EECS 216 Winter 2008
Lab 1: LTI Systems
Part II: In-lab & Post-lab Assignment

Professor Kim Winick

Department of Electrical Engineering & Computer Science

University of Michigan

© Kim Winick 2008

1   Laboratory Task Description

Unless otherwise specified, it will be assumed that the unit of time is the second and the unit of voltage is the Volt. When you are asked to measure the response of the circuit to a specified input, it will be assumed that you are being asked to measure the output for an initial condition of zero. Thus you must first short the circuit input (i.e., connect a wire across the circuit input) in order to discharge the capacitor resulting in Vin(0)=0. The capacitor will discharge completely in tens of milliseconds, and therefore the shorting wire can be disconnected after a second or two. **Also make sure that the oscilloscope is set for DC coupling for all of these measurements (procedure to do so is given below).**

In this lab you will be asked to measure the output response of the series RC circuit shown in Fig. 1.0.1 below for a variety of inputs, including a step, a combination of a step and a ramp, and a sinusoid. You will then compare your results to those computed as part of your pre-lab assignment.
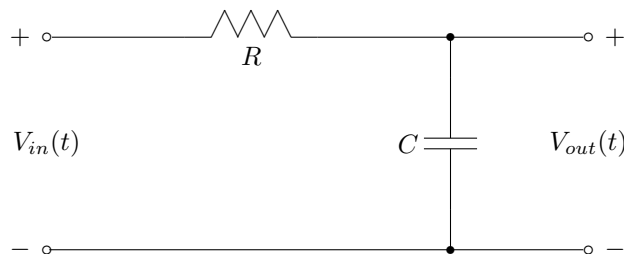

Figure 1.0.1: RC Circuit

When you make a measurement you should immediately qualitatively compare your measured result with the corresponding result calculated (and/or plotted) as part of your pre-lab assignment. You may notice that your measured results do not agree *exactly* with those that are computed. There are several factors that contribute to these discrepancies, and these factors include: (1) The signal generator does not produce the exact waveform requested. For example the rising edge of a unit step function produced by the signal generator will not be perfectly vertical nor will the value for $t > 0$ be exactly equal to 1. (2) The values of capacitance and resistance marked on the components are only nominal.

The actual values may deviate from these nominal values by as much as $\pm 10\%$. (3) The oscilloscope used in the laboratory is a digital instrument. The analog waveform collected by the oscilloscope is sampled in time and quantized in amplitude. The oscilloscope in your lab uses an 8-bit A/D (analog-to-digital) converter to quantize the amplitude, *i.e.*, each amplitude value is assigned a 8-bit binary number. For

example, if we consider an input voltage range between 0 V and 1 V, then any voltage between 0 V and 2^{-8} V would be assigned a value of 00000000, and any voltage between 2^{-8} V and 2×2^{-8} V would be assigned a value of 00000001, and any voltage between 3×2^{-8} V and 3×2^{-8} V would be assigned the value of 00000010, etc. This quantization process does not permit one to capture the actual values exactly, and thus the captured waveform will have a staircase appearance, rather than a smooth appearance. The staircase approximation to the waveform is commonly known as quantization noise or quantization error. Clearly this "noise" becomes smaller as the number of bits used by the A/D increases, but along with the bit increase comes a corresponding price increase for the instrument!

Most of you have taken EECS 215 and are familiar with the basic operation of oscilloscopes, function generators and power supplies. Prof. Ganago has prepared a tutorial on how to use the new Agilent oscilloscope (3000-series) and function generator 33220A which has been included as Appendix A in the lab manual. Below, step-by-step directions are given for the equipment settings needed to perform the indicated measurements. In addition, your GSI can assist you if you encounter difficulties.

For all of the measurements described below you must be certain that the input of the scope is set to be DC coupled. This setting allows a DC (*i.e.*, constant) signal component to pass through the scope input. If AC coupling is used instead, then a capacitor is placed in series with the scope input (automatically inside the scope). We know that a capacitor cannot pass a DC current, and thus DC signal components are blocked at the scope input. This mode of operation, of course, is inappropriate if you desire to measure signals, such as unit steps, that contain DC components.

The scope and signal generator are connected to the computer on your lab bench. Using the Scope Connect Software supplied by Agilent, it is possible to transfer data from the oscilloscope onto the computer for visualization and subsequent data processing, (using Matlab for example). You will be asked as part of this lab to store your measured waveforms on the computer for subsequent processing after you have left the lab. **Thus you must store your data in your own directory (or email it to yourself or place it on a USB memory stick) so that you may access it later via your CAEN account.** Your GSI will also come by before you leave the lab and will ask to see your collected waveforms in order to verify that you have successfully completed the specified tasks.

After you have completed the measurements and left the lab, you must produce the additional plots indicated in this write-up. You will be asked to hand-in these additional plots and the frequency response data collected in Section 2.5 as part of the Post-Lab assignment.

1.1   Equipment Set-up Instructions

RESET THE OSCILLOSCOPE

1. Power on the scope

2. Press **Save/Recall**

3. The top square softkey should say **Setups**. If it says **Waveforms** instead, press the top softkey to switch to the **Setups** menu.

4. Press the square softkey that is second from the top. It should say "Default Setup."

SET FOR DC COUPLING

1. Press the button labeled **1** to get the CH1 menu.

2. The top softkey should say **DC**. If it does not, press this key until **DC** is displayed.

3. Repeat for the button labeled **2** (CH2 menu).

SET FUNCTION GENERATOR FOR HIGH-Z OUTPUT

1. Power on the function generator.

2. Press the **Utility** button. It should light up.

3. The third softkey from the left should say "Output Setup". Press it.

4. Press the first softkey on the left until **High Z** is displayed.

5. Press the softkey on the right, for "Done."

SET UP HARDWARE

1. Build the RC Circuit shown in Fig. 1.0.1 of the lab manual with $R = 1\,\text{K}\Omega$ and $C = 1\,\mu\text{F}$.

2. Connect Function Generator to supply $V_{in}(t)$

3. Connect Ch1 to measure $V_{in}(t)$

4. Connect Ch2 to measure $V_{out}(t)$

---

2   In-Lab and Post-Lab Homework Assignment

Given that this is the first lab (*i.e.*, Lab 1) and that some of you may lack familiarity with the test and measurement equipment, you will be given rather detailed instruction regarding the measurement procedures.

2.1   Determine Unit Step Response & Compare with Theory

In-Lab:

- Assemble on a protoboard an RC circuit corresponding to Fig. 1.0.1 with $R = 1\,\text{K}\Omega$ and $C = 1\,\mu\text{F}$.

- Use the signal generator to generate a 1 V step.

- Use the oscilloscope to measure the response of the RC circuit to this input ($V_{in}(t) = u(t)$ V). Visually compare your result with the unit step response computed in the prelab assignment (4.1).

- Collect the data on the computer using the Scope Connect software.

Post-Lab:

- Plot this data together with the unit step response computed in the pre-lab assignment problem 4.1. For your plot use the Matlab axis command `axis([0 0.01 0 1])`.

Detailed Instructions for Part 2.2

---

1. Set the time and voltage bounds on the scope.

   Voltage: (rotate dials marked Vertical to change)
   Ch1 500 mV/div
   Ch2 500 mV/div
   Time: (rotate dials marked Horizontal to change) 5.00 ms/div

---

2. Set the trigger on the scope.

> Press the **Mode/Coupling** button. From top to bottom, settings should read:
> Mode: **Edge**
> Source: **CH1**
> Slope: (upward)
> Sweep: **Normal**
> Coupling: **DC**
> Press appropriate softkeys until trigger settings are as shown above.
> Rotate the **Level** dial to change the trigger level to 500 mV.

3. Set the function generator to produce a DC voltage.

> Press the **Utility** button. It should light up.
> Press the left softkey labeled "DC" until **On** is displayed.
> Rotate the dial until `+1.000 V_DC` is displayed.

4. Trigger output

> The **Output** button should not be lit. If it is lit, press it so that it turns off.
> Press the **Output** button to enable the output. It will turn on.
> The scope will detect the rising edge of the voltage and will display a step function in the input voltage `V_in(t)` in Channel1. Channel2 will have the step response of our circuit.
>
> If you do not see the image frozen on the scope, press **Output** on the function generator to zero the input voltage. Wait about a second for the output voltage of our circuit to converge to zero (this will not be displayed on the scope, so theres no way to check. Just wait a second after turning **Output** off to be sure the output voltage of our circuit is zero.) Then, press **Output** again to produce a step function for the circuit. If this still doesnt work after 2 or 3 tries, ask your GSI for help.

5. Capture the image in ScopeConnect

> Start Scope Connect software on the Windows machine.
> Click the icon that looks like a parallel port to connect the scope. (Note: if you get a response saying CANNOT CONNECT TO USB DEVICE, then go to the Tools pulldown menu, select options, and then select GPIB.)
> Double click the "Waveform 0" entry in the left menu to bring up an empty image of the scope.
> Click "Refresh."
> You should see the image of your scope.
> Click the "Export" Button.
> Save the image as "`lab1_part1.bmp`"

6. Capture data in ScopeConnect

> Double click the "Data 0" entry in the left menu to bring up a spreadsheet
> Make sure the "Screen" and "CH1" radio buttons are selected
> Click "Refresh"
> Click "Export"
> Save the data as "`lab1_part1_ch1.xls`"
> Select the "CH2" radio button.
> Click "Refresh"
> Click "Export"
> Save the data as "`lab1_part1_ch2.xls`"

7. Disconnect the scope by clicking the button with the red X. **If you do NOT do this, you cannot control the scope from its front panel!**

2.2 Determine the Pulse Response & Use Short Input Pulses to Approximately Determine the Impulse Response

In-Lab:

- Use the signal generator to generate a pulse of duration 1 ms and amplitude 100 mV.

- Use the oscilloscope to measure the response of the RC circuit to this input (*i.e.*, $V_{in}(t) = 0.1[u(t)?u(t?0.001s)]$ V). Visually compare your result to the pulse response computed in the Pre-Lab assignment (4.3).

- Collect your measurement data on the computer using the Scope Connect software.

- Repeat the above procedure for the following two pulses

    (i) duration 0.5 ms, amplitude 200 mV

    (ii) duration 0.1 ms, amplitude 1 V

Post-Lab:

- Plot your results together with the pulse responses and 10-4 $h(t)$ computed in the prelab (4.3) under the same conditions. For your plots use the Matlab axis command `axis([-1e-3, 10e-3, 0, 0.1])` (for all plots).

Detailed Instructions for Part 2.2

1. Set the time and voltage bounds on the scope.

    Voltage: (rotate dials marked Vertical to change)
    Ch1 50 mV/div
    Ch2 50 mV/div
    Time: (rotate dials marked Horizontal to change)
    500.0 us/div

2. Set the scope to trigger on an upward slope. Set the trigger voltage at 50 mV.

3. Set the function generator to produce the following periodic DC pulses:

    Width: 1ms
    Period: 1 s
    HiLevel: 100 mV
    LoLevel: 0V

4. Capture both an image and two data sets using Scope Connect.

5. Disconnect the scope by clicking the button with the red X.

6. Collect data and images for pulses of

    (i) width 0.5 ms, HiLevel 200 mV
    (ii) width 0.1 ms, HiLevel 1 V

2.3 Demonstrate Linearity

There is no In-Lab component to this section. Simply complete this task after you leave the lab.

Post-Lab:

Use the data collected in 2.2 together with linearity and time-invariance to compute the pulse response when the pulse duration is 0.5 ms and the pulse amplitude is 200 mV. Plot your result together with the result for Pre-Lab item 4.3. For your plot use the Matlab axis command `axis([0 0.01 0 1])`.

Helpful Hint: Your GSI will have uploaded to the course web site the MATLAB file `shift_216.m`. You may find it helpful in forming the shifted versions of signals. We make no guarantees that it will meet your specific needs; if necessary, modify it as needed.

2.4   Demonstrate Linearity & Time-Invariance In-Lab:

In-Lab:

- Use the function generator to generate the following input signal.

$$V_{in}(t) = [u(t) - u(t - 0.01)] + 200(t - 0.011)[u(t - 0.011) - u(t - (0.016)] \text{ V}$$

  This waveform has been pre-programmed into the generator by your GSI.

- Use the oscilloscope to measure the response, $V_{out}(t)$, of the RC circuit to this input. Visually compare your result with that obtained in prelab assignment 4.4.

- Collect the data on the computer using the Scope Connect software.

Post-Lab:

- Plot your result together with the corresponding results computed in the Pre-Lab assignment 4.4. For your Matlab plot use the axis command `axis([0 0.02 0 1.2])`.

Detailed Instructions for Part 2.4

---

1. Set the time and voltage bounds on the scope.

   Voltage: (rotate dials marked **Vertical** to change)

      Ch1 500 mV/div
      Ch2 500 mV/div

   Time: (rotate dials marked **Horizontal** to change)
      2.000 ms/div

2. Set the scope to trigger on a downward slope. Set the trigger voltage at 500 mV.

3. Load a saved waveform from the signal generator by pressing the **Arb** button.

   Press the softkey third from the right, labeled "Select Wform". Press the softkey second from the left, labeled "Stored Wforms" Make sure "ARB_1" is selected and press the softkey labeled "SELECT ARB"

   Change the following settings for the waveform:

      Period: 70 ms
      HiLevel: 1 V
      LoLevel: 0 V

4. Collect data and images

---

5. Have your GSI initial your pre-lab plot now indicating that you have completed this measurement successfully.

FYI: the waveform is defined by the following points. The waveform generator is interpolating between these points using a straight line. Thus the resulting waveform should be

$$V_{in}(t) = [u(t) - u(t - 0.01)] + 200(t - 0.011)[u(t - 0.011) - u(t - 0.016)] \text{ V}.$$

Point #	Time (ms)	Voltage (V)
1	0	1
2	10.00	1
3	10.01	0
4	11.00	0
5	16.00	1
6	16.01	0
7	69.99	0

2.5   Response to an Eternal Sinusoid & Frequency Response

In-Lab:

- Use the function generator to generate a sine wave of frequency 50 Hz with 10 Vpp.

- Use the oscilloscope to measure the response of the RC circuit. Observe that the output is sinusoidal after any transients die out.

- Find the peak-to-peak voltage and phase offset (in radians) of the output sinusoid relative to the input sinusoidal signal.

- Repeat for the following frequencies: 200 Hz, 500 Hz, 1 KHz and 5 KHz.

Post-Lab:

- Compare your results with the pre-lab calculation done for Pre-Lab problem 4-6.

Detailed Instructions for Part 2.5

1. Set the time and voltage bounds on the scope.

   Voltage: (rotate dials marked **Vertical** to change)

   Ch1 2.00 V/div
   Ch2 2.00 V/div

   Time: (rotate dials marked **Horizontal** to change)

   50.00 ms/div

2. Set the scope to trigger on a downward slope. Set the trigger voltage at 0 mV.

3. Using the function generator, create a sine wave at 50 Hz with 10 V peak to peak, with 0 V offset.

   Several periods of the sine wave should appear on the screen. Use the oscilloscope to measure the peak-to-peak amplitude of this sinusoid. What value do you get? Does it exactly equal the setting on the signal generator?

4. Measure peak to peak output voltage by pressing **Measure**, then pressing the top square softkey until **CH2** is selected. Press the second softkey from the top, labeled **Voltage**. Press it again to choose **Vpp**.

5. Write down the indicated peak to peak voltage in a table. Collect peak to peak voltages for those frequencies indicated in the lab manual. Change voltage and time scales as necessary to obtain accurate measurements.

6. Once more, create a sine wave at 50 Hz with 10 V peak to peak, with 0 V offset.

7. Set the time and voltage bounds on the scope.

> Voltage: (rotate dials marked **Vertical** to change)
>> Ch1 100 mV/div
>> Ch2 100 mV/div
> Time: (rotate dials marked **Horizontal** to change)
>> 500 μs/div

8. Measure time delay by pressing **Cursors**. The menu should display selections of **Manual** under "Mode" and **Time** under "Type." Select **CH1** under "Source," push the softkey of **CurA**, and rotate the lit dial to measure the zero crossing of the input waveform. Select **CH2** under "Source," push the softkey of **CurB**, and rotate the lit dial to measure the zero crossing of the output waveform. Record the value of ΔX in a table. Repeat the measurement procedure for the remaining frequencies given in the lab manual. Adjust time and voltage scales as necessary. Compute the phase shift.

	$V_{p-p}(\text{out})/V_{p-p}(\text{in})$	ΔX (seconds)	Phase shift ψ_d (radians)
$f = 50$ Hz			
$f = 200$ Hz			
$f = 500$ Hz			
$f = 1$ kHz			
$f = 5$ kHz			

Checklist for Post-Lab Assignment:

You should have MATLAB plots for each of 2.2, 2.3 and 2.4; be sure to use the correct axis command in each case. You should have a table for 2.5 that compares the theoretical and measured values of $|H(j2\pi f_c)|$ and $\angle H(j2\pi f_c)$ at the frequency points collected in the lab.

Note: If you do not recall how to read a spreadsheet with MATLAB, type "`help xlsread`" at a command prompt in MATLAB.