T2

RC filters and frequency response

In this laboratory,[1], you will analyse a simple RC circuit under square wave and sinusoidal excitation. You will use the modelling and computation package MATLAB to produce a mathematical simulation of this circuit, and observe the simulated results. These results will be compared in turn with observations of a real RC circuit.

Schedule:	
Preparation time :	1 hour
Lab time :	3 hours
Items provided:	
Tools :	Standard
Components :	Standard lab components
Equipment :	Standard bench
Software :	MATLAB
Items to bring:	
	Essentials. A full list is available on the Laboratory website at `https://secure.ecs.soton.ac.uk/notes/ellabs/databook/essentials/`

Before you come to the lab, it is essential that you read through this document and complete all of the preparation work in section 2. If possible, prepare for the lab with your usual lab partner. Only preparation which is recorded in your laboratory logbook will contribute towards your mark for this exercise. There is no objection to several students working together on preparation, as long as all understand the results of that work. Before starting your preparation, read through all sections of these notes so that you are fully aware of what you will have to do in the lab.

> **Academic Integrity** – *If you undertake the preparation jointly with other students, it is important that you acknowledge this fact in your logbook. Similarly, you may want to use sources from the internet or books to help answer some of the questions. Again, record any sources in your logbook.*

[1]

Revision History		
November 29th, 2012	mcf	Version 1 (based on material from Part I lab by bmah, July 2004).
September 2nd, 2013	mcf	updated.

1 Aims, Learning Outcomes and Outline

This exercise aims to provide you with:

- an opportunity to apply the principles of complex number representations to analysis of simple electronic circuits
- further experience of the MATLAB symbolic computation and numerical modelling software
- experience of software simulations of simple electronic circuits
- an opportunity to compare the results of mathematical analysis with software simulations and with practical observation

Having successfully completed the lab, you will be able to:

- analyse simple passive linear electronic circuits using phasors
- use MATLAB to simulate a passive linear electronic circuit in both switched DC and AC contexts
- demonstrate correspondence between the results of algebraic analysis of transfer functions, MATLAB simulations and the real-world behaviour of a circuit

---

2 Preparation

It is essential that you complete all the work in this section before you come to the lab. If at all possible, prepare for the lab with your usual lab partner. Only preparation which is recorded in your laboratory logbook will contribute towards your mark for this exercise. There is no objection to several students working together on preparation, as long as all understand the results of that work. Before starting your preparation, read through all sections of these notes so that you are fully aware of what you will have to do in the lab.

2.1 Understanding of concepts.

Using the information in your course texts in Mathematics and Circuit Theory, and on the ECS Website, go through the following work in your logbook, writing it up as you go:

- Write down the equation relating the charge stored in a capacitor to the voltage across it. Differentiate this equation to relate instantaneous capacitor current to capacitor voltage.
- Using the expression for complex voltage and current and the differential relationship above, derive the impedance of an ideal capacitor.
- The time constant for an RC circuit is $\tau = RC$. Explain how you would determine the time constant of the waveform shown in Figure 3.
- Ensure that you are familiar with the MATLAB content of ELEC1200 problem sheet 2 and 3. Review the indicative solutions.

2.2 Relationship of the simulated behaviours to laboratory measurements

- Given that signal generator controls and readouts usually give the amplitude of sinusoidal outputs in Vrms, what generator setting is required to produce an (open-circuit) voltage with a peak value of 1.0V?
- If you are using an oscilloscope to observe signals in a circuit where the resistance is of the order of kΩ, the capacitance is of the order of tens of nF and the frequencies to be observed are below 100kHz, will you need to use your $\times 10$ high-impedance probe, or will the $\times 1$ leads available on the bench be adequate? Give reason(s) for your answer in your logbook.

2.3 Values and preliminary analysis

- Using the ECS Website and/or the introductory section of the Laboratory Handbook, write in your logbook the nearest preferred values (NPVs) to a 0.1μF capacitor and a 1.59kΩ resistor assuming that capacitors are limited to E6 (20% tolerance) values and resistors to E12 (10% tolerance) values. Using standard circuit analysis techniques and your work in section 2.1 above, show in your logbook that equation (1) of section 3.1 below represents the relationship between input and output voltages for the circuit of Figure 1 below.
- Work out in your logbook the magnitude and phase of the output voltage in Figure 1 when the input voltage is a 1V peak sinusoid of frequencies 100Hz, 500Hz, 1kHz, 5kHz and 10kHz. Produce a table showing the predicted magnitude and phase and leave columns for the simulated (from Sections 3.2, 3.3) and observed (from Section 3.4) results.

---

3 Laboratory Work

3.1 Initial analysis of an RC circuit

Figure 1 shows a simple RC circuit, $R = 1.59\text{k}\Omega$, and $C = 0.1\mu\text{F}$

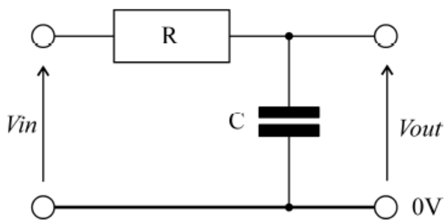

Figure 1: Simple RC circuit

To analyse how this circuit works over different frequencies, you need to work out the voltage network function of the circuit (i.e. $\frac{V_{\text{out}}(j\omega)}{V_{\text{in}}(j\omega)}$) using f as the frequency of operation (units of Hertz) related to ω the angular frequency of operation (units radians per second) by $\omega = 2\pi f$.

$$\frac{V_{\text{out}}(j\omega)}{V_{\text{in}}(j\omega)} = \frac{\frac{1}{j\omega C}}{R + \frac{1}{j\omega C}} = \frac{1}{1 + j\omega CR} = \frac{1}{1 + j2\pi fCR}. \quad (1)$$

3.2 Using MATLAB to produce plots of magnitude and phase against frequency

Use `ode23` to run a simulation to determine V_{out} the response of the RC circuit if V_{in} corresponds to a sine wave of 1V amplitude and a frequency of 1kHz. Look at the solutions to ELEC1200 problem sheet 2 to recall how to do this. To insert the AC source define the function:

```
function dv=rc(t,v)
dv(1) = -X * v(1)+ Y*sin(Z.*t)
```

for suitable numerical values of X, Y and Z which you should determine by writing down the differential equation relating V_{out} to V_{in}.

Simulate the system on an appropriate timescale, $[T0, T1]$, and with appropriate initial condition V:

```
[t vout] = ode23(@rc, [T0, T1],V);
```

and plot V_{out}:

```
plot(t,vout)
```

Experiment with a few values of the initial condition V, and then fix V at the value that produces the steady state response (i.e. with no transient).

On the same figure (use `hold on`) plot V_{in}:

```
vin=cos(2*pi*1000*t);
plot(t,vin)
```

By zooming in as appropriate, measure the phase shift and the amplitude change between V_{in} and V_{out}. To read values on the plots, try the MATLAB command `ginput`. This will turn the mouse arrow into a cursor symbol on the plot. Mark on the graph the point you wish to read by left-clicking (left-clicking at several points will store several sets of coordinates). Press RETURN and the coordinates of the point(s) clicked will appear in the command window. The first number represents the x-axis value and the second number represents the y-axis value. To obtain greater resolution of this data, investigate the graphic displays zoom facility and the `format long` command.

Label the figures appropriately (axes etc.) and print for your log book, along with a record of the waveforms amplitude, phase, and frequency. You might consider placing text on this display using the commands:

```
gtext ('vin') % place the word input
gtext ('vout')
```

MATLAB will show a cross-hair cursor allowing you to place the origin of text precisely with a mouse click, so that you can produce input and output plots with the curves labelled e.g. as in Figure 2.

Does the numerical simulation correspond to the calculations in Section 3.1?

3.3 Extension to multiple frequencies

Change the frequency of the simulated circuit and produce simulations at 100Hz, 500Hz, 5kHz and 10kHz. For at least some of these additional frequencies print out the appropriately annotated MATLAB plot and stick it into your logbook. As part of your preparation you should have produced a table showing the predicted and simulated magnitude and phase for all five frequencies used for preparation work (100Hz, 500Hz, 1kHz, 5kHz and 10kHz), with columns left ready for adding the results of simulations run in section 3.2, here and in section 3.4 – use this table for recording the results of your simulations. Write in your logbook a cross-reference to the page where this table can be found. If you did not do that part of the preparation, do it now!

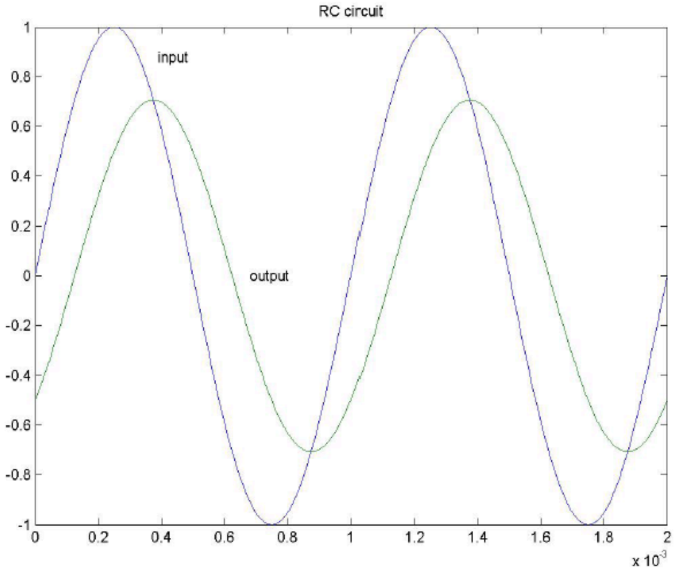

Figure 2: Example labelled figure

3.4 Real-world AC circuit observations

Build the circuit of Figure 1 using the nearest preferred values of resistor and capacitor determined in your preparation (section 2.3). Apply a 1V peak, 1kHz sine wave from a signal generator and display the input and output waveforms on the oscilloscope.

Measure the output waveform amplitude and phase, and compare with your simulation results writing the values observed into the table created as part of your preparation, and giving a cross reference to the page on which this table can be found. Is there any difference between simulation and observation? If so, suggest reasons for this. (Note: measuring relative phase is made easier by varying the oscilloscope timebase so that the waveform occupies an integer number of graticule divisions. Use the Var or Vernier adjustment, and return it to standard calibration setting after use.) For this section of the exercise you must use standard oscilloscope observation technique: for each value taken from the instrument, state the number of divisions of voltage/ time and the scaling of voltage/time per division. Do not use any of the oscilloscopes built-in parameter measurement facilities.

Repeat the above measurements and comparisons with the signal generator producing 1V peak sinusoids of 100Hz, 500Hz, 5kHz and 10kHz. As before, record the data from these observations in the table produced as part of your preparation.

From the above observations, can you suggest uses for the circuit of Figure 1? An ideal low pass filter cuts out all frequency content above a certain level, and allows all frequency content below this level to pass through. To what extent is the RC circuit a low pass filter?

3.5 Measurement of the time constant RC

In addition to numerical computation, MATLAB can also do symbolic computation. Try:

```
x=dsolve('Dx= -(1/(R*C))*x +(1/(R*C))', 'x(0)=0')
```

Explain the syntax using `help` etc. and verify the resulting answer by substitution into the corresponding differential equation. MATLAB has created a *symbolic object* called `vout`. We first convert this to a string using `vectorize`, and then use the `eval` command to evaluate the resulting string as a MATLAB command. We can then plot the response:

```
t = [0:10^(-5):10^(-3)];
vout = eval(vectorize(x));
plot(t,z)
```

You should obtain something like Figure 3. Calculate the time constant of the circuit (using your preparation work from section 2.3) and check that this agrees with the time constant derived from the MATLAB plot. As previously make sure you label the graph, axes etc. appropriately.

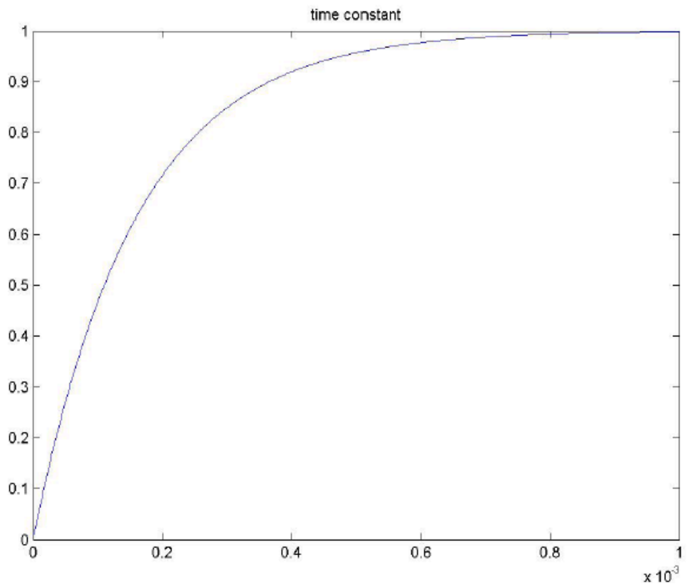

Figure 3: Step response

We now want to determine the RC constant experimentally. One simple way to do this is to drive the circuit with a square wave. First look at this in simulation. A simple way to generate a square wave is to use the `sign` function, e.g.:

```
t = [0:10^(-6):10^(-2)];
plot(t,sign(sin((2*pi*200).*t)))
```

By appropriately modifying your simulation, use `ode23` to simulate the circuit driven by a square wave of frequency 200Hz.

Apply a 200Hz square wave signal to your RC circuit. Observe the output waveform and measure its time constant. How does this compare with the value determined from the MATLAB simulation shown in Figure 3 and from your theoretical prediction using the C and R values in section 2.3? Measure and explain the behaviour at (100Hz, 500Hz, 1kHz, 5kHz and 10kHz). Compare to the simulation.

Revisit your predictions of the phase shift and amplitude change for the circuit driven by the AC source at 100Hz, 500Hz, 1kHz, 5kHz and 10kHz based on your experimentally determined value of the time constant RC.

---

4 Optional Additional Work

4.1 A CR circuit

Consider Figure 4 below, in which the capacitor and resistor of Figure 1 have been swapped. Using the same techniques as in section 2.3 determine the frequency- domain network functions. Hence predict the output magnitude and relative phase for 1V peak sinusoidal inputs of frequencies 500Hz, 1kHz and 5kHz. Simulate the circuit using MATLAB and compare the results with your predictions. Build the circuit (or simply rearrange the connections of the existing real-world circuit) and observe its behaviour at 500Hz, 1kHz and 5kHz, comparing this with the predictions and simulations. Does the CR circuit have a filter like behaviour? How might it be used?

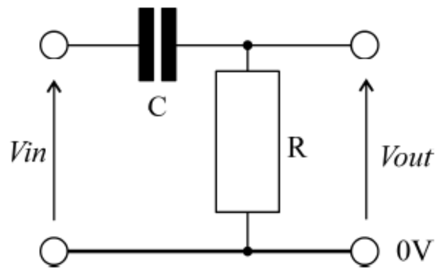

Figure 4: CR circuit

4.2 An LC circuit

Set up a MATLAB simulation for an LC circuit. Show the behaviour both close to, and far away from the resonant frequency. Show also the behaviour at the resonant frequency. Does the simulation correspond to your expectations from the theory?