Lab 3 – First- and Second-Order Circuits

XMUT203 Analogue Circuits and Systems

Topics

Laboratory 3 – Frequency Responses

1. Resonant Frequency.
2. Q-factor.
3. Parameter Sensitivity.
4. Extension.

Introduction

1. Introduction

- In this lab, you will use LTSpice to explore some of the properties of RLC resonant circuits.
- You will also be introduced to some advanced LTSpice tips and tricks that should be useful as you continue your studies in XMUT203.

Background

2. Background

- Resonance is a common topic that pops up throughout Electronic and Computer Systems Engineering.
- It plays a central part in filter theory (XMUT220, XMUT301, and XMUT303), is used in oscillator and positive feedback circuits (XMUT303, ECEN403) and even features prominently in mechatronics, often as the result of mechanical, rather than electrical, interactions (ECEN425).
- We are also familiar with resonance in our everyday lives: musical instruments have specially designed sound boxes to help their strings resonate, while near-field payment systems such as Snapper, payWave, and PayPass rely on resonance to power embedded microchips that otherwise would have no power source.

Background

- We will use LTspice to illustrate some of the properties of resonant circuits in this lab.
- Recall that you can use LTspice on the computers in Lab 1.
- Alternatively, you can install LTspice on your own machine by downloading the relevant package from:

  http://linear.com/ designtools/software/#LTspice

- It is cross-platform, available free of charge, and does not have a node limit.
- Should you get stuck and require help with LTSpice, there is a wealth of information available online.
- In particular, there is an official getting started guide at:

  http://cds.linear.com/docs/ en/software-and-simulation/LTspiceGettingStartedGuide.pdf.

Background

- Simon Bramble also maintains a series of excellent tutorials on LTspice).
- Part 1 is most relevant for the work we will do in XMUT203:

  (http://simonbramble.co.uk/lt_spice/ltspice_lt_spice. htm

Resonant Frequency

3. Resonant Frequency

- Resonance in AC circuits is a condition in which a specific frequency determined by the values of the resistance, capacitance, and inductance.
- For series resonance, the condition of resonance is determined by minimum impedance and zero phase.
- The resonance of a series RLC circuit occurs when the inductive and capacitive reactances are equal in magnitude but cancel each other because they are 180 degrees apart in phase.
- Parallel resonance on the other hand is the opposite of series resonance where maximum impedance occurred at zero phase, and current gain rather than voltage gain as in the series resonance will be observed.

Resonant Frequency

- The sharp minimum or maximum in impedance which occurs is useful in tuning applications.
- The sharpness of the minimum or maximum depends on the value of R and is characterized by the "Q" of the circuit.

Exercise 1 – Series Resonant Circuit

Exercise 1 – Series Resonant Circuit

- Consider the LTspice model of a series RLC circuit shown in figure 1.
- Notice that it contains an independent voltage source, set to give an AC signal with an amplitude of 1 and a phase of 0: AC 1 0.

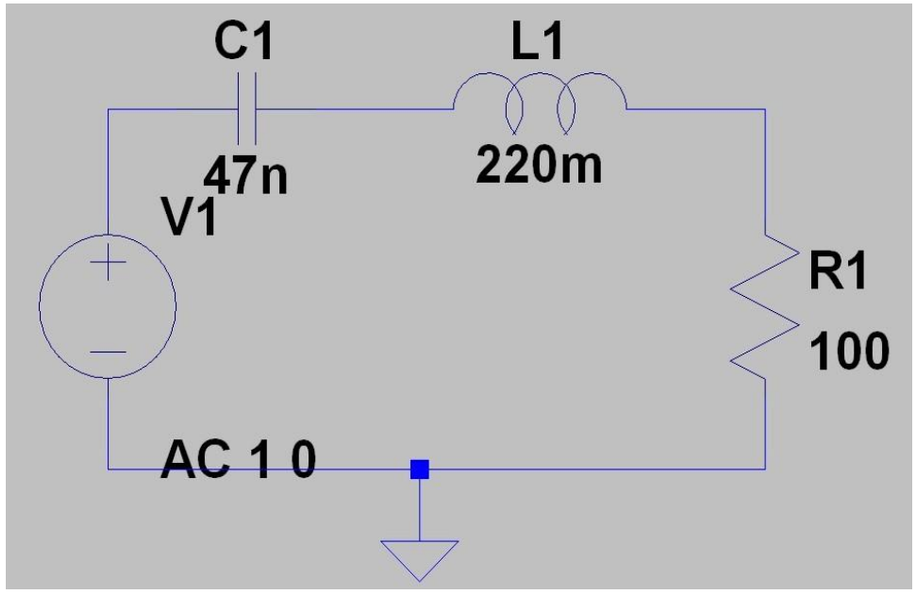

Figure 1: Series RLC circuit

Exercise 1 – Series Resonant Circuit

1. Calculate the transfer function of this circuit, i.e. the ratio of the voltage across the resistor, R1, compared to the source voltage, V1.
2. Knowing the circuit transfer function, derive an expression for the resonant frequency of the circuit, i.e. the frequency at which the imaginary component of the transfer function is minimized.
3. Recreate the circuit shown in Fig. 1 in LTspice and run an AC analysis on it. Start with 3 points per decade and choose your start and end frequency to be roughly three decades either side of your expected resonant frequency.
4. Plot the voltage across R1 and use cursors[1] to compare the resonant frequency of your simulated circuit to the resonant frequency you calculated.

Exercise 1 – Series Resonant Circuit

5. Does your simulated frequency response look similar to the one shown in Fig. 2? Do your measurements of resonant frequency agree with your calculations?
6. If not, try increasing the number of discrete frequencies modelled in your simulation. You can do this by increasing the number of points per decade in the "Edit Simulation Cmd" dialogue box.
7. How many points are necessary to get a transfer function as smooth as the one shown in Fig. 2? How many points are necessary to get agreement between your calculated and measured values of resonance?

- Before you start the next section, save your LTspice model file and create a copy to work on. You will need the original again later in the lab.

Q-Factor

4. Q-Factor

- The Q- or quality-factor of a resonant circuit is a measure of how much energy is stored in the circuit during each AC cycle.
- Specifically, it is the ratio of energy stored to energy dissipated during each cycle.
- For a series RLC circuit,

$$Q = \frac{1}{R}\sqrt{\frac{L}{C}} \qquad (Eq.\,2.1)$$

- In the following figure, solid line is magnitude, dotted is phase, but none of that matters because there aren't any axes labels or coordinates on this figure.

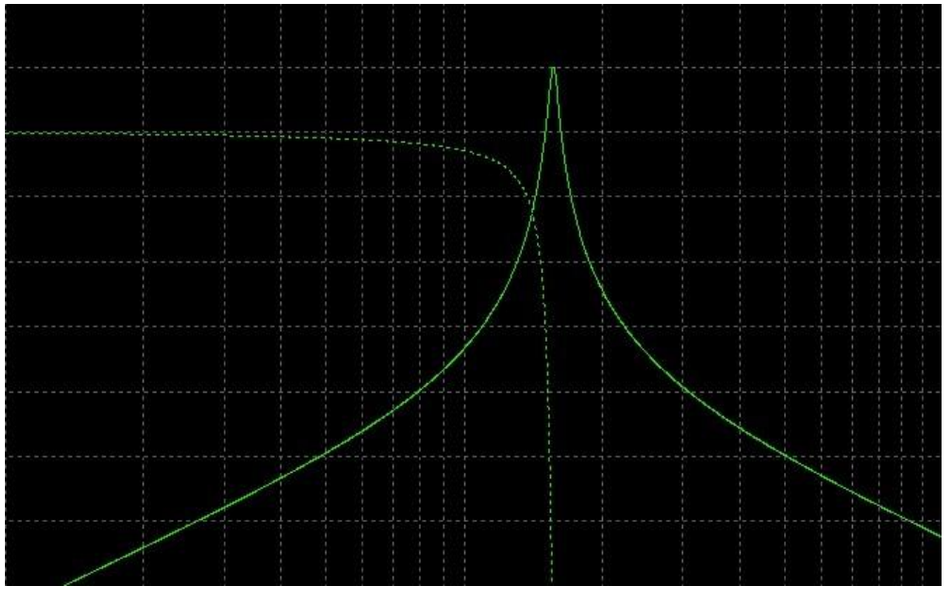

Figure 2: The shape of the frequency response of a resonant circuit.

Q-Factor

- The form of the equation changes slightly for a parallel resonant circuit, to

$$Q_{parallel-cct} = R\sqrt{\frac{C}{L}} \qquad (Eq.\,2.2)$$

- For either circuit, we can measure Q as the ratio of the resonant frequency, f_{res}, to the bandwidth[2], B, of the circuit:

$$Q = \frac{f_{res}}{B} \qquad (Eq.\,2.3)$$

- Practically, circuits with high Q factor tend to have smaller bandwidths and thus higher selectivity.

Exercise 2 – Q-Factor of Series Resonant Circuit

Exercise 2 – Q-Factor of Series Resonant Circuit

1. Calculate the Q factor for the circuit shown in Fig. 1. Compare this value to a measured value based on the transfer function you simulated in the previous section. Does your simulated value agree with your calculations?
2. Calculate the value of the resistor, R1, required to give you Q values of 0.25, 0.5, 0.707, 1, and 10.
3. Change the independent voltage source to provide a DC amplitude of 1 V and run a transient analysis on your model, making sure to select the "Start external DC supply voltage at 0 V" checkbox.

   A stop time of about 100 ms should be long enough for this particular analysis. Note the characteristics of the voltage waveform across R1 when the resistor has a value of 100 Ω.

Exercise 2 – Q-Factor of Series Resonant Circuit

4. Now, vary the simulated value of R1 according to the values you calculated to give predetermined Q. Observe the changes in the voltage waveform across R1.

   Hint: rather than examining waveforms for each value of R1 individually, it may be useful to calculate all of the curves in a single simulation run.

   a. To do this, set the value of R1 to {R}: this tells the model that R1 is equal to the value of the variable, R.

   b. Next, click on the .op button in LTspice to enter a SPICE directive:

      ```
      .step param R list 1 10 100.
      ```

      Note the leading period! This directive tells the model that the parameter R will have three separate values of 1, 10, and 100, which are specified as a list.

   c. Rerun your analysis.

Exercise 2 – Q-Factor of Series Resonant Circuit

5. What is the effect of increasing Q on the transient response of your circuit?

   By definition, we say that circuits with low Q factor, i.e. $Q < 0.5$, are *overdamped*.

   Similarly, circuits with $Q > 0.5$ are considered *underdamped*, and circuits with $Q = 0.5$ are *critically damped*.

5. Based on the voltage waveforms you observed across R1, what do you think these terms (e.g. overdamped, underdamped, critically damped) mean in the context of RLC circuits?
6. Are there any properties that you can say belong specifically to overdamped or underdamped systems?

Parameter Sensitivity

5. Parameter sensitivity to component variations in resonant circuits

- While simulation with ideal components can guide the design of electronic circuits, we need to be mindful that real components do not have exact values and that the results we obtain in the real world can be quite different from those we simulate on a PC.
- For example, we are already well aware that resistors have particular tolerances associated with them, as specified by the fourth or fifth (depending on the resistor) colour band on their body.
- We know that resistors with a gold tolerance band may vary by up to 5%, while resistors with a brown band can be expected to vary by only 1% from their nominal value.
- Fortunately, LTspice can help us out in some cases where we expect component variations to be critical to parameters in our design.

Exercise 3 – Sensitivity Analysis of Resonant Circuit

Exercise 3 – Sensitivity Analysis of Resonant Circuit

- For this part of the lab, we are going to use the LTspice model constructed at the start of the lab to run sensitivity analysis.
- We will run Monte Carlo analyses, which allow us to determine the variation we can expect in parameters such as resonant frequency and Q factor from variations in the nominal values of our R1, L1, and C1 components.

1. We will explore the effect of variations in capacitance first. Change the value of the capacitor in your model to:

   ```
   {mc(47n,0.10)}
   ```

   Here `mc` is short for Monte Carlo, which tells the model that the component value of the capacitor is nominally 47 nF with a uniformly distributed random variation in the range ±5%.

Exercise 3 – Sensitivity Analysis of Resonant Circuit

2. Next, add a SPICE directive to complete several runs of the simulation, rather than just one:

   ```
   .step param run 1 100 1
   ```

   Here the numbers after the parameter, run, are the start value, 1, the end value, 100, and the increment: 1.

   Notice that we could have used the list syntax from the previous section, but this would have been tedious:

   ```
   .step param run list 1 2 3 4 5 6 7...
   ```

3. Run your simulation and estimate the range of resonant frequencies produced.

   Similarly, estimate the range of bandwidths, and calculate the range of Q factors.

Exercise 3 – Sensitivity Analysis of Resonant Circuit

Hint: to get the best estimates you may wish to increase the number of frequencies modelled around the resonant frequency.

So that the addition of frequency points does not slow your simulations down too much, consider reducing the overall range of frequencies you simulate for this question.

4. What is the relationship between changes in the capacitor component value, C, and the parameters f_{res} and Q-factor? For example, does f_{res} vary with C, C^2, $1/C$ or $C^{1/2}$?
5. Repeat your analysis separately for changes in L and R. Do the variations in f_{res} and Q-factor make sense given the expression for f_{res} you derived in Section 3 and the equation given for $Q_{series-cct}$ in Section 4, i.e. Eq. (2.1)?

Exercise 3 – Sensitivity Analysis of Resonant Circuit

- In the comparisons you have just performed you have been assessing with the sensitivity, S, of a parameter, P, to a component, C.
- In this case the components are the values of the resistor, capacitor, and inductor in your RLC circuit and the parameters are f_{res} and Q, but we can use the same concept to assess a range of components and parameters far beyond this narrow example.
- Formally, we define the sensitivity of a parameter to a component, S_c^p, as:

$$S_c^p = \frac{\left(\frac{dP}{P}\right)}{\left(\frac{dC}{C}\right)} \quad (2.4)$$

Exercise 3 – Sensitivity Analysis of Resonant Circuit

- This means that, for example, the sensitivity of a parameter that varies by 10% for a component that changes by 10% is 1, and the sensitivity of a parameter that varies by 1% for a component variation of 10% is 0.1.
- Generally smaller variations are better when we are dealing with unintended sensitivities.

Extension

6. Extension

- You might notice that different arrangement of the circuit components e.g. parallel, series, parallel-series, etc., might change the overall behavior of the system.
- As a result the parameters associated with the given circuit e.g. f_{res}, Q-factor, etc., might change as well if we have different component configuration.

Exercise 4 – Parallel Resonant Circuit

Exercise 4 – Parallel Resonant Circuit

- If you finish the lab and have time to spare, consider some of the following questions:

  1. How does the transfer function for a series RLC circuit that you calculated in Section 3 change if the circuit contains a resistor, capacitor, and inductor all in parallel across a voltage source?

     At resonance, is the voltage across the parallel circuit at a maximum or a minimum?

     At frequencies far away from the resonant frequency, is the voltage high or low?

Exercise 4 – Parallel Resonant Circuit

2. What is the sensitivity of f_{res} when both L1 and C1 are subject to variation?

   How does this compare to the sensitivities of f_{res} to just L1 or C1 in isolation?

3. What happens to Q-factor in your series RLC circuit as R1 shrinks to zero?

   What does this do to the step response 3 of the circuit?

   What would happen if Q-factor were negative? How could you practically make Q-factor negative?

Conclusion

7. Conclusion

- This laboratory introduced various LTSpice techniques that allow a series RLC circuit to be modelled.
- We have seen that simulations will agree closely with calculations provided they are set up correctly: often it is not obvious to know if this is the case.
- We have also seen that aspects of a model can be given several different values by a single LTSpice simulation.
- This last property should be of special interest to us as we begin to use LTSpice in more and more of our engineering courses.

Footnote

Footnote:

1. Click on the name of the trace at the top of the plot to bring up the cursor dialogue box.
2. Recall that the bandwidth of a circuit is the range of frequencies for which the gain is within 3 dB of the maximum gain.
3. You calculated the step response of your RLC circuit in Section 4.