CHAPTER

13

Analysis of Analog Circuits and Models

13.1 GOALS OF THIS CHAPTER

In Chapter 12 we defined the basic players that are featured in the next three chapters: electrical and mechanical analog elements. We found that they can be distinguished by the way they treat energy: they supply it, store it, or use (dissipate) it. They also establish relationships between the potential energy variables and the kinetic energy variables, that is, between the force and movement variables. In electrical circuits, voltage is the potential energy variable, whereas current is the kinetic energy variable; in mechanical systems, force is the potential energy variable and velocity is the kinetic energy variable. In this chapter, we learn how to analyze collections of these elements. Collections of electrical elements are called "networks" or "circuits,"[1] whereas collections of mechanical elements are simply "mechanical systems." Often the circuit or mechanical system we are analyzing is meant to represent a real circuit or mechanical system, but occasionally it may be the analog representation of a physiological mechanism. For example, the Windkessel models introduced in Chapter 1 (Section 1.4.5.1) are circuits meant to represent the mechanical load on the heart.

In circuit or mechanical system analysis, we find representations for one or more of the system's variables. For example, in Example 12.5 we found the voltage across a simple circuit and in Example 12.9 we found the velocity of a mass in a mechanical system. We may only want to find one voltage or current in a circuit, or one force of velocity, but the tools we develop here enable us to find any variable in the system and/or to construct a transfer function for a circuit or mechanical system. Once we have the transfer function, we can use any of the techniques developed in previous chapters, such as Bode plots to get the frequency characteristic or the inverse Laplace transform to find the time domain behavior of the system.

Having defined the players in the last chapter, we now need to find the rules of the game: the rules that describe the interactions between elements. For both mechanical and electrical elements, the rules are based on two fundamental conservation laws: (1) conservation of energy and (2) conservation of mass or charge. For electrical elements, these rules are termed

[1] The terms "circuits" and "networks" are used interchangeably.

https://doi.org/10.1016/B978-0-12-809395-5.00013-8

"Kirchhoff's voltage law" (KVL) for conservation of energy and "Kirchhoff's current law" (KCL) for conservation of charge. For mechanical systems, we only use conservation of energy which manifests as an extension of Newton's famous force law, $F = Ma$ and as illustrated in Example 12.9. With these rules and some basic matrix techniques, we can analyze any linear network or mechanical system.

A unique behavior of systems covered in this chapter is the phenomenon of "resonance." Certain circuits or mechanical systems respond most favorably to a small set of frequencies. Such systems are called "resonant circuits" or "resonant systems" and can be useful in selecting out a specific frequency or range of frequency. Resonance is a common, and usually undesirable, feature in mechanical systems.

Specific goals of this chapter include:

- Provide definitions of circuit conversation laws: Kirchhoff's law,
- Show how to analyze single-loop circuits: mesh analysis,
- Show how to analyze multiple loop circuits using matrices: mesh analysis,
- Show how to analyze one node and multiple node circuits: nodal analysis,
- Define the mechanical system conservation law, Newton's force law, and demonstrate its application to more complex mechanical systems,
- Describe resonance circuits and resonant mechanical systems.

13.2 CONSERVATION LAWS: KIRCHHOFF'S VOLTAGE LAW

KVL is based on the conservation of potential energy: the total potential energy in a closed loop must be zero. Since voltage is a measure of potential energy, the law implies that all voltage increases or decreases around a closed loop must sum to zero. Simply stated, what goes up must come down (in voltage):

$$\sum_{\text{Loop}} v = 0 \tag{13.1}$$

To do anything useful, electrical elements must have current flowing through them. Otherwise, the voltage across the element, any element, is zero, and for all practical purposes the element does nothing and can be ignored. But current can only flow in a loop; it cannot simply fall out of the end of an unconnected element. So, by reverse logic, all elements that do anything must be connected in a closed path, a loop. KVL applies to all elements in the circuit and allows us to write an equation for all electrical elements connected in a loop. Figure 13.1 illustrates KVL. It also gives an example of a "useless" element that is connected to the loop but is not itself in a loop. Since no current can flow through this element, the voltage across the element, V_4, must be zero. The voltage on one side of this element is the same as the voltage on the other side, so it might as well not be there. (The story changes if something is connected to that element that creates a closed loop for that element. Then current flows through that element and voltage V_4 is no longer zero.)

More complicated circuits may contain a number of loops, and some elements may be involved in more than one loop, but KVL still applies. The analysis of a circuit with any number of loops is a straightforward extension of the analysis for a single loop.

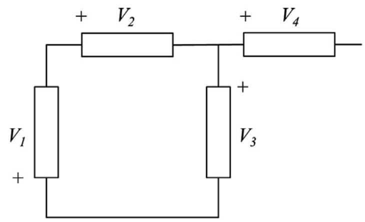


FIGURE 13.1 Illustration of Kirchhoff's voltage law. Three generic elements form a loop. The three voltages must sum to zero: $V_1 + V_2 + V_3 = 0$. This circuit also includes an element that is not part of a loop. Since this element is not in a loop there is no current through it and no voltage across it. So V_4 is zero and the voltage on either side of this element is the same. The element acts like a wire: it does nothing, and the circuit behaves as if it were not there.

Although all circuits can be analyzed using only KVL, in some situations the analysis is simplified by using the other conservation law that is based on the conservation of kinetic energy, that is, conservation of charge. This law is known as KCL and states that the sum of currents into a connection point (otherwise known as a "node") must be zero:

$$\sum_{\text{Node}} i = 0 \tag{13.2}$$

In other words, what goes in must come out (with respect to charge at a connection point). For example, consider the three currents going into the node in Figure 13.2. According to KCL the three currents must sum to zero: $i_1 + i_2 + i_3 = 0$. Of course we know that one, or maybe two, of the currents is actually flowing out of the node, but this just means that one (or two) of the current values will be negative.

In the analysis of some electronic circuits, both KVL and KCL are applied, but to analyze the networks covered in this chapter only one of the two rules is required at any given time. As mentioned previously, network analysis involves the determination of the network's variables or transfer function. In the case of networks, the transfer function is the ratio of a

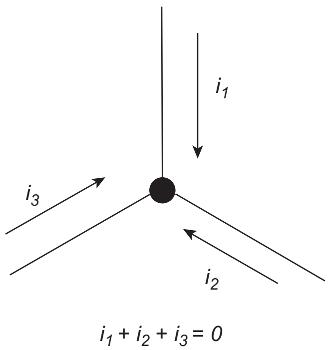


FIGURE 13.2 Illustration of Kirchhoff's current law. The sum of the three currents flowing into the connection point, or node, must be zero. In reality, one (or two) of these currents is flowing out of the node, which means that one (or two) current(s) has negative values.

specific output variable to a specific input variable, usually both voltages. Although either rule can be used to analyze the networks in this chapter, the rule of choice is the one that results in the least number of equations.

"Mesh analysis" is the term given to the analysis that uses KVL. This terminology makes more sense if you know that the word "mesh" is technical jargon for a circuit loop. Mesh analysis using KVL generates one equation for each loop in the circuit. "Nodal analysis" is the term used for the circuit analysis approach that uses KCL, and it generates one equation for each node in the circuit minus one node. (In all circuits one node provides a reference and is assumed to be at 0.0 V. A node that is assumed to be at 0.0 V is said to be "grounded.") If multiple equations are generated by either method they must be solved simultaneously. The method that leads to the fewest number of simultaneous equations is an obvious choice, particularly if we must find the solutions manually. However, if we use MATLAB to solve the circuit equations, it matters little how many equations we need to solve, so we could stick with just one approach, usually KVL. Nonetheless, some electronic networks require the use of both rules, so we learn to use them both.

13.2.1 Mesh Analysis: Single Loops

Mesh analysis employs KVL (Equation 13.1) to generate the circuit equations. In mesh analysis you write equations based on voltages but solve for currents. Once you find the loop currents, you can go back and find any of the voltages in the loop by applying the basic voltage/current definitions given in Chapter 12. (As you might guess, in nodal analysis it is the opposite: you write an equation(s) based on currents, but end up solving for node voltages.) Mesh analysis can be done using an algorithm that provides a step-by-step procedure that can be applied to almost any circuit.[2] In our now classic learn-by-doing approach, an example of this mesh analysis algorithm is given in Example 13.1.

EXAMPLE 13.1

Find the voltage across the capacitor in the network of Figure 13.3. Note that the source is sinusoidal, so phasor analysis is used.

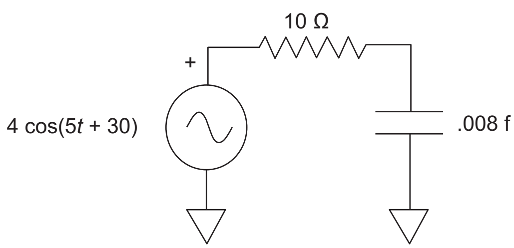


FIGURE 13.3 Single mesh (loop) circuit used in Example 13.1.

[2]Circuits containing electronic elements such as transistors, or integrated circuits, require a more complicated analysis procedure, but the basic ideas are the same.

Solution: We have already solved a similar network in Chapter 12 (Example 12.5) using an extended version of Ohm's law. However, the algorithmic approach used here is more general and can be applied to more complicated networks.

The circuit has one mesh (i.e., loop) and three nodes. Nodal analysis requires the simultaneous solution of two equations, whereas mesh analysis requires the solution of only one equation, making it the analysis of choice. Remember that by KVL what goes up must come down, but the trick is to keep accurate tabs on the direction, or "polarity," of the voltage changes: up or down corresponding to voltage increases or decreases. This bookkeeping problem is simplified by the algorithmic approach described here. The steps are presented in considerable detail for this example, but are easily applied to a wide range of network problems.

Step 1. Apply a transformation to the network so that all elements are represented in either the phasor or Laplace domain. Table 12.5 can be used to get the phasor or Laplace representations of the various elements. Since we are dealing with a sinusoidal stimulus, we will use the former. The sources are represented as phasor constants, $V_s \angle \theta$, whereas passive elements are given their respective phasor impedances: R Ω, $j\omega L$ Ω, or $1/j\omega C$ Ω. Putting the Ω symbol on the circuit diagram is a good way to show that the elements have been transformed into phasor (or Laplace) notation. Sometimes voltage sources use root mean square, but peak values will be used in this text as is more common. It really does not matter as long as you are consistent and know which units are being used.

Step 2. This step consists of defining the mesh current (or currents if more than one loop is involved). The loop in this example is closed by the two grounds since the two grounds are at the same voltage and are actually connected. The mesh current goes completely around the loop in either a clockwise or counterclockwise direction, theoretically your choice. To be consistent, in this book we always assume that the mesh current travels clockwise around the mesh. Maybe the current is traveling in the opposite direction, but that is not a problem because then the value we find for this current will be negative. Defining the current direction automatically defines the voltage polarities for the passive elements, since by definition current must flow into the positive side of a passive element. Recall, the polarity of a voltage source is defined by the element itself. After completion of these two steps, the circuit looks as shown in Figure 13.4.

Step 3. Apply KVL. We mentally go around the mesh (again clockwise) summing the voltages, but it is an algebraic summation. We assign positive values if there is an increase in voltage and negative values if there is a decrease in voltage. Start at the lower left corner (below the source) and

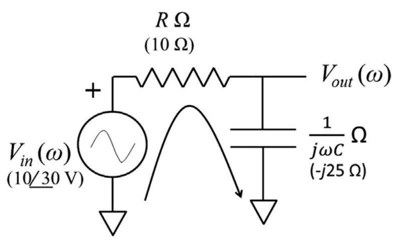


FIGURE 13.4 The circuit in Example 13.1 after Steps 1 and 2. The passive elements and source have been converted to their phasor representation (indicated by the Ω symbol). The direction of mesh current, $I(\omega)$, is arbitrary, but has been assigned as clockwise, a convention used throughout this book. If actual current flow is counterclockwise, $I(\omega)$ will turn out to have a negative value.

proceed around the loop in a clockwise direction. Traversing the source leads to a voltage rise, so this entry is positive; the next two components have a voltage drop (from + to −), so their entries are negative:

$$V_S - V_R - V_C = 0$$

Substituting in the element impedances, $V_R = R$, $V_C = \frac{1}{j\omega C} I(\omega)$, this equation becomes:

$$V_S - RI(\omega) - \frac{1}{j\omega C} I(\omega) = 0$$

$$V_S - I(\omega)\left(R + \frac{1}{j\omega C}\right) = 0$$

In fact, to develop this mesh equation we could have started anywhere in the loop and gone in either the clockwise or counterclockwise direction, but again for consistency, and since it really does not matter, we will always go clockwise and always begin at the lower left corner of the circuit.

Step 4. Solve for the current. Put the source (or sources, if more than one) on one side of the equation and the terms for the passive elements on the other. Then solve for $I(\omega)$.

$$V_S(\omega) = (R + Z_C)I(\omega)$$

$$I(\omega) = \frac{V_S}{R + Z_C}$$

Step 5. Solve for any voltages of interest. In this problem, we want the voltage across the capacitor. The voltage−current relationship for a capacitor is $V_C(\omega) = Z_C(\omega)I(\omega)$. Substituting our solution for $I(\omega)$ into this equation:

$$V_C = Z_C(\omega)I(\omega) = \frac{Z_C(\omega)V_S}{R + Z_C(\omega)}$$

Sometimes you can leave the solution in this form, with variables for element values, for example, when you do not know the specific values, when several different values may be used in the circuit, or when the problem will be solved on the computer. In this case we have specific values for our elements, so we can substitute in the values for R, Z_C, and V_S and solve:

$$V_C = \frac{(-j25)4\angle 30}{10 - j25} = \frac{(25\angle -90)4\angle 30}{27\angle -68} = \frac{100\angle -60}{27\angle -68} = 3.7\angle 8 \text{ V}$$

The next example applies this five-step algorithm to a slightly more complicated single-loop circuit.

EXAMPLE 13.2

Find the general solution for V_{out} for the circuit in Figure 13.5. Since it contains a resistor, inductor, and capacitor, it is sometimes called an "RLC" circuit. The arrows on either side of V_{out} indicate that this output voltage is the voltage across the capacitor. Again the two ground points are effectively connected.

Steps 1 and 2 lead to the circuit shown in Figure 13.6 below. Passive elements are shown with units in ohms to help remind us that we are now in the phasor domain.

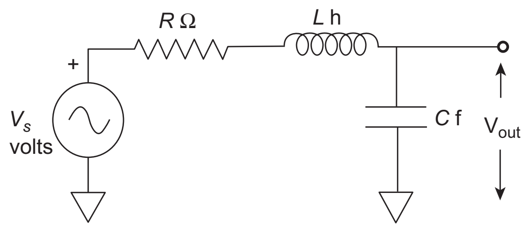


FIGURE 13.5 The "RLC" network used in Example 13.2.

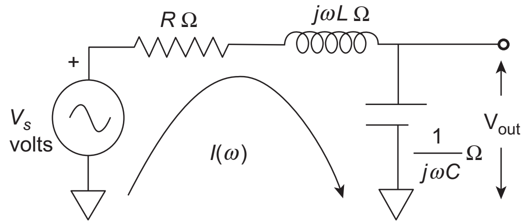


FIGURE 13.6 The network in Figure 13.5 after replacing the passive elements by their impedances, representing the source as a phasor voltage, and assigning the mesh current $I(\omega)$ as clockwise.

Step 3. Now write the basic equation going around the loop:

$$V_S(\omega) - RI(\omega) - j\omega LI(\omega) - \frac{1}{j\omega C}I(\omega) = 0$$

$$V_S(\omega) - I(\omega)\left(R + j\omega L + \frac{1}{j\omega C}\right) = 0$$

Step 4. Solve for $I(\omega)$:

$$V_S(\omega) - I(\omega)\left(R + j\omega L + \frac{1}{j\omega C}\right)$$

To clean things up a bit, clear the fraction in the denominator and rearrange the right-hand side into real and imaginary parts. (Recall $j^2 = -1$.)

$$I(\omega) = \frac{V_S(\omega)j\omega C}{R(j\omega C) + j\omega L(j\omega C) + 1} = \frac{V_S(\omega)j\omega C}{1 - \omega^2 LC + j\omega RC} \text{ A}$$

Step 5. Now find the desired voltage, the output voltage, V_{out}. Use the same strategy used in the last example: multiply $I(\omega)$ by the capacitance impedance, $1/j\omega C$:

$$V_{out}(\omega) = \frac{V_S(\omega)j\omega C}{1 - \omega^2 LC + j\omega RC}\left(\frac{1}{j\omega C}\right) = \frac{V_S(\omega)}{1 - \omega^2 LC + j\omega RC} \text{ V} \tag{13.3}$$

To find a specific value for V_{out} it is necessary to put in specific values for R, L, and C as well as for V_S and ω. The source defines ω. However, by using some of the tools developed in Chapter 6, much can be learned from just the form of the equation. For example, Equation 13.3 is a second-order equation in phasor notation. By equating coefficients, we could even determine values for ω_n and δ in terms of R, L, and C without actually solving the equation. This is done in a later example.

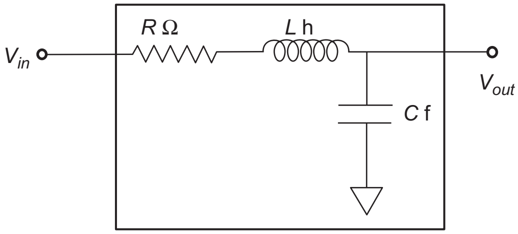


FIGURE 13.7 The network of Figure 13.6 viewed as an input–output system with V_S now represented as an input, V_{in}, and V_{out} as the output. The transfer function for this system is $TF(\omega) = V_{out}(\omega)/V_{in}(\omega)$.

The network in Figure 13.6 can also be viewed as an input–output system having a transfer function where the voltage source, V_S, becomes the input voltage and V_{out} is the output. When thought of in these terms, the network might be drawn as in Figure 13.7.

If the network is thought of as an input–output system, then the transfer function for this network is $TF(\omega) = V_{out}(\omega)/V_{in}(\omega)$. To repeat the caveat stated in the Laplace transform chapter, Chapter 7, the term "transfer function" should technically only be used for a function that is written in terms of the Laplace variables. However, the concept is so powerful that it is used to describe almost any input–output relationship, even qualitative relationships. To find the transfer function for the system shown in Figure 13.7, simply divide both sides of Equation 13.3 by $V_S(\omega)$.

$$\frac{V_{out}(\omega)}{V_{in}(\omega)} = \frac{1}{1-\omega^2 LC + j\omega RC} \tag{13.4}$$

This is clearly the transfer function of a second-order system (compare Equations 6.42 and 6.43). Since this transfer function is in the phasor domain, it is limited to sinusoidal functions or general periodic functions if we bring in the Fourier transform. However, this circuit could just as easily be analyzed using Laplace domain variables as illustrated in the next example.

EXAMPLE 13.3

Find the Laplace domain transfer function for the system/network shown in Figure 13.8.

Solution: The analysis of this network in the Laplace domain follows the same steps for the phasor domain analysis, except that in Step 1 the elements are represented by their Laplace representations: R, sL, and $1/sC$.

Step 1. The elements' values are replaced by the Laplace domain equivalents. These modified element values carry the unit of ohms to show they are impedances.

Step 2. The mesh current is assigned, again clockwise, but as a Laplace variable: $I(s)$. After the application of these two steps, the circuit appears as shown in Figure 13.9.

Step 3. Writing the loop equation:

$$V_{in}(s) - RI(s) - sLI(s) - I(s)/sC = 0$$
$$V_{in}(s) - I(s)(R + sL + 1/sC) = 0; \quad V_{in}(s) = I(s)(R + sL + 1/sC)$$

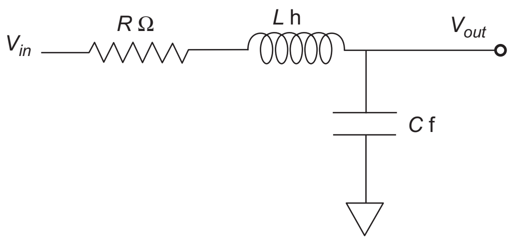


FIGURE 13.8 In Example 13.8, we solve for the Laplace transfer function of this network, $V_{out}(s)/V_{in}(s)$.

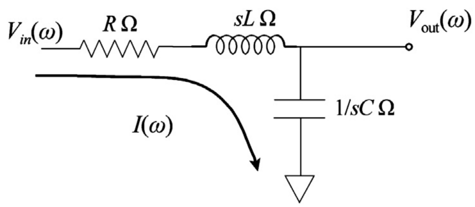


FIGURE 13.9 The circuit of Figure 13.8 with elements represented in Laplace notation and the mesh current assigned.

Step 4. Solving for $I(s)$:

$$I(s) = \frac{V_{in}(s)}{R + sL + 1/sC}$$

Step 5. Solving for $V_{out}(s) = I(s)/sC$:

$$V_{out}(s) = \frac{V_{in}(s)sC}{R + sL + 1/sC}\left(\frac{1}{sC}\right) = \frac{V_{in}(s)}{sRC + s^2CL + 1}$$

Rearranging into standard Laplace format where the highest power of s (in this case the s^2) has a coefficient of 1.0:

$$V_{out}(s) = \frac{V_{in}(s)/CL}{s^2 + R/L^s + 1/LC}$$

To find the transfer function, just divide by $V_S(s)$:

$$TF(s) = \frac{V_{out}(s)}{V_{in}(s)} = \frac{1/CL}{s^2 + R/L^s + 1/LC} \tag{13.5}$$

This is the Laplace transform transfer function of a second-order system having the same format as Equation 7.31, repeated here.

$$TF(s) = \frac{\omega_n^2}{s^2 + 2\delta\omega_n s + \omega_n^2} \tag{13.6}$$

Equating coefficients to solve for ω_n in terms of R, L, and C:

$$\omega_n^2 = 1/LC; \quad \omega_n = \sqrt{\frac{1}{LC}} \tag{13.7}$$

Solving for δ:
$2\delta\omega_n = \frac{R}{L}$; $\delta = \frac{R}{2L\omega_n}$ Substituting in for ω_n

$$\delta = \frac{R}{2L\left(1/\sqrt{LC}\right)} = \frac{R\sqrt{LC}}{2L} = \frac{R}{2}\sqrt{\frac{C}{L}} \tag{13.8}$$

The values of ω_n and δ are of particular importance in resonant systems as described in the section on resonance.

There are two implicit assumptions in the development of the transfer function of Equation 13.5: (1) the input, V_{in}(s), is an ideal voltage source, that is, V_{in}(s) will produce whatever current is necessary to maintain its prescribed voltage and (2) nothing is connected to the output, nothing meaning that no current flows out of the output terminals. Another way to state the second assumption is that $V_{out}(s)$ is connected to an "ideal load." (An ideal load is one where the load, Z_L, is infinite, which means there is really no load at all.) Although these assumptions may not always be true, in many real circuits input and output conditions are surprisingly close to these idealizations.

13.2.2 Mesh Analysis: Multiple Loops

Any single-loop circuit can be solved using the five-step process and a large number of useful circuits consist of only a single loop. Nonetheless, it is not difficult to extend the approach to contend with two or more loops, although the complex arithmetic can become tedious for three loops or more. Again, this is not really a problem since MATLAB can handle the necessary math for a large number of loops without breaking a sweat. The following example uses the five-step approach to solve a two-loop network and indicates how larger networks can be solved.

EXAMPLE 13.4

An extension of the Windkessel model used in Example 12.4 is shown in Figure 13.10. In this version, an additional resistor has been added to account for the resistance of the aorta, so the model is now a two-mesh circuit. In this example, we find the relationship between $v(t)$, which represents the pressure in the left heart, and the current $i(t)$, which represents the blood flow in the aorta. This is done using phasor notation in conjunction with an extension of the approach developed for single-loop circuits. For this circuit $R_A = 0.79$ mmHg s/mL, $R_p = 0.0033$ mmHg s/mL, and $C = 1.75$ mL/mmHg.

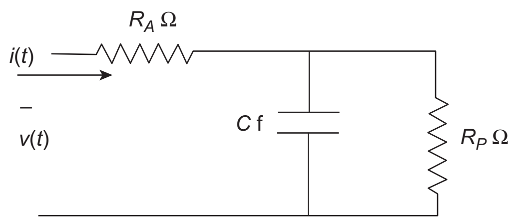


FIGURE 13.10 A two-mesh (two-loop) representation of the Windkessel model that is analyzed in Example 13.4.

Solution: To find the phasor relationship between v_{in} and i_{in}, follow the same five-step plan, but with modifications to Step 2. Since this problem is solved manually, we substitute in the actual component values to simplify the complex algebra.

Step 1. Represent all elements by their equivalent phasor or Laplace representations. This step is always the same in any analysis. Note that the capacitor impedance in phasor notation becomes:

$$\frac{1}{j\omega C} = \frac{1}{j1.75\omega} = \frac{0.57}{j\omega}$$

Step 2. Define the mesh currents. This step is essentially the same as for single-loop circuits. The only trick is that the mesh currents go around each loop and are defined as limited to their respective loop. These currents also are defined as going in the same direction, clockwise or counterclockwise. Again, for consistency, we always define mesh current in the clockwise direction. Of course, real currents are not limited to an individual loop in such an organized fashion. Mesh currents are artificial constructs that aid in solving multiloop problems. Nonetheless, the two mesh currents do account for all of the currents in the circuit. For example, the current through the capacitor would be the difference between the two mesh currents, $I_1(\omega) - I_2(\omega)$. This is not a problem as long as this difference in current is used when solving for the voltage across that capacitor.

Steps 1 and 2 lead to the circuit in Figure 13.11.

Step 3. Apply KVL around each loop, keeping in mind that the voltage drop (or rise) across the resistor shared by both meshes will be due to two currents, and since the currents are flowing in opposite directions their voltage contributions will have opposite signs: $I_1(\omega)$ produces the usual voltage drop, but $I_2(\omega)$ produces a voltage gain since it flows into the bottom of the resistor (again, mentally going clockwise around the loop). Each loop produces a separate equation.

Mesh 1: KVL following standard procedure, beginning in the lower left-hand corner.

$$V(\omega) - 0.79I_1(\omega) - \frac{0.57}{j\omega}I_1(\omega) + \frac{0.57}{j\omega}I_2(\omega) = 0$$

The capacitor produces a voltage term that is due to both I_1 and I_2, and the term produced by I_2 is positive because mesh current I_2 produces a voltage rise when going around the loop clockwise.

Mesh 2: KVL using the same procedure:

$$\frac{0.57}{j\omega}I_1(\omega) - \frac{0.57}{j\omega}I_2(\omega) - 0.033I_2(\omega) = 0$$

Now the capacitor contributes a voltage rise from current I_1 as we mentally go around the second loop clockwise, in addition to a voltage drop from I_2.

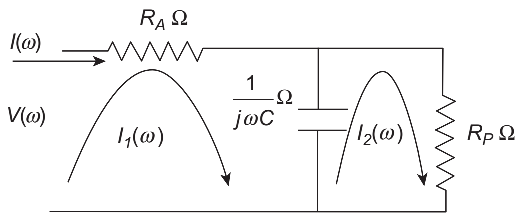


FIGURE 13.11 Windkessel model components have been converted to impedances and the mesh currents defined.

Step 4. Solve for the current(s). Rearranging the two equations, placing current on the right side and sources on the left, and separating the coefficients of the two current variables gives us two equations to be solved simultaneously. Pay particular attention to keeping the signs straight.

$$\begin{aligned} V(\omega) &= \left(0.033 + \frac{0.57}{j\omega}\right) I_1(\omega) - \frac{0.57}{j\omega} I_2(\omega) \\ 0 &= -\frac{0.57}{j\omega} I_1(\omega) + \left(0.79 + \frac{0.57}{j\omega}\right) I_2(\omega) \end{aligned}$$

With only two equations, it is possible to solve for the currents using substitution, avoiding matrix methods. However, with more than two meshes, matrix methods are easier and lend themselves to computer solutions. (The solution of a three-mesh circuit using MATLAB is given in the following discussion.) The spacing used in the above-mentioned equations facilitates transforming them into matrix notation:

$$\begin{vmatrix} V(\omega) \\ 0 \end{vmatrix} = \begin{vmatrix} 0.033 + \dfrac{0.57}{j\omega} & -\dfrac{0.57}{j\omega} \\ -\dfrac{0.57}{j\omega} & 0.79 + \dfrac{0.57}{j\omega} \end{vmatrix} \begin{vmatrix} I_1(\omega) \\ I_2(\omega) \end{vmatrix} \tag{13.9}$$

Solve for $I(\omega) \equiv I_1(\omega)$, using the method of determinants (Appendix G):

$$I_1(\omega) = \frac{\begin{vmatrix} V(\omega) & -\dfrac{0.57}{j\omega} \\ 0 & 0.79 + \dfrac{0.57}{j\omega} \end{vmatrix}}{\begin{vmatrix} 0.033 + \dfrac{0.57}{j\omega} & -\dfrac{0.57}{j\omega} \\ -\dfrac{0.57}{j\omega} & 0.79 + \dfrac{0.57}{j\omega} \end{vmatrix}} = \frac{V(\omega)\left(0.79 + \dfrac{0.57}{j\omega}\right)}{0.026 + \dfrac{0.02}{j\omega} + \dfrac{0.45}{j\omega} + \left(\dfrac{0.57}{j\omega}\right)^2 - \left(\dfrac{0.57}{j\omega}\right)^2}$$

$$I_1(\omega) = \frac{V(\omega)\left(0.79 + \dfrac{0.57}{j\omega}\right)}{0.026 + \dfrac{0.47}{j\omega}} = \frac{V(\omega)(0.57 + j0.79\omega)}{0.47 + j0.026\omega}$$

$$I_1(\omega) = \frac{0.47}{0.57}\,\frac{V(\omega)(1 + j1.39\omega)}{1 + j0.055\omega} = \frac{0.82V(\omega)(1 + j1.39\omega)}{1 + j0.055\omega}$$

Even this relatively simple two-mesh circuit involves considerable complex arithmetic, but it is easy to solve these problems using MATLAB.

Step 5. In this case we want the cardiac pressure as a function of blood flow. In this model, pressure is analogous to the input voltage, $V(\omega)$, and flow to the current, $I(\omega)$. To find $V(\omega)$ as a function of $I(\omega)$, we just factor out $V(\omega)$ and then invert the equation:

$$\frac{V(\omega)}{I(\omega)} = \frac{1.22(1 + j0.055\omega)}{1 + j1.39\omega} \tag{13.10}$$

The relationship between cardiac blood flow, $i(t)$ in this model, and cardiac pressure, $v(t)$ here, can be used to determine the pressure given the flow or vice versa. It is only necessary to have a quantitative description of the flow or the pressure. The pressure or flow need not be sinusoidal (which would not be very realistic); as long as it is periodic it can be decomposed into sinusoids using the Fourier transform. This approach is illustrated in the next example.

EXAMPLE 13.5

Use the three-element Windkessel model as described by Equation 13.10 to find the cardiac pressure given the blood flow. Use a realistic pulsatile waveform for blood flow as given by Equation 13.11. (This waveform is also used in Problem 14 of Chapter 12.) It might seem more logical to have cardiac pressure as the input and blood flow as the output, following the causal relationship, but in certain experimental situations we might measure blood flow and want to use the model to determine the cardiac pressure that produced that flow.

$$i(t) = \begin{cases} I_o \sin^2\left(\dfrac{\pi t}{T_1}\right) & 0 \leq t < T_1 \\ 0 & T_1 \leq t < T \end{cases} \tag{13.11}$$

Solution: To find the pressure we use the standard frequency domain approach: convert $i(t)$ to the frequency domain using the Fourier transform, multiply $I(\omega)$ by Equation 13.10 to get $V(\omega)$, and then take the inverse Fourier transform of $V(\omega)$ to get $v(t)$, the pressure. It is good to remember that in this approach you are solving for $V(\omega)$ a large number of times, each at a different frequency, i.e., the frequencies determined by the Fourier transform of $i(t)$. The fact that the overall answer can be obtained as just the sum of all the individual solutions is possible because superposition holds for this linear system.

Solutions are obtained for the pressure waveform over one cardiac cycle, which we assume takes 1.0 s. We make $f_s = 1.0$ kHz, so we need a 1000-point time vector to construct a 1.0-s blood flow waveform given in Equation 13.11. We assume that $I_o = 500$, the period of the heartbeat, T, is 1 s, and T_1 in Equation 13.11 is 0.3 s; in other words, blood flow occurs only during approximately one-third of the cardiac cycle (the "systolic" time period).

After we take the Fourier transform of $i(t)$ to get $I(\omega)$, we multiply it by the transfer function, $TF(\omega)$, given in Equation 13.10. To implement this transfer function, we construct a frequency vector, ω, that is of the same length as $i(t)$ (and, hence, the same length as $I(\omega)$) and has the same range as $I(\omega)$, from 1 to f_s Hz. After multiplying $I(\omega)$ by $TF(\omega)$ (point by point), we take the inverse Fourier transform. Although the output of the inverse Fourier transform should be real, computational errors can produce nonzero imaginary values, so we take the real part when plotting.

```
% Example 13.5 Plot the cardiac pressure from the Windkessel model.
%
fs = 1000;              % Sampling frequency
t = (1:1000)/fs;        % Time vector. One sec long
T1 = .3;                % Period of blood flow in sec
it = round(T1*fs);      % Index of blood flow
```

```
w = 2*pi*(1:fs);                          % Frequency vector in radians
i = 500*(sin(pi*t/T1)).^2;                % Define blood flow waveform, i(t).
                                            Equation 10.11
i(it:1000) = 0;                           % Zero period when no blood flow
subplot(2,1,1);
plot(t,i,'k');                            % Plot blood flow waveform
  .......labels and title.......
I = fft(i);                               % Take Fourier transform
TF = 1.22*(1+j*0.055*w)./(1+ j*1.39*w);   % Pressure/flow relationship
V = TF.*I;                                % Find V (in phasor)
v = ifft(V);                              % Find v(t). Inverse FT
subplot(2,1,2);
plot(t,real(v),'k');                      % Plot pressure waveform
....... Labels.......
```

Results: The plots produced by this program are shown in Figure 13.12. Cardiac pressure follows flow during the systolic period, but when flow stops (when the aortic valve closes) pressure falls slowly but does not become zero. A more complicated four-element Windkessel model is analyzed in one of the problems.

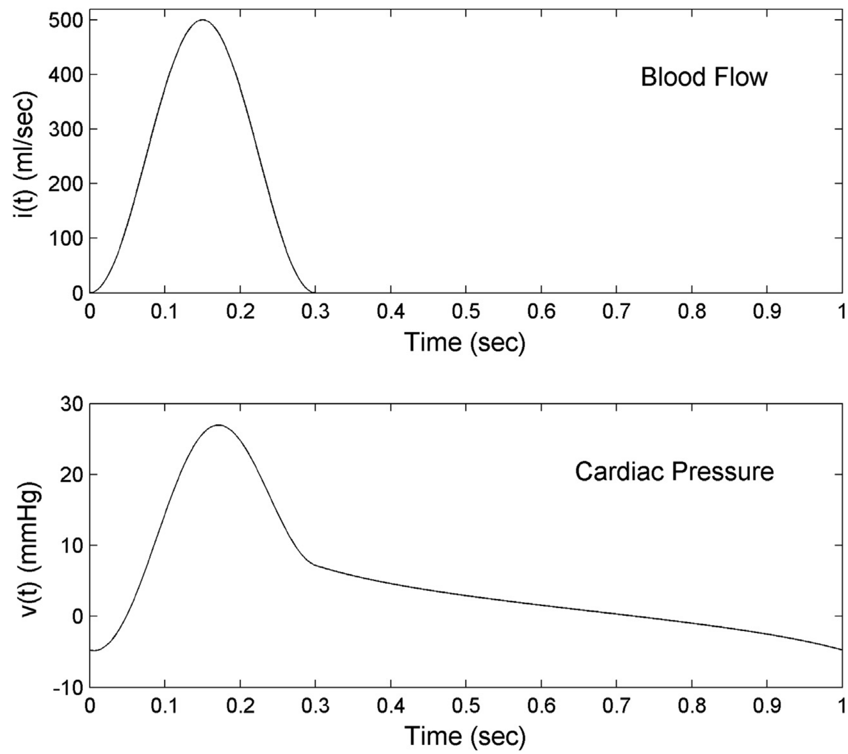


FIGURE 13.12 Pressure in the aorta left heart (lower curve) as a function of time as determined from the three-element Windkessel model. The upper curve shows the assumed blood flow pattern that produces this pressure.

13.2.2.1 *Shortcut Method for Multimesh Circuits*

There is a shortcut that makes it possible to write the matrix equation directly from inspection of the circuit. Regard the matrix equation, Equation 13.9, derived for the circuit in Figure 13.10 and repeated here:

$$\underbrace{\begin{vmatrix} V(\omega) \\ 0 \end{vmatrix}}_{\boldsymbol{V}} = \underbrace{\begin{vmatrix} 0.033+\dfrac{0.57}{j\omega} & -\dfrac{0.57}{j\omega} \\ -\dfrac{0.57}{j\omega} & 0.79+\dfrac{0.57}{j\omega} \end{vmatrix}}_{\boldsymbol{Z}} \underbrace{\begin{vmatrix} I_1(\omega) \\ I_2(\omega) \end{vmatrix}}_{\boldsymbol{I}}$$

This equation has the general form of $\boldsymbol{V} = \boldsymbol{Z}\ \boldsymbol{I}$ as indicated by the brackets. The left-hand side contains only the sources; the right-hand side contains a matrix of impedances multiplied by a vector containing the mesh currents. The source column vector contains the source in Mesh 1 in the upper position and the source in Mesh 2 in the lower position, which, in Example 13.4, is 0.0 because there are no sources in Mesh 2. The current column vector consists of just the two mesh currents. The impedance matrix also relates topographically to the circuit: the upper left entry is the sum of impedances in Mesh 1, the lower right is the sum of impedances in Mesh 2, and the off-diagonals (upper right and lower left) contain the negative of the sum of impedances common to both loops. In this circuit there is only one element common to both elements, the capacitor, so the off-diagonals contain the negative of this element, but other circuits could have multiple elements common to the two meshes. Putting this verbal description into mathematical form:

$$\begin{vmatrix} \Sigma V_S \text{ Mesh } 1 \\ \Sigma V_S \text{ Mesh } 2 \end{vmatrix} = \begin{vmatrix} \Sigma Z \text{ Mesh } 1 & -\Sigma Z \text{ Mesh } 1\&2 \\ -\Sigma Z \text{ Mesh } 1\&2 & \Sigma Z \text{ Mesh } 2 \end{vmatrix} \begin{vmatrix} I_1 \\ I_2 \end{vmatrix} \tag{13.12}$$

In this equation the impedance sums, $\Sigma\ Z$, are additions; there is no need to worry about signs except if the impedance itself is negative, as for a capacitor. However, the summation of voltage sources, the $\Sigma\ V_S$, still requires some care, since the summations must take the voltage source signs into consideration. For example, the source in Mesh 1 ($\Sigma\ V_S$ Mesh 1) in Example 10.4 has a positive sign because it represents a voltage rise when going around the loop clockwise. If more than one source appears in a mesh, these sources are algebraically summed using the rule for keeping track of signs.

This shortcut rule also applies to circuits that use Laplace representations. The approach can easily be extended to circuits having any number of meshes, although the subsequent calculations become tedious for three or more meshes unless computer assistance is used. The extension to three meshes is given in the following example, but the solution is determined using MATLAB.

13.2.3 Mesh Analysis: MATLAB Implementation

EXAMPLE 13.6

Solve for V_{out} in the three-mesh network in Figure 13.13. This circuit uses realistic values for R, L, and C.

Solution: Follow the steps used in the previous examples, but use the shortcut method in Step 3. In Step 4, solve for the currents using MATLAB, and in Step 5 solve for V_{out} using MATLAB.

Steps 1 and 2. Figure 13.14 shows both the original circuit and the circuit after Step 1 and 2 with the elements represented in phasor representation and the phasor currents defined.

The impedances for L and C are determined as:

$$Z_L = j\omega L = j2\pi fL = j2\pi 10^4\left(10\times 10^{-3}\right) = j628\ \Omega$$

$$Z_{C2} = \frac{1}{j\omega C_2} = \frac{1}{j2\pi f C_2} = \frac{1}{j2\pi 10^4\left(0.022\times 10^{-6}\right)} = \frac{1}{j1.38\times 10^{-3}} = -j723\ \Omega$$

$$\text{Similarly}: Z_{C2} = \frac{1}{j2\pi f C_2} = \frac{1}{j2\pi 10^4\left(0.01\times 10^{-6}\right)} = j1592\ \Omega$$

Step 3. The matrix equation for Step 3 is only a modification of Equation 13.11 where the voltage and current vectors have three elements each and the impedance matrix is extended to a 3×3 matrix written as:

$$\begin{vmatrix} \Sigma V_S \text{ Mesh } 1 \\ \Sigma V_S \text{ Mesh } 2 \\ \Sigma V_S \text{ Mesh } 3 \end{vmatrix} = \begin{vmatrix} \Sigma Z \text{ Mesh } 1 & -\Sigma Z \text{ Mesh } 1\&2 & -\Sigma Z \text{ Mesh } 1\&3 \\ -\Sigma Z \text{ Mesh } 1\&2 & \Sigma Z \text{ Mesh } 2 & -\Sigma Z \text{ Mesh } 2\&3 \\ -\Sigma Z \text{ Mesh } 1\&3 & -\Sigma Z \text{ Mesh } 2\&3 & \Sigma Z \text{ Mesh } 3 \end{vmatrix} \begin{vmatrix} I_1 \\ I_2 \\ I_3 \end{vmatrix} \tag{13.13}$$

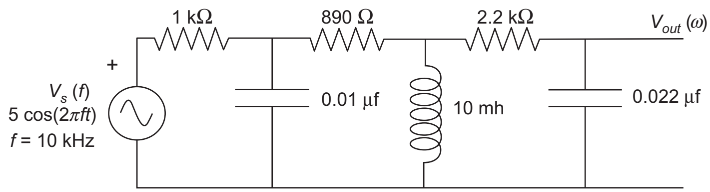


FIGURE 13.13 A three-mesh circuit analyzed in Example 13.6.

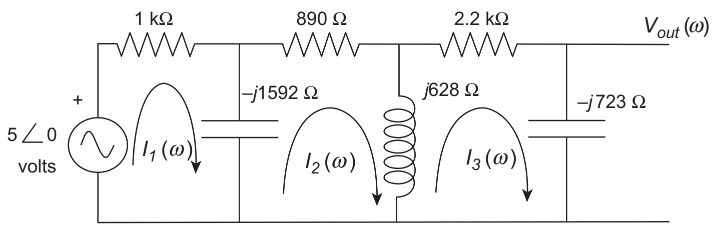


FIGURE 13.14 The circuit shown in Figure 13.13 after the mesh currents have been assigned, the voltage source has been converted to phasor notation, and the elements have been converted to their equivalent impedances.

where Σ Z Mesh 1, Σ Z Mesh 2, and Σ Z Mesh 3 are the sums of impedances in the three meshes; Σ Z Mesh 1 & 2 is the sum of impedances common to Meshes 1 and 2; Σ Z Mesh 1 & 3 is the sum of impedances common to Meshes 1 and 3; and Σ Z Mesh 2 & 3 is the sum of impedances common to Meshes 2 and 3. Note that all the off-diagonals are negative sums and the impedance matrix has symmetry about the diagonal. Such symmetry is often found in matrix algebra, and a matrix with this symmetry is termed a "Toplitz matrix." In this particular network there are no impedances common to Meshes 1 and 3, but one of the problems has a three-mesh circuit in which all three meshes have at least one element in common.

Applying Equation 13.13 to the network in Figure 13.14 gives rise to the matrix equation for this network:

$$\begin{vmatrix} 5 \\ 0 \\ 0 \end{vmatrix} = \begin{vmatrix} 1000 - j1592 & j1592 & 0 \\ j1592 & 890 - j964 & -j628 \\ 0 & -j628 & 2200 - j95 \end{vmatrix} \begin{vmatrix} I_1(\omega) \\ I_2(\omega) \\ I_3(\omega) \end{vmatrix}$$

Note that the $-j964$ in the middle term is the algebraic sum of $j628 - j1592$ and the $-j95$ in the lower right is the sum of $j628 - j723$.

Steps 4 and 5. To find V_{out} we need only the current through the final 0.022-μF capacitor, $I_3(\omega)$. Solve for this current, then the desired voltage is $-j723\ I_3(\omega)$. The MATLAB program does both. First it defines the voltage vector and impedance matrix and then solves the matrix equation for the current vector. The current, `I(3)`, is then used to find the output voltage.

```
% Example 13.6 Solution of a three-mesh network
%
V = [10 0 0]';            % Note the use of the transpose symbol, '
% Define the impedance matrix
Z= [1000 - 1i*1592, 1i*1592, 0;...
    1i*1592, 890-1i*964, -1i*628;...
    0, -1i*628, 2200-1i*95];
I = Z\V                   % Solve for the currents
Vout = I(3)*(-723*1i)     % Find and output the requested voltage
Vmag = abs(Vout)          % also as magnitude and phase
Vphase = angle(Vout)*306/(2*pi)
```

Results: The output of this program for the three-mesh currents is:

```
I =
  0.0042 + 0.0007i
  0.0037 - 0.0029i
  0.0008 + 0.0011i
```

The output voltages are:

```
  Vout =  0.7908 - 0.5659i
  Vmag =  0.9724 V phase =  -30.2476 deg
```

Again, MATLAB accepts either `i`, `1i`, or `j` to represent a complex number, but outputs only using `i`. (In newer versions of MATLAB, the symbol `1i` is recommended to represent an imaginary number as this gives improved speed and stability.)

The time domain output is determined directly from the phasor output given earlier. Recall that $\omega = 10^4$ rad/s:

$$V_{out}(t) = 0.97\cos\left(2\pi 10^4 t - 30.2\right) \text{ V}.$$

Analysis: In the MATLAB program, the voltage vector is written as a row vector, but it should be a column vector as in Equation 13.13, so the MATLAB transpose operator (single quote) is used.[3] The second line defines the impedance matrix using standard MATLAB notation. The third line solves for the three currents by matrix inversion, implementing the equation $\boldsymbol{I} = \boldsymbol{Z}^{-1}\,\boldsymbol{V}$ using the backslash (\) operator to invert the impedance matrix. The fourth line multiplies the third mesh current, $I_3(\omega)$, by the capacitor impedance to get V_{out}. The next two lines convert V_{out} from real and imaginary components into the more conventional polar form.

[3]It could just as easily been entered as a column vector directly (`V = [10; 0; 0]`); the transposed row vector was just the whim of the programmer.

If MATLAB can solve one matrix equation, it can solve it again and again. By making the inputs sinusoids of varying frequency, we can generate the Bode plot of any network. The next example treats a network as an input–output system where the source voltage is the input and the voltage across one of the elements is the output. MATLAB is called upon to solve the problem many times over at different sinusoidal frequencies and for three component values.

EXAMPLE 13.7

Plot the magnitude Bode plot in hertz of the transfer function of the network in Figure 13.15 with inputs and outputs as shown. Plot the Bode plot for three values of the capacitance: 0.01, 0.1, and 1.0 μf. The three capacitors all have the same values and the three resistors are all 2.0 kΩ.

Solution: We could first manually determine the frequency domain transfer function and then plot that function for the three capacitance values. However, that would require some tedious

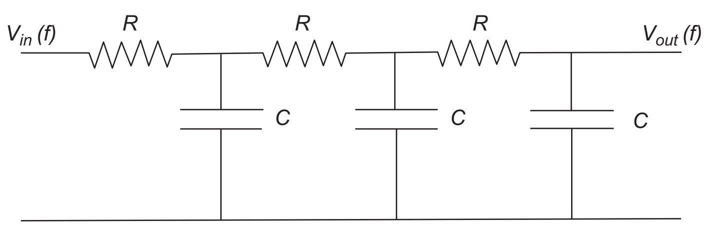


FIGURE 13.15 A network that can be viewed as an input–output system where the transfer function is $TF = V_{out}/V_{in}$. The Bode plot (i.e., system frequency spectrum) of this system is found in Example 13.7. The three resistors are all 2.0 kΩ. The three capacitors also have the same value, but the Bode plot is determined for three different capacitance values: 0.01, 0.1, and 1.0 μf.

algebra, particularly as we have to leave the capacitance values as variables. It is easier for us to rely more heavily on MATLAB. Set up the KVL matrix equations and the output equation, solve these equations in MATLAB over a range of sinusoidal frequencies, and plot the output voltages for these frequencies.

Follow the five-step procedure to generate the KVL matrix problem, then find the output in terms of the third mesh current. Use MATLAB to solve for this output current from the KVL matrix, then solve for V_{out}, the voltage across the output capacitor. Assume V_{in} is an ideal sinusoidal source that varies in frequency. Select a frequency range that includes any interesting changes in the Bode plot. This might require trial and error, so, to begin, let us try a large frequency range of between 10 Hz and 80 kHz. To keep the computational time down, use 20-Hz intervals. Use one loop to vary the frequency and another outside loop to change the capacitor values.

Steps 1 and 2. At this point you can implement these steps by inspection. The capacitors become $1/j\omega C$ and the mesh currents are $I_1(\omega)$, $I_2(\omega)$, and $I_3(\omega)$. No need to redraw the network.

Step 3. Again the KVL equation for this circuit can be done by inspection.

$$\begin{vmatrix} V_{in}(s) \\ 0 \\ 0 \end{vmatrix} = \begin{vmatrix} R+1/j\omega C & -1/j\omega C & 0 \\ -1/j\omega C & R+2/j\omega C & -1/j\omega C \\ 0 & -1/j\omega C & R+2/j\omega C \end{vmatrix} \begin{vmatrix} I_1(\omega) \\ I_2(\omega) \\ I_3(\omega) \end{vmatrix}$$

Steps 4 and 5. These steps are implemented in the following MATLAB program noting that:

$$V_{out}(\omega) = 1/j\omega C^{I_3(\omega)}$$

The MATLAB implementation of the KVL and output equations is shown in the following discussion. The matrix equation and its solutions are placed in a loop that solves the equation for 4000 values of frequency ranging from 10 to 80 kHz. In this example, frequency is requested in hertz. The desired voltage V_{out} is determined from mesh current I_3. A Bode plot is constructed by plotting 20 log $|V_{out}|$ against log frequency since $V_{in} = 1$ at all frequencies. This loop is placed within another loop that repeats the 4000 solutions for the three values of capacitance.

```
% Example 13.7 Solution of a 3-mesh network
R = 2000;                         % Resistor values
C = [1, 0.1, 0.01]*1e-6;          % Capacitance values
V = [ 1; 0; 0];                   % Mesh voltages: Vin = 1
for i = 1:3                       % Capacitance value loop
  for k = 1:4000                  % Frequency loop
    f(k) = 10 + (k-1)*20;         % Determine frequency: 10- 80 kHz
    w = 2*pi*f(k);                % Calculate frequency in radians
    Xc = 1/(j*w*C(i));            % Calculate capacitance impedance
    Z = [R+Xc, -Xc, 0; -Xc, R+2*Xc, -Xc; 0, -Xc, R+2*Xc];  % Mesh equation
    I = Z\V;                      % Solve for currents
    Vout(k) = Xc*I(3);            % Solve for Vout
  .....label capacitor value.....
  end
  semilogx(f,20*log10(abs(Vout))); hold on;  % Bode plot
end
```

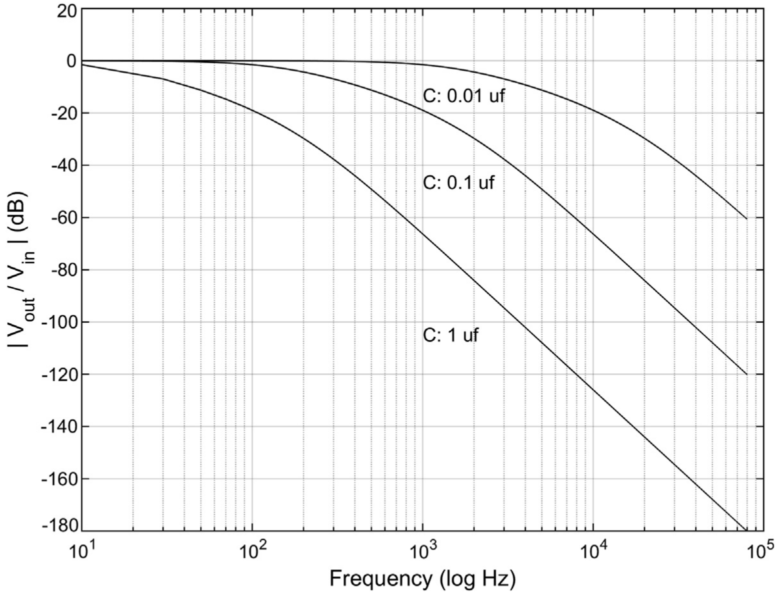


FIGURE 13.16 The Bode plot of the network shown in Figure 13.16 where V_{out} is the output and V_{in} the input. The frequency characteristics are shown for three capacitor values as indicated. These plots show that the network is a low-pass filter with a slope of 60 dB/decade. Increasing the capacitor values decreases the cutoff frequency.

The Bode plots generated by this program are shown in Figure 13.16. The plots of the three curves look like low-pass filters with downward slopes of 60 dB/decade, each having a different cutoff frequency (i.e., −3 dB attenuation points). Based on the Bode plot slope, this network is a third-order low-pass filter with a cutoff frequency that varies with the value of C. In fact, the cutoff frequency is approximately $1/2\pi RC$. Usually filters above first-order are constructed using active elements (see Chapter 15), but the circuit of Figure 13.15 is easy to construct and is occasionally used as a convenient third-order filter in real-world electronics.

13.3 CONSERVATION LAWS: KIRCHHOFF'S CURRENT LAW—NODAL ANALYSIS

KCL can also be used to analyze circuits. This law, based on the conservation of charge, was given in Equation 13.2 and is repeated here:

$$\sum_{\text{Node}} i = 0 \tag{13.14}$$

KCL is best suited for analyzing circuits with many loops but only a few connection points. Figure 13.17 shows the Hodgkin–Huxley model for the nerve membrane. The three voltage–resistor combinations represent the potassium membrane channel, the sodium membrane channel, and the chloride membrane channel, whereas C is the membrane capacitance. Analyzing this circuit requires four mesh equations (V is actually a voltage source) but only

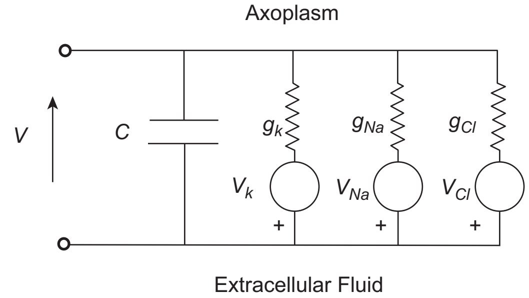


FIGURE 13.17 The Hodgkin–Huxley model for nerve membrane. The three voltage–resistor combinations represent the potassium membrane channel (K), the sodium membrane channel (Na), and the chloride membrane channel (Cl), whereas C is the membrane capacitance. This circuit contains four meshes, but only one independent node (assuming the bottom node is ground).

one nodal equation. In this model, most of the components are nonlinear, at least during an action potential, so the model cannot be solved analytically. Nonetheless, the defining equation(s) would be generated using nodal analysis and can be solved using computer simulation.

Another example of a circuit appropriate for nodal analysis is shown in Figure 13.18. This circuit has four meshes and mesh analysis would give rise to four simultaneous equations. This same circuit has only two nodes (marked A and B; again ground points do not count) and requires solving only two nodal equations. If MATLAB is used, then solving a four-equation problem is really not all that much harder than solving a two-equation problem; it is just a matter of adding a few more entries into the voltage vector and impedance matrix. However, when circuits are used as models representing physiological processes as in Figure 13.17, the more concise description given by nodal equations is of great value.

The circuit in Figure 13.18 contains a current source, not the familiar voltage source. This is because nodal analysis is an application of a current law, so it is easier to implement if the sources are current sources. A similar statement could be made about mesh analysis: mesh analysis involves voltage summation and it is easier to implement if all sources are voltage sources. The need to have only current sources may seem like a drawback to the application of nodal analysis, but we see in Chapter 14 that it is easy to convert voltage sources to equivalent current sources and vice versa, so this requirement is not really a handicap. In this

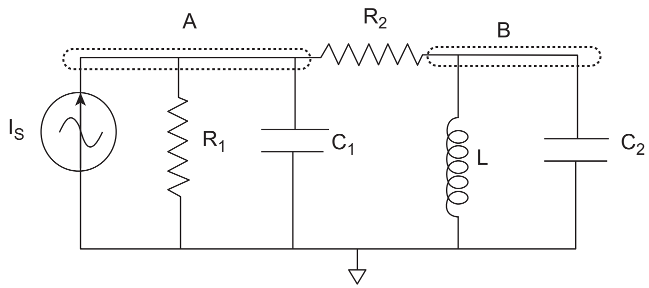


FIGURE 13.18 A circuit with four meshes and two nodes. The power source in this circuit is a current generator, I_S.

chapter, all nodal analysis examples use current sources, but the technique can be applied equally well to voltage sources after the simple conversion described in the next chapter.

Analyzing circuits using nodal analysis follows the same five-step procedure as that used in mesh analysis. In fact, Steps 4 and 5 are the same. Step 1 could also be the same, but often elements are converted to $1/Z$, instead of simply Z. Inverse impedance, $Y = 1/Z$, is termed "admittance" since it describes how current is admitted as opposed to impeded. In Step 2 the node voltages are assigned rather than the mesh currents and in Step 3 the equations are generated using KCL.

The equations developed from KCL have a sort of inverse symmetry with those of mesh analysis. In mesh analysis, we are writing matrix equations of the form:

$$\boldsymbol{v} = \boldsymbol{Zi} \tag{13.15}$$

where $\boldsymbol{v}$ is a voltage vector, $\boldsymbol{i}$ is a current vector, and $\boldsymbol{Z}$ is an impedance matrix (Equations 13.12 and 13.13). In nodal analysis we are writing matrix equations in the form:

$$\boldsymbol{i} = \boldsymbol{Yv} \tag{13.16}$$

where $\boldsymbol{Y}$ is a matrix, termed the "admittance matrix" containing the inverse impedances. The terms $\boldsymbol{v}$ and $\boldsymbol{i}$ are vectors as in Equation 13.15, although they occupy different positions in the equation.

EXAMPLE 13.8

Find the voltage V_A in the circuit in Figure 13.19.

Solution: This circuit requires two mesh equations (after conversion of the current source to an equivalent voltage source as explained in Chapter 14), but only one nodal equation. Moreover, it conveniently contains a current source, making nodal analysis even easier. There are four currents flowing into or out of the single node at the top of the circuit labeled A. The current in the current source branch is 0.1 cos ($2\pi 10t$), and the current in the other three branches is equal to the voltage, V_A, divided by the impedance of the branch, i.e., $I(\omega) = V_A(\omega)/Z_{Branch}(\omega)$. By KCL, these four currents sum to zero.

After applying Steps 1 and 2, the network becomes as shown in Figure 13.20. If we define $V_A(\omega)$ as a positive voltage, then the current through the passive elements flows downward as shown owing to the voltage–current polarity rule for passive elements. The frequency in radians is $\omega = 2\pi f = 2\pi 10 = 62.8$ rad/s.

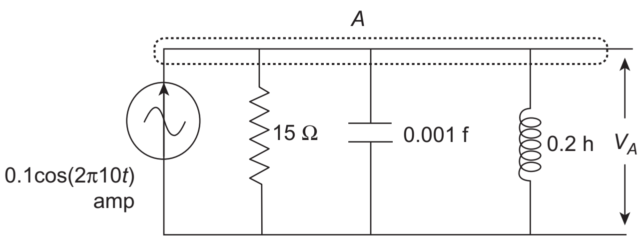


FIGURE 13.19 The network analyzed in Example 13.8. This network contains only one node and can be analyzed with a single KCL equation.

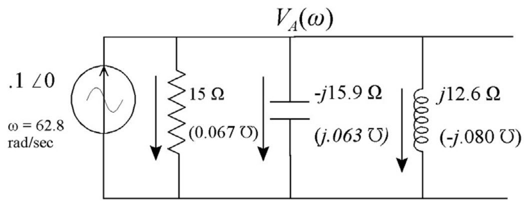


FIGURE 13.20 The circuit of Figure 13.19 after assigning nodal currents and converting the components values to impedances and admittances. (The units for admittance are called mhos; that is, ohms spelled backward. The symbol for admittance is the inverted symbol for ohms, ℧.)

Step 3 applies KCL: the four currents sum to zero. As with mesh analysis, care has to be taken that the signs are correct. The current source flows into node A so it is positive, but the other three currents flow out so they are negative.

$$i_S(\omega) - i_R(\omega) - i_C(\omega) - i_L(\omega) = 0 \quad \text{(KCL)}$$

$$I_S - \frac{V_A(\omega)}{R} - \frac{V_A(\omega)}{1/j\omega C} - \frac{V_A(\omega)}{j\omega L} = 0$$

$$0.1 + \frac{V_A(\omega)}{15} + \frac{V_A(\omega)}{-j15.4} + \frac{V_A(\omega)}{j13} = 0$$

Now we can solve this single equation for $V_A(\omega)$. The equation is easier written in terms of admittances: $Y = 1/Z$. The values of the admittances are shown in parentheses in the circuit given previously. Using admittances:

$$I_S + Y_R V_A(\omega) + Y_C V_A(\omega) + Y_L V_A(\omega) = 0$$

$$I_S + V_A(\omega)(1/R + j\omega C + 1/j\omega L) = 0$$

$$0.1 + V_A(\omega)(0.067 + j0.065 - j0.077) = 0$$

$$V_A(\omega) = \frac{0.1}{0.067 + j0.065 - j0.077} = \frac{0.1}{0.067 - j0.012}$$

$$V_A(\omega) = \frac{0.1}{0.068\angle - 10} = 1.47\angle 10 \ \text{V}$$

Moving to multinodal systems, we go directly to the shortcut, matrix equation. If we apply KCL to circuits with multiple nodes, we find that the equations fall into a pattern similar to that of mesh analysis, except that they have the form of Equation 13.16: $\boldsymbol{i} = \boldsymbol{Yv}$. The admittance matrix consists of the summed admittances (i.e., $1/Z$'s) that are common to each node along the diagonal and the summed admittances between nodes on the off-diagonals. This general format is shown here for a three-node circuit:

$$\begin{vmatrix} \Sigma I_1 \\ \Sigma I_2 \\ \Sigma I_3 \end{vmatrix} = \begin{vmatrix} \Sigma Y \ \text{Node 1} & -\Sigma Y \ \text{Node 1\&2} & -\Sigma Y \ \text{Node \&3} \\ -\Sigma Z \ \text{Mesh 1\&2} & \Sigma Y \ \text{Node 2} & -\Sigma Y \ \text{Node 2\&3} \\ -\Sigma Y \ \text{Node 1\&3} & -\Sigma Y \ \text{Node 2\&3} & \Sigma Y \ \text{Node 3} \end{vmatrix} \begin{vmatrix} V_1 \\ V_2 \\ V_3 \end{vmatrix} \tag{13.17}$$

The application of Equation 13.17 is straightforward and follows the same pattern as in mesh analysis. An example of nodal analysis to the two-node circuit is given in Example 13.9.

EXAMPLE 13.9

Find the voltage, v_2, in the circuit shown in Figure 13.21. This circuit, like the one in Figure 13.18, has two nodes having voltages v_1 and v_2.

Solution: Apply nodal analysis to this two-node circuit. Follow the step-by-step procedure outlined earlier, but in Step 3 write the matrix equation directly, modifying Equation 13.17 for two nodes. Implement Step 4 to solve for V_B using MATLAB.

Step 1. Convert all the elements to phasor admittances. Note that $\omega = 20$ rad/s.

Step 2. Assign nodal voltages. This has already been done in the circuit. The circuit after modification by these two steps is shown in Figure 13.22.

Step 3. Generate the matrix equations directly following a reduced version of Equation 13.15. Note that inductors now have $-j$ values, whereas conductors have $+j$ values. Also note that the two nodes share two components, so the shared admittance will be the sum of the admittances from each component:

$$\sum Y_{\text{nodel }1,2} = .004 - j.007$$

Hence the KCL circuit equation becomes:

$$\begin{vmatrix} 0.5 \\ 0 \end{vmatrix} = \begin{vmatrix} 0.01 + 0.004 + j0.01 - j0.007 & -0.004 + j0.007 \\ -0.004 + j0.007 & 0.004 - j0.005 - j0.007 + j0.04 \end{vmatrix} \begin{vmatrix} V_1 \\ V_2 \end{vmatrix}$$

$$\begin{vmatrix} 0.5 \\ 0 \end{vmatrix} = \begin{vmatrix} .014 + j.003 & -0.004 + j0.007 \\ -.004 + j.007 & 0.004 - j0.028 \end{vmatrix} \begin{vmatrix} V_1 \\ V_2 \end{vmatrix}$$

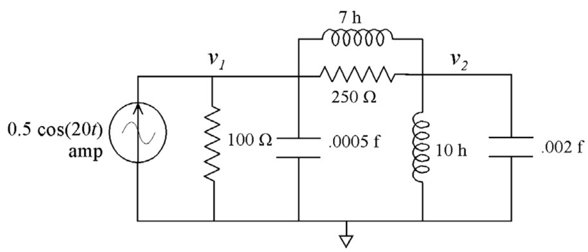


FIGURE 13.21 Two-node circuit analyzed using KCL matrix equations in Example 13.9.

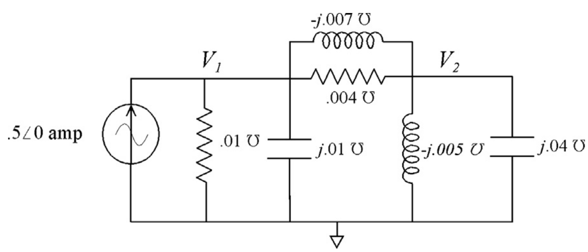


FIGURE 13.22 The circuit shown in Figure 13.21 after assignment of nodal currents and converting element values to admittances. Note that the component values are given as admittances in mhos (℧).

Step 4. This matrix equation can easily be solved using MATLAB as illustrated by the following code.

```
% Example 13.9  Solution of two-node matrix equation
%
I = [.5; 0];                              % Assign current vector
Y11 = 0.01 +.004 + 1i*.01 -1i*.007;       % Assign admittances
Y12 = (.004-1i*.007);
Y22 = 0.004 - 1i*.005 - 1i*.007 + 1i*.04;
Y1 = [Y11 -Y12; -Y12 Y22];                % Admittance matrix
V = Y\I;                                  % Solve for voltages
Magnitude = abs(V(2))                     % Magnitude and phase of V2
Phase = angle(V(2))*360/(2*pi)
```

The output gives the magnitude and phase of V_2 as:

```
Mag = 8.9234;  Phase  -149.3235
```

Converting to the time domain:

$$v_2(t) = 8.92\cos(20t - 149)\ \mathrm{V}$$

The approach used in Example 13.9 can be extended to three-node or even higher-node circuits without great difficulty. A three-node problem is given at the end of the chapter. Nodal analysis applies equally well to networks represented in Laplace notation. The basic five-step approach can also be applied to the analysis of lumped-parameter mechanical systems as described in the next section.

13.4 CONSERVATION LAWS: NEWTON'S LAW—MECHANICAL SYSTEMS

The analysis of lumped-parameter mechanical systems also uses a conservation law, one based on the conservation of energy. In mechanical systems, force is potential energy, since force acting through a distance produces work: $W = \int F dx$. The mechanical version of KVL (the electrical law based on the conservation of energy) states that the sum of the forces around any one connection point must be zero. This is a form of the classic law associated with Newton:

$$\sum_{\text{Point}} F = 0 \tag{13.18}$$

In this application, a connection point includes all connections between the mechanical elements that are at the same velocity (just as a node is all the points at the same voltage).

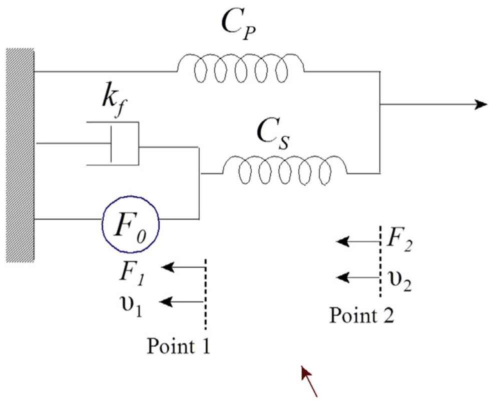


FIGURE 13.23 Linear mechanical model of skeletal muscle. F_0 is the force produced by the active contractile element, C_P and C_S are the parallel and series elasticities, and k_f is the viscous damping associated with the tissue.

Figure 13.23 shows one version of the linear model for skeletal muscle. Skeletal muscle has two different elastic elements: a parallel elastic element, C_p, and a series elastic element, C_s. (In mechanical systems, the symbol C is used to indicate a component's compliance, the inverse of elasticity, $1/k_e$.) The force generator, F_0, represents the active contractile element, and k_f represents viscous damping inherent in the muscle tissue. The muscle model has two connection points that may have different velocities, labeled Point 1 and Point 2. The positive force is defined inward, reflecting the fact that muscles can generate only contractile force. This is the reason they are so often found in agonist–antagonist pairs.

Since this system has two different velocities, its analysis would require the simultaneous solution of equations. The system is the mechanical equivalent of a two-mesh electrical circuit. The two equations would be written around Points 1 and 2: the sum of forces around each point must be zero. The graphic on the left represents a zero-velocity point or a solid wall, the analog of a ground point in an electrical system.

The muscle model will be analyzed in Example 13.11, but for a first example of the application of Newton's law (Equation 13.18) we turn to a less complicated, single equation system with a single velocity point as shown in Figure 13.24.

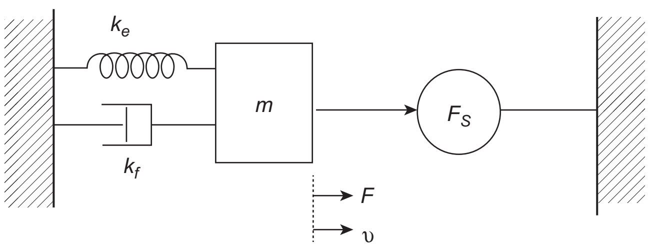


FIGURE 13.24 A simple mechanical system consisting of a force generator, F_S, mass, m, friction element, k_f, and elasticity, k_e. The velocity of the mass, υ, is found using Equation 13.18 in Example 13.10.

EXAMPLE 13.10

Find the velocity and displacement of the mass in the mechanical system shown in Figure 13.24. The force is $F_S(t) = 5\cos(2t + 30)$ dynes. The following parameters also apply:

$$k_f = 5\ s/\text{cm}; \quad k_e = 8\ \text{dyn/cm}; \quad m = 3\ \text{g}.$$

Solution: In this example, all units are in the cgs metric system and therefore are comparable. This is not always the case in mechanical systems; in Example 13.11, conversion of units is required. To analyze this system, we follow the same five-step plan developed for electric circuits.

Step 1. Convert the variables to phasor notation and represent the passive elements by their phasor impedances. Since $\omega = 2$:

$$Z_f = k_f = 5; \quad Z_e = \frac{k_e}{j\omega} = \frac{8}{j2} = -j4; \quad Z_m = j\omega\, m = j2(3) = j6$$

Step 2. Assign variable directions. In mechanical systems, we use the convention of assigning the force and velocity in the same direction, but the direction (right or left) is arbitrary (in this example it is to the right). This is analogous to assigning currents as counterclockwise and keeping track of voltage polarities by going in the same direction. By assigning force and velocity in the same direction, the polarity of passive elements is always negative just as in electric circuits, so the equations look similar.

Step 3. Apply Newton's law about the force−velocity point (next to the mass):

$$\sum F = 0; \quad F_S(\omega) - k_f v(\omega) - \frac{k_e}{j\omega} v(\omega) - j\omega m v(\omega) = 0$$

$$5\angle 30 - v(\omega)(5 - j4 + j6) = 0$$

The first three steps follow a path parallel to that followed in the KVL analysis, whereas the last two steps are essentially identical: solve for the velocity (analogous to current), then any other variable of interest such as a force or, in this case, a displacement.

Step 4. Solve for the phasor velocity:

$$v(\omega) = \frac{5\angle 30}{5 - j4 + j6} = \frac{5\angle 30}{5 + j2} = \frac{5\angle 30}{5.39\angle 21.8} = 0.93\angle 8.2\ \text{cm/s}$$

Step 5. Solve for displacement. Since $x(t) = \int v dt$, integration in the phasor domain is division by $j\omega$; so $x(\omega) = v(\omega)/j\omega$:

$$x(\omega) = \frac{v(\omega)}{j\omega} = \frac{0.93\angle 8.2}{j2} = \frac{0.93\angle 8.2}{2\angle 90} = 0.465\angle -81.8\ \text{cm}$$

Both $v(\omega)$ and $x(\omega)$ can be converted to the time domain if desired:

$$v(t) = 0.93\ \cos(2t + 8.2)\ \text{cm/s}$$
$$x(t) = 0.4610\ \cos(2t - 81.8)\ \text{cm}$$

The next example is more complicated in two ways: (1) the component values are not given and must be carried as variables through the algebra, and (2) there are two summation points in the problem, so two equations are required that must be solved simultaneously.

EXAMPLE 13.11

Find the force out of the skeletal muscle model in Figure 13.23.

Solution: After converting to the phasor domain, write Newton's law (Equation 13.18) around points 1 and 2. Solve for $v_2(\omega)$, then $F_2(\omega) = v_2(\omega)Z_{C_P}(\omega) = v_2(\omega)1/j\omega C_P$. In this solution the algebra is a bit tedious because the various parameters remain as variables, but the procedure is otherwise straightforward.

Steps 1 and 2. The force and velocity assignments are given in Figure 13.22. The various components become:

$$F_o \to F_o(\omega); \quad k_f \to k_f; \quad C_P \to \frac{1}{j\omega C_P} \quad C_S \to \frac{1}{j\omega C_S}$$

The equation around Point 1 is:

$$F_o - k_f v_1(\omega) - \frac{1}{j\omega C_s}(v_1(\omega) - v_2(\omega)) = 0$$

Note that the force generated by the elastic element C_s depends on the difference in velocities between Points 1 and 2. The force that is generated by $v_1(\omega) - v_2(\omega)$ is in the opposite direction of F_1, which accounts for the negative sign in front of this term.

Step 3. The equation around Point 2 is:

$$0 - \frac{1}{j\omega C_P}v_2(\omega) - \frac{1}{j\omega C_s}(v_2(\omega) - v_1(\omega)) = 0$$

Rearranging to separate coefficients of $v_1(\omega)$ and $v_2(\omega)$:

$$F_o = \left(k_f + \frac{1}{j\omega C_s}\right)v_1(\omega) - \frac{1}{j\omega C_s}v_2(\omega)$$

$$0 = \frac{1}{j\omega C_s}v_1(\omega) + \left(\frac{1}{j\omega C_s} + \frac{1}{j\omega C_p}\right)v_2(\omega)$$

Step 4. Solving $v_2(\omega)$:

$$v2(\omega) = \frac{\begin{vmatrix} F_o & -\dfrac{1}{j\omega C_s} \\ 0 & \dfrac{1}{j\omega C_s} + \dfrac{1}{j\omega C_p} \end{vmatrix}}{\begin{vmatrix} k_f + \dfrac{1}{j\omega C_s} & -\dfrac{1}{j\omega C_s} \\ -\dfrac{1}{j\omega C_s} & \dfrac{1}{j\omega C_s} + \dfrac{1}{j\omega C_p} \end{vmatrix}}$$

$$= \frac{F_o\left(\dfrac{1}{j\omega C_s} + \dfrac{1}{j\omega C_p}\right)}{\dfrac{k_f}{j\omega C_s} + \dfrac{k_f}{j\omega C_p} + \left(\dfrac{1}{j\omega C_s}\right)^2 + \dfrac{1}{(j\omega)^2 C_p C_s} - \left(\dfrac{1}{j\omega C_s}\right)^2}$$

Canceling the two terms, noting that $j^2 = -1$, and multiplying through by $(j\omega)^2$:

$$v_2(\omega) = \frac{j\omega F_o\left(\frac{1}{C_s}+\frac{1}{C_p}\right)}{\frac{1}{C_pC_s}+j\omega\left(\frac{k_f}{C_s}+\frac{k_f}{C_p}\right)} = \frac{j\omega F_o(C_s+C_p)}{1+j\omega(k_fC_s+k_fC_p)}$$

Step 5. Now to find the force at the output, multiply $v_2(\omega)$ by the impedance of the parallel elastic element, C_P.

$$F(\omega) = v_2(\omega)\left(\frac{1}{j\omega C_p}\right) = \frac{j\omega F_o(C_s+C_p)}{1+j\omega(k_fC_s+k_fC_p)}\left(\frac{1}{j\omega C_p}\right) = \frac{F_o(C_s+C_p)/C_p}{1+j\omega(k_fC_s+k_fC_p)}$$

In the next chapter we find that this equation, if viewed as a transfer function $F(\omega)/v_2(\omega)$, has the same general properties as those of a low-pass filter. Although real muscle can be fairly well described by the elements in Figure 13.23, the equation does not take into account the component nonlinearities. Nonetheless, this linear analysis provides a starting point for more complicated models and analyses.

The final example solves a two-equation system that includes a mass. This somewhat involved example also demonstrates how to convert various measurement units to the cgs system.

EXAMPLE 13.12

Find the velocity of the mass in the system shown in Figure 13.25. Assume that $F_S(t) = 0.001$ cos(20t) N and that the following parameter assignments apply:

$$m = 1.0\text{ oz};\quad k_{f1} = 200\text{ dyn} - \text{s/cm};$$
$$k_{e1} = 6000\text{ dyn}/s;\quad k_{e2} = 0.05\text{ lbs/in}.$$

Solution: We first need to convert F_S, m, and k_{e2} to cgs units. Converting F_S is relatively straightforward since it is already in MKS metric units. Use the conversion factors in Appendix D:

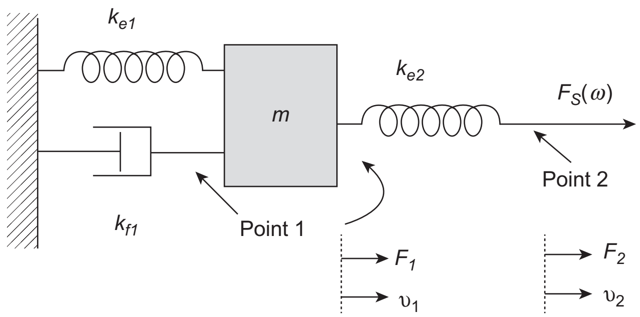


FIGURE 13.25 The mechanical system used in Example 13.12. This system has two independent velocities at points 1 and 2, so as with a two-mesh or two-node circuit, it requires two equations for analysis.

$1\ \text{N} = 10^5$ dyn. Hence $F_S = 100\ \sin\ (20t)$ dyn. To convert m from English units (oz) to cgs metric units, use the conversion factors in Appendix D in conjunction with dimensional analysis:

$$m = 1.0\ \text{oz}\frac{1\ \text{lb}}{16\ \text{oz}}\frac{1\ \text{kg}}{2.2046\ \text{lb}}\frac{1000\ \text{gm}}{1\ \text{kg}} = 28.4\ \text{gm}$$

In the English system the pound is actually a measure of mass but is often used as a measure of force as is the case here. The assumption is that a pound weight is equal to the force produced by a pound accelerated by gravity (i.e., $F = mg$). So it is not surprising that the cgs equivalent of 1 lb is 4103.109 gm[4], a measure of mass; however, the cgs equivalent of a 1-lb weight is 4.448 N, a measure of force. The cgs system is not without some ambiguity: grams are sometimes used to represent a weight as well as a mass. In cgs, the earth's gravitational constant, g, is equal to 980.6610 cm/s^2, so a gram weight is 980.6610 dyn. (Similarly a kilogram weight is 9.807 N). Applying the conversion factors from pounds weight to dynes and inches to centimeters:

$$k_{e2} = 0.05\frac{\text{lb}}{\text{in}}\frac{4.448\ \text{N}}{\text{lb. wt}}\frac{10^5\text{dyn}}{1\ \text{N}}\frac{1\ \text{in}}{2.54\ \text{cm}} = 8756\ \text{dyn/cm}$$

This is a lot of work to go through before we even get to Step 1, but real-world problems often involve messy situations like inconsistent units.

Step 1. Following the conversion to cgs units, the determination of the phasor impedances is straightforward. With $\omega = 20$ rad/s:

$$Z_{f1} = k_{f1} = 200\ \text{dyn/cm};\quad Z_m = j\omega m = j20(28.4) = 568\ \text{dyn/cm}$$
$$Z_{e1} = \frac{k_{e1}}{j\omega} = \frac{6000}{j20} = -j300\ \text{dyn/cm};\quad Z_{e2} = \frac{k_{e2}}{j\omega} = \frac{8756}{j20} = -j438\ \text{dyn/cm}$$

Step 2. The force and velocity directions have already been assigned with F_1 and F_2 as positive to the right. After the first two steps, the phasor representation of the system is as shown in Figure 13.26.

In writing the equations about the two points, we must take into account the fact that the spring and friction on the right side of the mass have nonzero velocities on both sides. Therefore, the net

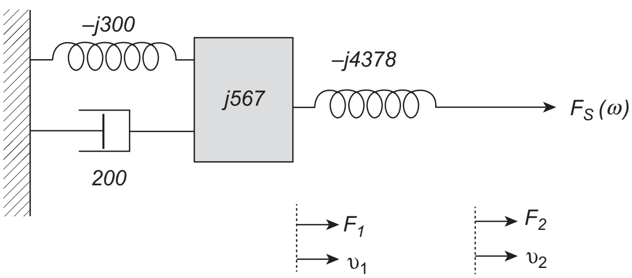


FIGURE 13.26 The mechanical system in Figure 13.25 after the component values have been converted to mechanical impedances. Before converting to impedances, it was first necessary to convert all the units to the cgs metric system.

velocity across these two elements is $v_2 - v_1$. Thus the force across the right-hand spring is $k_{e2}/j\omega$ $(v_2 - v_1)$. With this in mind, the equation around Point 2 becomes:

$$F_S - \frac{k_{e2}}{j\omega}(v_2 - v_1) = F_S - (-j438)(v_2 - v_1) = 0$$

For Point 2, the force exerted by this spring has the same magnitude, but is opposite in direction (i.e., positive with respect to F_1). Thus the equation for the force around Point 1 is:

$$\frac{k_{e2}}{j\omega}(v_2 - v_1) - \left(j\omega m - k_f - \frac{k_{e1}}{j\omega}\right)v_1 = 0$$
$$-j438(v_2 - v_1) - j568v_1 - 200v_1 - (-j300)v_1 = 0$$

Multiplying through by -1 and rearranging the two equations as coefficients of v_1 and v_2:

$$0 = (200 + j568 - j300 - j438)v_1(\omega) - (-j438)v_2(\omega)$$
$$100 = -(-j438)v_1(\omega) - j438v_2(\omega)$$

and in matrix notation:

$$\begin{vmatrix} 0 \\ 100 \end{vmatrix} = \begin{vmatrix} 200 - j170 & j438 \\ j438 & -j438 \end{vmatrix}\begin{vmatrix} v_1(\omega) \\ v_2(\omega) \end{vmatrix}$$

Solving for v_2, the velocity, this time manually:

$$v_2(\omega) = \frac{\begin{vmatrix} 200 - j170 & 0 \\ j438 & 100 \end{vmatrix}}{\begin{vmatrix} 200 - j170 & j438 \\ j438 & -j438 \end{vmatrix}} = \frac{20000 - j17000}{-j85600 - 74460 + 191844}$$

$$v_2(\omega) = \frac{20000 - j17000}{117384 - j87600} = \frac{26249\angle -40}{146470\angle -36.7} = 0.18\angle -3.3 \text{ cm/s}$$

Of course, this solution could have been more easily obtained using MATLAB.

[4]To make matters even more confusing there are two units of mass in the English system termed pounds and abbreviated lbs. The most commonly used pound is termed the commercial or "avoirdupois pound," whereas a less commonly used measure is the "troy" or "apothecary pound." To convert: 1 troy lb. = 0.822857 avoirdupois lb. In this text only avoirdupois pounds are used, but conversions for both can be found in Appendix D.

13.5 RESONANCE

Resonance is a frequency-dependent behavior characterized by a sharp increase (or decrease) in some system variable(s) over a limited range of frequencies. It can occur in almost any feedback system, including mechanical and electrical systems as well as chemical and molecular systems. Often it is beneficial and exploited as in proton resonance used in magnetic resonance imaging, optical resonance used in spectroscopy for identifying molecular systems, or electrical resonance used to isolate frequencies or generate sinusoidal signals. However, it can also be undesirable, particularly in mechanical systems. For example, shock

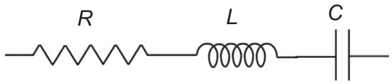


FIGURE 13.27 A series RLC circuit that can exhibit the properties of resonance at a select frequency.

absorbers are friction elements placed on cars to increase damping and reduce the resonant properties of automotive suspension systems. Resonance is discussed in terms of mechanical and electrical systems since these are the systems we have been studying, but the basic concepts are applicable to other systems.

13.5.1 Resonant Frequency

In electrical and mechanical systems resonance occurs when the impedance of an inertial-type element (mass or inductor) equals and cancels the impedance of a capacitive-type element. Consider the impedance of a series RLC circuit shown in Figure 13.27.

In the frequency (phasor) domain, the impedance of this series combination is:

$$Z(\omega) = R + j\omega L + \frac{1}{j\omega C} = R + j\left(\omega L - \frac{1}{\omega C}\right)\ \Omega \tag{13.19}$$

At some value of ω, the capacitor's impedance will be equal to the inductor's impedance and the two impedances will cancel. This will leave only the resistor to contribute to the total impedance. To determine the frequency at which this cancellation takes place, simply set the impedances equal and solve for frequency:

$$\begin{gathered}\omega_o L = \frac{1}{\omega_0 C}; \quad \omega_o L(\omega_o C) = 1; \quad \omega_o^2 = \frac{1}{LC} \\ \omega_o = \frac{1}{\sqrt{LC}}\ \text{rad/s}\end{gathered} \tag{13.20}$$

where ω_o is the "resonant frequency." Note that this is the same equation as for the undamped natural frequency, ω_n, in a second-order representation of the RLC circuit (see Equation 13.7). If we plot the magnitude of the impedance in Equation 13.19, we get a curve that reaches a minimum value of R at $\omega = \omega_o$ and increases on either side, Figure 13.28. The sharpness of the curve relates to the bandwidth of the resonant system as discussed in the next section.

13.5.2 Resonant Bandwidth, Q

When a system approaches the resonant frequency, the system variables (voltage—current or force—velocity) will increase (or decrease) to a maximum (or minimum). The sharpness of that curve depends on the energy dissipation element (resistance or friction). Figure 13.29 shows an RLC circuit configured as an input—output system. In the next example, we find and plot the Bode plot of the transfer function of this system for different values of R and show how the sharpness of the resonant peak depends on R: as R decreases the sharpness increases.

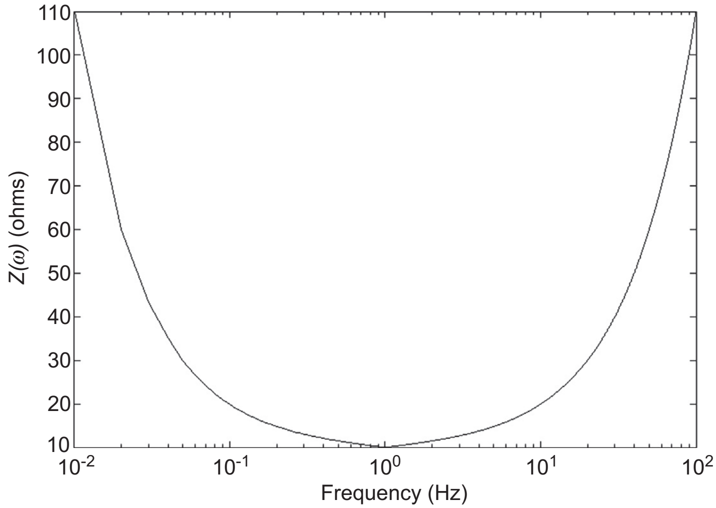


FIGURE 13.28 A plot of impedance versus frequency for a series RLC circuit as given in Equation 13.19. The impedance reaches a minimum at $\omega = \omega_0$.

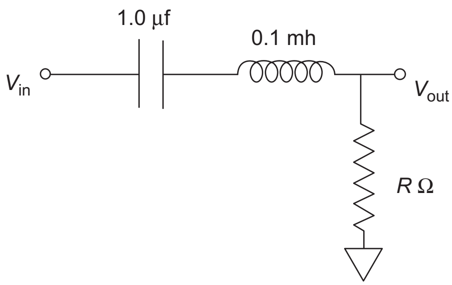


FIGURE 13.29 An RLC circuit configured as an input–output system. This Bode plot (i.e., system frequency spectrum) of this system is determined for different values of R in Example 13.13.

EXAMPLE 13.13

Plot the Bode plot of the transfer function of the network shown in Figure 13.28 for four values of resistance: 0.1, 1.0, 10, and 100 Ω. Plot the Bode plot in radians per seconds (for variety).

Combining **Step 1** (conversion), **Step 2** (current assignment), and **Step 3** (KVL), the equation for the circuit current becomes:

$$V_{in}(\omega) = I(\omega)\left(R + j\omega L + \frac{1}{j\omega C}\right)$$

Steps 4 and 5. Since we will be using MATLAB to plot the Bode plot, we can leave the L and C as variables. Solving for $I(\omega)$, then finding $V_{out}(\omega)$:

$$I(\omega) = \frac{V_{in}(\omega)}{R + j\omega L + 1/j\omega C}$$

$$V_{out}(\omega) = I(\omega)R = \frac{RV_{in}(\omega)}{R + j\omega L + 1/j\omega C}$$

The transfer function becomes:

$$TF(\omega) = \frac{V_{out}(\omega)}{V_{in}(\omega)} = \frac{R}{R + j\omega L + 1/j\omega C} \tag{13.21}$$

Rearranging into the standard Bode plot format with the lowest coefficient of ω equal to 1 (in this case the constant):

$$\begin{aligned} TF(\omega) &= \frac{V_{out}(\omega)}{V_{in}(\omega)} = \frac{j\omega RC}{j\omega RC + j^2\omega^2 LC + 1} = \frac{j\omega RC}{j\omega RC - \omega^2 LC + 1} \\ TF(\omega) &= \frac{V_{out}(\omega)}{V_{in}(\omega)} = \frac{j\omega RC}{1 - \omega^2 LC + j\omega RC} \end{aligned} \tag{13.22}$$

Note that since we will be solving the transfer function equation on the computer it would have been just as easy to use Equation 13.21 directly; the extra work of putting it into standard Bode plot form was just an exercise.

The resonant frequency of the transfer function is $\omega_o = 1/\sqrt{LC} = 1 \times 10^5$ rad/s and the Bode plot should include a couple of orders frequency above and below this frequency, say, from 10^3 to 10^7 rad/s. Solving Equation 13.22 over this frequency range leads to the following MATLAB program.

```
% Example 13.13 Bode plot of the TF of an RLC circuit with 4 different values of R.
%
R= [0.1 1 10 100];            % Resistance values
L = 10^-4;                    % Inductance value
C= 10^-6;                     % Capacitance value
w = (1000:1000:10000000);     % Frequency range
for k = 1:length(R)
  TF = (j*w*R(k)*C)./(1 - w.^2* L*C + j*w*R(k)*C);  % TF equation
  TF = 20*log10(abs(TF));     % Convert to dB
  semilogx(w,TF,'k');         % Plot as Bode plot
  ....... text labels......
end
  ....... labels and axis.......
```

Results: This program produces the graph shown in Figure 13.30.

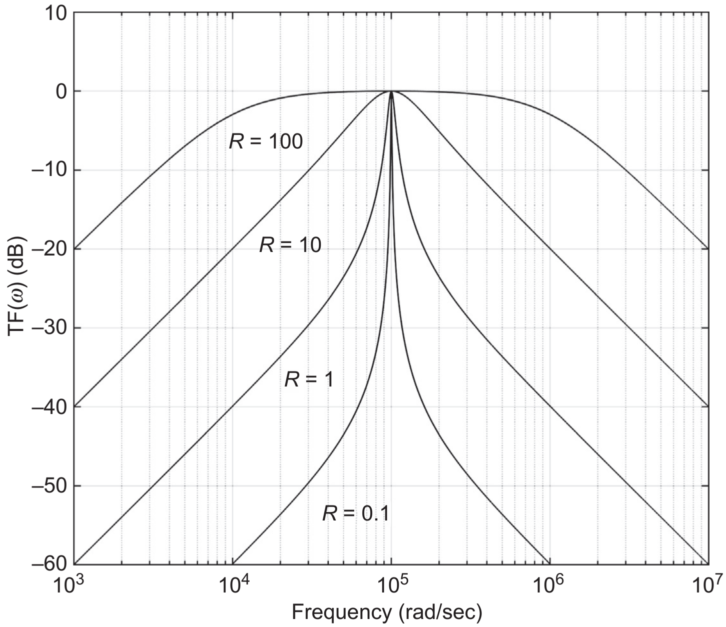


FIGURE 13.30 The Bode plot (system frequency spectrum) of the RLC system shown in Figure 13.29 for four different values of R. This input–output system shows a resonance at 10,000 rad/s for all values of R, but the resonance peak increases dramatically with decreasing values of R.

Figure 13.30 shows that the resonance peak of the transfer function occurs at the same frequency for all values of R. This is expected since the resonant frequency is a function of only L and C, Equation 13.18. Specifically, the peak occurs at:

$$\omega_o = 1/\sqrt{LC} = 1\Big/\sqrt{10^{-4}10^{-6}} = 100,000 \text{ rad/s}.$$

Peak sharpness increases as R decreases. In the configuration shown in Figure 13.29, the RLC network is a band-pass filter; however, if the resistor and capacitor are interchanged, the circuit becomes a low-pass filter as used in previous examples. Nonetheless, this arrangement of an RLC network still has a resonant peak at a frequency determined by L and C (Equation 13.18), and the sharpness of the peak varies with R in a similar manner. The behavior of a low-pass RLC network is examined in a problem.

The sharpness of the frequency curve around the resonant frequency is an important property of resonant systems. This characteristic is often described by a number known as "Q," which is defined as the resonant frequency divided by the bandwidth:

$$Q = \frac{\omega_o}{BW} \tag{13.23}$$

where ω_o is the resonant frequency and BW is the bandwidth using the standard definition: the difference between the high-frequency cutoff and the low-frequency cutoff (−3 dB points). Occasionally, Equation 13.23 is referred to as "selectivity." To confuse things further, Q also has an alternate, more fundamental definition.

Classically, Q is defined in terms of energy storage and energy loss to describe how close energy storage elements such as inductors and capacitors approach the ideal. Ideally such elements should only store energy, but real elements also dissipate energy owing to parasitic resistance. In this context, Q is defined as the energy stored over the energy lost in one cycle:

$$Q = 2\pi \frac{\text{Energy stored}}{\text{Energy dissipated}} \tag{13.24}$$

Using Equation 13.24 we can calculate Q for a single element such as an inductor. Assume an inductor has not only inductance L h, but also a parasitic series resistance of R Ω. Since inductors are constructed as wire coils, often with quite a bit of wire, inductors inevitably have some resistance. (Inductors are the least ideal of the passive electrical elements.) The energy lost in the wire's resistance over one sinusoidal cycle is equal to the power integrated over the cycle, and the power is just vi, or for a resistor, $R\, i^2$. Assuming a sinusoidal current through the resistor, $i_R(t) = I\sin(\omega t)$, the energy lost over one cycle becomes:

$$E_{lost} = \int_{Cyc} vidt = \int_0^{2\pi} R(I\sin(\omega t))^2\, dt = \frac{2\pi RI^2}{2\omega}\ \text{J}$$

The energy stored in an inductor is also the integral of vi over one cycle. Since the parasitic R is effectively in series with the inductor, the current through the inductor is the same as through the resistor $i_L(t) = I\sin(\omega t)$ and, from the definition of an inductor (Equation 12.15), the voltage is L times the derivative of the current:

$$v_L(t) = L\frac{di}{dt} = \frac{d(I\sin(\omega t))}{dt} = \omega I\cos(\omega t)$$

$$E_{Stored} = \int_{Cyc} vi\, dt = \int_0^{2\pi} \omega I^2 \cos(\omega t)\sin(\omega t)dt = \frac{\omega LI^2}{2\omega}$$

Plugging in the two energies into Equation 13.24, the Q of an inductor becomes:

$$Q_L = \frac{2\pi\left(\dfrac{\omega LI^2}{2\omega}\right)}{\left(\dfrac{2\pi RI^2}{2\omega}\right)} = \frac{\omega L}{R} \tag{13.25}$$

where L is the inductance, ω the frequency in rad/s, and R the parasitic resistance.

Similarly, it is possible to derive the Q of a capacitor having a capacitance value C and a parasitic resistance R as:

$$Q_C = \frac{1}{\omega RC} \tag{13.26}$$

However, in a circuit that includes both an inductor and a capacitor, most of the parasitic resistance is from the inductor. Any contribution from the capacitor is usually ignored and Equation 13.25 can be used to find the Q.

Based on these definitions, it is possible to derive Equation 13.23 from the definition of bandwidth. Returning to the RLC circuit in Figure 13.29, the transfer function of this circuit is given in Equation 13.21 is:

$$TF(\omega) = \frac{R}{R + j\omega L + 1/J\omega C} = \frac{R}{Z(\omega)} = \frac{1}{Z(\omega)/R}$$

where $Z(\omega)$ is the series R, L, C impedance.

$Z(\omega)/R$ can also be written as:

$$\frac{Z}{R} = \frac{R + j\omega L - \dfrac{j}{j\omega C}}{R} = 1 + j\left(\frac{\omega L}{R} - \frac{1}{\omega C}\right); \quad \text{multiplying both imaginary terms by } \frac{\omega_o}{\omega_o}$$

$$\frac{Z}{R} = 1 + j\left(\frac{\omega}{\omega_o}\left(\frac{\omega_o L}{R}\right) - \frac{\omega_o}{\omega}\left(\frac{1}{\omega_o C}\right)\right)$$

This allows us to substitute the definition of Q into the equation for Z/R:

$$\frac{Z}{R} = 1 + j\left(\frac{\omega Q}{\omega_o} - \frac{\omega_o Q}{\omega}\right) = 1 + jQ\left(\frac{\omega}{\omega_o} - \frac{\omega_o}{\omega}\right) \tag{13.27}$$

At the cutoff frequencies, $|TF(\omega_{H,L})| = 0.707 \ |TF(\omega = \omega_o)|$. However, at the resonant frequency, $Z(\omega = \omega_o) = R$ and $TF(\omega = \omega_o)$. Hence, at the cutoff frequencies $|TF(\omega_{H,L})| = 0.707$.

So to find the bandwidth (which is just the difference between the cutoff frequencies), set $\left|TF(\omega)\right| = 0.707\left|TF(\omega)\right|$ so $\left|\frac{Z}{R}\right| = \left|\frac{1}{TF(\omega)}\right| = \frac{1}{.707} = 1.414$ and solve for ω_H and ω_L. Setting the magnitude of Equation 13.27 to $1.414 = \sqrt{2}$, we get:

$$\left|1 + jQ\left(\frac{\omega}{\omega_o} - \frac{\omega_o}{\omega}\right)\right| = \sqrt{2}; \quad \left(\text{For}\,|1 + jB| = \sqrt{2}\,,\ B = \pm 1\right)$$

$$Q\left(\frac{\omega}{\omega_o} - \frac{\omega_o}{\omega}\right) = \pm 1$$

There are two solutions to the equation for $+1$ and -1:

$$\omega_L = \omega_0\left(1 - \frac{1}{2Q}\right); \quad \omega_H = \omega_0\left(1 + \frac{1}{2Q}\right)$$

$$BW = \omega_H - \omega_L = \frac{\omega_0}{Q}; \quad Q = \frac{\omega_0}{BW}$$

We can also relate Q to the standard coefficients of a second-order underdamped equation. Again referring to the RLC circuit in Figure 13.29, the transfer function can be easily determined in terms of the Laplace variable, s. To make the solution more general, we continue using variables L and C to represent the inductance and capacitance:

Steps 1, 2, and 3:

$$V_{in}(s) = I(s)(R + sL + 1/sC)$$

Steps 4 and 5:

$$I(s) = \frac{V_{in}(s)}{R + sL + 1/sC}; \quad V_{out}(s) = I(s)R = \frac{RV_{in}(s)}{R + sL + 1/sC}$$

Solving for the transfer function, $V_{out}(s)/V_{in}(s)$:

$$TF(s) = \frac{V_{out}(s)}{V_{in}(s)} = \frac{R}{R + sL + 1/sC}$$

Putting the transfer function equation into standard Laplace domain format where the highest power of s has a coefficient of 1:

$$TF(s) = \frac{V_{out}(s)}{V_{in}(s)} = \frac{sR}{sR + s^2L + 1/C} = \frac{sR/L}{sR/L + s^2 + 1/LC}$$

Rearranging and comparing with the standard form of a second-order equation analyzed in Chapter 7 (Equation 7.31):

$$TF(s) = \frac{R/L^s}{s^2 + R/Ls + 1/LC} = (RCs)\frac{1/LC}{s^2 + R/Ls + 1/LC} = (RCs)\frac{\omega_n^2}{s^2 + 2\delta\omega_n s + \omega_n^2}$$

where $\omega_n = \omega$. Equating coefficients:

$$\begin{aligned} &2\delta\omega = \frac{R}{L}; \quad \delta = \frac{R}{2\omega L}; \quad \text{and} \quad Q = \frac{\omega L}{R} \\ &\delta = \frac{1}{2Q}; \quad Q = \frac{1}{2\delta} \end{aligned} \tag{13.28}$$

The relationship between Q and δ is amazingly simple.

The characteristics of resonance and the various definitions and relationships described apply equally to mechanical systems. In fact, most mechanical systems exhibit some resonant behavior and often that behavior is detrimental to the system's performance (consider a car with bad shock absorbers). An investigation of the resonant properties of a mechanical system is given in the next example.

EXAMPLE 13.14

Find the Q of the mechanical system shown in Figure 13.31. The system coefficients are $k_f = 6$ dyn s/cm; $m = 8$ g; $k_e = 10$ dyn/cm.

Solution: Q can be determined directly from δ in transfer function using Equation 13.28. So we need to find the transfer function $v(s)/F_s(s)$. Applying the standard analysis based Equation 13.18:

$$F_s(s) - v(s)(k_f + ms + k_e/s) = 0; \quad v(s) = \frac{F_s(s)}{k_f + ms + k_e/s}$$

$$\frac{v(s)}{F_s(s)} = \frac{s/m}{s^2 + \frac{k_f}{m}s + \frac{k_e}{m}} = \frac{s/m}{s^2 + 2\delta\omega_n s + \omega_n^2}$$

Equating coefficients:

$$\omega_n = \sqrt{\frac{k_e}{m}} = \sqrt{\frac{10}{8}} = 1.1 \text{ rad/sec}; \quad 2\delta\omega_n = \frac{k_f}{m}; \quad \delta = \frac{k_f}{2m\omega_n} = \frac{k_f}{2\sqrt{k_e m}} = \frac{6}{2\sqrt{10(8)}} = 0.335$$

$$Q = \frac{1}{2\delta} = 1.49$$

The next example uses MATLAB to explore the ways in which the Bode plot and impulse response of a generic second-order system vary for different values of Q.

EXAMPLE 13.15

Plot the frequency characteristics and impulse response of a second-order system in which $Q = 1$, 10, and 100. Use MATLAB and assume a resonant frequency, ω_n^2 of 1000 rad/s. Note that it could be either a mechanical or electrical system; the equations would be the same.

Solution: Begin with the standard second-order Laplace transfer function equation, but substitute Q for δ. Convert this equation to the time domain by taking the inverse Laplace transform to get the impulse response. (Recall, the inverse Laplace transform of the transfer function is the impulse response.) Then convert the transfer function equation to the phasor domain to get the Bode plot equation. Use MATLAB to plot both the Bode plot and the time response.

The standard Laplace second-order equation is:

$$TF(s) = \frac{\omega_n^2}{s^2 + 2\delta\omega_n s + \omega_n^2};$$

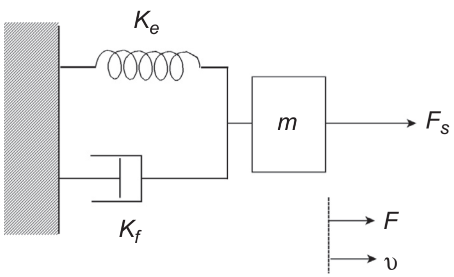


FIGURE 13.31 A mechanical system consisting of a mass, friction, and elasticity used in Example 13.14.

Substituting in $\delta = \frac{1}{2Q}$ and $\omega_n = 1000$:

$$TF(s) = \frac{\omega_n^2}{s^2 + \dfrac{\omega_n}{Q}s + \omega_n^2} = \frac{10^6}{s^2 + \dfrac{1000}{Q}s + 10^6}$$

To find the impulse response, take the inverse Laplace transform of the transfer function. Since $Q \geq 1$, δ will be ≤ 0.5, so the system is underdamped for all values of Q and entry 15 in the Laplace transform table can be used:

$$x(t) = \frac{\omega_n}{\sqrt{1-\delta^2}}\left[e^{-\delta\omega_n t}\sin\left(\omega_n\sqrt{1-\delta^2}t\right)\right] = \frac{10^3}{\sqrt{1-\dfrac{1}{4Q^2}}}\left[e^{-1000/2Q}\sin(1000\sqrt{1-\dfrac{1}{4Q^2}}t\right]$$

To find the frequency response, convert to phasor and plot for the requested values of Q:

$$TF(\omega) = \frac{10^6}{(j\omega)^2 + j\omega\dfrac{1000}{Q} + 10^6} = \frac{1}{1-\left(\dfrac{\omega}{10^3}\right)^2 + \dfrac{j\omega}{Q10^3}}$$

Plotting of the frequency and time domain equations is done in the following program.

```
% Example 13.15 Frequency response and impulse response of a
%  second-order system.
%
wn = 1000;                          % Define resonant frequency
w = (100:10:10000);                 % Define a frequency vector for Bode plot
t = (10ˆ-5:10ˆ-5:.2);               % Define a time vector
Q = [1 10 100];                     % Define Q's
for k = 1:length(Q)                 % Calculate and plot the frequency plots
  TF = 1./(1-(w/1000).ˆ2 + j*w/(Q(k)*1000));  % Frequency equation
  TF = 20*log10(abs(TF));           % Convert to dB
  semilogx(w,TF,'k');               % Bode plot
end

  .......labels and axis.......

%
% Now construct the impulse responses
figure;
for k = 1:3                         % Cal. and plot the time response
  d =sqrt(1-1/(4*Q(k)ˆ2));          % Define square root of 1-δ2
  x = (wn/d)*(exp(-wn*t/(2*Q(k)))).*sin(wn*d*t);
  subplot(3,1,k);                   % Plot separately for clarity
    plot(t,x,'k');
    ylabel('x(t)');
end
  .......labels.......
```

This program generates the following plots. Figure 13.32 reiterates the message in Figure 13.30, that high-Q systems can have very sharp resonance peaks.

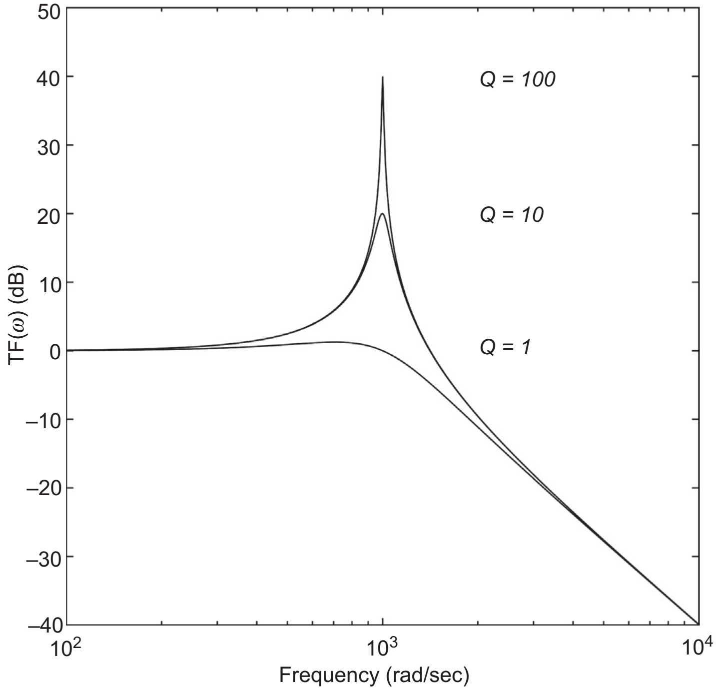


FIGURE 13.32 The frequency response of the transfer function of a second-order system to three different values of Q. The impulse response from this system is shown in Figure 13.33 for the same values of Q.

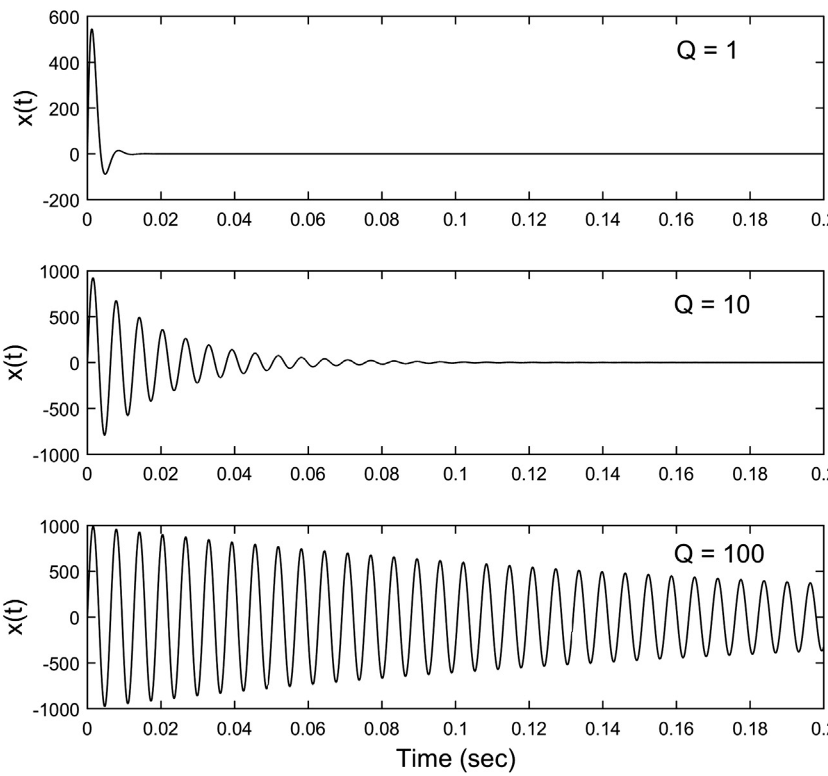


FIGURE 13.33 The impulse response of a second-order system with three values of Q. The frequency response of the transfer function of this system with the three values of Q is shown in Figure 13.32.

In the time domain, sharp resonance peaks correspond to sustained oscillations at the resonance frequency that diminishe slowly, Figure 13.33.

A common example of a high-Q mechanical system is a large bell. Striking a bell is like applying a mechanical impulse and the tone continues to sound long after it is struck. In fact, sustained oscillation is a characteristic of high-Q systems and is sometimes referred to as "ringing" even in electrical systems.

If a high-Q circuit is used as a filter, it can be quite selective with regard to the frequencies it allows to pass through. This is demonstrated in the final example.

EXAMPLE 13.16

Demonstrate the filtering characteristic of a high-Q system. Generate a 1.0-s chirp waveform that continually increases in frequency from 1 Hz to 1 kHz. (Recall, a chirp waveform increases in frequency with time usually linearly.) Assume a sampling frequency of 5 kHz.

Modify the RLC circuit of Figure 13.29 so that the resonant frequency, f_o, is 100 Hz and the circuit has a Q of 10. Assume the capacitor value is the same as that shown in Figure 13.29 (i.e., 1.0 μf), and adjust L to get the desired resonant frequency. Simulate the operation of passing the chirp waveform through the filter and plot the output. Repeat for two other values of resonant frequency: $f_o = 250$ and 500 Hz.

To simulate the operation the circuit performs on the signal, find the circuit's impulse response and convolve that response with the input to get the simulated output.

Solution: First we are requested to design the network to have the desired resonant frequency and Q. The resonant frequency is entirely determined by L and C. Since we already have a value for C, we can determine the value of L from Equation 13.21.

$$\omega_0 = 2\pi f_0 = \frac{1}{\sqrt{LC}}; \quad f = \frac{1}{2\pi\sqrt{LC}}$$

$$\text{Squaring both sides}: f_0^2 = \frac{1}{4\pi^2 LC}$$

$$\text{Solving for } L: L = \frac{1}{4\pi^2 f_0^2 C} = \frac{1}{4\pi^2 (10^4)(10^{-6})} = 2.53 \text{ h} \quad \text{when } f_0 = 100 \text{ Hz}$$

For the other two resonant frequencies:

$$L = \frac{1}{4\pi^2 f_0^2 C} = \frac{1}{4\pi^2 (250^2)(10^{-6})} = 0.41 \text{ h} \quad f_0 = 250 \text{ Hz}$$

$$L = \frac{1}{4\pi^2 (500^2)(10^{-6})} = 0.1 \text{ h} \quad f_0 = 500 \text{ Hz}$$

To find the resistor values required to give the desired Q, we can use Equation 13.28:

$$Q = \frac{\omega_0 L}{R} = \frac{2\pi f_0 L}{R}; \quad R = \frac{2\pi f_0 L}{Q}$$

$$R = \frac{2\pi(100)(2.53)}{10} = 158.9\ \Omega \quad f_0 = 100 \text{ Hz}$$

TABLE 13.1 Resonant Frequencies and Respective Component Values of the Circuit Given in Figure 13.28

Resonant Frequency (Hz)	R (Ω)	L (h)	C (μf)
100	158.9	2.53	1.0
250	64.4	0.41	1.0
500	31.4	0.10	1.0

Solving for the other two values of L and R, the component values for the three resonant frequencies are summarized in Table 13.1.

Now that we have the circuit designed we need to find the impulse response. We can get this from the inverse of the Laplace transfer function. In Example 13.11 we find the transfer function of the network in Figure 13.29. Substituting in s for $j\omega$ in Equation 13.21 we get:

$$TF(s) = \frac{R}{R + Ls + 1/CS} = \frac{RCs}{RCs + CLs^2 + 1} = \frac{R/Ls}{s2 + R/Ls + 1/CL} \tag{13.29}$$

To find the inverse Laplace transform, we note that $Q = 10$ in all cases, thus $\delta = 1/2\,Q = 0.05$, so the system is underdamped. We can use entry 13 of the Laplace transform table (Appendix B) where:

$$b = R/L; \quad \alpha = R/2L; \quad c = 0$$
$$\alpha^2 + \beta^2 = 1/LC; \quad \beta^2 = 1/LC - \alpha^2 = 1/LC - \alpha^2; \quad \beta = \sqrt{1/LC} - \alpha$$

Taking the inverse Laplace transform, the impulse response becomes:

$$\delta(t) = e^{-\alpha t}\left[\left(\frac{-(R/L)\alpha}{\beta}\right)\sin \beta t + R/L \cos \beta t\right] \tag{13.30}$$

where α and β were defined earlier (it is easier to program if we leave these as variables and substitute in their specific values in the MATLAB routine). We could combine this equation into a single sinusoid with phase, but since we are going to program it in MATLAB further reduction is not necessary.

To construct the chirp signal, we define a time vector, t, that goes from 0 to 1 in steps of T_s where $T_s = 1/f_s = 1/5000$. This time vector is then used to construct a frequency vector of the same length that ranges from 1 to 1000. The chirp signal is then constructed as the sine of the product of f and t (i.e., `sin(pi*f.*t)`) so that the frequency increases linearly as t increases. (Note: use π, not 2π, to make the final frequency 1000 Hz).

We then program the impulse response using the above-mentioned equation and the various values of R, L, and C. Convolving the input signal with the impulse response produces the output signal. The solution of Equation 13.30 is performed three times in a loop for the various values of R, L, and C and plotted.

```
% Example 13.16 Demonstrate the filter characteristics of a high-Q system.
%
fs = 5000;                          % Sample frequency
Ts = 1/fs;                          % Sample interval
C = 0.000001;                       % Capacitor value (fixed)
R = [158.9, 64.4, 31.4];            % Resistor values
L = [2.53, 0.41, 0.1];              % Inductor values
f_0 = [100, 250, 500];              % Plot labels
t = (0:Ts:1);                       % Define time vector (0 - 5 sec; Ts = .0002)
f = t*1000;                         % Frequency goes from 1 to 1000 Hz
xin = sin(pi*f.*t);                 % Construct "chirp"
%
for k = 1:3                         % Calculate impulse response
  alpha = R(k)/(2*L(k));            % Define alpha and beta, then the
                                      impulse response
  beta = sqrt(1/(L(k)*C)) - alpha;
  h  =  exp(-alpha*t).*((-R(k)*alpha)/(L(k)*beta)*sin(beta*t)  +  (R(k)/L(k))
*cos(beta*t));
  xout = conv(h,xin);               % Filter vin
  xout = xout(1:length(t));         % Remove extra points
  subplot(3,1,k);                   % Plot separately
    plot(t,xout,'k');
  ....... Labels and title.......
end
%
figure;                         % Plot spectrum of chirp
XIN= abs(fft(xin));                 % FFT
f1 = (1:length(t))*fs/length(t);    % Construct frequency vector for
                                      plotting
plot(f1(1:2500),XIN(1:2500),'k');   % Plot only valid points
   .......labels......
```

Analysis: The initial portion of the program defines the sampling characteristics, the component values, and the chirp. (If you have a sound system on your computer, try typing `sound(xin)` in MATLAB after executing this code. The resulting chirp sound is both startling and amusing.)

A loop is used to evaluate the impulse response for the three component configurations and find the circuit's output to the chirp input using convolution. This output is then plotted and is shown in Figure 13.34. The RLC circuit functions as a band-pass filter with a center frequency that is equal to the resonant frequency. The chirp signal's frequency depends on time and is approximately equal to the time in milliseconds. The signal appears at the output only when the chirp frequency corresponds to the circuit's resonant frequency. So when the resonant frequency is 100 Hz, the chirp signal passes through the filter at approximately 100 ms, Figure 13.34 (upper curve), and when it is

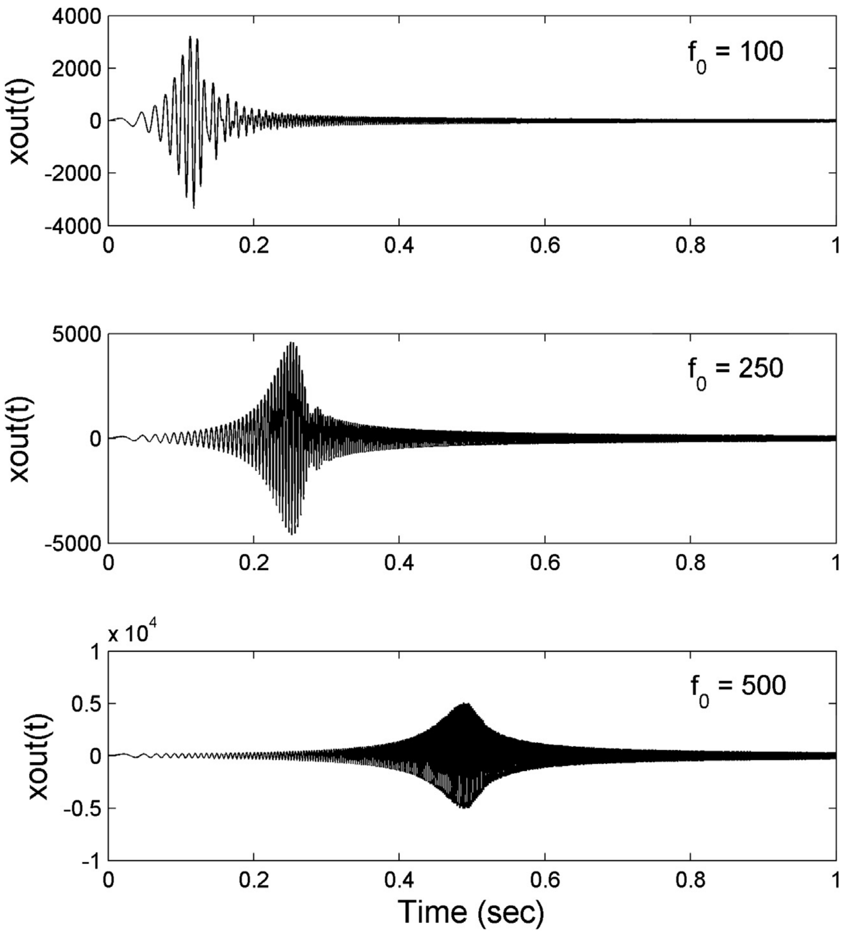


FIGURE 13.34 The output of a chirp signal after passing through a high-Q second-order system. The chirp ranges in frequency between 10 and 1000 Hz over a 1.0-s time period, so the instantaneous frequency is approximately equal to the time in milliseconds. The chirp signal passes through the system at a time when its frequency corresponds with the resonant frequency of the system.

500 Hz, it passes through the filter at approximately 0.5 s, Figure 13.34 (lower curve). The selective ability of a high-Q filter is well demonstrated in this example. If Q increases, the selectivity also increases as is demonstrated in one of the problems.

The last section of the program plots the frequency spectrum of the chirp signal using the standard `fft` command. The spectrum is seen in Figure 13.35 to be relatively flat up to 1000 Hz as expected.

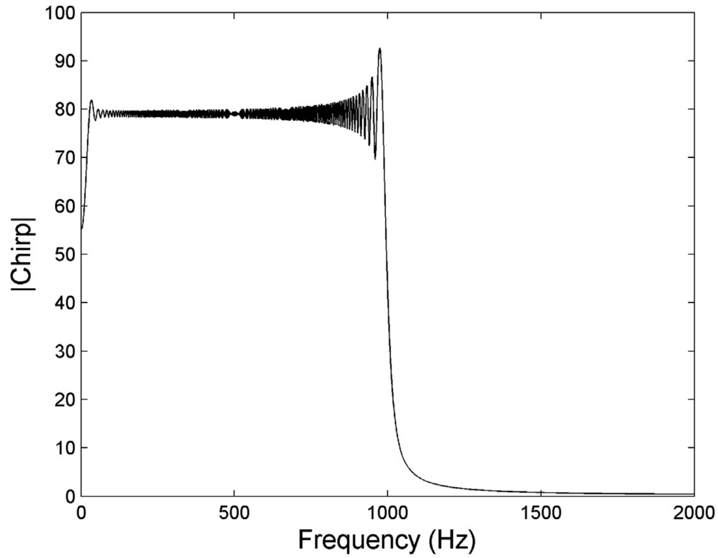


FIGURE 13.35 The spectrum of the chirp signal used in Example 13.16.

13.6 SUMMARY

Conservation laws are invoked to generate an orderly set of descriptive equations for any collection of mechanical or electrical elements. In electric circuits, the law of conservation of energy leads directly to KVL, which states that the voltages around the loop must sum to 0.0. Combining this rule with the phasor representation of network elements leads to an analysis technique known as mesh analysis. In mesh analysis, matrix equations are constructed that have the form of Ohm's law: $\boldsymbol{v} = \boldsymbol{Z}\boldsymbol{i}$, where $\boldsymbol{v}$ is a voltage vector, $\boldsymbol{i}$ is a current vector, and $\boldsymbol{Z}$ an impedance matrix. These equations are solved for the mesh currents, $\boldsymbol{i}$, and can be used to determine any voltage in the system.

The law of conservation of charge leads to KCL, which states that the sum of voltages into a node must be 0.0. As with KVL, it can be used to find the voltages and currents in any network. The application of KCL leads to a matrix equation of the form $\boldsymbol{i} = \boldsymbol{v}/\boldsymbol{Z}$, or $\boldsymbol{i} = \boldsymbol{v}\boldsymbol{Y}$, where $\boldsymbol{Y}$ is the admittance matrix consisting of inverse impedances. This equation can be solved for the node voltages, $\boldsymbol{v}$. From the node voltages any current in the circuit can be found. This analysis, termed nodal analysis, leads to fewer equations in networks that contain many loops but only a few nodes.

The conservation law that applies to mechanical systems is Newton's law, which, keeping with the sum to 0.0 idea, states that the forces on any element must sum to 0.0. Again using the phasor representation of mechanical elements, this law can be applied to generate equations of the form $\boldsymbol{F} = \boldsymbol{Z}\boldsymbol{v}$, where $\boldsymbol{F}$ is a force vector, $\boldsymbol{v}$ is a velocity vector, and $\boldsymbol{Z}$ is a matrix of mechanical impedances. These equations are solved for velocities and these velocities can be used to determine all of the forces in the system.

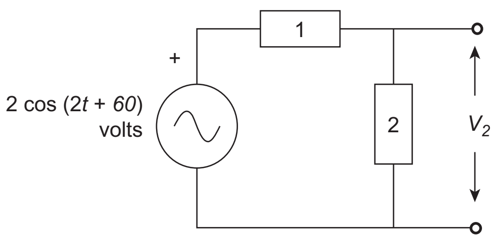
These conservation laws, and the analysis procedures they lead to, allow us to develop equations for even very complex electrical or mechanical systems. Large systems may generate equations that are difficult to solve analytically, but MATLAB makes short work out of even the most challenging equations.

Resonance is a phenomenon commonly found in nature. In electrical and mechanical systems, it occurs when two different energy storage devices have equal, but oppositely signed, impedances. During resonance, energy is passed back and forth between the two energy storage devices. For example, in an oscillating mechanical system, the moving mass stretches the spring transferring the kinetic energy of the mass to potential energy in the spring. Once the spring is fully compressed (or extended), the energy is passed back to the mass in terms of momentum as the spring recoils. Without friction this process would continue forever, but friction removes energy from the system, so the oscillation gradually decays. Electrical RLC circuits behave in exactly the same way, passing energy between the inductor and capacitor while the resistor removes energy from the system. (Recall, both friction and resistance remove energy in the form of heat.)

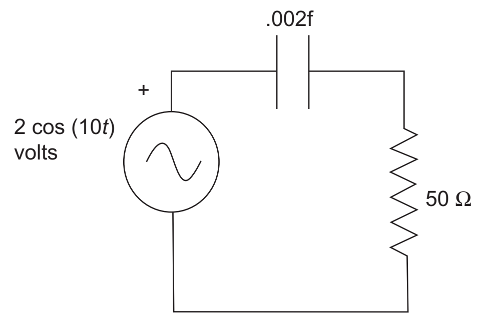
The quantity Q is a measure of the ratio of energy storage to energy dissipation of a system. The Q of a system is inversely proportional to the damping factor, δ. In the frequency domain, a higher Q corresponds to a sharper resonance peak. In the time domain, higher Q systems have longer impulse responses: they continue "ringing" long after the impulse because energy is only slowly removed from the system. Such systems are useful in frequency selection: with their sharp resonant peaks they are able to select out a narrow range of frequencies from a broadband signal. Standard systems analysis techniques such as Bode plots and the transfer function can be used to describe the behaviors of these systems.

PROBLEMS

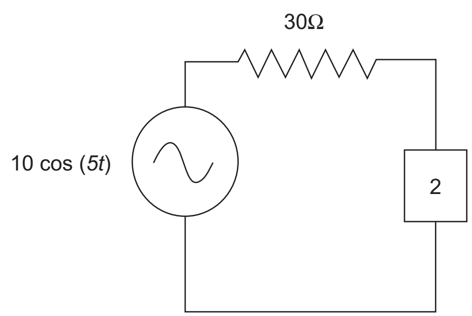
1. In the following circuit, the voltage across element 1 is $2\cos(2t + 60)$. What is the voltage, V_2, across element 2? Assume the plus side of element 1 is to the right and define positive V_2 as upward.



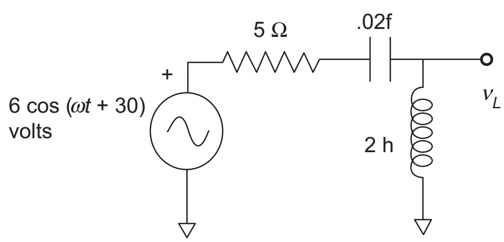
2. Find the voltage across the 50-Ω resistor.



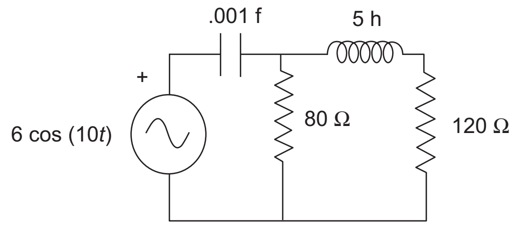
3. The loop current (clockwise) in the following circuit is 0.2 cos (10t – 103). What is element 2 (i.e., is it an R, L, or C) and what is its value?



4. Find the voltage across the inductor, v_L, for $\omega = 10$ and 20 rad/s.

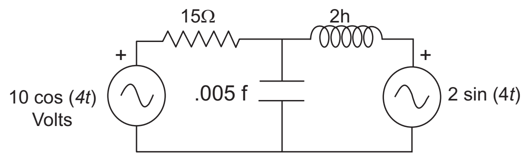


5. Find the voltage across the 5-h inductor in the following network.

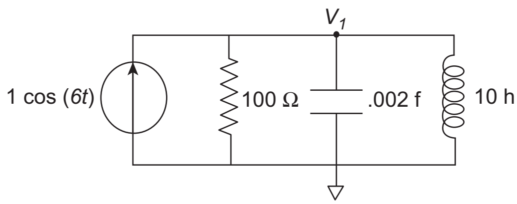


6. What is the voltage in the center 80-Ω resistor in the network given in Problem 5? (Note the total current through the resistor is $i_R = i_1 - i_2$.)

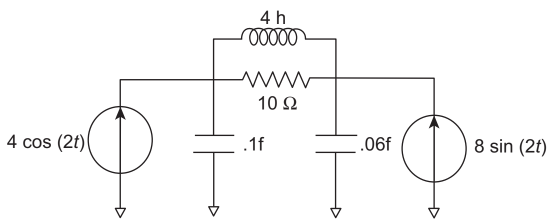
7. Find the current through the 2-h inductor in the following circuit. Is the voltage source, V_2, supplying energy or storing energy?



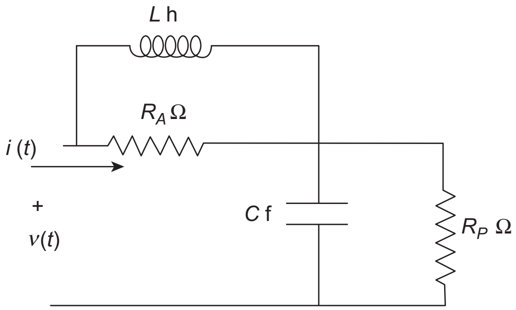
8. Find V_1.



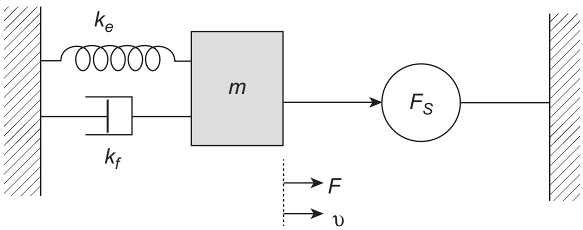
9. Find the voltage across the 10-Ω resistor.



10. The following circuit is a four-element Windkessel model. The element values in this model of the cardiovascular system are $R_A = 0.76$ mmHg/mL/s, $C = 1.75$ mL/mmHg, $R_p = 0.33$ mmHg/mL/s, and $L = 0.005$ mmHg/mL. Write the three-mesh equation and use the frequency domain approach of Example 13.5 to find the pressure in the aorta (i.e., $v(t)$) assuming the blood flow as given in Equation 13.11. As in Example 13.5, this problem requires both manual calculations and MATLAB.



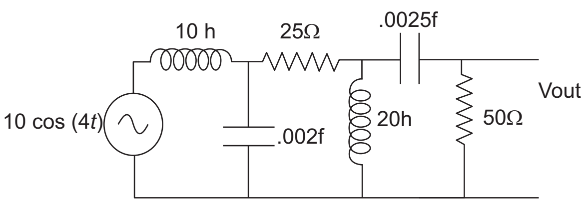
11. The following mechanical system has $k_f = 8$ dyn s/cm; $k_e = 12$ dyn/cm; $m = 2$ g; $F_s(t) = 10\cos(2t)$ dyn.



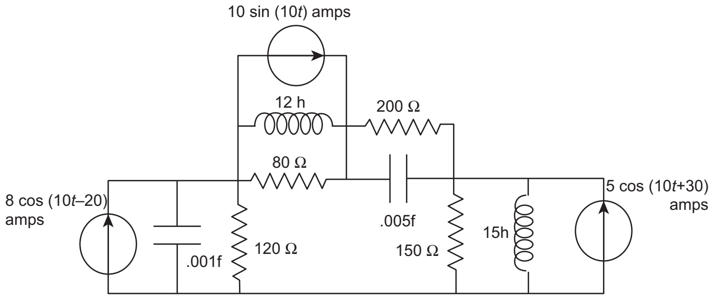
Find the length of the spring when $t = 0.10$ s. (Hint: solve for $x(\omega)$ where $x(\omega) = v(\omega/j\omega)$. Then convert to time domain $x(t)$ and solve for $t = .10$ s.)

12. In the mechanical system of Problem 11, what must be the value of k_f to limit the maximum velocity to ± 1 cm/s?

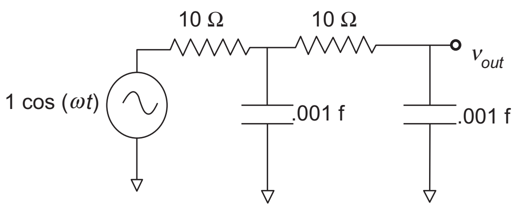
13. Find V_{out} using MATLAB. (Hint: Example 13.7 provides a framework for the code required to solve this three-mesh circuit, but you should let MATLAB do most of the work.)



14. Find the voltage across the right-hand current generator, using MATLAB and nodal analysis. Remember, 10 sin(10t) = 10 cos(10t − 90). This is a three-node circuit, so the admittance matrix has the same format as the impedance matrix in Problem 13. (Hint: You can solve this entirely in MATLAB. First define the current sources directly in MATLAB, then directly define the admittance matrix. Solve the resultant equation for $\boldsymbol{V}$. The voltage across the right-hand current generator should be `V(3)` in the voltage vector.)

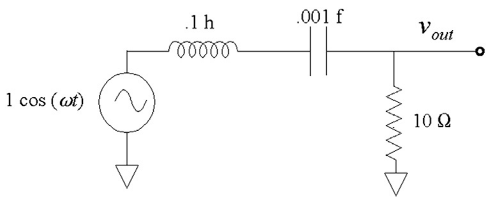


15. Find v_{out} for the following circuit for frequencies ranging between 1 and 1000 rad/s. Plot the magnitude of v_{out} as a function of frequency. (Hint: Set up ω as a vector and then directly write the impedance matrix in MATLAB keeping frequency as a variable. Solve for v_{out}. The vector ω should be a vector range between 1 and 1000 rad/s in increments of 1 rad/s. Then plot `abs`($v_{out}(\omega)$).)

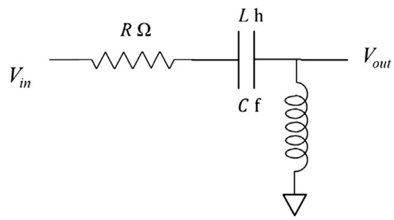


16. Repeat problem 15 for the mechanical system shown in Problem 11. That is, find v_{out} (which is now a velocity) for that mechanical system for frequencies ranging between 1 and 1000 rad/s. Plot the magnitude of v_{out} as a function of frequency. Use the parameter values: $k_f = 10$ dyn s/cm; $k_e = 1100$ dyn/cm; $m = 1.10$ g; $F_S(t) = \cos(\omega t)$.

17. Find and plot $V_{out}(\omega)$ over a range of frequencies from 11 to 10,000 rad/s. Plot V_{out} in dB as a function of log ω (i.e., on a semi-log axis).



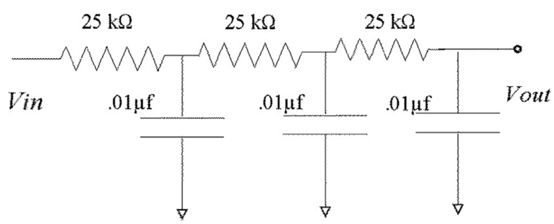
18. Use the graphical technique (not MATLAB) developed in Chapter 6 to plot the magnitude Bode Plot for the following RLC circuit. Set the value of the components: $L = 1$ h, $C = 0.0001$ f, and $R = 10\ \Omega$.



19. Use MATLAB to plot the magnitude transfer function of the circuit in Problem 18 for values of $R = 1$, 10, 70, and 500 Ω. Assume that $L = 1$ h and $C = .0001$ f. Frequency, ω, should range between 10 and 1000 rad/s. Use a small enough frequency increment to show the details of the frequency plot. If you are a reasonably skilled MATLAB programmer, you can use different colors for the four plots. How does the resistance affect the shape of the frequency plot?

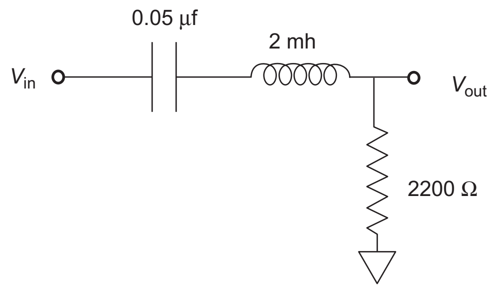
20. Use MATLAB to plot the Bode Plot (magnitude and phase) for the transfer function for the mechanical system in Problem 10 for values of $k_e = 2$ dyn/cm, m = 3 g, and $k_f = 1$, 10, 70, and 500 dyn s/cm. How does the friction element affect the shape of the frequency plot?

21. Use MATLAB to plot the magnitude Bode plot of the transfer function of the following circuit . Note the realistic units used for the elements. What type of filter is it? Plot between 100 and 10^6 rad/s, but plot in hertz.



22. Reverse the resistors and capacitors for the network of Problem 21 and plot the magnitude Bode plot. How does it compare with that of the original network?

23. Plot the Bode Plot (magnitude and phase) for the transfer function of the following RLC circuit. Reverse the position of the resistor and capacitor and replot. That is, put the resistors across the top and connect the capacitor to ground. How does this change the frequency characteristics of the filter?



24. Plot the Bode plot of the network used in Example 13.16 from the transfer function. Note that the transfer function equation was derived in Example 13.13 and is given by Equation 13.21. Plot the Bode plot for the three configurations of component values given in Table 13.1 over a frequency range of 1 to 10,000 Hz.

25. Plot the Bode plot of the network used in Example 13.16 as in Problem 24, but use the time domain approach and the impulse response. Plot the Bode plot for the three configurations of component values over a frequency range of 1 to 1000 Hz. Plot in decibel versus log frequency and take care to construct the correct frequency vector. (Hint: The code in Example 13.15 already determines the impulse response; you just need to add the code that determines the Bode plot and plots frequency response. Recall, the spectrum is the Fourier transform of the impulse response.)

26. Modify the code of Example 13.16 so that the simulated circuit has a resonant frequency of 400 Hz and plot the circuit's response to the chirp for three different values of Q: 2, 10, and 20. Note the effect of increasing and decreasing the filter's selectivity.