Lab 6: Transient Response of a 1st Order RC Circuit

Theory and Introduction

Goals for Lab 6 - The goal of this lab is to explore the transient behavior of a 1st order circuit. To do so, you will build a circuit that will cause an LED to flash at a frequency of your choosing.

Theory

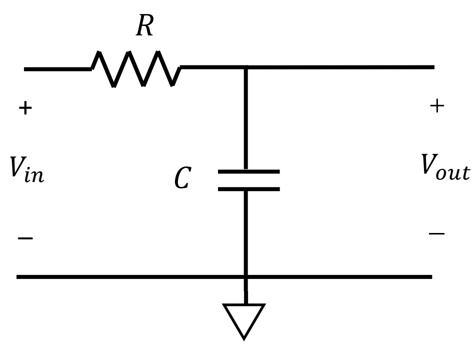


Figure 6.1 – A 1st Order RC Circuit.

In the RC circuit shown in Figure 6.1, a resistor and a capacitor form a first order circuit. Using the techniques described in class and in the textbook, the differential equation that describes this circuit is found to be

$$\frac{dV_{out}}{dt} + \frac{V_{out}}{RC} = \frac{V_{in}}{RC}.$$

Assuming that V_{in} is a constant, and that initially the output voltage is $V_{out}(0) = V_o$, the solution to this differential equation is

$$V_{out}(t) = V_{in} + (V_o - V_{in})e^{-t/RC}.$$

The voltage transitions from a starting value of V_o to a final value of V_{in} in an exponential fashion. The graph of such a behavior is shown in Figure 6.2. The product, $\tau = RC$, which is given in units of seconds, is called the *time constant* of the circuit and determines the rate at which the output voltage transitions from its starting to ending values. Note that by time, $t = 5\tau$ (five time constants), the output voltage has very nearly reached its final value.

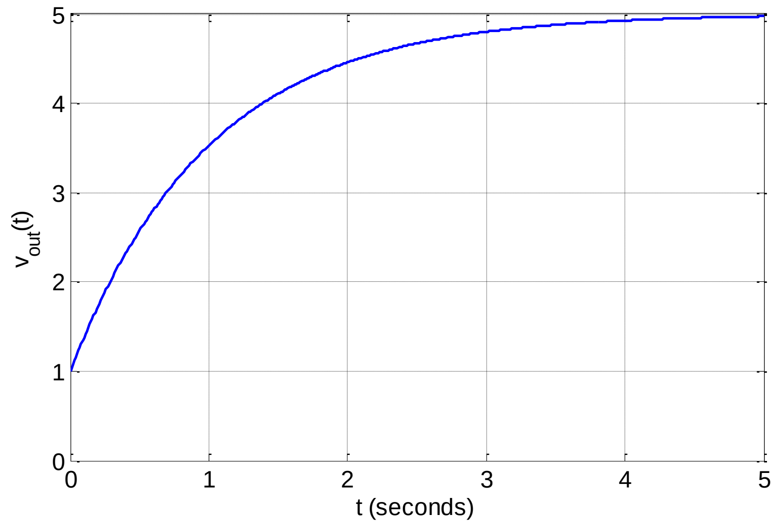


Figure 6.2 – Transient response of a 1st Order RC Circuit; $V_o = 1, V_{in} = 5, \tau = 1.$

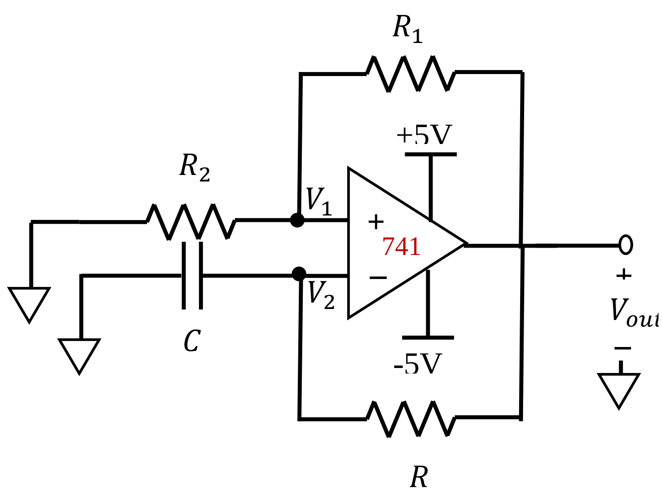


Figure 6.3 – An Op-amp based oscillating circuit

In this lab, we are going to use the circuit shown in Figure 6.3. The analysis of this circuit proceeds as follows. First, using the fact that the current flowing into the non-inverting terminal must be zero, it follows that the voltages V_1 and V_{out} must be related by a voltage divider,

$$V_1 = V_{out}\frac{R_2}{R_1 + R_2}.$$

To simplify future notation, let γ be the voltage divider, $\gamma = \frac{R_2}{R_1+R_2}$, so that $V_1 = \gamma V_{out}$. Next, using the fact that the current flowing into the inverting terminal must also be zero, the voltages V_2 and V_{out} are found to be related by

$$\frac{dV_2}{dt} + \frac{V_2}{RC} = \frac{V_{out}}{RC}.$$

Note that V_1 and V_2 are related to V_{out} by two very different equations. As a result, we would expect that $V_1 \neq V_2$. If the two inputs to an op-amp are not equal, the op-amp will saturate. So, at this point it should be pretty clear that this op-amp circuit is operating in saturation mode. When $V_2 < V_1$, the output of the op-amp will be $V_{out} = +V_{sat}$, and when $V_2 > V_1$, we get $V_{out} = -V_{sat}$.

Case 1: Consider first the case when $V_2 < V_1$. The voltage V_1 will be constant and equal to $V_1 = \gamma V_{sat}$. The voltage V_2 will be given by the solution to the differential equation,

$$\frac{dV_2}{dt} + \frac{V_2}{RC} = \frac{V_{sat}}{RC}.$$

Depending on the initial condition, this solution will be of the form

$$V_2(t) = V_{sat} + (V_2(0) - V_{sat})e^{-t/RC}.$$

This behavior will look like that in Figure 6.2 where it approaches a final value of $V_2(t) \to V_{sat}$. However, since V_1 is smaller than V_{sat}, at some point $V_2(t)$ will grow to the point where it becomes larger than V_1. At that point we switch to the other case.

Case 2: When $V_2 > V_1$ we now get $V_1 = -\gamma V_{sat}$ and V_2 will be of the form

$$V_2(t) = -V_{sat} + (V_2(0) + V_{sat})e^{-t/RC}.$$

Now $V_2(t)$ will decrease over time and approach a final value of $V_2(t) \to -V_{sat}$. Again, since V_1 is now larger than $-V_{sat}$, at some point $V_2(t)$ will decrease to the point where it becomes smaller than V_1. At that point we switch back to the first case.

So this circuit is going to switch back and forth (oscillate) between the two cases described above. The voltages involved are all periodic and will behave as shown in Figure 6.4. If we take time $t = 0$ to be the beginning of a period as shown in Figure 6.4, the starting voltage for V_2 will be

$V_2(0) = -\gamma V_{sat}$. The first switching time will occur when $V_2(t) = \gamma V_{sat}$. We can determine that time by solving for t in the equation

$$V_2(t) = V_{sat} + (-\gamma V_{sat} - V_{sat})e^{-t/RC} = \gamma V_{sat}.$$

After a little algebra, this is found to be

$$t = -\tau \ln\left(\frac{1-\gamma}{1+\gamma}\right).$$

This time represents ½ of a period of each of the periodic waveforms. The resulting frequency (1/period) is then

$$f_o = -\frac{1}{2\tau \ln\left(\frac{1-\gamma}{1+\gamma}\right)}.$$

For example, for the values given in Figure 6.4, we would expect the frequency to be $f_o =$ 0.621Hz and therefore the period is 1.601sec.

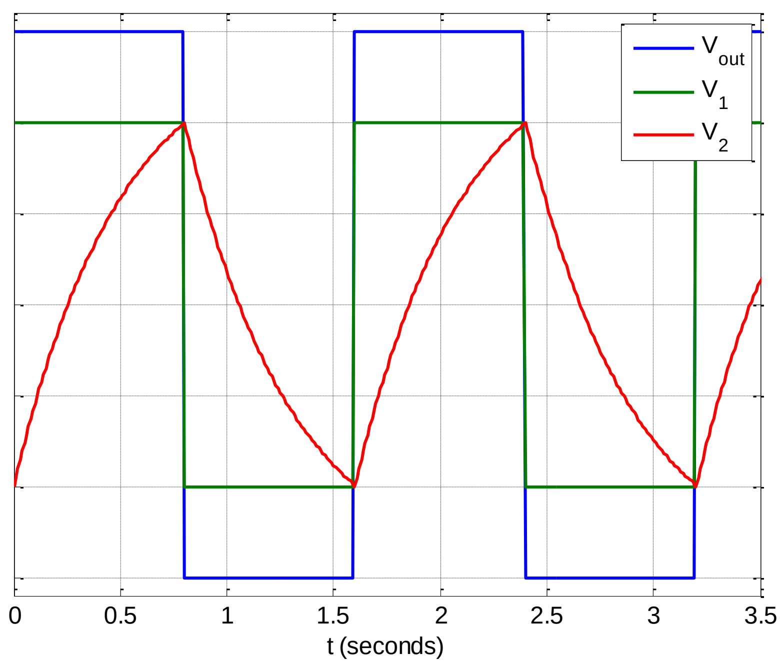


Figure 6.4 – Waveforms in the oscillating op-amp circuit; $\gamma = \frac{2}{3}, \tau = 0.5sec.$

Prelab

A. Design the circuit of Figure 6.3 to oscillate at a frequency as near to $f_o = 0.5$ Hz as you can get it. Choose component values for R, C, R_1, R_2 that are available to you in your lab kit. Also, try to choose the values of R_1 and R_2 so they sum to a value as close as possible to $10k\Omega$. Note, the resistors (or the capacitor) do not necessarily have to be a single component. You can use parallel/series combinations to get the component values closer to where you would like them. **Specify your chosen component values and then calculate your expected values for the following quantities related to the voltage $\boldsymbol{V_2(t)}$:**

a. **Actual frequency of oscillation.** This should be near 0.5Hz but probably not exactly 0.5Hz.

b. **Peak-to-peak voltage.** For this, you will need to know the saturation voltage of the op-amp. Use your measured value from Lab 4.

c. **Root-Mean-Square (RMS) Voltage.** For a periodic signal $x(t)$ with period T_o, the RMS value is given by

$$X_{rms} = \sqrt{\frac{1}{T_o}\int_{T_o} x^2(t)dt},$$

where the integral is taken over any interval of length T_o.

Make sure to show all of your work.

Note: It is very unlikely that your design will end up exactly the same as somebody else's. Please do your own work.

B. Simulate your design using SPICE**. Turn in the transient waveforms at $\boldsymbol{V_1}$, $\boldsymbol{V_2}$, and $\boldsymbol{V_{out}}$.** Consult your TA if you need help setting up the transient response in SPICE. A few pointers are listed below.

Transient Simulation in LTSPICE

Once you have created your schematic diagram of the circuit in SPICE, it is fairly straightforward to run a transient simulation. When you click on the "running man" an "Edit Simulation Command" box will pop up. Choose the "Transient" tab. In this case, the only parameter you should need to enter is the “Stop time.” Since you will be creating periodic waveforms with a period of 2 seconds and you will want to observe a few periods, you probably want to choose a stop time of something like 6-10 seconds. Then click OK. A window with an empty plot will open next to your schematic. This is where your waveforms will appear. You will need to tell SPICE where you want to measure a voltage. To do so, hover the mouse over your schematic and you will see a probe icon. Click on the point where you want to measure voltage and the waveform will appear in the plot window. You will probably see a waveform that is zero for all time. This is because SPICE is assuming there are no initial conditions (all voltages are zero initially). With no voltage everywhere at time zero, nothing will ever happen in this circuit. You will need to specify a non-zero initial condition. To do this, choose “SPICE directive” from the “Edit” menu. An “Edit Text on Schematic:” box will pop up. In the text window, enter something like “.ic V(n001)=1” This is telling SPICE to use an initial condition of 1V at node 001. You may want to use a different node number (or different initial voltage) depending on how your nodes are labelled. When you click OK you will want to left click in the schematic window to place the text SPICE directive in your schematic. You should see the text you entered on your schematic now. Run the simulation again with the new initial condition. Hopefully you are starting to see something like the waveforms in Figure 6.4.

Note: When you put your schematic together in SPICE, please make sure that you have hooked up the correct components to the positive and negative op-amp input terminals. For some reasons, students frequently hook things up backward and can’t figure out why the waveforms don’t look like they are supposed to.

Procedure

Equipment and parts needed

A selection of ¼-W resistors
A selection of capacitors
1 – Red LED
1 – Green LED
1 – 10kΩ potentiometer
1– potentiometer turner (a small slotted screwdriver should be suitable for this purpose)
1 – 741 OPAMP

Task 1 – Flashing LED Circuit.

Build the circuit shown in Figure 6.5. Here we have added some LEDs to the output of the circuit you studied in the prelab. Use the component values you designed in the prelab. If you have chosen your component values correctly, the two LEDs should flash on and off alternately about once every two seconds.

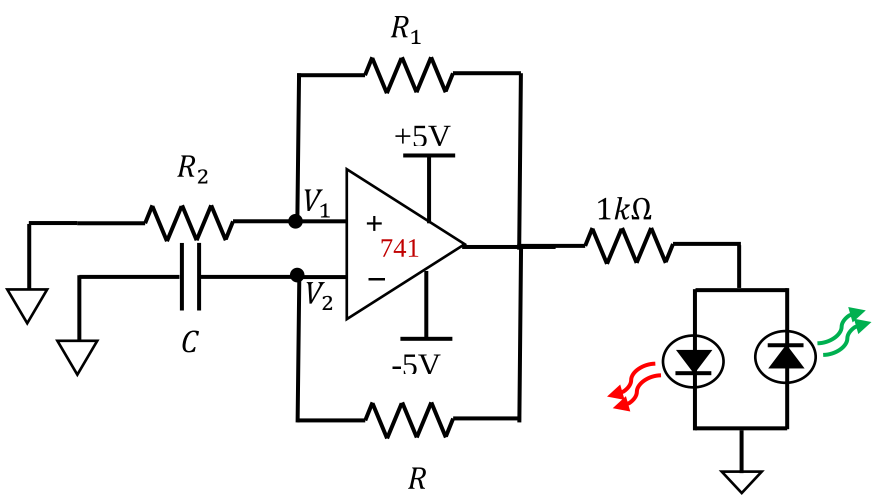


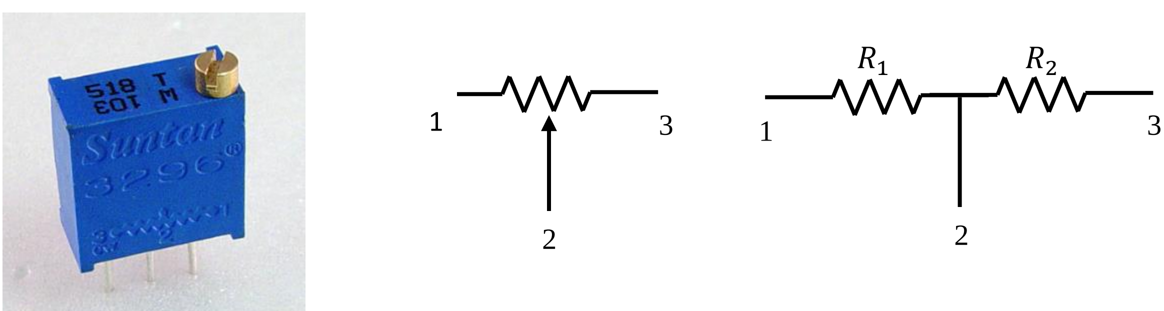
Figure 6.5 – A Flashing LED Circuit

Use the bench oscilloscope or the one on your AD2 to display both $V_1(t)$ and $V_2(t)$. Adjust your vertical scale so that the waveforms occupy about 70% (as close as you can) of the full scale and adjust the horizontal (time) scale so that approximately two periods of the waveforms are shown. If you need help learning how to use the oscilloscope, do not hesitate to ask your TA.

A. Once you have the waveforms displayed the way you like, save a screenshot of the waveforms for use in your lab report.
B. Use the scope to measure each of the following quantities for both $V_1(t)$ and $V_2(t)$:
   a. Actual frequency of oscillation.
   b. Peak-to-peak voltage.
   c. Root-Mean-Square (RMS) Voltage.
C. In your lab report, compare the measured values with the design values from your prelab. Be sure to point out any differences and give an explanation for the major cause(s) of any differences you see. To give an intelligent explanation here, you might want to measure the actual component values and the actual saturation voltage of your op-amp.

Task 2 – Potentiometers

In order to make it easy to adjust the parameters of the waveforms created by the circuit in Figure 6.5, you will now replace the two resistors, R_1 and R_2, that formed the voltage divider with a potentiometer (also known as a variable resistor or "pot" for short). Figure 6.6(a) shows a picture of typical potentiometer. You can envision the potentiometer as a resistance laid out in a linear fashion with a slider that can be moved anywhere within the length of the resistive material (see Figure 6.6(b)). The slider essentially splits the resistance into two parts as shown in Figure 6.6(c). The two resistances R_1 and R_2 must sum to a constant value which is usually printed on the package. For example, a $10k\Omega$ pot must have $R_1 + R_2 = 10k\Omega$. By connecting all three terminals up to a circuit, the potentiometer basically functions as a variable voltage divider. You can also use the potentiometer as a variable resistor by leaving terminal 3 (or terminal 1) disconnected.



(a) (b) (c)

Figure 6.6 – (a) A typical potentiometer, (b) its schematic, and (c) an equivalent circuit.

A. Replace the resistors R_1 and R_2 from the circuit in Task 1 with a $10k\Omega$ potentiometer. Your circuit should now look like that shown in Figure 6.7. Adjust the potentiometer until the circuit oscillates at a frequency of 1Hz (You can check the frequency of oscillation on the oscilloscope). Remove the pot from the circuit and measure the values of R_1 (between terminal 1 & 2 of the potentiometer) and R_2 (between terminal 2 & 3 of the potentiometer). What is the corresponding value of the voltage division ratio, γ? Note: If you are using the DMM in the lab, it is straightforward to measure the resistances. If you are using the AD2 you may just want to directly measure the voltage division ratio. Put the pot back in the circuit. Record the waveforms you see on the scope to include in your lab report.

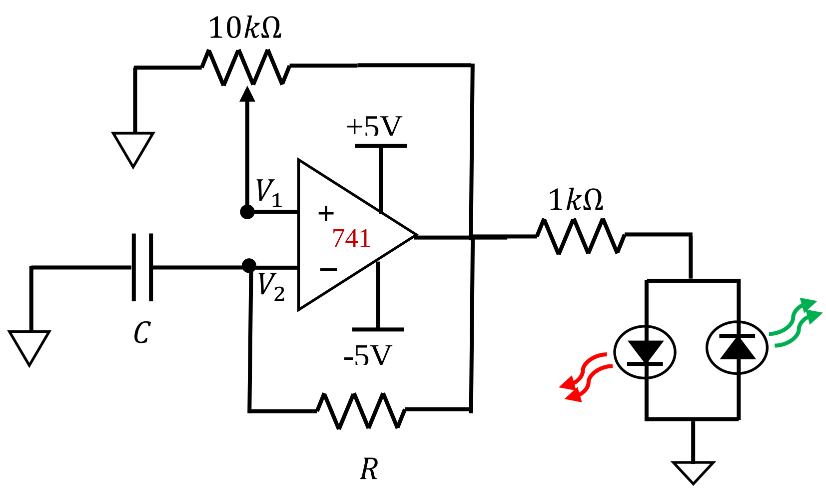


Figure 6.7 – Modified Flashing LED Circuit for Task 2.

B. Replace the resistor R with one at half its value. You can easily do this by placing another identical resistor in parallel. Measure and record the new frequency of oscillation. Record the waveforms you see on the scope to include in your lab report.

C. Again, adjust the potentiometer until the circuit oscillates at 1Hz. As in part A, remove the pot from the circuit and measure the voltage division ratio. Record this value to include in your lab report. Record the waveforms you see on the scope to include in your lab report.

Before you leave the lab…

Your TA will show you how to use the bench oscilloscope to make some of the same measurements you did with the scope on your AD2. You should bring your data sheet to be signed by the TA and you should also be prepared to show your TA how you made your various measurements.

Lab Report

1. Title page, as always
2. Report write up:
    A. Procedure – Write in your own words what you did in lab
    B. Measured Data – Include tables of values you measured throughout the lab.
    C. Measured Waveforms - Be sure that you label your screen shots and intersperse comments and screenshots through your document.
    D. Sample Calculations –Demonstrate how you performed any calculations needed.
    E. Discussion –Be sure to answer any questions posted in the procedure section. Also, discuss differences between theoretical values calculated in the prelab and actual values measured in the lab. Definitely comment on what you think are the major sources of any significant differences.