

ECE 220 Laboratory 5

Step Response and Frequency Response

Michael W. Marcellin

Please follow all rules, procedures and report requirements as described at the beginning of the document entitled ECE 220 Laboratory 1.

Always wear your safety glasses when performing your lab experiments.

Prelab Part 1

Consider the RC circuit shown below in Figure 1. We will examine the step response and natural response of this circuit. Recall the FIFE equation from class:

$$x(t) = x(\infty) + (x(t_0) - x(\infty))e^{-(t-t_0)/\tau}$$

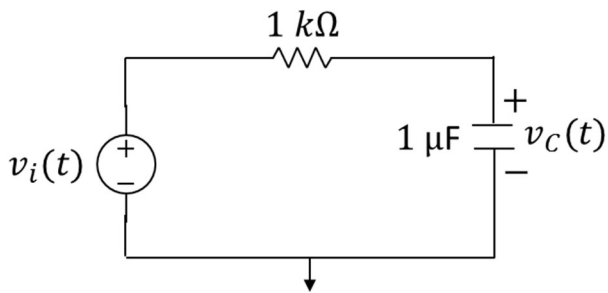


Figure 1. RC step response and natural response.

Suppose $t_0 = 0$, $v_C(t_0) = 0$ V, and $v_i(t) = 5$ V for $t \geq t_0$.

- a) Write an equation for $v_C(t)$ for $t \geq 0$ (step response).
- b) About when will $v_C(t)$ reach its final value?

Now assume $t_0 = 10$ ms, $v_C(t_0) = 5$ V, and $v_i(t) = 0$ V for $t \geq t_0$.

- c) Write an equation for $v_C(t)$ for $t \geq 10$ ms (natural response).
- d) About when will $v_C(t)$ reach its final value?

Now, assume that $v_i(t)$ is the square wave shown in Figure 2 below.

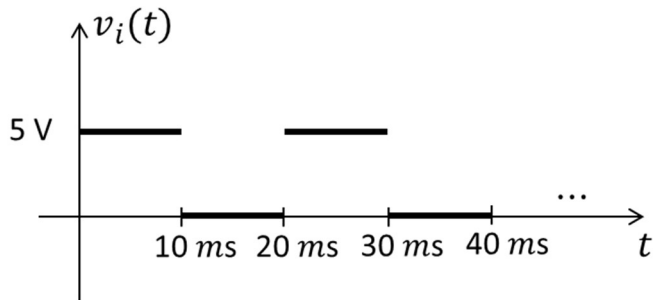


Figure 2. Square wave.

Assume $v_C(0) = 0$ V. Then your equation from a) above is good for $0 \leq t < 10$ ms. Now, from your answer to b) above, $v_C(t)$ reaches 5 V well before 10 ms. So, your equation from c) above is good for $10 \leq t < 20$ ms.

We will verify this using PSpice. Start a new project as usual, and draw the circuit shown below in Figure 3. We use a new part in this circuit called VPULSE to generate the square wave of Figure 2. This part can generate a very general periodic pulse sequence. Set the parameters as shown in the figure. The meanings of the parameters are as follows:

PER is the period

V1 is the “first” voltage

V2 is the “second” voltage (usually higher than V1)

TD is the Time Delay before switching from V1 to V2

PW is the Pulse Width (time voltage stays at V2 before returning to V1)

TR is the Rise Time (time voltage takes to transition from V1 to V2)

TF is the Fall Time (time voltage takes to transition from V1 to V2)

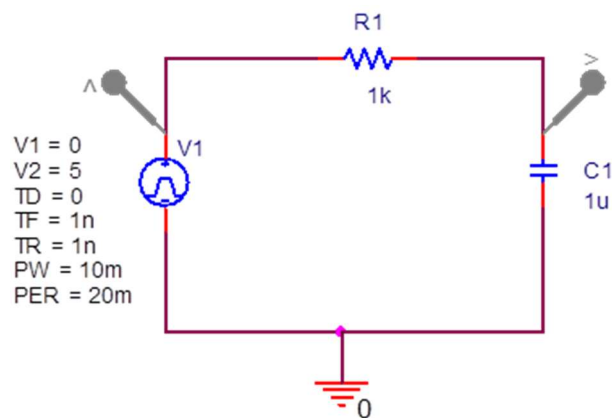


Figure 3. RC circuit with square wave input.

Double click carefully on the capacitor symbol itself (not the label or value). The property editor spreadsheet opens. Scroll left or right until you find the column labeled IC. This is the initial condition for the capacitor voltage. Set the cell below IC to 0. Click on the tab labeled PAGE 1 to return to your schematic.

Create a new simulation profile and set Analysis Type to Time Domain (Transient).
Set the parameters as follows:

Run to time: 40 *m*
Start saving data after: 0
Maximum step size: 0.1 *m*

Add voltage markers as shown in the figure and run your simulation. Verify that the resulting curves agree with your answers to a)-d) above.

e) print your graph.

Prelab Part 2

Consider the RC circuit shown below in Figure 4.

a) Use complex phasor analysis to calculate the steady state output voltage $v_o(t)$ for three different values of f . Specifically, $f = 10$ Hz, $f = 1$ kHz, and $f = 100$ kHz.

Don't forget that $\omega = 2\pi f$, so that the three different inputs are $v_i(t) = \cos(20\pi t)$, $v_i(t) = \cos(2000\pi t)$, and $v_i(t) = \cos(200000\pi t)$. If you have done this correctly, you should get something fairly close to $v_o(t) = \cos(20\pi t)$, $v_o(t) = \frac{1}{\sqrt{2}} \cos(2000\pi t - 45^\circ)$, and $v_o(t) = 0.01 \cos(200000\pi t - 90^\circ)$, for the three different inputs, respectively.

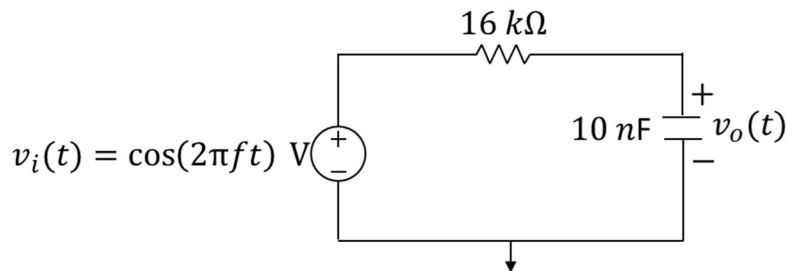


Figure 4. Steady state response for RC circuit.

Note that the only thing that changed in your three calculations is the frequency of the cosine input (10 Hz, 1 kHz, 100 kHz). In particular, the circuit is exactly the same in all three cases. From your calculations, we can see that the 10 Hz cosine passes through the circuit essentially unaltered. On the other hand, the 100 kHz cosine essentially does not pass through the circuit. From this, we conclude that the circuit is a “low pass” filter. That is, low frequency sinusoids pass through the circuit, while high frequency sinusoids are “filtered out.”

We will verify your results for the $f = 1$ kHz case using PSpice. Start a new project and draw the circuit of Figure 5. As in previous labs, use the part VSIN to generate the input voltage $v_i(t)$. This part generates $\sin(2\pi ft)$ by default. We can get $\cos(2\pi ft)$ by setting the parameter $TD = \frac{-1}{4f}$. For the example here with $f = 1$ kHz, we need $TD = \frac{-1}{4000} = -0.25$ ms. To set this

value, click carefully on the VSIN part (not the labels or values). The property editor spreadsheet opens. Scroll left or right until you see the column labeled TD. Enter -0.25 m in the cell below TD. To understand this choice for TD, see the appendix.

Create a new simulation profile. Set the Analysis Type to Time Domain (Transient), and enter the following parameters:

Run to time: 6.5 ms

Start saving data after: 5 ms

Maximum step size: $1\text{ }\mu\text{s}$ (don't forget that u means "micro" in PSpice)

The parameters above will collect 1.5 ms of data starting at time $t = 5\text{ ms}$. We wait until 5 ms have elapsed before collecting data so that the circuit is in steady state (transient response has died away).

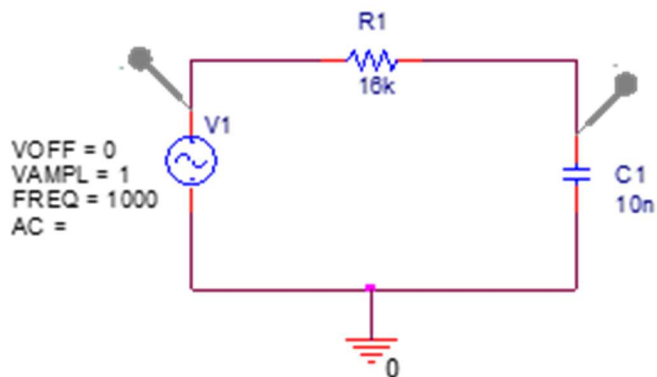


Figure 5. PSpice RC circuit.

Place voltage markers as shown in the figure to measure the input voltage and output (capacitor) voltage. Run the simulation.

The resulting figure should show two sinusoids. Verify that they appear as expected. Checking the amplitudes is easily done by inspection. The phase of the output voltage is a little trickier. We can obtain it as follows: First note that the input voltage crosses through 0 V (with negative slope) at 5.25 ms . The output voltage passes through 0 V (with negative slope) a short time later. You can determine this time fairly accurately using the PSpice cursor.

Click Trace->Cursor->Display

Now, in the lower left corner of the graph (the black part of the screen) is a legend which shows symbols for the two voltage curves (traces). Click carefully on the legend symbol (not the label next to it) for the output voltage curve. Now, click anywhere on the graph to initially place the PSpice cursor. Then, drag the cursor left/right with the mouse (or use the left/right cursor keys on your keyboard) to move the PSpice cursor to the place where the output voltage is closest to 0 V . Hint: Watch the values in the cursor window in the lower right corner of your screen. Once you have the cursor in the correct place,

Click Plot->Label->Mark

This will label the (x, y) coordinates of the PSpice cursor on the screen.

As an example, Figure 6 below shows what this looks like for the first zero crossing of the input voltage. Your task is to mark the first zero crossing of the output voltage.

b) print your graph showing the appropriate mark.

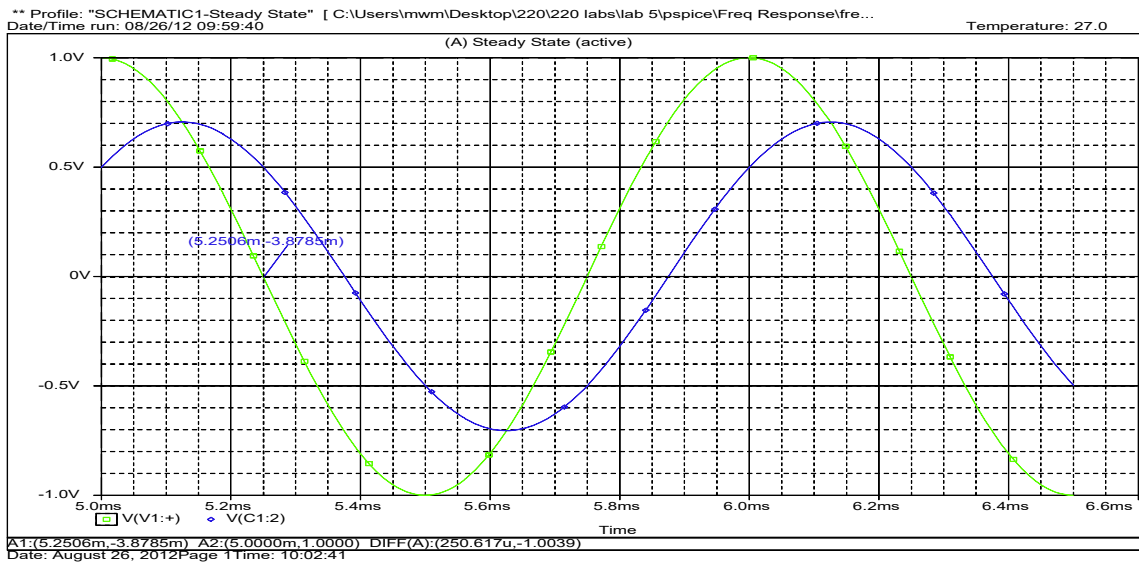


Figure 6. Voltage marking example.

Now, back to calculation of the output voltage phase. As mentioned above, and as shown in Figure 6, the input voltage crosses through 0 V at 5.25 ms. Your graph shows the time at which the output voltage crosses through 0 V a short time later. The difference between these two times is the time delay, Δt . You can find the phase delay of the output voltage, relative to the input voltage, using unit analysis, as

$$\Delta\theta \text{ (degrees)} = \Delta t \text{ (seconds)} \times f \left(\frac{\text{cycles}}{\text{second}} \right) \times 360 \left(\frac{\text{degrees}}{\text{cycle}} \right)$$

c) Calculate $\Delta\theta$. Verify that your answer agrees with your calculations from part a).

Prelab Part 3

We discovered above that changing the frequency of the input cosine changes both the magnitude and phase of the output cosine. That is, the magnitude and phase of the output cosine are both functions of frequency, say $A(f)$ and $\theta(f)$, respectively. The function $A(f)$ is called the magnitude response of the circuit, while the function $\theta(f)$ is called the phase response of the circuit. We now consider making plots of these two functions. From the resulting graphs, we can easily see how the magnitude and phase of the output cosine will behave as a function of the input cosine frequency. We could create these graphs by computing

more points by hand (as we did in Part 2 above), but this would be rather tedious. Let's get PSpice to generate these graphs for us.

Start a new project in PSpice. Draw the circuit shown below in Figure 7. We use another new part called VAC. This is an AC voltage source for which we can easily perform a sweep over a range of frequencies. We use another new part to create the node labeled v_o . To place this part, click

Place->Power

A new window opens.

Select the part named VCC_CIRCLE. As always, don't worry about the library name.

Enter V_o in the box labeled "Name:"

Click OK.

Don't forget that you can press r to rotate the part before you place it in the location shown on the figure.

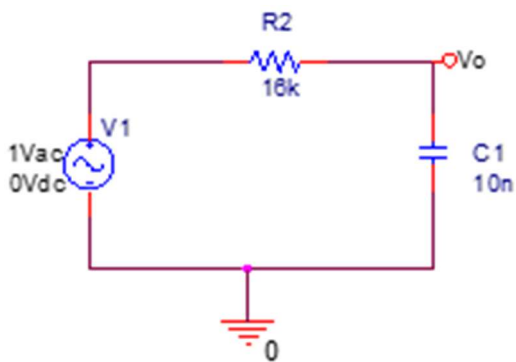


Figure 7. Low pass filter frequency response.

Create a new PSpice simulation profile. This time, we choose AC Sweep/Noise as the Analysis Type. Choose Logarithmic/Decade Sweep Type, and enter the following parameters:

Start Frequency: 1

End Frequency: 100k

Points/Decade: 101

Click OK

Run the simulation.

The resulting graph should be blank. Click Trace->Add Trace. A new window opens. On the left window, scroll if needed, then click on $V(V_o)$ and click OK. You should now see the magnitude response of the low pass filter. That is, the graph shows the amplitude of the output cosine as a function of f . As expected, low frequency sinusoids pass through the filter essentially unmodified. High frequency sinusoids are essentially eliminated. In between is a transition region where the sinusoids are allowed to pass through the filter to varying extents. Examine the graph carefully and make sure you understand its significance. In particular, verify the points that you have previously computed. Specifically, the magnitude should be about 1 at 10 Hz, 0.707 at 1 kHz, and 0.01 at 100 kHz (see your answer to Part 2a)).

- a) Observe and record the magnitude resp's onse at 3 kHz.
- b) Print your graph.

Close the graph.

The magnitude response is usually plotted in decibels (dB). To convert the voltage v_o from volts into dB, you use the following equation: $v_o (dB) = 20\log(v_o)$. The log is base 10. At first this may seem confusing. The original v_o has units of volts (V). After taking the log and multiplying by 20, the result has units of decibels (dB). Both of these values are still representative of the voltage and so the name is usually not changed. Both are still called v_o . You can distinguish between the two by the units. For example, $v_o = 1.0$ V is the same thing as $v_o = 0$ dB. Similarly, $v_o = 0.01$ V is the same thing as $v_o = -40$ dB.

Run the simulation again. The graph should again be blank. Click Plot->Add Plot to Window. You should now have two boxes where we will put individual graphs. Click on the top and bottom boxes, and note that you can select one of the two boxes as indicated by SEL>>. Select the top box. We will now plot the magnitude response (in dB) for our low pass filter in this top box. This will be exactly the same graph as printed in part b), but in units of dB.

Click Trace->Add Trace

A new window opens. On the right side, click DB(). Note that this is different from D(). Now, on the left side, click V(Vo). The Trace Expression at the bottom of the window should now say DB(V(Vo)). That is, PSpice will compute in dB, the amplitude of the voltage V at the node labeled Vo. Click OK. You should now see the magnitude response of the low pass filter in dB. Verify that the value is 0 dB (1.0 V) at 10 Hz, and -40 dB (0.01 V) at 100 kHz, as expected.

Now select the bottom box in your graph. Click Trace->Add Trace

A new window opens. On the right side click P(). Then on the left side click V(Vo). The Trace Expression at the bottom of the window should now say P(V(Vo)). That is, PSpice will compute the phase P of the voltage V at the node labeled Vo. Click OK. The bottom window should now contain a graph of the phase response. The phase response indicates the phase by which the output cosine will be shifted, relative to the input cosine. Verify that the value of the phase response is essentially 0° at 10Hz and -90° at 100 kHz, as expected from your calculations in Part 2a).

- c) Observe and record the magnitude (in dB) and phase (in degrees) at 1 kHz.
- d) Print your graph.

Prelab Grading

Part 1

- a) 2 pts
- b) 1 pt
- c) 2 pts
- d) 1 pt
- e) 3 pts

Part 2

- a) 3 pts
- b) 3 pts
- c) 2 pts

Part 3

- a) 2 pts
- b) 2 pts
- c) 2 pts
- d) 2 pts

25 total pts

Lab 5 Data Sheet

Please follow all rules, procedures and report requirements as described at the beginning of the Lab 1 Data Sheet.

Lab Grading: Data Sheet = 15 pts; Explanation = 10 pts.

Lab Part 1

Pulse response circuit working: Y / N (circle one)

6 pts

Lab Part 2

$v_i =$	$v_o =$	$\Delta t =$	$\Delta \theta =$
---------	---------	--------------	-------------------

4 pts

Frequency response circuit working: Y / N (circle one)

5 pts

Lab Part 3

No experiments.

Lab Experiments

Lab Part 1

Build the circuit shown in Figure 1. Use the function generator to make $v_i(t)$ equal to the square wave shown in Figure 2. Set the peak-to-peak voltage to 5 V. You will need to set a DC offset (level shift) so that the voltage switches between 0 V and 5 V, rather than -2.5 V and 2.5 V. Use channel 1 of the scope to monitor the square wave, and channel 2 to monitor the capacitor voltage. Set the scope to trigger on channel 1. As usual, use positive slope triggering. Set the triggering level a bit above 0 V so that the square wave begins at 5 V on the scope screen. Verify that the circuit behaves as expected according to your graph from Part 1 e) of the prelab.

Have the TA verify that your circuit is working, and initial the results on your data sheet.

Online students: take a screenshot showing the screen of your oscilloscope.

Lab Part 2

Build the circuit shown in Figure 4. Set the frequency of the cosine to 1 kHz. Use channel 1 of the scope to monitor $v_i(t)$, and channel 2 to monitor $v_o(t)$. Set the scope to trigger on channel 1 with positive slope and level 0 V. Measure the amplitude of $v_i(t)$ and $v_o(t)$. Also measure the time delay of $v_o(t)$ with respect to $v_i(t)$. Record these three values on your data sheet. Finally, compute the phase delay of $v_o(t)$ with respect to $v_i(t)$. Record this on your data sheet as well. Try changing the frequency of the cosine to several different values and make sure the results are as you expect. In particular, the circuit should “pass” low frequency cosines through, and “filter out” high frequency cosines.

Have the TA verify that your circuit is working, and initial the results on your data sheet.

Online students: take a screenshot showing the screen of your oscilloscope.

Lab Part 3

No experiments.

Appendix

In this appendix, we derive the appropriate value for the parameter TD so that the VSIN part results in a voltage of $\cos(2\pi ft)$.

From class, we know that $\sin(2\pi ft) = \cos(2\pi ft - 90^\circ)$. Conversely, $\cos(2\pi ft) = \sin(2\pi ft + 90^\circ)$. Since $90^\circ = \pi/2$ radians, we have

$$\cos(2\pi ft) = \sin\left(2\pi ft + \frac{\pi}{2}\right) = \sin\left(2\pi f\left(t + \frac{1}{4f}\right)\right).$$

The VSIN part in PSpice uses TD to implement a Time Delay in the sin as follows:

$$\sin(2\pi f(t - TD)).$$

Comparing the two equations above, we see that the appropriate choice is

$$TD = \frac{-1}{4f}.$$