# Exponential Waveforms

## Purpose:  To measure the step response of circuits containing dynamic elements such as capacitors.

## Equipment Required:
• 1 - HP 54xxx Oscilloscope
• 1 - HP 33120A Function Generator
• 1 - Protoboard
• 1 - Capacitance Meter (This is available in the instrument room)
• 1 - 0.1-µF capacitor
• 1 - 1.1-k Ω resistor
• 1 - 0.47-µF capacitor

## A. Prelab:
**1.  Reading**
Read Sections 7-1 to 7-3 in the text. *

**2. Electronics  Workbench  Simulation**
Assume the output resistance of the function generator ($R_{TH}$ in Fig. 1) is 50 Ω . Do an EWB transient analysis of the circuit of Figure 1. Use an 8 V PP square wave source with no DC offset. Note that there is a square wave source in the passive parts bin which is identical to the one shown in Figure 1; you do not need to use the function generator in EWB. You will need to determine an appropriate source frequency so that the voltage across the capacitor reaches its final value before the source changes to the next voltage level. The results of your computer circuit analysis should look similar to Figure 2.  At low frequencies (i.e., 50Hz) the capacitor voltage waveform will look essentially like the input — a square wave.  At higher frequencies the waveform will look like that shown in Figure 3. Finally, as the frequency increases still higher the waveform will look like that shown in Figure 2.  Print out the results and include them in your lab report.  See Appendix I for an explanation of how to perform a transient analysis in EWB.
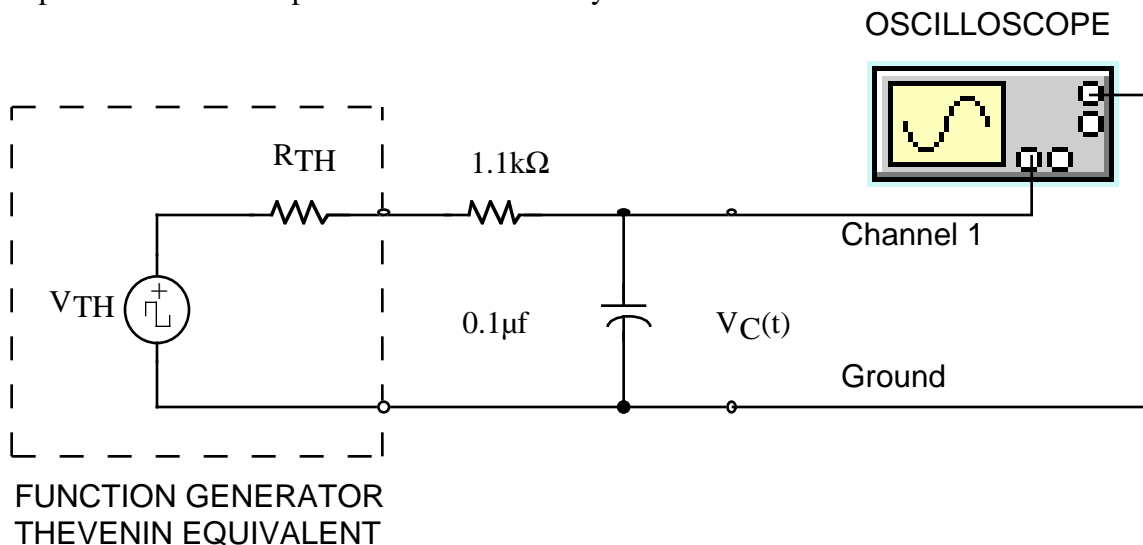


**Figure  1.  Driven  RC  Circuit**

## 3. Background

The equation describing the output $v_C(t)$ of the circuit of Figure 1 is of the form:

$$v_C(t) = (IC - FC)e^{-\frac{t}{T_C}} + FC \qquad (1)$$

where IC is the initial value (initial condition) of the voltage, FC is the final value (final condition) of the voltage, and $T_C$ is the circuit time constant. Assume the function generator produces a square wave with a peak-to-peak voltage of 8 V, no DC offset ($V_{avg} = 0V$), and a period $T_O$ greater than $10 T_C$. Under these circumstances the initial condition IC and the final condition FC will be equal to the maximum and minimum voltages of the source. In this case, the assignment of +4 V or –4 V to the initial condition IC depends on whether the charging cycle or the discharging cycle are being observed, as shown in Figure 2. For the discharging cycle the initial condition IC = +4 V and the final condition FC = -4 V. The opposite is true for the charging cycle, IC = -4 V and FC = +4 V.



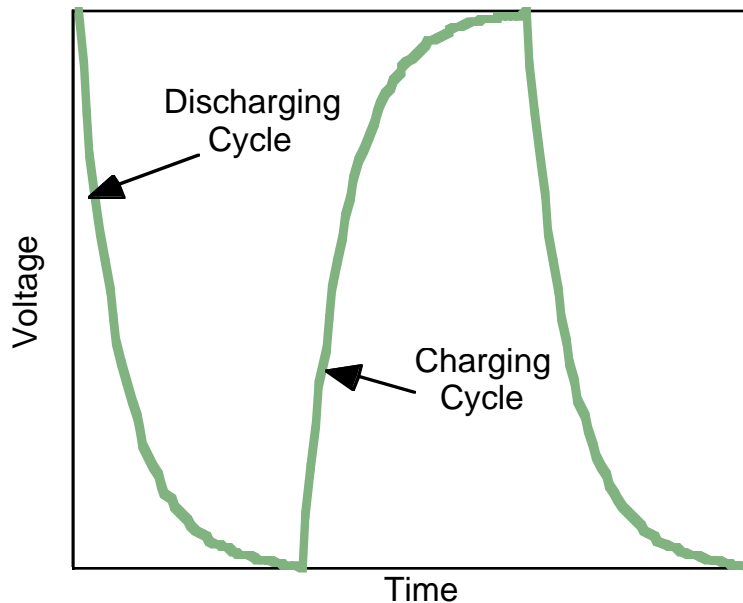**Figure 2. RC Charge/Discharge Cycle**

## B. Lab Procedure:

Determine the output resistance of the laboratory function generator, $R_{TH}$. This value can be determined from the HP instrument specifications available on the ENGR 210 Web page. If you cannot find this data you may assume it is 50Ω but should state that assumption in Data Table 1. Record $R_{TH}$ and your data source in Data Table 1. The Thévenin equivalent model of the function generator is shown in Figure 1.

1. Build the circuit shown in Figure 1 on your protoboard. **Before constructing the circuit** measure and record the EXACT value of your resistor and capacitor. Use the DMM to measure the value of your resistor. Record this value in Data Table 2. Use the digital capacitance meter available in the instrument room to measure the capacitance of the capacitor you use in your circuit. Also record this value in Data Table 2.

2. Function generator

Set the function generator to produce a square wave, with a frequency of 50 Hz, no DC offset and a peak-to-peak voltage of 8 V. Be sure that the output of the function generator is set to high impedance when you set the output of the function generator.

3. Oscilloscope

   a. Set the oscilloscope to display the waveform $v_c(t)$ produced by this circuit. The display should be similar to a square wave but with slightly rounded edges as shown in Figure 3.
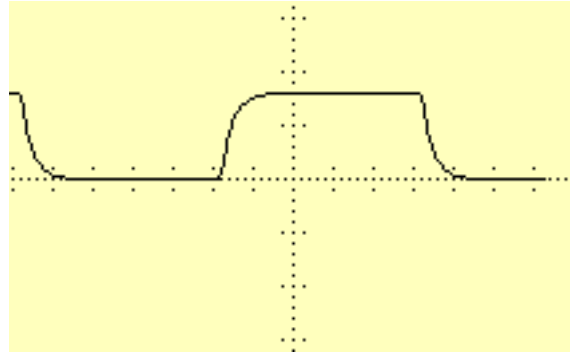


**Figure 3. Oscilloscope display of RC response to square wave.**

   b. Adjust the scope's horizontal and vertical position knobs and the trigger slope and level so that the waveform is completely visible, but fills the scope face. Line up the start of the exponential waveform with the vertical grid line on the left edge of the scope face. Set the horizontal scale (time base) so the trace looks something like Figure 4. It is not necessary that you be able to see 5 time constants; but it is good practice to expand the display so that at least 50% of the scope face is used for measurements (as shown in Figure 4).
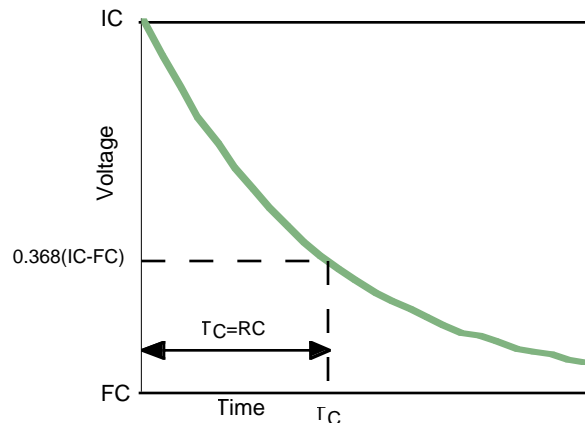


**Figure 4. Oscilloscope display of exponential decay waveform**

   c. <u>Method 1 of MEASURING the time constant</u> $T_C$ — definition: This method uses no calculations and uses the definition of the time constant to measure it using the just as you did in Lab 6. Begin by adjusting the voltage cursor V1 to 0.368 of the voltage difference (FC-IC). Set the first time cursor t1 to the beginning of the decay, i.e. the initial value of the waveform at the left side of the display. Then adjust the second time cursor t2 to correspond to where the V1 cursor and the decaying waveform intersect. The value of deltat displayed on the scope screen is the time constant $T_C$. Record this value in Data Table 3.

d.  <u>Method 2 of MEASURING the time constant</u> $T_C$ — curve fitting: **Use Benchlink Scope as described in Appendix 2 to make a printout or bitmap file of the oscilloscope display.** This display will be analyzed as part of your lab writeup. Because you will use this display to determine $T_C$ it is very important that you adjust the voltage sensitivity to get a display similar to that shown in Figure 4. You can use the oscilloscope's vertical positioning adjustment to accurately place the waveform's start at the left top of the screen.

3.  Charging cycle
    Set up the scope to display the charge cycle, i.e. the part of the waveform when the voltage is increasing. Determine $T_C$ from this waveform using the cursors. Remember that the deltat measured by the oscilloscope MUST correspond to the period in which the vertical voltage increases to 0.632 of its final value. Record this value of deltat in Data Table 4. **Use Benchlink to make a printout of this oscilloscope waveform.**

4.  Capacitors in series and parallel.
    For these measurements you can use either a charge or discharge cycle.
    a.  Put a 0.47μF capacitor in parallel with the 0.1μF in your circuit. The combination of these capacitors is $C_{eff}$. Using the cursor based method measure the new $T_C$ caused by this combination of capacitors. Record this value of $T_C$ in Data Table 5. **Use Benchlink to record the waveform you used to make your measurements.**
    b.  Place the 0.47 μF and 0.1 μF capacitors in series. Using the scope cursors determine the value of $T_C$ and record it in Data Table 5. **Use Benchlink to record the waveform you used to make your measurements.**

# DATA AND REPORT SHEET FOR LAB 7

Student Name (Print): _____ Student ID: _____

Student Signature: _____ Date: _____

Student Name (Print): _____ Student ID: _____

Student Signature: _____ Date: _____

Student Name (Print): _____ Student ID: _____

Student Signature: _____ Date: _____

Lab Group: _____

**Data Table 1. Function Generator Characteristics**
Generator Impedance:          _____          $\Omega$
Source of Information:          _____
_____

**Data Table 2. Actual value of circuit components**

|   | Measured value |
|---|---|
| R |   |
| C |   |

**Data Table 3. Cursor Measurements (Discharge cycle)**

Measured $T_C$ [deltat from the oscilloscope]    _____    milliseconds

**Data Table 4. Cursor Measurements (Charging cycle)**

Measured $T_C$ [deltat from the oscilloscope]    _____    milliseconds

**Data Table 5. Parallel $C_{eff}$ measurement**

|   | Measured $T_C$ [deltat from the oscilloscope] |
|---|---|
| Parallel $C_{eff}$ | milliseconds |
| Series $C_{eff}$ | milliseconds |

**Data analysis:**

1. Include an EWB Simulation of the circuit of Figure 1.

2. Include the Benchlink® Scope prinout of your actual circuit performance from part 3(d).

3. From your Benchlink® Scope printout in 2. estimate the actual $T_C$ of the circuit by the following two methods. Show all calculations as well as any measured points on your plot and their time/voltage coordinates.

   a. Determine (IC - FC) from your printout. This value of (IC -FC) should be the peak-to-peak voltage of the source and should be near 8 V. Locate the voltage 0.368*(8 V) = 2.94 V on your printout. This should typically be about one third of the way up from the bottom of the oscilloscope display (as shown in Figure 4). This voltage level is significant because one time constant has elapsed when the exponential has decayed to 36.8% of the difference between the initial voltage and the final voltage. Determine this time $T_C$ from your plot.
   b. Equation 1 can be used to solve for the time constant $T_C$ by measuring the voltage and time coordinates of the waveform at two points, as shown in Figure 5.
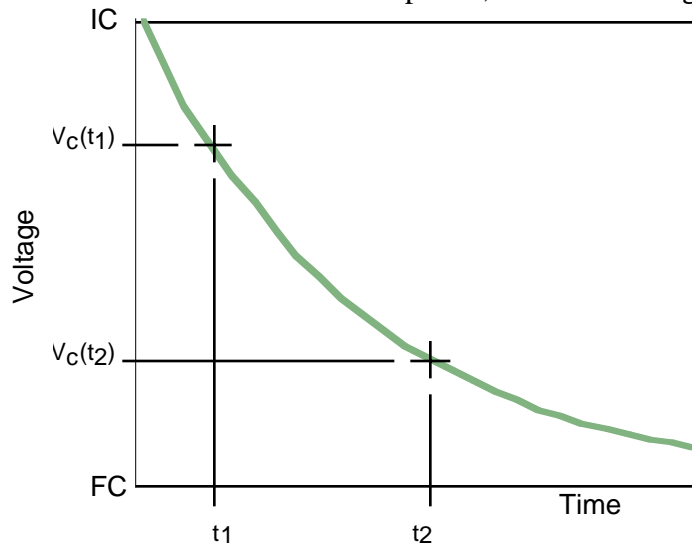


**Figure 5. Measuring $T_C$ by curve fitting to two points**

   c. Moving the last term in Equation 1 to the left of the equal sign and dividing these two measurements gives

$$\frac{v_C(t_1) - FC}{v_C(t_2) - FC} = \frac{(IC - FC)e^{-\frac{t_1}{T_C}}}{(IC - FC)e^{-\frac{t_2}{T_C}}} = e^{\frac{t_2 - t_1}{T_C}} \qquad (2)$$

Solving for the time constant $T_C$ is accomplished by taking the natural logarithm of both sides and rearranging to produce:

$$T_C = \frac{t_2 - t_1}{\ln\left(\frac{v_C(t_1) - FC}{v_C(t_2) - FC}\right)} \qquad (3)$$

Solve for the time constant $T_C$ using Equation (3).

4. In the textbook, we have seen that the time constant is given by the equation $T_C = RC$. Use this equation, either value of $T_C$ computed in Question 3, and the actual value of your resistor plus the actual output resistance of the function generator to compute the capacitance, C. Compute the percentage error between the capacitance value you computed and the capacitance measured with the capacitance meter. The nominal capacitance value is $0.1\mu F \pm 20\%$.

5. Include your Benchlink waveform from Part 3.

6. Include your Benchlink waveforms from Part 4.


**Questions:**
1. Does your value for $T_C$ in Data Table 5 agree with the formula for capacitors in parallel? Show why or why not.

2. Does your value for $T_C$ in Data Table 5 agree with the formula for capacitors in series? Show why or why not.

3. Which method of computing the time constant in (See Data Analysis, 3(a) and 3(b)) do you feel is the more accurate? What are the sources of error?

4. How did your calculation of C from the measured time constant (See Data Analysis, 4) compare with the value measured with the capacitance meter?

5. Are the measured time constants for the charge (Data Table 3) and the discharge (Data Table 4) cycles the same?

*Reference: Roland E. Thomas and Albert J. Rosa, The Analysis and Design of Linear Circuits 2/e, Prentice Hall, (New Jersey, 1998)

# Appendix 1
# Transient Analysis Using EWB

## Example — transient analysis of a series RLC circuit using EWB

### a) Using EWB
Electronics Workbench is on the network and can be run from any computer directly connected to CWRUnet. To run Electronics Workbench from the software library using any non-circuits lab machine:
1. Open up Network Neighborhood.
2. Double-click on Entire Network (On NT machines go into the Network or Compatible Network selection).
3. Find the computer called 'software-lib' and double click on it.  If you are prompted to login type "guest" for user name and leave the password blank.
4. Select (click once on) the folder called 'vol1'.  Electronics Workbench can be found in Library/Windows95-NT Shortcuts/Departments/ElectricalEngineering
5. Under the File menu, choose 'Map Network Drive'.
6. In the dialog box that appears, choose S for the drive and make sure the Reconnect at Logon box is checked so that you don't have to go through this process again.
7. Hit OK.

### Notes:
1. The most current version of Keyaccess (version 5.0) must be running to access the program.
2. There is no need to copy any files from the software library to one's own computer. The program can only be run directly from the software library.
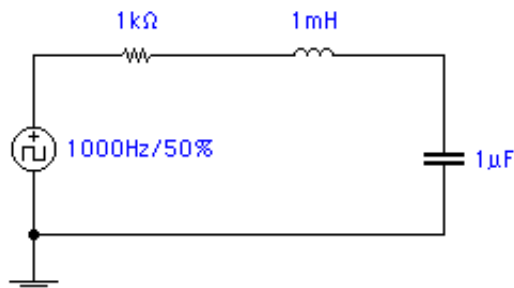


**Figure 6.** Finished Series RLC Circuit

### b) Building the Circuit
To build the circuit shown in Figure 6 you must place a pulse source (note that EWB has many different kinds of sources and you will need to choose the correct one), a resistor, an inductor, a capacitor, and a ground on the workspace. Complete the circuit as shown in Figure 7.  Note that there is a connector between the ground and the pulse signal source. You can drag a connector from the parts bin over the wire between the ground and signal source.  Or, if you drag a wire from the bottom of the capacitor over the wire between the ground and signal source you will see a dot appear.  EWB automatically creates a connector when you place the end of a wire over another wire.  To make this connector simply release the mouse button. However you do it, you should get the circuit shown in Figure 7.  Note that a connector was added between the inductor and capacitor and assigned the label Vo.
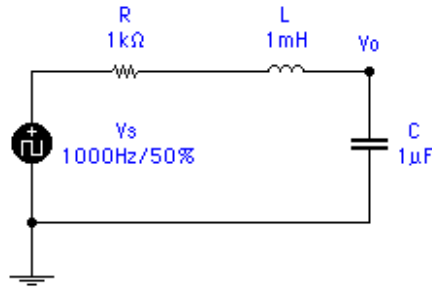
**Figure 7.** Labeled Circuit

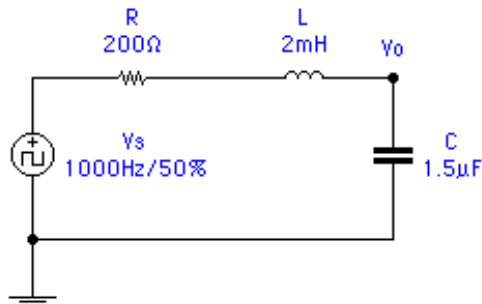Now change the values in the circuit of Figure 7 to those in Figure 8.



**Figure 8.** Labeled Circuit with Final Values

### c) Attaching instruments

The EWB oscilloscope has the connections shown in Figure 9. The channel A and channel B inputs are at the bottom of the icon. The ground is at the upper right. This ground connection should be connected to the ground of your simulated circuit otherwise you may get erroneous measurements. There is also an external trigger input connection immediately below the oscilloscope ground connection but you will rarely use that connection.
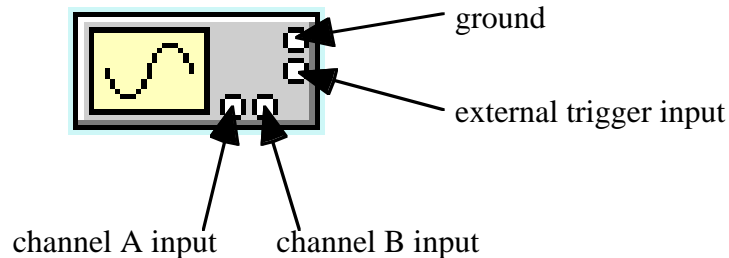


**Figure 9. EWB oscilloscope connections**

You will want to add an oscilloscope to the circuit of Figure 8 to measure the input and output waveforms. Click on the oscilloscope icon on the instrument shelf, and drag it to your circuit. Note that the icon will disappear from the instrument shelf as you can only use one oscilloscope in your circuit. Insert a connector between VS and R and connect Channel A of your oscilloscope to it. Connect Channel B of the oscilloscope to Vo. Finally, connect a ground to the top right hand connector as shown in Figure 10. You can select and move the oscilloscope the same way you select or move a component such as a resistor.
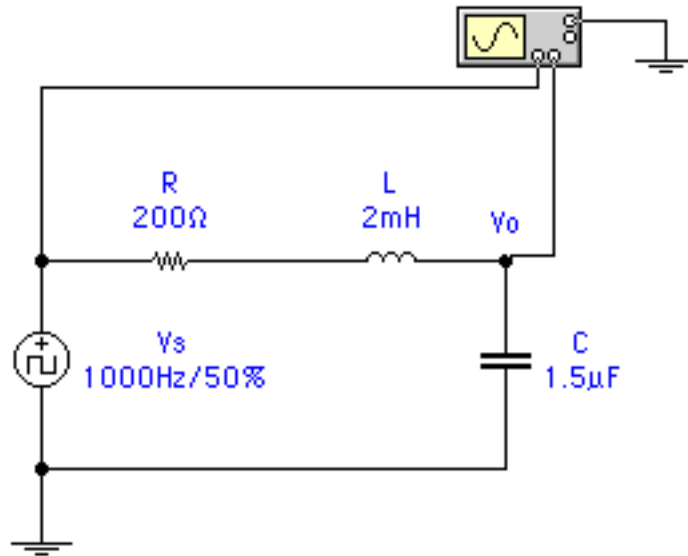
**Figure 10.** RLC circuit with attached oscilloscope

**d) Using the EWB oscilloscope**
To control and use an instrument such as an oscilloscope in EWB you must access its instrument panel. The easiest way to do this is to simply double-click on the oscilloscope icon. You can also select the oscilloscope icon and choose **Zoom** from the **Circuit** menu. Select the various instrument options by clicking the appropriate buttons on the instrument panel to change values or units.
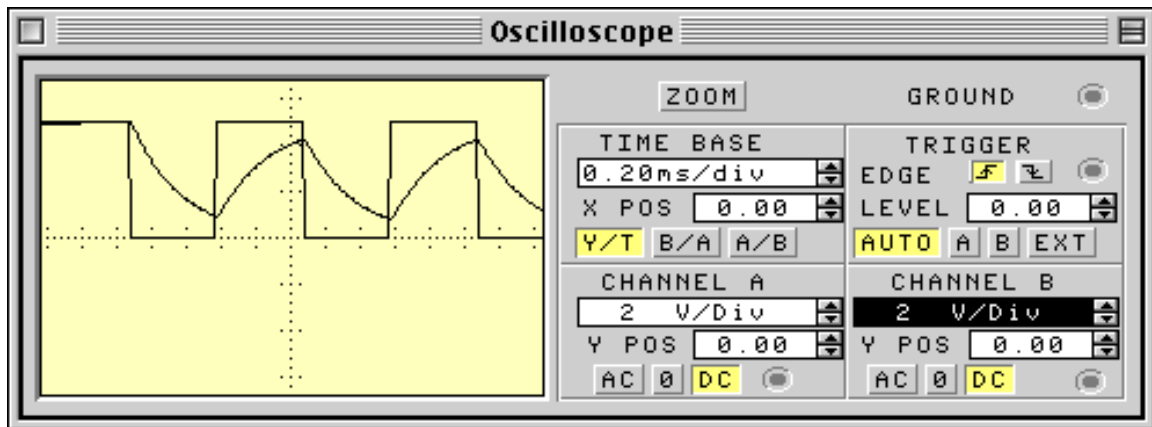


**Figure 11.** Oscilloscope Control Panel

For transient measurements you will typically want both Channel A and B inputs set to DC. The Trigger set to AUTO and the Time Base set to Y/T. This last setting shows the input voltages as a function of time. Unlike a real oscilloscope the EWB scope has a ZOOM button which will change the oscilloscope into a larger, more detailed display. This display allows you to scroll the waveform backwards in time to see any events you missed. It also has two cursors which allow you to make accurate time measurements from the oscilloscope waveform.

**e) Circuit simulation**
Clicking the power switch activates or begins the EWB simulation; however, to understand what your oscilloscope and other instruments will be displaying you have to understand what the SPICE engine is computing. A **Transient Analysis** in EWB starts with the circuit's initial conditions and computes the time dependent response of the circuit. To do a transient analysis you must open the **Analysis Options** dialog box from the **Circuit** menu. Select **Transient Analysis** in the **Analysis Options** dialog box. The oscilloscope will then display the transient response of the circuit when the power switch is turned on. EWB computes the initial conditions for the circuit from its DC operating point.

There are several EWB options which you may find useful for Transient Analysis:
b. Often a Transient Analysis results in many consecutive oscilloscope screens of waveforms which are updated too quickly for you to watch the circuit behavior. If this happens you can open the **Analysis Options** menu and select **Pause after each screen** by placing an **X** in the box to the right. This will cause the simulation to pause every time the oscilloscope display is full. You can then examine the oscilloscope display at your leisure. You can then go to the **Circuit** menu and choose **Resume** which will cause the simulation to continue until the oscilloscope screen is full again.
c. The Tolerance setting in the **Analysis Options** dialog box controls the accuracy of the simulation results. The default setting is 1 %, which gives you the fastest simulation speed but the lowest level of accuracy. To change the level of accuracy, click **Tolerance**. Reducing the tolerance (i.e., to 10%) decreases the time needed to simulate a circuit.
d. If your trace on the oscilloscope looks ragged you can change the number of calculated points for the display by selecting **Analysis Options** and changing the **Time domain points per cycle**. Typically this is 100 but increasing it to 400 or more will improve the quality of the displayed waveform.

**f) RLC transient response**
You can see the waveforms of the input and output voltages on the oscilloscope, as in Figure 12. These waveforms are shown with the inputs set to DC (a requirement for most transient analysis), the time base set to 0.2 ms/div, and the channels set to 5 V/div. The Vs input wire was set to red and the Vo input wire to green to get the oscilloscope displays shown. Note that these are for the circuit of Figure 10 with the pulse generator output set to 10 volts. You can set the color of a wire by double clicking on it to bring up a color selection palette.
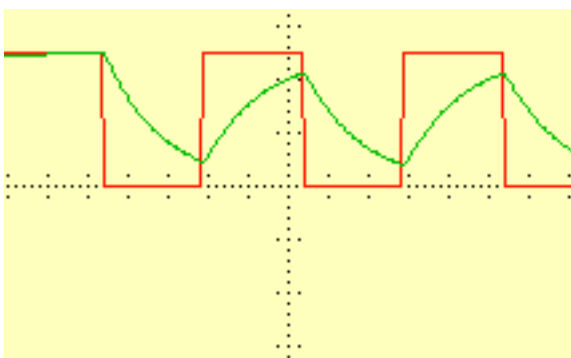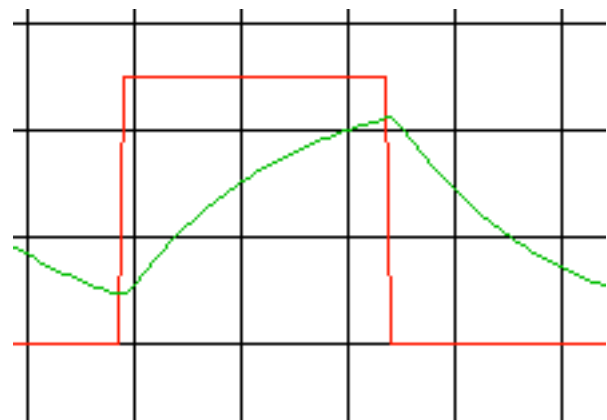


(a) Input and output waveforms                 (b) Input and output waveforms zoomed
**Figure 12.** Transient analysis of series RLC circuit

# Appendix 2
# Using Benchlink® Scope to print HP scope waveforms

The Benchlink® Scope software uses the HPIB board inside the PC to read the image that the oscilloscope displays on its screen. This picture file is then displayed on the PC and can be copied, printed and saved as a bit map for use in any program. This software only works with HP 54xxx series scopes like we have in the laboratory.

To use Benchlink® Scope software to capture a waveform.
8. On the PC, from the Programs menu, go to Benchlink® suite and choose the Benchlink® Scope program.
9. As Benchlink® opens up it will search for the oscilloscope and (if successful) will display the window shown below:



10. If you get an error message make sure that the scope is turned on.
11. Next, click on the **Image** menu and choose **New....**



12. Click OK on the dialog box when you are ready to capture the scope image.



   e.
   f.
13. The oscilloscope image is then displayed on the screen as a bitmap. Press F7 to update this image as often as you wish. You can Print or Copy this image or save it as a file.