ECE 332 Lab 7d

RLC Circuit Design

This Laboratory Exercise is to be an individual effort. There should be NO collaboration with anyone except the course instructors or technicians. Each student must completely document any assistance received from others as well as any references used. If no help is received, state *Documentation: None.*

You have had enough lab experience, both with simulation and with use of the myDAQ virtual instruments on actual circuits, and you are now ready to undertake the design of a circuit. In this exercise, you will design an RLC circuit to meet certain specifications, prove the design through simulation in Multisim, and then build and test the design in the laboratory.

Goals

This lab continues your work on the simple RLC series circuit, building on what you have already learned about simulation of circuits and about laboratory measurement. The goal of this lab is to introduce you to the engineering design process as applied to circuit design. When you have completed this lab, you should be able to

- Apply the engineering design process of Figure 1.
- Design a series RLC circuit to meet certain step-response specifications.
- Simulate your design using Multisim.
- Build your circuit on a protoboard using the parts available and connect it to myDAQ.
- Test the performance of your design using the myDAQ oscilloscope to make measurements.
- Prepare a formal report on the performance of your design

Engineering Design

Figure 1 is one representation of the process of doing engineering design. It begins with the statement of a problem in the form of specifications for a particular system. Theory, generally in mathematical form, helps develop ideas of what is possible and how the system could actually work.

Once you have a foundation based on theory, you can design what you think will be the solution to the problem. To keep costs down and to save time, you simulate this design to see if it actually provides a solution to the problem. Notice that Design and Simulation are iterative processes. Trying the design via simulation, correcting and improving the design, then trying it again–you keep iterating until you have a good solution.

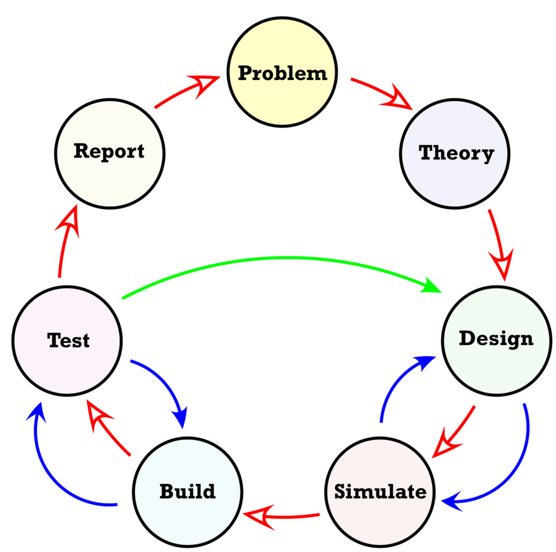


Figure 1: Engineering Design

Only when the simulation says the design has a good chance of working do you expend the time and money to build the actual circuit. Testing obviously has to prove the design. Again notice the iterative nature of Build and Test. This is done until the design actually does meet the specifications.

There's another path for iteration, from Test back to Design. This comes about when testing shows the design itself is faulty. This is an expensive step that you don't want to have to take. Good design work at the beginning helps prevent this.

Finally, your work is of no value until you report on what you have designed so that others, who perhaps will have to build and install 500 of your circuits in flight-line aircraft, can proceed.

To put all this into one statement, engineering design is not a hit-or-miss process but rather an orderly approach to the solution of a problem within the constraints of time and cost.

Academic Direction

- Work individually without collaboration. You *MAY NOT REFER TO ANY OTHER CADET'S WORK OR SEEK ASSISTANCE FROM ANY OTHER CADET.*
- You may only discuss the prelab with any DFEC instructor or technician.
- Complete the pre-lab assignment before coming to the actual lab session. You will use your pre-lab work to implement your design. Pre-lab instructions are later in this document.
- Complete this lab and turn in the lab report by the due date shown in the course syllabus. Instructions for the report are at the end of this document.
- This lab is to reinforce the material you cover in the classroom, so questions on this lab may appear in quizzes, graded reviews, and the final examination.
- You must complete the lab to pass this course.
- If you need to make up this lab, you must do it within 48 hours of the due date. Check with your instructor or e-mail our technician, Ms. Elmore at susan.elmoreusafa.edu to make arrangements for access.

Theory

We have already developed some of the theory surrounding the series RLC circuit (and in general of all linear second-order systems) in Lab 4s. You should look back at that lab as a review. The solution to the differential equation stated in that lab is the sum of the natural and the forced responses:

$$v_o(t) = v_N(t) + v_F(t)$$

Since the input, $v_i(t)$, will be a step function with an amplitude of A, the forced response part of the final solution will always be:

$$v_F(t) = A$$

The natural response of the RLC circuit you are designing is an underdamped response. This means $\zeta < 1$. Look back at Lab 4s if you don't remember this. The solution to the homogenous equation has two complex roots:

$$s_1, s_2 = -\zeta\omega_o \pm j\omega_o\sqrt{1-\zeta^2} = -\alpha \pm j\beta$$

where $\alpha = \zeta\omega_o$ and $\beta = \omega_o\sqrt{1-\zeta^2}$. The total solution is:

$$v_o(t) = K_1 e^{-\alpha t}\cos(\beta t) + K_2 e^{-\alpha t}\sin(\beta t) + A$$

for $t \geq 0$.

Your task in this lab is to select R, L, C, and A to meet the design specifications you are given. Note here R is the total resistance in the circuit, including the parasitic resistance of the inductor.) Figure 2 shows the circuit.

A note on resistance R: You must generally include the source resistance–the source's Thévenin resistance–in the total resistance R. In this case, the source resistance of the myDAQ source AO 0 is very small because that output comes from an op-amp. This is not true, however, if the source current exceeds myDAQ's specification of 2 mA maximum.

Equipment

You will have available:

- Limited values of resistors, potentiometers, capacitors, and inductors. There is a table on the course web site under Laboratory Exercises.
- Protoboard and wires.
- myDAQ running on your own laptop.

Specifications

Design a series RLC circuit driven by a step function that meets the specifications in the following table. The last two digits of your SSN determine which set is yours.

Last Digits of SSN	$\zeta \pm 10\%$	$f_o \pm 10\%$ (kHz)	Final Value $\pm 10\%$ (V)
00 - 15	0.15	2.5	2
16-32	0.2	2.5	1.5
33-49	0.1	5.0	2.5
50-65	0.15	5.0	1.5
66-82	0.1	7.5	2.5
83-99	0.15	7.5	2

You are designing this circuit to provide a particular step response, so you should assume everything starts at t = 0 and the circuit at that time is in the zero state (i.e., initial conditions are both 0).

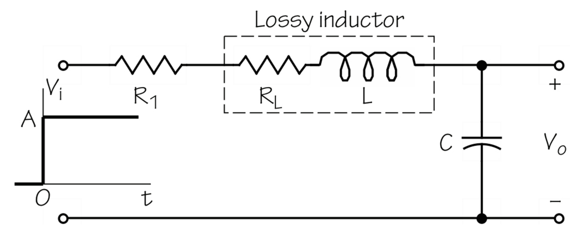


Figure 2: RLC Circuit

Prelab

A. Tasks

The following tasks are to be completed before you can start working on your circuit in the laboratory:

1. Using your assigned specifications, develop the complete equation for $v_o(t)$ in the form given on the previous page.
2. Select parts for R, L, and C from the list of available lab parts, remembering that R is the total resistance in the circuit and therefore must include the parasitic resistance of the inductor (which you probably won't know until you measure it in lab). See the note on parasitic resistance below.
3. Simulate with Multisim the circuit you design. Be sure the square wave steps from 0 to A volts.
4. Measure, using your simulation results, ζ and f_o, and demonstrate they meet your assigned specifications. See Lab 4s for a reminder on how to do this.
5. Measure the rise time and the overshoot from the simulation results:

   The rise time is the time it takes for the response to go from 10% to 90% of its final value. here, your initial value is 0 and your final value is A, so the rise time is the time from 0.1A to 0.9A.

   The overshoot is the precentage by which the first peak of the damped sine wave exceeds the final value. Here, your final value is A, so if the peak is 1.3 A, the overshoot is (1.3A - 1.0A)100% = 30%.

6. Using the results of your simulation measurements, rewrite the complete equation for $v_o(t)$.

7. Use Matlab to plot this complete equation for $v_o(t)$ and show the plot matches the result you got from Multisim.

Parasitic Resistance

Inductors have resistance and this resistance will figure into your circuit design. Pay attention to the relationship between the damping factor ζ and R and L:

$$\zeta = \frac{R}{2}\sqrt{\frac{C}{L}}$$

Your goal is to achieve a particular damping factor. If you choose a small value of L, R must likewise be small for a particular ζ. It is possible the parasitic resistance of the inductor, R_L in Figure 2, will be more than the total resistance your design requires. That means you can't build what you designed and need to start again with a different inductor.

Prelab Report

Your prelab report is an informal report that shows you are ready to start the actual lab. You should have the worksheet at the end of this lab completed. It is to be submitted at the start of the lab session.

Here is a checklist of what is to be in the prelab report:

- Response equation $v_o(t)$ from specifications.
- Circuit drawing of your design with all part values.
- Measured data from Multisim.
- Calculated circuit parameters from simulated data.
- Prove your circuit parameters are within the specified tolerances.
- Updated response equation based on your circuit's actual parameters.
- Matlab plot to verify Multisim result.

Lab Work

The goal of the lab work is to show that your circuit design meets specifications, but you may need to make adjustments in your design as you go through the tests.

1. Measure the parasitic resistance, R_L, of the inductor before you start building your circuit and adjust your choice of the value of the rest of the resistance R_1.

2. Build the circuit and include the proper resistance. You may choose to use a potentiometer that you set to the design value of R_1 you have calculated.

3. You may tweak the resistance by adjusting the pot if you wish.

4. If you can't get the specified parameters to workout within $\pm$ 10%, you may need to choose different parts. Be sure to record what you have done so you can include this information in your final report.

5. Set the Function Generator so the length of the square wave is long enough to allow the wiggles of the damped sine wave to show properly.

6. Set the amplitude and offset of the square wave so that it steps from 0 to A volts.

7. When the scope is displaying the proper waveform, collect data by measuring on the scope just as you did in Lab 5v.

8. Verify you have met the specifications (your grade depends on, among other things whether you met all of them).

Post Lab Report

The post-lab report is to be formal, which means laid out professionally and printed. This will be discussed in class. Here are some pointers:

- No handwriting is allowed anywhere in the report, including on such additions as computer printouts.
- Your lab grade will be based mostly on the report.
- Use well-written, properly-edited English.
- 80% of your grade is for technical content, 20% for effective communication including organization, completeness, spelling, readability, and neatness.
- Be sure to include any references needed.
- Graphics must be professionally drawn and embedded in your report.
- Label all graphics.
- Include a Documentation section. Document all help received.
- Submit a hard copy of your report to your instructor.

The End

By now, you have done enough lab work that you are familiar with both Multisim and with the use of myDAQ to make laboratory measurements. You have built circuits using the protoboard. You'll be expected to carry these skills to other labs in this course and to other courses that you may take.

You've also done some engineering design to create a circuit that meets certain specifications, carrying this design through the engineering design process. While doing this, you've gotten some experience at working with real components and some of the problems they present. You've especially encountered the problem of parasitic resistance.

Prelab Worksheet

Name ___________________________________

Write the equation for the output voltage vased on your specifications

You should have values for your constants

$v_o(t)$ = ____

Draw your designed circuit below. The designed circuit is what you simulated and plan to build. This means you must use real-world values for all components. Include all values in the space below.

Simulate your design in Multisim. Record your measurements below.

T __________________ x_1__________________ x_2__________________

Rise Time _______________ Overshoot_______________

Calculate the actual values for your simulated circuit parameters.

ω_d __________________ ζ__________________ ω_o__________________

Based on your calculated values, what are your error margins compared to your specifications?

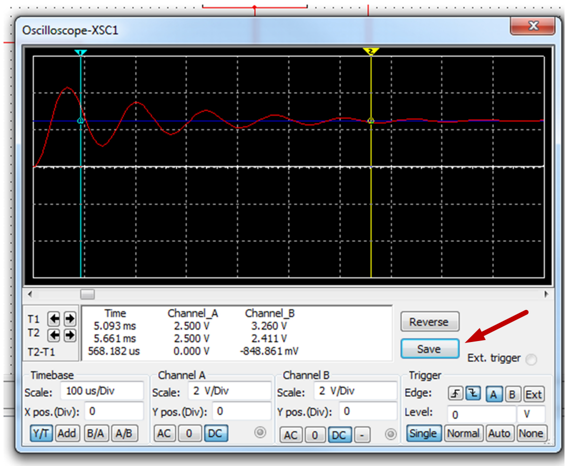
ζ __________________ ω_o__________________

Based on your calculated values from Multisim, what is the circuit's actual output voltage equation?

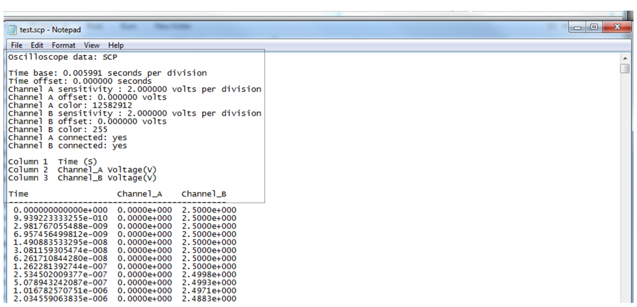
$v_{actual}(t)$ = ____

Use Matlab to plot the actual voltage output and compare this to your simulated results.
Both plots should be on the same graph as shown on the next page.

Some of you may have trouble getting both graphs on one Matlab plot. To do this, you will have to output the scope data to a data file. Click on the *Save* button, shown next to the red arrow below.



Choose *Save as* an Oscilloscope data file (*.scp)–this is the default setting. Pick whatever name you wish. To import this Matlab requires you first open the file in Microsoft Notepad. You need to delete all the header file information–everything in the rectangular box in the figure below.

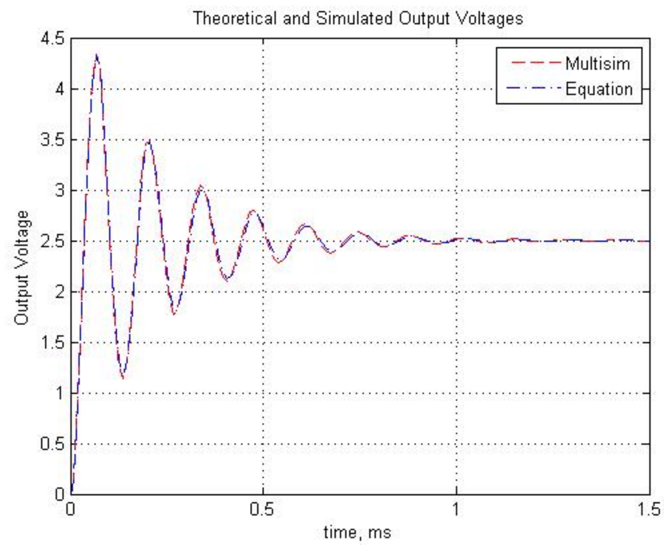


```
test.scp - Notepad
File Edit Format View Help
Oscilloscope data: SCP

Time base: 0.005991 seconds per division
Time offset: 0.000000 seconds
Channel A sensitivity : 2.000000 volts per division
Channel A offset: 0.000000 volts
Channel A color: 12582912
Channel B sensitivity : 2.000000 volts per division
Channel B offset: 0.000000 volts
Channel B color: 255
Channel A connected: yes
Channel B connected: yes

Column 1  Time (S)
Column 2  Channel_A Voltage(V)
Column 3  Channel_B Voltage(V)

Time                    Channel_A    Channel_B
--------------------------------------------------
 0.000000000000e+000   0.0000e+000   2.5000e+000
 9.939223333255e-010   0.0000e+000   2.5000e+000
 2.981767055488e-009   0.0000e+000   2.5000e+000
 6.957456499812e-009   0.0000e+000   2.5000e+000
 1.490883533295e-008   0.0000e+000   2.5000e+000
 3.081159305474e-008   0.0000e+000   2.5000e+000
 6.261710844280e-008   0.0000e+000   2.5000e+000
 1.262281392744e-007   0.0000e+000   2.5000e+000
 2.534502009377e-007   0.0000e+000   2.4998e+000
 5.078943242087e-007   0.0000e+000   2.4993e+000
 1.016782570751e-006   0.0000e+000   2.4971e+000
 2.034559063835e-006   0.0000e+000   2.4883e+000
```

Once this is done, save the file as a .txt file. Now it is ready to load into Matlab. To do this, open Matlab. Ensure the .txt file you created is in the working directory. Type the following code to load the data:

```
load yourfilename.txt
```

The data is now loaded in Matlab. Type *whos* to see the variable name–usually, it will be the same name as the filename. You should now be able to plot this scope data. Once you've done this, you can write the Matlab code to plot the equation for the output voltage. You should get something like the figure below.



Note both plots are on the same graph. Once you have this, print out a copy and turn in with your prelab worksheet.