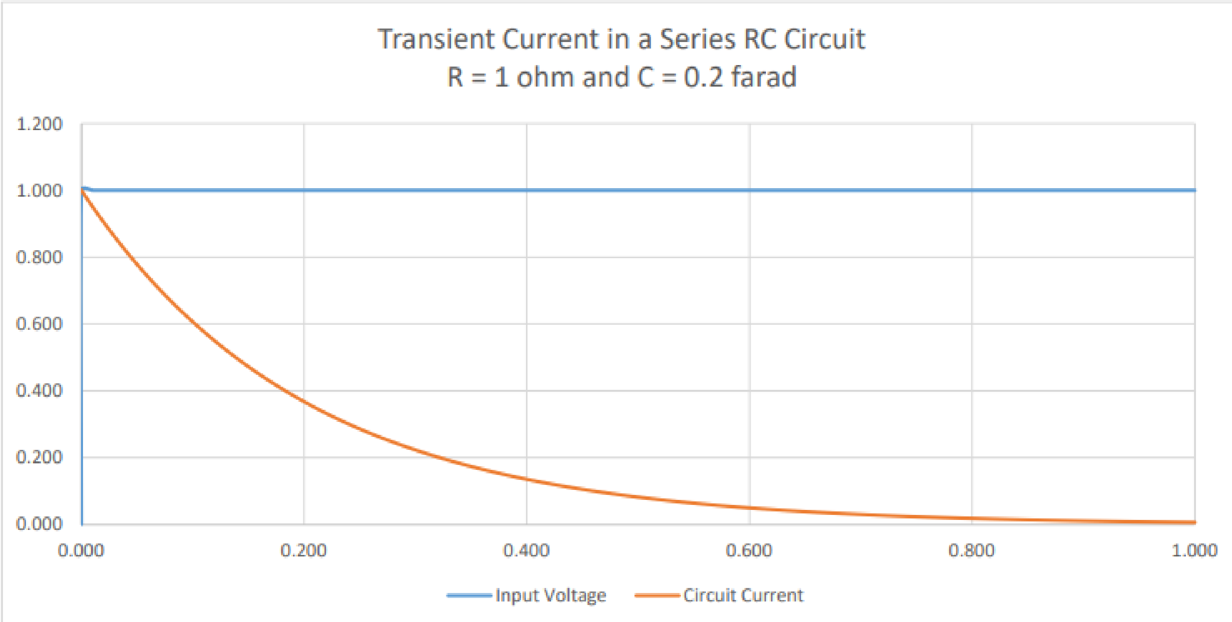



Figure 2.9 RC circuit set up to demonstrate Transient effect on Multisim for step up voltage

From Figure 2.9, we see that the source voltage is constant at 3.3 V, but current should decrease and eventually fall to 0 A. Additionally, voltage across resistor should be 0 V when switched off. It will spike to 3.3 V, and then gradually fall to 0 V. On the other hand, the voltage across capacitor will increase as time passes. This is similar to the DC Steady State, as more time is allowed, capacitor stores up charge, and current flow stops.

The step-down voltage graph will be shown in the results section and explained in that section too.

The last section of this lab report will focus on RLC circuits. We will be observing effects on current and voltage in an RLC circuit in the DC Steady State, AC Interactive, AC Sweep, and Transient Analysis. Let us first take a quick look at the circuit for the DC Steady State.

**DC Steady State (RLC)**

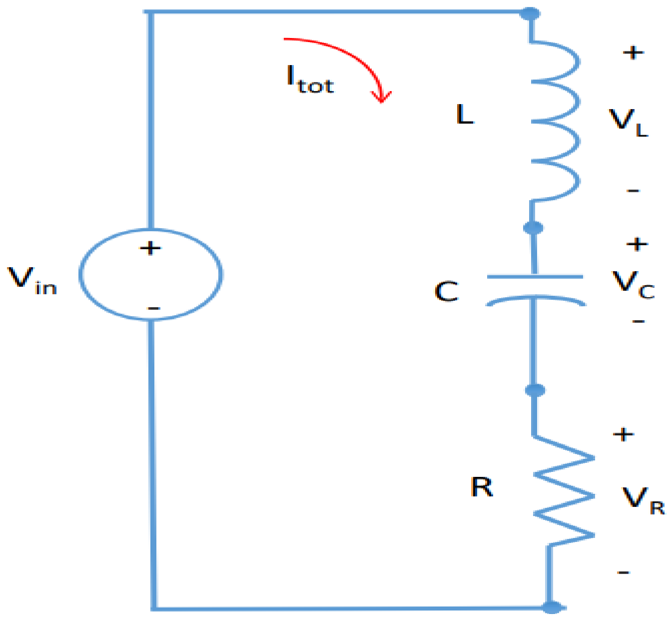



Figure 3.1 RLC circuit set up to demonstrate DC Steady State

From Figure 3.1, we see that a DC supply is connected to a capacitor, resistor, and inductor in series. The supply is kept on for a long time, and the effects on voltage and current are recorded.

The equations below will help us to understand the theoretical results that would be observed.

$$V_{in} = V_L + V_C + V_R$$

According to KVL, the equation should look like this. The component voltages add up to source voltage. However, as DC current is flowing for a long time, the inductor acts like any other connecting wire and the capacitor stores charge. As capacitor continues to store charge, current decreases and soon falls to zero. As a result, the equation appears as follows:

$$V_C = V_{in}$$

The resistor has no voltage drop across it as there is no current.

Now, let us observe the circuit in Multisim. A DC supply is connected to a resistor of 560 Ω, capacitor of capacitance 5700 pF, and inductor of inductance 220 µH in series. Voltmeters are used to record source and all component voltages, and ammeter records source current. Reference points need to be set up to measure p.d. and then DC OP mode is selected and simulation started. The circuit should look like this:

| Quantity | Magnitude |
|---|---|
| Voltage across resistor ($V_R$) / V | 0.0 |
| Voltage across capacitor ($V_C$) / V | 3.3 |
| Current across  circuit ($I_{tot}$) / A | 0.0 |
| Voltage across source ($V_{in}$) / V | 3.3 |
| Voltage across inductor ($V_L$) / V | 0.0 |

Table 3 Expected DC Steady State output for RLC circuits

From Table 3, we observe that voltage across inductor and resistor is 0 V. The current is also 0 A. However, the capacitor voltage is equal to the source voltage. This is the expected results.

Now, we can move on to AC Interactive analysis. The circuit is as shown below:

**AC Interactive (RLC)**

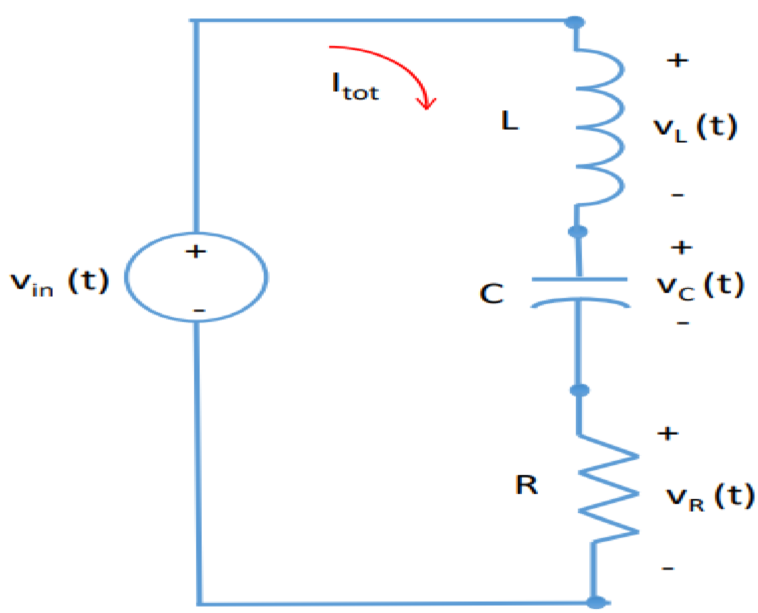



Figure 3.2 RLC circuit set up to demonstrate AC Interactive Analysis

From Figure 3.2, we see the same set up with resistor, inductor, and capacitance, but this time the source is an AC supply. Now, to understand the concept of what happens in this analysis, we have to first look at the equations, and know what the resonance frequency is.

$$v_{in}(t) = L\frac{di_{tot}(t)}{dt} + \frac{1}{C}\int i_{tot}(t)\, dt + \mathrm{R}\, i_{tot}(t)$$

If we let

$$v_{in}(t) = V_o \sin(\omega t)$$

Then the solution is:

$$i_{tot}(t) = \frac{V_o}{\sqrt{R^2 + \left(\frac{1}{\omega C} - \omega L\right)^2}} \sin(\omega t + \theta_{RLC})$$

$$with \quad \theta_{RLC} = \operatorname{atan}\left[\frac{\left(\frac{1}{\omega C} - \omega L\right)}{R}\right]$$

Where,
$I_{tot}$ (t) is source current
$V_{in}$ (t) is source voltage
ω is 2πf
C is capacitance
R is resistance
L is inductance

The first equation is made following KVL. For the resistor, the component voltage is shown as the product of resistance and source current. For capacitor, it is the product of the integral of $i_{tot}$ (t) and inverse of capacitance. For inductor, it is the product of rate of change of current and inductance. When it is rearranged, it gives the second to last equation. Now, we must determine the resonance frequency. The procedure is as below:

$$\left(\frac{1}{\omega C} - \omega L\right) = 0$$

Then we get

$$\theta_{RLC} = \operatorname{atan}[0] = 0\, deg$$

and

$$i_{tot}(t) = \frac{V_o}{R}\sin(\omega t)$$

The effect of the inductor and capacitor cancel each other. This is called a resonance and occurs at the frequency

$$\omega = \frac{1}{\sqrt{LC}} \quad or\ since\ \omega = 2\pi f\ we\ get \quad f = \frac{1}{2\pi\sqrt{LC}}$$

Now, considering this equation and taking our problem in hand, we have a capacitor of 5700 pF, and inductor of inductance 220 µH. Therefore, our resonance frequency is 142 kHz (3 sf).

The interesting thing here is for frequencies below resonance, current is leading. However, for frequencies above resonance frequency, current is lagging.

The equations have both $\omega L$ and $\omega C$. Depending on which is greater, the $\theta$ is either positive or negative. As a result, the phase  is either negative or positive. The phase is positive for frequencies below resonance and negative for frequencies above resonance.

Now, we can move on to Multisim. The circuit is set up with the previously mentioned data for the components. It is connected to AC supply in series. Now, the frequency for AC needs to be set below resonance frequency at first. I have decided to take 30 kHz in this case. Voltmeters are set across source and resistor. The ammeter records source current. The AC interactive mode is switched on, and graph should show up as in Figure 2.3.

From Figure 2.3, we saw that current was leading at low frequency. The voltage across resistor was also proportional to the current, and varied in phase with it. The results should appear just as in Figure 2.3. The analysis is better demonstrated in the results section.

Let us see a case of frequencies above 142 kHz. We just have to change the AC supply frequency to above resonance frequency on Multisim. I decided on a frequency of 600 kHz. The results are produced as soon as AC Interactive mode is switched on. Figure 1.3 in RL circuits demonstrates the expected graph.

From Figure 1.3, we can see the current is lagging. The voltage across resistor is in phase with the current, but out of phase with source voltage. The maximum amplitude of source current appears after the maximum amplitude of source voltage. The details of the actual graph will be provided in the results section.

We can now move on to AC Sweep analysis for RLC circuits.

**Band Pass Filter**

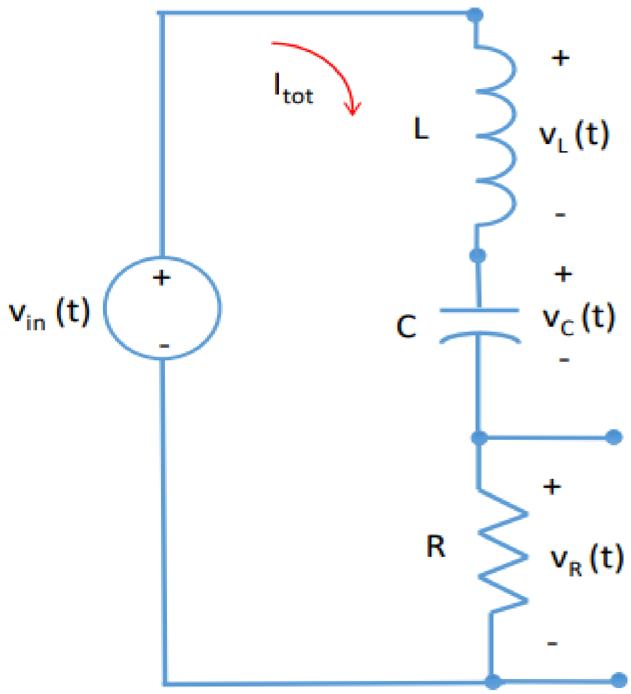



Figure 3.3 RLC circuit set up to show band pass filter

From Figure 3.3, we observe an AC supply of 3.3 V connected to the three components in series. A voltmeter is placed across resistor to observe the band pass filter. The equations for this portion of the analysis are beyond the scope of this course. Therefore, we directly go into Multisim analysis.

Now, let us observe the circuit in Multisim. The AC supply is provided 3.3 V. R = 560 Ω, L = 220 µH, and C = 5700 pF are to be used for the three components. Voltmeter is set across resistor, and AC Sweep mode is turned on.

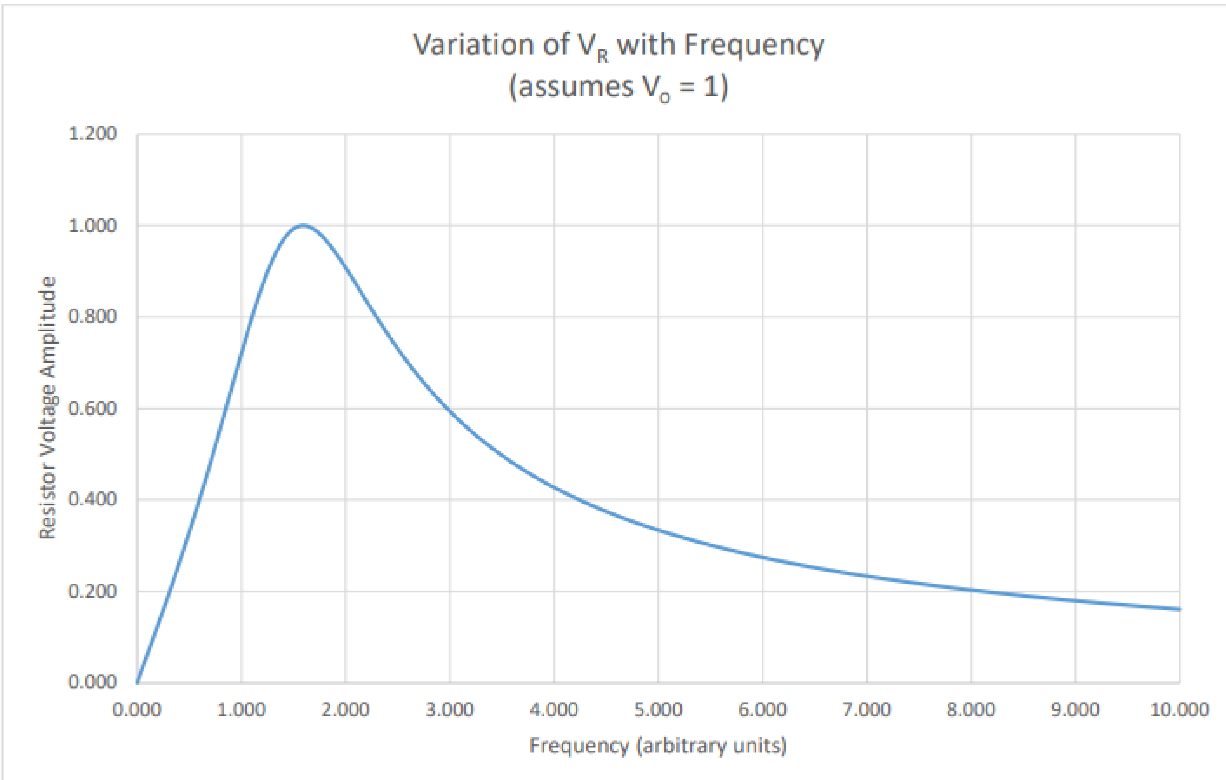



Figure 3.4 RLC circuit set up to show band pass filter on Multisim

Figure 3.4 are not the actual results, rather expected results (Just like all the other graphs provided till now). From Figure 3.4, we can observe that with increase in frequency, voltage across resistor increases, reaches a maximum value at resonance frequency, and then decreases as frequency increases above resonance frequency. The Multisim analysis should be similar to this.

The final part of the report concerns the RLC circuit in the Transient Analysis. Only the step-up voltage will be observed, but at different capacitance values of capacitor. Let us move on to the circuit diagram.

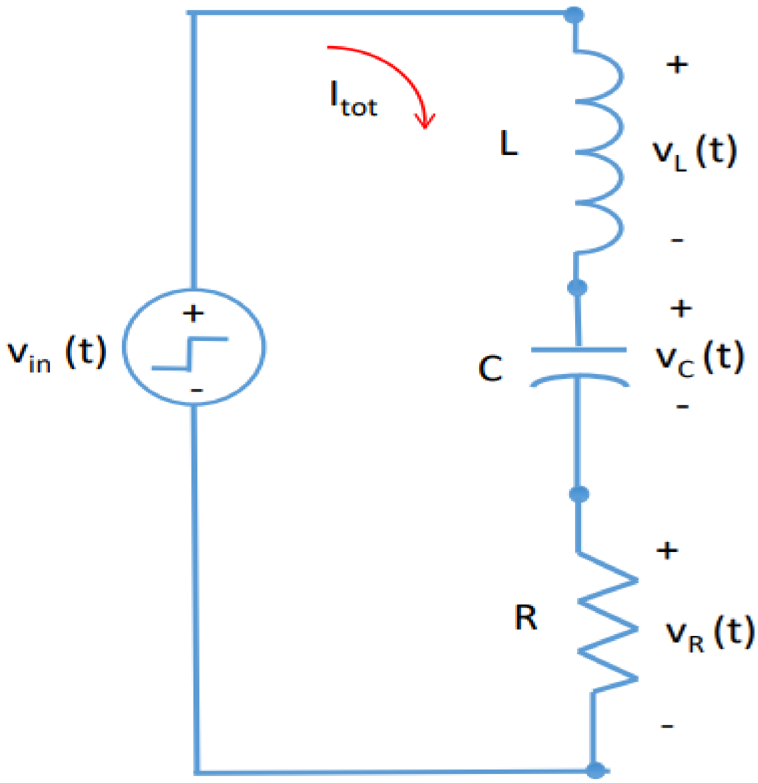



Figure 3.5 RLC circuit set up for step-up voltage

From Figure 3.5, we observe what happens when voltage is stepped up from 0 to 3.3 V across the three components. The equations are beyond the scope of this course, and so we will directly proceed to Transient Analysis on Multisim. The experiment will be conducted with three different capacitors.

The first capacitor value is set to C = 5700 pF. The resistor has value of 560 Ω, and inductor of 220 µH. The step up voltage will increase from 0 V to 3.3 V.

We can expect to observe the changes in voltage across the three different components when voltage steps up. It will also show changes in source voltage and source current. Most importantly, it should show that the capacitor voltage will soon reach source voltage.

In the next experiment, the circuit is set up as is. We just change the capacitance to C = 570 pF. The Multisim Transient Analysis is below.

We can expect to see the changes in voltage across source, resistor, capacitor, and resistor. It should show us the changes to source current. This should show that the capacitor voltage will soon reach source voltage. It also should show that voltage across other components soon falls to zero, including source current.

In the next experiment, the circuit is set up as is. We just change the capacitance to C = 57 pF. The Multisim Transient Analysis is below.

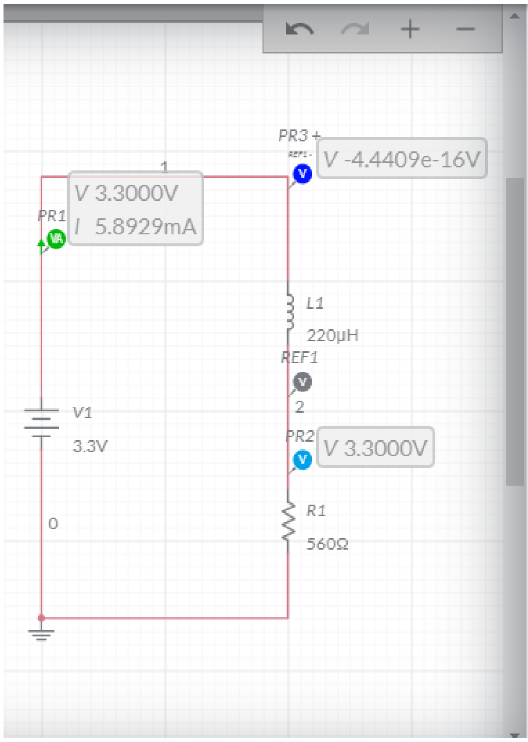
We can see the changes in voltage across source, resistor, capacitor, and inductor. It should show us the changes in source current.. Most importantly, it should show that the capacitor voltage will soon reach source voltage. It should also show that voltage across other components soon falls to zero, including source current. An interesting thing to note would be that the lower the capacitance, the greater the fluctuations in component voltages.

The graphs have not been presented here. The analysis in the results section will display the actual graphs, and explain the real situation for the last three experiments.

This was the final experiment for the report, and all results were recorded and stored.

## Results

In the results section, we will look at each graph from each experiment described in the procedures and results section. We will observe all the RL circuits on Multisim first. Let us begin with the RL circuit in DC Steady State.




DC OP 1

| Signal | Value |
|---|---|
| PR1: V(1) | 3.3000V |
| PR2: V(2) | 3.3000V |
| PR1: -I(V1) | 5.8929mA |
| PR3: V(1)-V(2) | -4.4409e-16V |

Figure 4.1 A series RL circuit set up on Multisim and its corresponding results

As it can be seen, the theory is supported in real time experiments. The voltage across resistor is 3.3 V, while voltage across inductor is so small that it is negligible. Source voltage remains constant at 3.3 V.

We will observe the AC Interactive analysis on Multisim next. As I stated before, it has an AC supply, and demonstrates lagging current. Here is the circuit and corresponding graph:

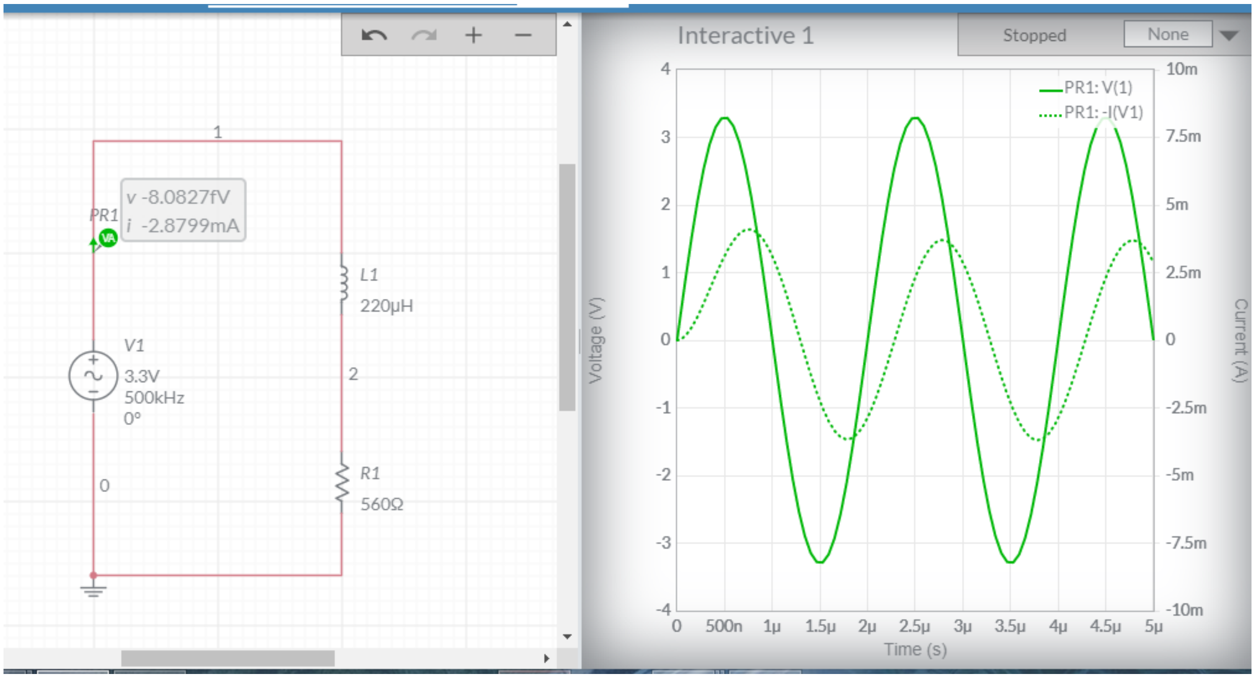



Figure 4.2 An RL circuit set up with AC supply in Multisim and the results in graphs

In Figure 4.2, we see the circuit is set up and simulation started. The corresponding graph is displayed. The current is the green dotted line, and voltage is the green solid line. The graph representing current is out of sync and demonstrates a lag and also decline in value as time passes. The sinusoidal appearance is due to the AC supply.

Next, we can look at the actual Multisim analysis of low pass filter in RL circuits.

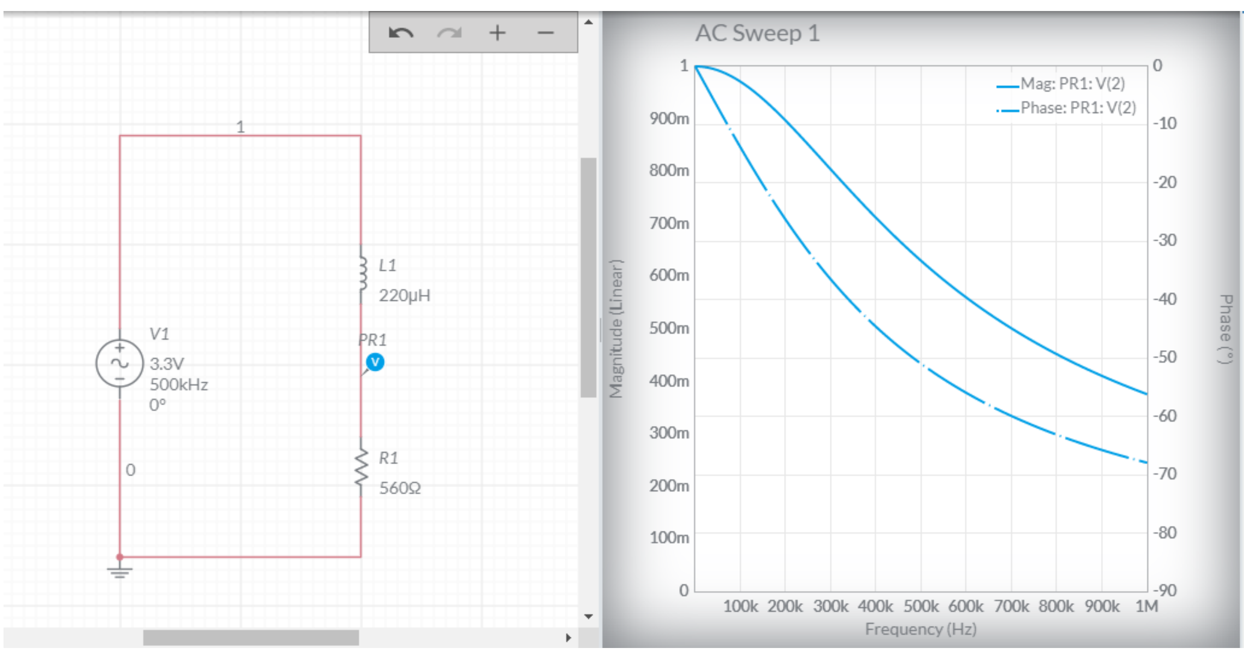



Figure 4.3 RL circuit in Multisim showing low pass filter

From Figure 4.3, we observe that as frequency increases from 0 Hz to 1 MHz, the magnitude of voltage decreases. This shows that at low frequency, resistor has a higher voltage drop than at higher frequencies. This happens as inductor allows high voltage drop across resistor in low frequencies, but it opposes high frequencies as a higher voltage is induced across inductor. The voltage across inductor increases with increase in rate of change in current.

RL circuits are also capable of demonstrating high pass filter characteristics. The actual experimental results on Multisim are below:

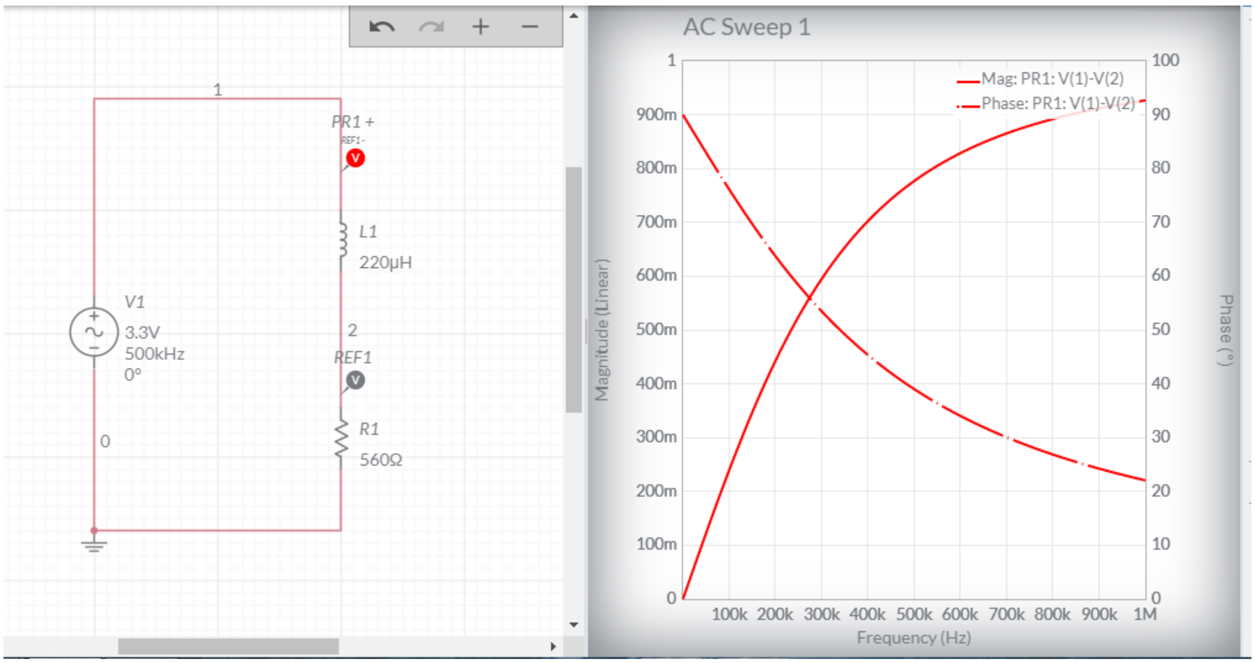



Figure 4.4 RL circuit with high pass filter characteristic on Multisim

In Figure 4.4, voltage across inductor is 0 V when frequency is 0 Hz, but as frequency increases the voltage across it increases. According to Faraday's Law, the rate of change of current is proportional to the voltage induced across the inductor. Therefore, as frequency increases, the rate of change of current increases, and the voltage across inductor increases.

RL circuits also demonstrate notable changes when voltage is stepped up or down in Transient Analysis. The step-up case is observed first.

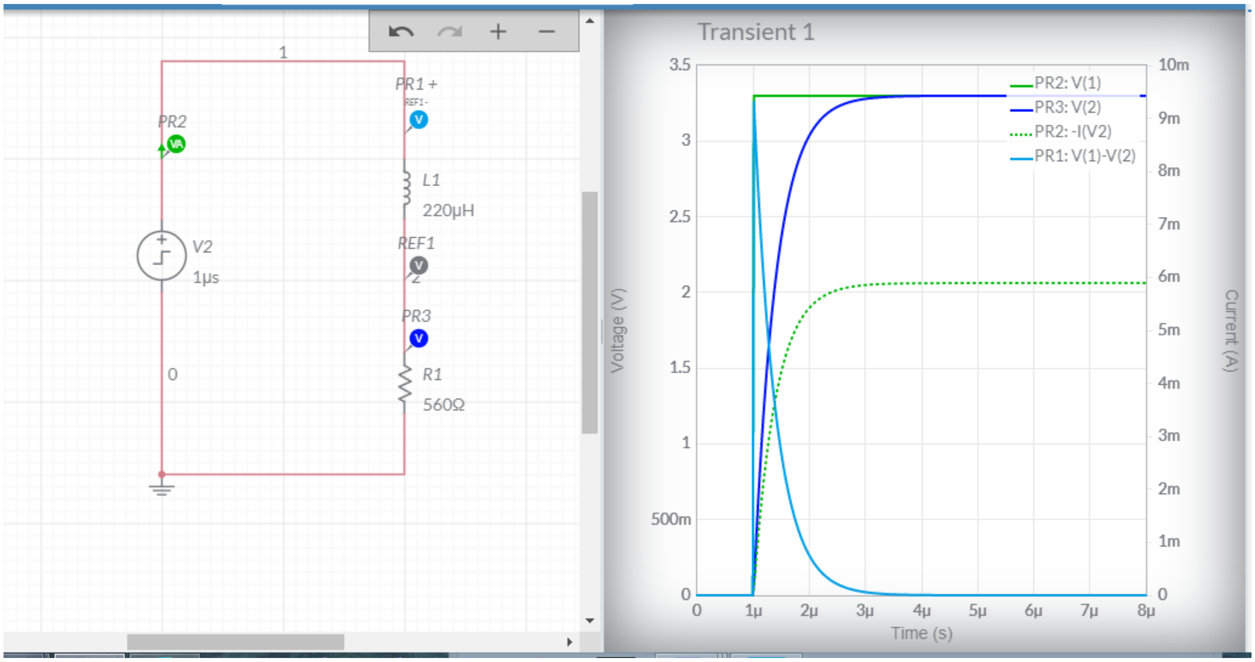



Figure 4.5 Transient effects when voltage is stepped up

From Figure 4.5, we see that the graph shows a sudden increase in source voltage to 3.3 V at 1 µs. The voltage across inductor climbs to 3.3 V instantly, and then slowly continues to fall to zero. The voltage across resistor remains zero, but slowly increases as current is supplied for longer time. The source voltage remains constant at 3.3 V. The current increases and then becomes constant. By 4 µs, component voltages and source current become constant.

The step-down case is observed now.

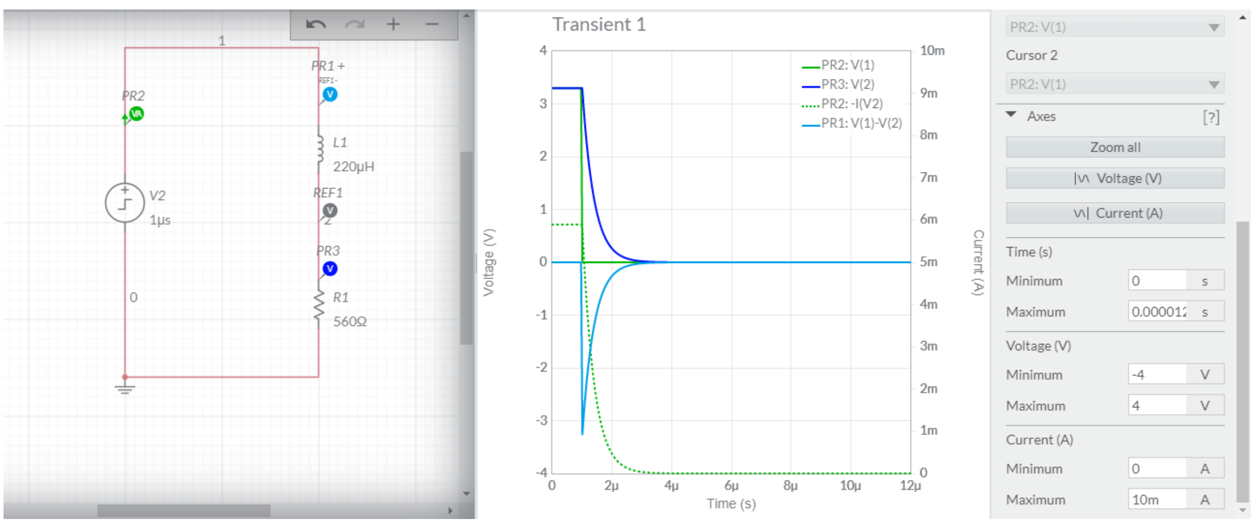



Figure 4.6 Transient effects when step down voltage is applied

From Figure 4.6, we see that source voltage and $V_R$ are equal and constant. As soon as voltage source is switched off, $V_R$ decreases to zero, and inductor voltage spikes and then drops to zero too. The current and source voltage also drop to zero after some time.

The next section concerns the actual results from all the experiments with RC circuits. Let us start with RC circuit in DC Steady State on Multisim.

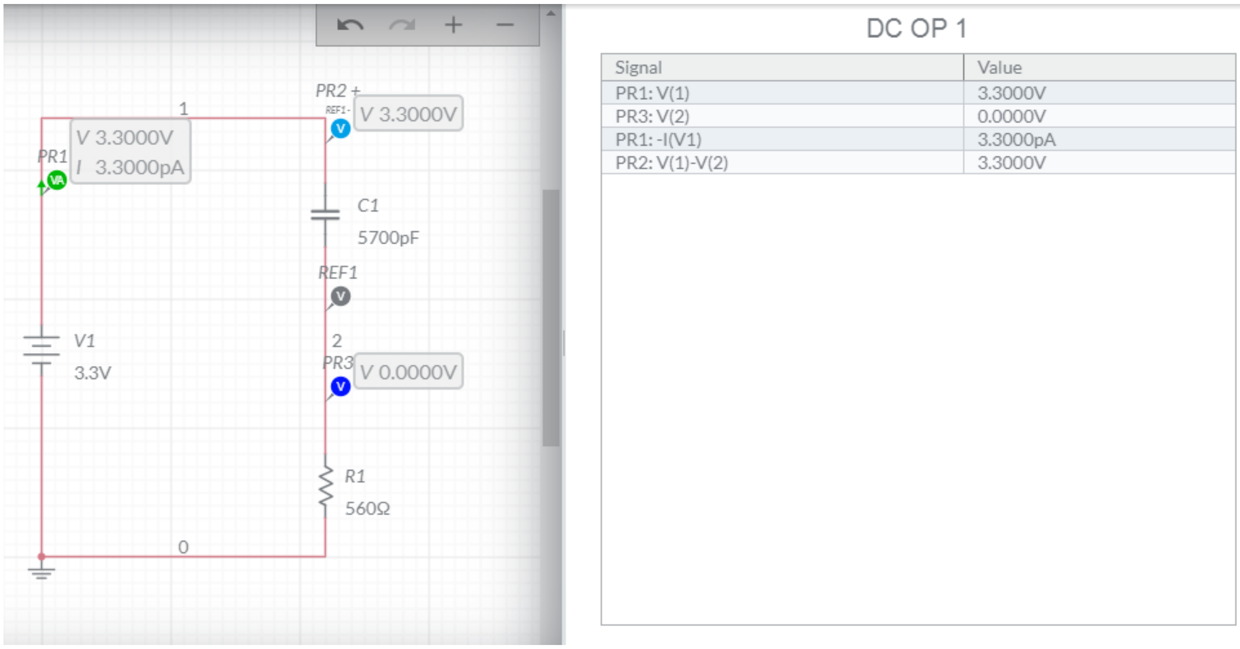



Figure 5.1 Circuit design and experiment results in Multisim for DC Steady State

From Figure 5.1, we see the circuit set up, and the corresponding table of results. It supports the theory explained before that DC current applied for a long time will result in source voltage and voltage drop across capacitor to be equal. The results show $V_C$ to be 3.3 V and source voltage to be 3.3 V too. $V_R$ is 0 V and circuit current is also 0 A.

We can now move on to graphs relating to AC supply. We will start with the RC circuit in AC Interactive mode on Multisim.

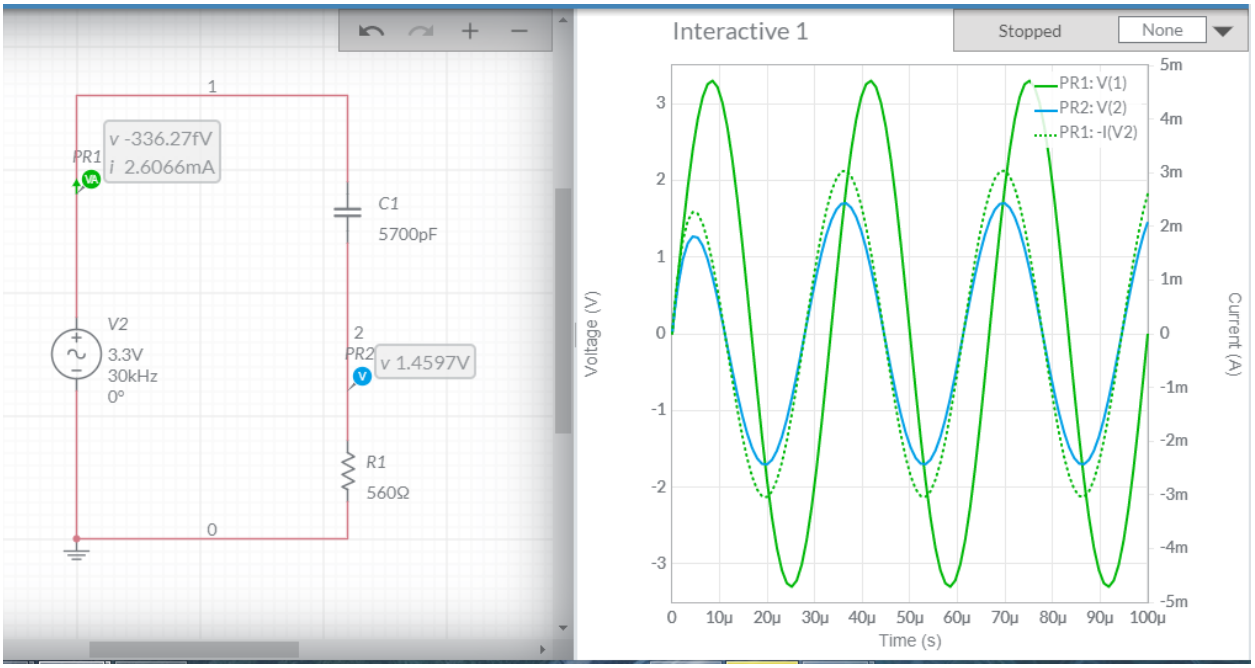



Figure 5.2 RC circuit in Multisim set up as AC Interactive to demonstrate leading current

From Figure 5.2, we observe the leading current. The green dotted line reaches its maximum amplitude before the source voltage can reach its maximum amplitude. This means current is leading. Moreover, the voltage across resistor is also proportional to that current as it was explained in expected results to be.

The next RC circuit incorporating AC supply will be observed in AC Sweep mode on Multisim to demonstrate high and low pass filter. Let us begin with the high pass filter first.

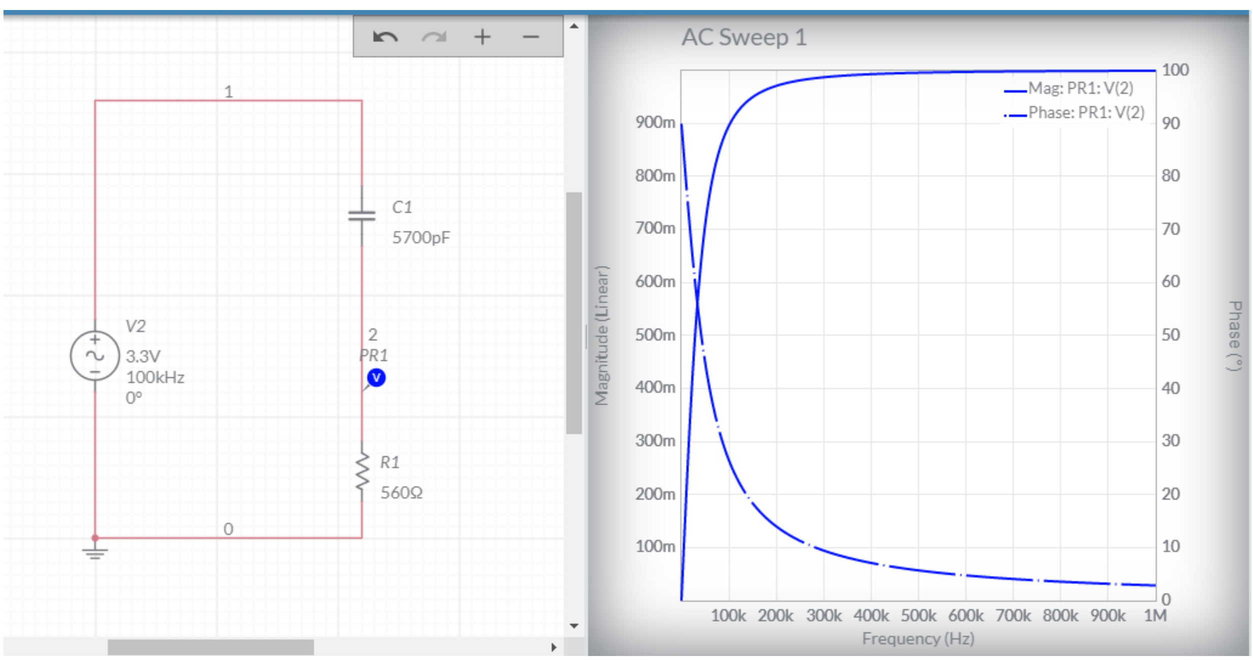



Figure 5.3 RC circuit set up as AC Sweep in Multisim to demonstrate high pass filter

From Figure 5.3, we observe that the blue solid line, representing $V_R$ is present. The voltage across resistor increases as frequency increases from 0 Hz to 1 MHz. It shows the high pass characteristic. The capacitor fails to store charge effectively when current keeps switching from positive to negative to positive again, and so higher the rate of change of current, the lower the voltage across capacitor and higher across resistor.

The low-pass filter can also be observed if we simply add a voltmeter across the capacitor.

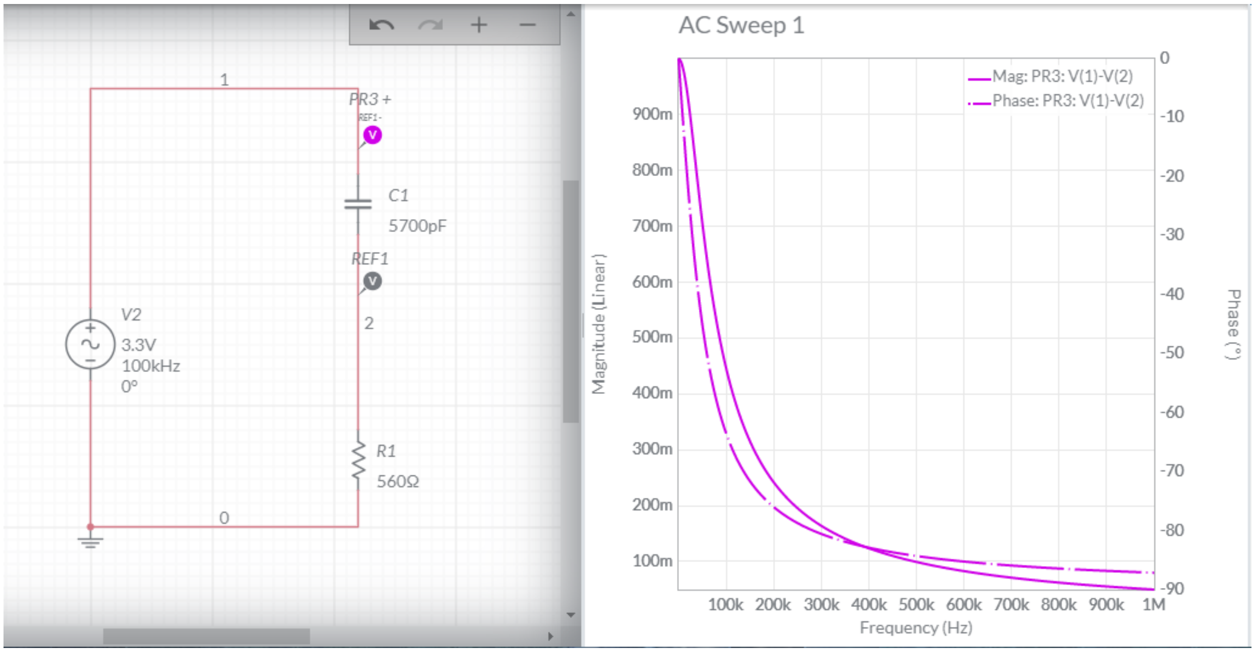



Figure 5.4 RC circuit set up as AC Sweep in Multisim to demonstrate low pass filter

From Figure 5.4, we observe that an increase in frequency leads to a decrease in voltage across capacitor. The graph ultimately falls to magnitude of 0 as frequency continues to increase. However, it is very high when frequency is 0 Hz. It gives a sense of how it permits high voltage drop at low frequencies.

We can now move on to step-up and down voltage to observe the Transient Analysis on Multisim. We can first observe the step-up voltage.

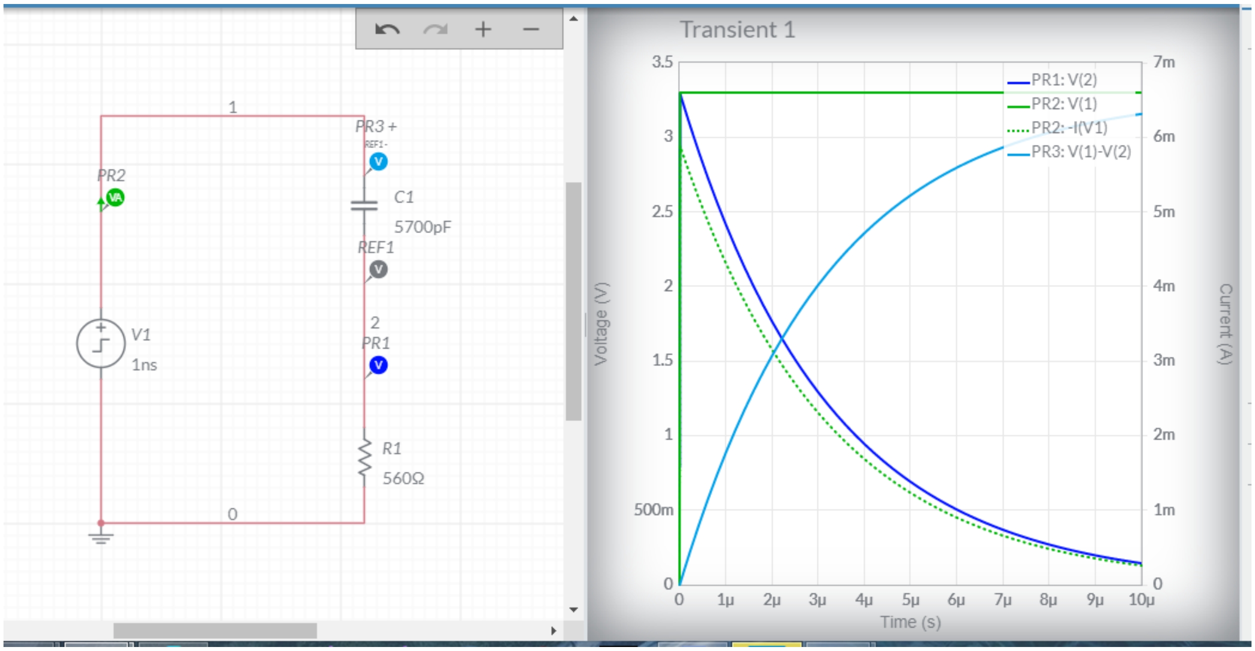



Figure 5.5 RC circuit set up to demonstrate Transient effect on Multisim for step up voltage

From Figure 5.5, we observe the effects when voltage is suddenly stepped-up. Voltage across resistor decreases from 3.3 V As time passes beyond 10 µs, it will eventually drop to 0 V. However, the voltage at start is 0 V across capacitor, but it increases to 3.3 V as time passes by. Circuit current also decreases and will fall to zero.

The step-down impact is discussed next.

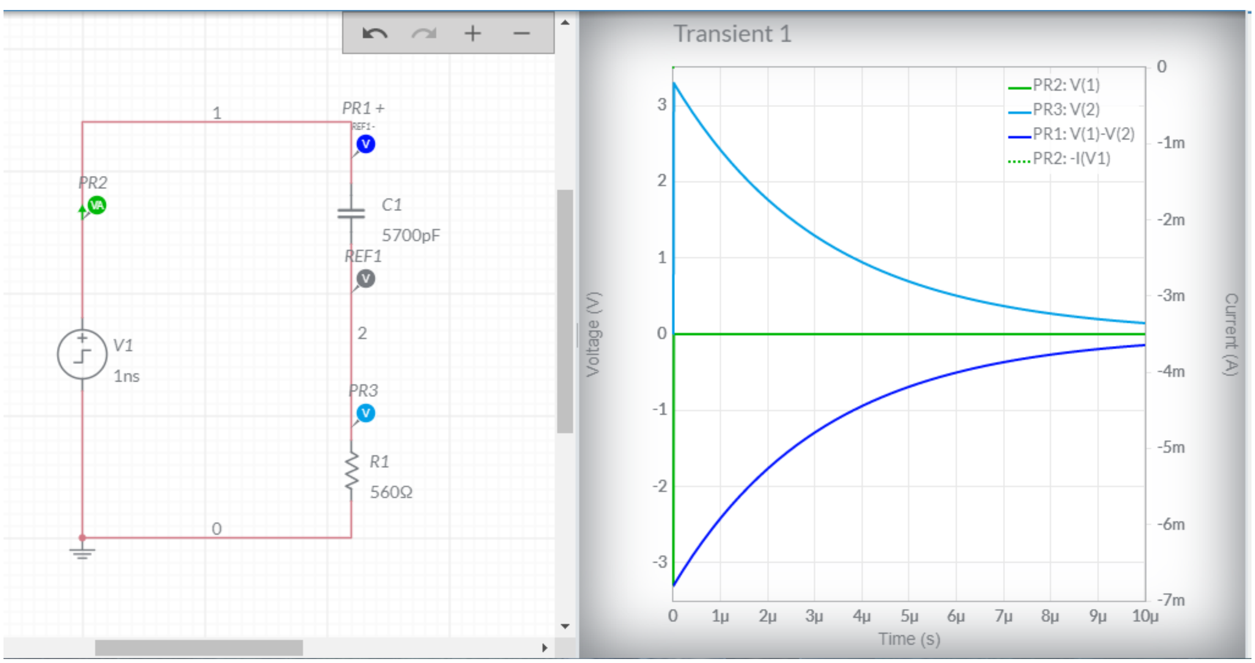



Figure 5.6 RC circuit set up to demonstrate Transient effect on Multisim for step down voltage

From Figure 5.6, it is seen that circuit voltage and circuit current drops to zero almost instantaneously, but capacitor and resistor voltage takes time to fall to zero. As explained in the expected results section, this happens as capacitor needs time to discharge its stored charge. The theory is supported in the actual experiments.

The final part of the results section talks about RLC circuits. We can start by observing the DC Steady State.

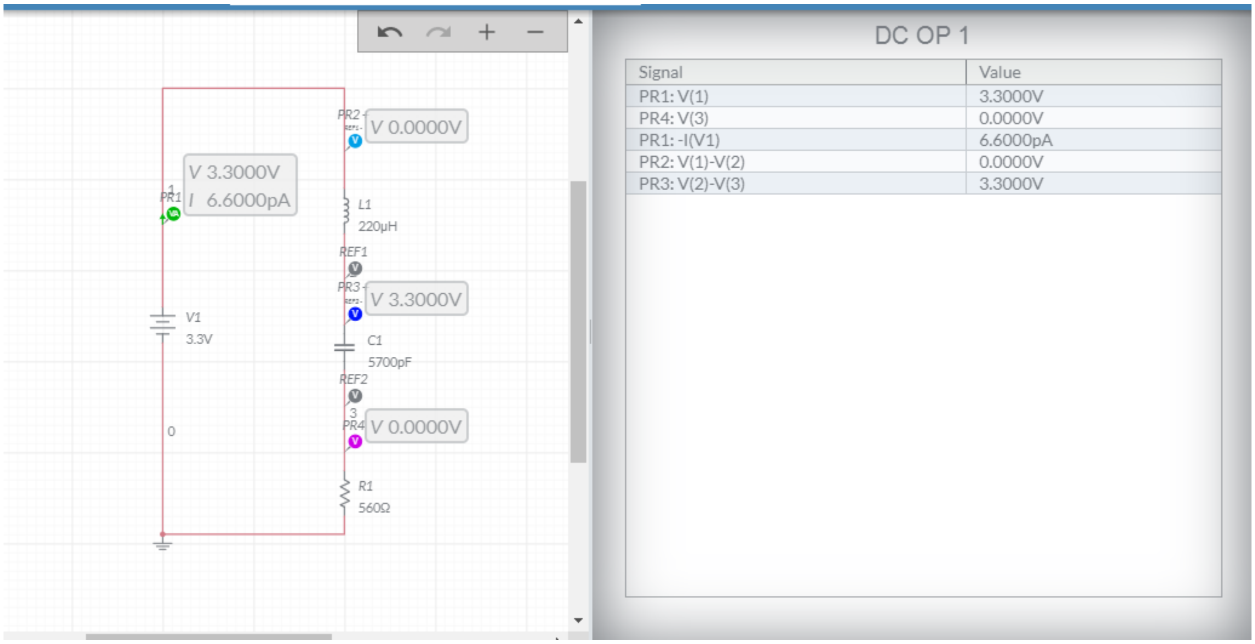



Figure 6.1 RLC circuit set up to demonstrated Steady State on Multisim

From Figure 6.1, the circuit is observed in DC Steady State. The results match the criteria set up in Table 3 in Expected Result section. The capacitor has 3.3 V across it while the other components have 0 V across them. The current is so small that it is considered to be 0 A. The source voltage remains at 3.3 V.

The next experiment that will be discussed will be in the AC Interactive mode displaying lagging current for frequencies below resonance frequency. We had calculated in the procedure section that our resonance frequency is 142 kHz. Now, we observe 30 kHz frequency which is below 142 kHz.

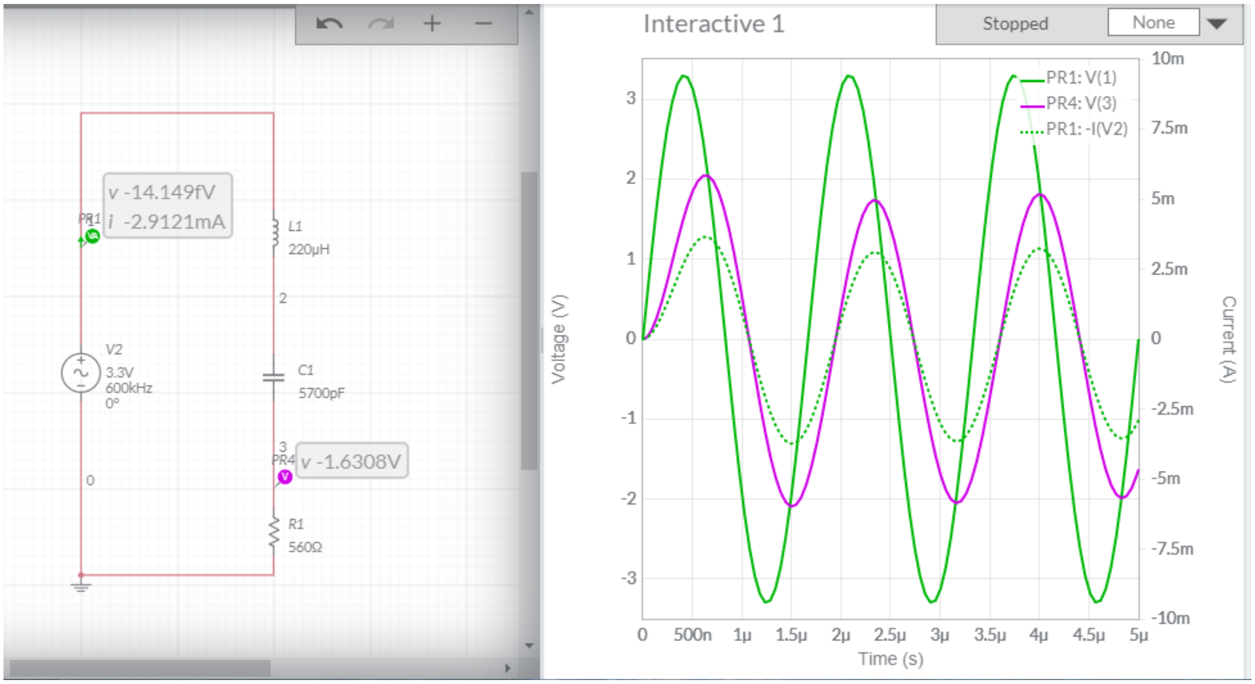



Figure 6.2 RLC circuit set up to demonstrate lagging current on Multisim for freq below resonance

From Figure 6.2, we observe the lagging current. The green dotted line appears to be behind the green solid line representing source voltage. The resistor voltage is about 1.6 V which is half of 3.3 V supply voltage. The resistor voltage is also in sync with current.

We observe frequencies above 142 kHz now. It is taken to be 600 kHz.

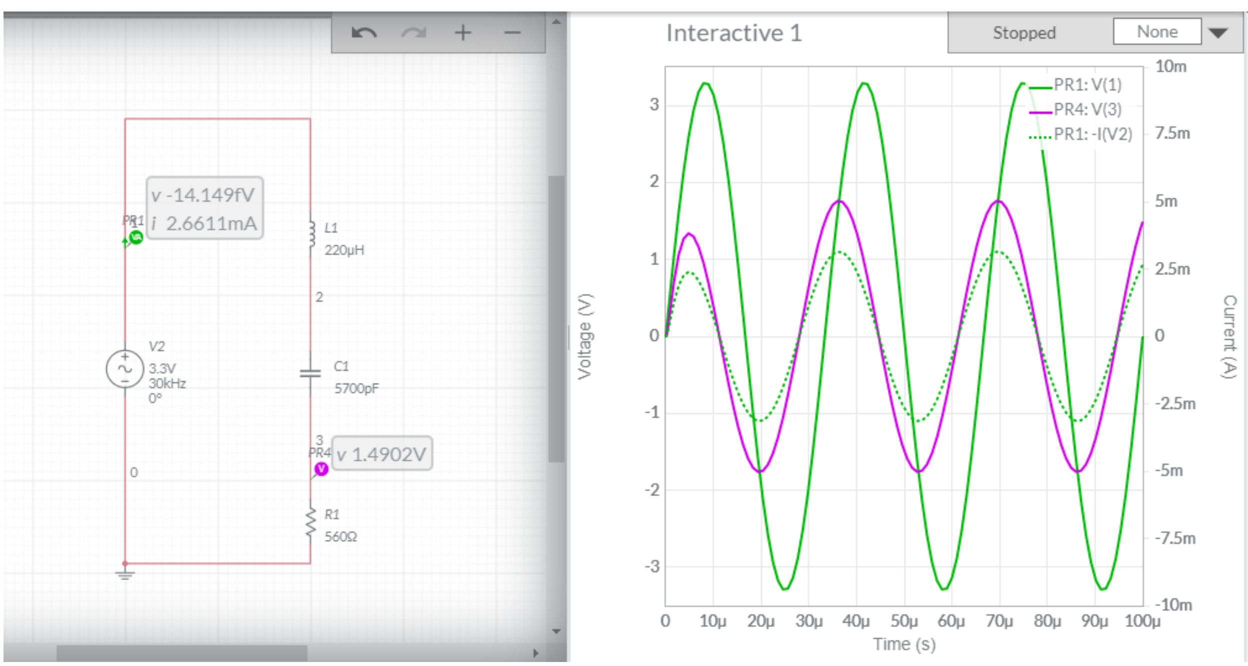



Figure 6.3 Circuit set up to demonstrate leading current on Multisim for freq above resonance

From Figure 6.3, we observe the leading current. The green dotted line appears to be before the green solid line representing source voltage. The resistor voltage is about 1.6 V which is half of 3.3 V supply voltage. The resistor voltage is also in sync with current.

For the next part, we observe the band pass filter in AC Sweep mode.

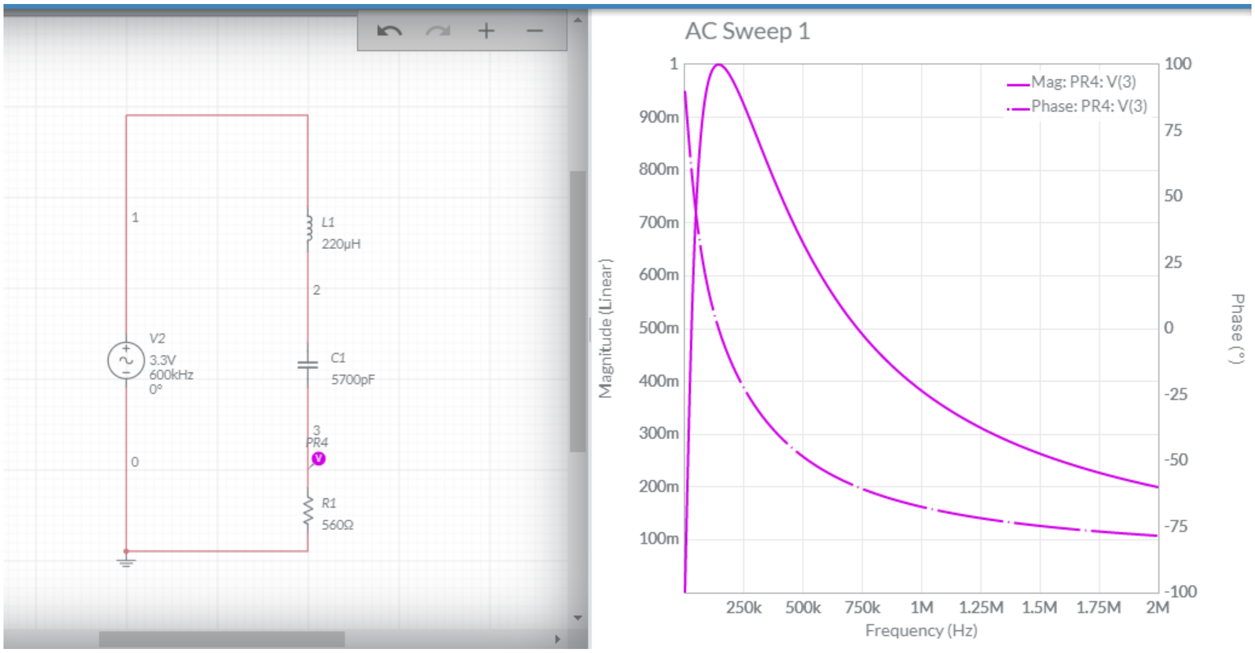



Figure 6.4 RLC circuit set up to demonstrate band pass filter on Multisim

From Figure 6.4, we observe peak voltage, or source voltage to be equal to resistor voltage at the resonance frequency of 142 kHz. It falls to below a ratio of 1 for frequencies above or below resonance. The phase decreases with increase in frequency.

The last part of the results section deals with RLC circuits in Transient Analysis for different values of capacitance. The first experiment is for C = 5700 pF.

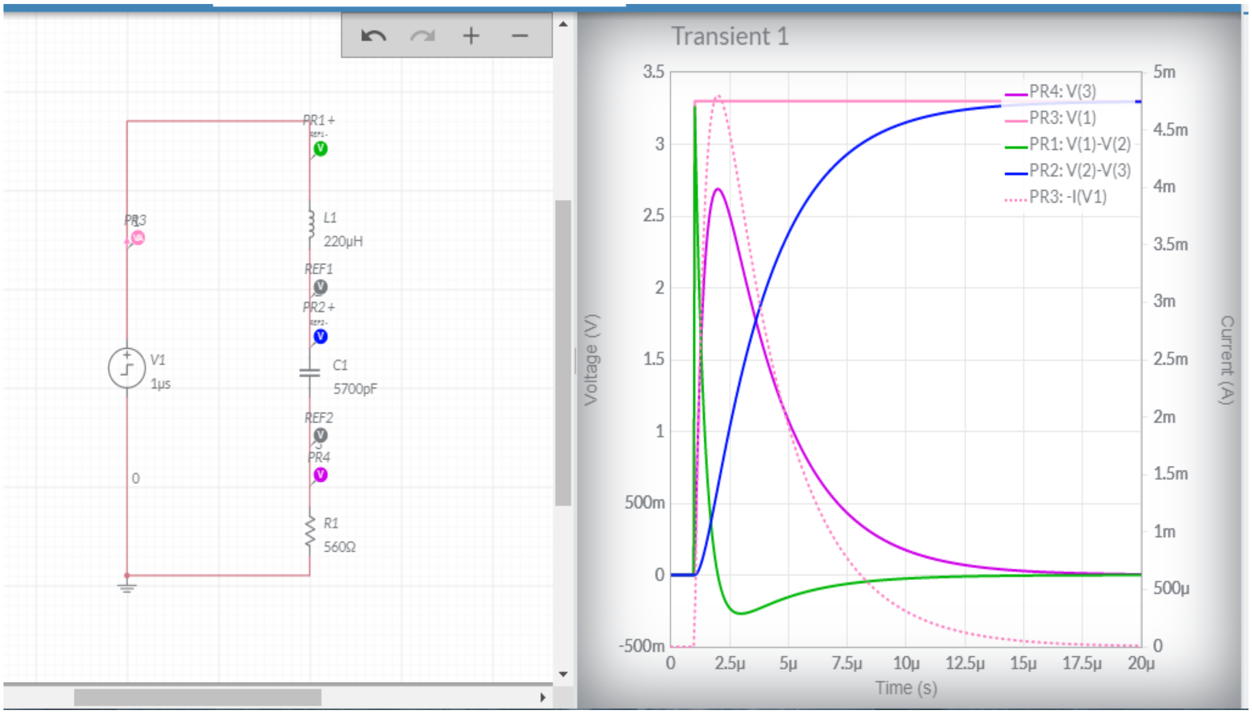



Figure 6.5 RLC circuit set up to demonstrate Transient effect on Multisim for step up voltage for C = 5700 pF

From Figure 6.5, we observe sudden changes in component voltages and current. However, after the fluctuations, we observe inductor and resistor voltage to drop to 0 V. Only the capacitor voltage shows a steady increase from 0 to 20 µs until it reaches 3.3 V (source voltage). This happens as the source voltage is DC, and DC current for long time will eventually result in capacitor storing charge, and having voltage drop to be 3.3 V.

The next circuit we observe will be for the capacitance value of C = 570 pF.

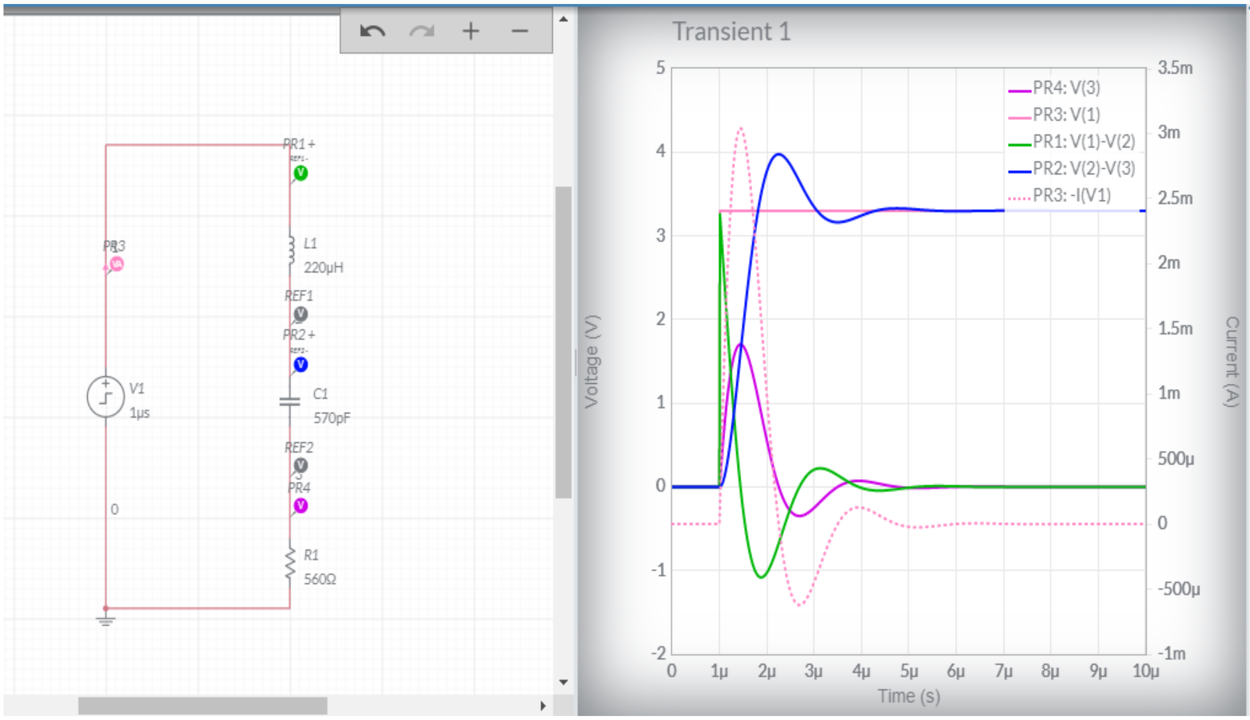



Figure 6.6 RLC circuit set up to demonstrate Transient effect on Multisim for step up voltage for C = 570 pF

From Figure 6.6, we observe sudden changes in component voltages and current. However, there are more fluctuations than Figure 6.5. We still observe inductor and resistor voltage to drop to 0 V. The capacitor voltage shows an increase in voltage from 0 to approximately 4.0 V, a slight decrease, and then finally constant at 3.3 V between t = 0 to 5 µs. The results are as expected.

The next circuit we observe will be for the capacitance value of C = 57 pF.

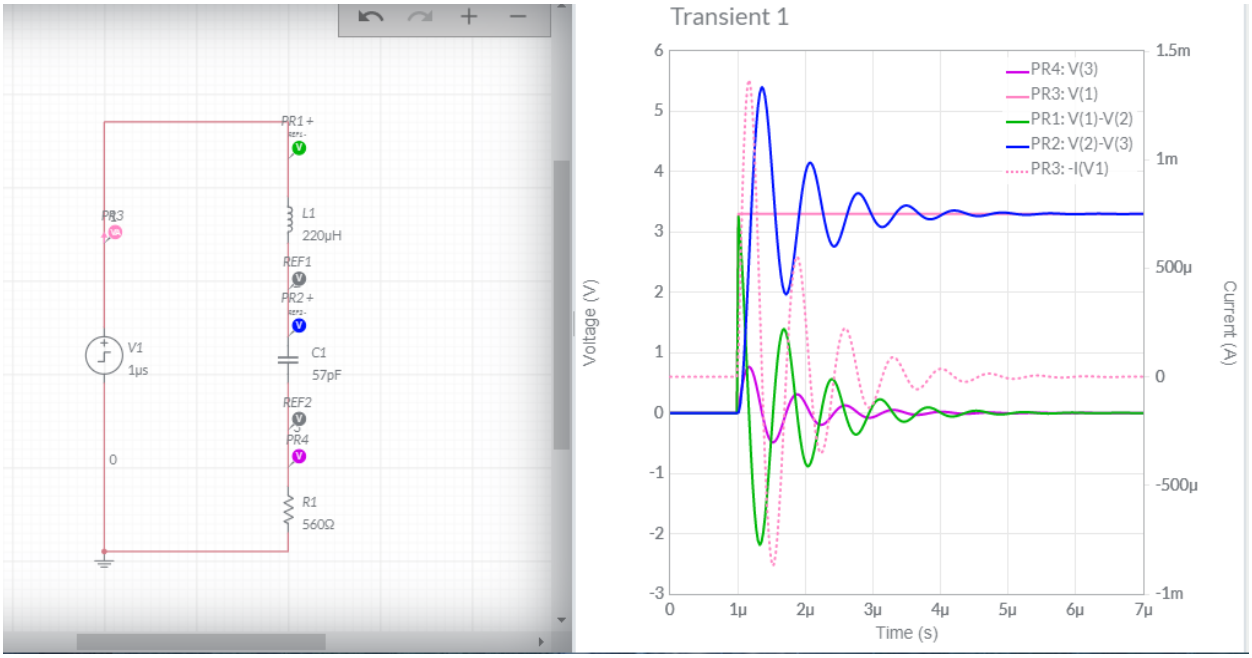



Figure 6.7 RLC circuit set up to demonstrate Transient effect on Multisim for step up voltage for C = 57 pF

From Figure 6.7, we observe sudden changes in component voltages and current. However, the fluctuations are greatest here. We observe inductor and resistor voltage to drop to 0 V, and capacitor voltage to finally become constant at 3.3 V by roughly 6 µs.

The results have by far matched all the expected results. We have looked at quite a number of circuits throughout this lab report.

## Discussion and Conclusion

The objectives of the lab exercise were met. The results match with the expected results. There were a few conceptual insights that were revealed to me. For instance, I saw how the graph for $V_R$ is proportional to the source current due to the equation V = I * R. Moreover, I learnt why a phase difference occurs instead of just remembering the fact that current lags or leads. There were no difficulties regarding system construction or measurement as it is quite simple to form a circuit in Multisim. If I did this lab work again, I would love to see the equations for Transient Analysis and AC Sweep for RLC circuits. The graphs themselves were surprising that means the equations are even more interesting.

## References/Appendices

The report is solely my work. I did not work with another teammate.

There are no outside sources used except for the notes that were provided in CE 1100 class under Professor James M. Florence.