NI myDAQ AND MULTISIM PROBLEMS FOR CIRCUITS

Fawwaz T. Ulaby and Michel M. Maharbiz

By Ed Doering

Contents

Chapter 1

Introduction

This supplement to *Circuits* by Ulaby and Maharbiz contains 40 additional end-of-chapter problems designed for three-way solution: analytical, simulation, and measurement. After solving the problem analytically the student continues by solving the same problem with NI Multisim and then once again with NI myDAQ computer-based instrumentation and circuit components. By iterating on each dimension of the problem until all three agree students "triangulate on the truth" and develop confidence in their analytical and laboratory skills.

Each problem requests at least one common numerical value for comparison among the three methods. The percent difference between simulated and analytical results as well as measured-to-analytical results indicates the degree to which the student has achieved a correct solution. Normally simulation and analytical results agree to within a percentage point, and measurements often agree with analytical results to within five percent.

The problems are organized as four per chapter for Chapters 2 through 11 of *Circuits*. The table of contents indicates the associated section number of the textbook in parentheses. Each problem contains the problem statement and sufficient detail to guide the student through the simulation and physical measurement steps. Short video tutorials are linked to each problem to provide detailed guidance on Multisim techniques and ELVISmx computer-based instruments for the myDAQ.

This document is fully hyperlinked for section and figure references, and all video links are live hyperlinks. Opening the PDF version of this document is the most efficient way to access all links, and clicking a video hyperlink automatically launches the video in a browser. Within the PDF, use `ALT+leftarrow` to navigate back to a starting point.

1.1 Resources

- Appendix A details the parts list required to implement all of the circuits and includes links to component distributors.
- Appendix B describes how to implement a variable voltage source and two styles of current sources with the LM317 adjustable voltage regulator. Many of the circuits require a DC voltage other than the standard ±15V and 5V power supplies offered by the NI myDAQ. The adjustable voltage source pictured in Figure B.3 on page 165 should be constructed at the beginning of the term and left in place for subsequent circuits.
- Appendix C describes the Texas Instruments TL072 dual operational amplifier used in many of the circuits. The op amp is frequently used as a voltage follower to strengthen the 2 mA current drive of the myDAQ analog outputs. Appendix D describes the Intersil DG413 quad analog switch used in many of the transient response problems.
- Appendix E details a laboratory technique to measure time constants while Appendix F explains how to measure amplitude and phase shift for sinusoidal signals.
- Appendix G lists all of the available video links.

1.2 Goals for Student Deliverables

Students should document their work in sufficient detail so that it could be replicated by others. Present your work on the "Analysis" section as you would on a standard problem set. Be sure to include a "Given" section with your own drawing of the circuit diagram, a "Find" section that lists the requested results for the problem, a detailed solution process, and a clearly-identified end result. Do all of this work on engineering green paper or in a lab book or as otherwise required by your instructor.

The "Simulation" section presents your work to set up the circuit simulation in NI Multisim and the simulation results you used to obtain meaningful information. Create a word processing document that contains an organized set of screenshots with highlights and annotations as well as text to lead the reader through the screenshots. Include the circuit schematic and dialog box setup parameters for information not already visible on the schematic – circle parameters that you entered or changed away from default values. Also include simulation results, again circling control settings that you changed and highlighting regions where you obtained information. Figure 1.1 illustrates a screenshot from NI Multisim properly highlighted to indicate control settings that were adjusted away from default values as well as regions on the screen where measurements were obtained. Interpret the simulation results by writing them in standard form including units, and write any additional calculations that were necessary to reach an end result for simulation.

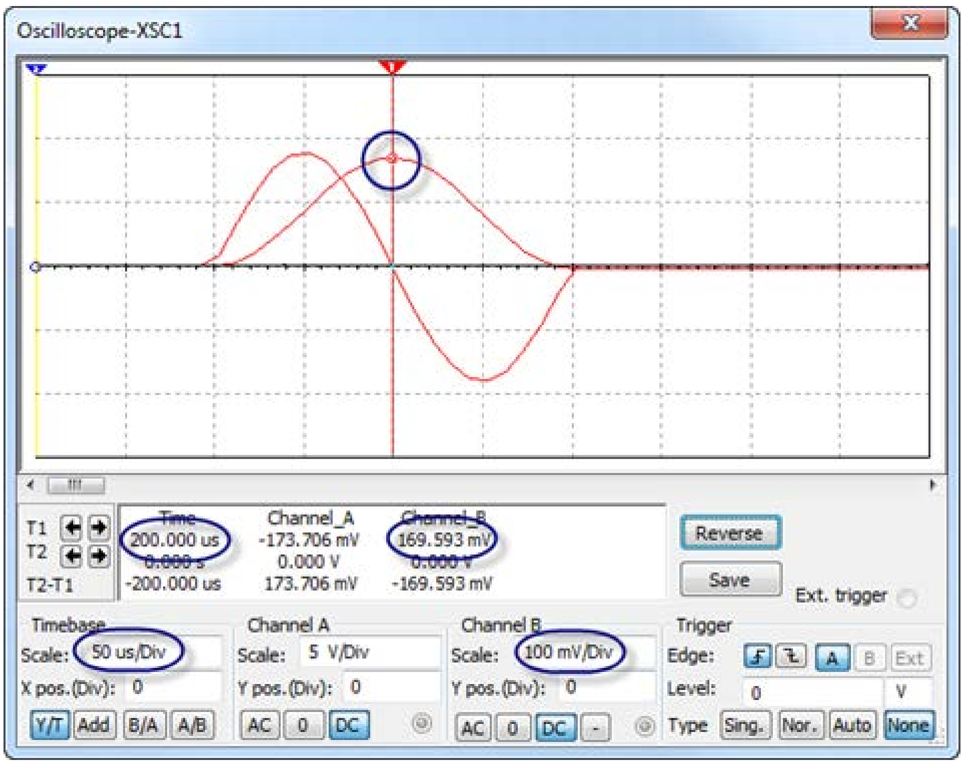


Figure 1.1: NI Multisim screenshot showing proper markings to indicate control settings adjusted away from default values as well as regions where measurement was obtained.

NOTE: Screen shots in Microsoft Word 2010 can be easily captured and highlighted as follows:

1. Select “Insert” tab and then “Screenshot,”
2. Choose the desired window or select “Screen Clipping” to define an arbitrary region,
3. Select “Shapes,” and
4. Place circles or boxes to highlight important values.

The “Measurement” section presents your work to set up the physical circuit and NI ELVISmx signal generators and measurement instruments. This section also includes your measurement results. Follow the general guidelines for the “Simulation” section. Your instructor may require a photo of your breadboard circuit and myDAQ connections along with your student ID when you work on the problem outside of scheduled class time. Also include a schematic diagram showing all myDAQ connections.

Finally, the “Summary” section compares the requested numerical results from each of the three methods. Tabulate three results for each requested numerical quantity (analytical, simulation, and measurement) and tabulate two percentage differences for each requested numerical quantity:

- Simulation-to-Analytical: $[(X_S - X_A)/X_A] \times 100\%$
- Measurement-to-Analytical: $[(X_M - X_A)/X_A] \times 100\%$

1.3 Student Deliverables Checklist

1. Engineering paper or lab book – submit directly to instructor:
   (a) Analysis
      i. "Given / Find" section including original circuit
      ii. Detailed solution
      iii. End result clearly identified
   (b) Simulation – interpreted results from simulation screen shots
   (c) Measurement
      i. Circuit schematic with myDAQ connections
      ii. Interpreted results
   (d) Results comparison table
2. Word processor document – submit electronically to instructor:
   (a) Simulation screen shots
      i. Circuit schematic
      ii. Dialog box parameters with circles around entered or modified control values
      iii. Simulation results marked up to highlight key results
   (b) Photo of circuit on breadboard and myDAQ connections (if required)
   (c) Measurement screen shots
      i. ELVISmx signal generator instruments with circles around entered or modified values
      ii. ELVISmx measurement instruments marked up to highlight key results and circles around entered or modified control values

1.4 Acknowledgements

I gratefully acknowledge contributions from the following individuals:

- Tom Robbins (NTS Press) for his editorial support throughout this project,
- Erik Luther (National Instruments) for his enthusiastic support of the NI myDAQ product for engineering education,
- David Salvia (Penn State University) for his helpful suggestions regarding the design of this project, and
- Rose-Hulman students in Electrical Systems ES203 (Spring 2011) who offered much helpful feedback on their experience with selected problems.

Ed Doering
Department of Electrical and Computer Engineering
Rose-Hulman Institute of Technology
Terre Haute, IN 47803

doering@rose-hulman.edu

Chapter 2

Resistive Circuits

2.1 Kirchhoff's Laws (2-3)

Determine currents I_1 to I_3 and the voltage V_1 in the circuit of Figure 2.1 with component values $I_{\mathrm{SRC}} = 1.8$ mA, $V_{\mathrm{SRC}} = 9.0$ V, $R_1 = 2.2$ kΩ, $R_2 = 3.3$ kΩ, and $R_3 = 1.0$ kΩ.

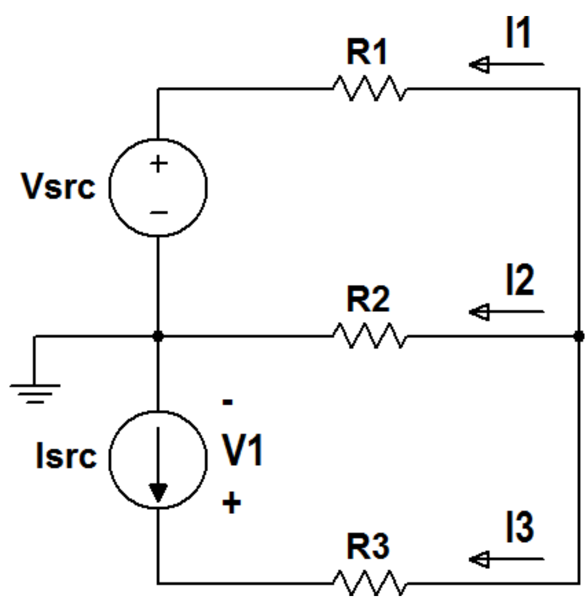


Figure 2.1: Circuit for Problem 2.1

NI Multisim Measurements

Enter the circuit of Figure 2.1 on the preceding page into NI Multisim and measure the currents I_1 to I_3 and the voltage V_1.

- Place components from the "Virtual Components" palette
- Place a Simulate → Instruments → Measurement Probe for each current
- Place a Simulate → Instruments → Multimeter to measure the voltage V_1
- Use interactive simulation Simulate → Run

NI Multisim video tutorials:

- Find commonly-used circuit components:
  `http://youtu.be/G6ZJ8C0ja9Q`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`
- Measure DC voltage with a voltmeter:
  `http://youtu.be/XLyslyikUws`

NI myDAQ Measurements

Build the circuit of Figure 2.1 on the previous page. Use the myDAQ DMM (digital multimeter) to measure the currents I_1 to I_3 and the voltage V_1.

- Implement the voltage source V_{SRC} according to the circuit diagram of Figure B.2 on page 164.
- Measure V_{SRC} with the myDAQ DMM voltmeter and adjust the potentiometer to set the voltage as close to 9.0 volts as possible.
- Implement the current source I_{SRC} according to the circuit diagram of Figure B.4 on page 166. Use a 680 Ω resistor for the adjustment resistor R.
- Measure I_{SRC} with the myDAQ DMM ammeter and confirm that the current is close to 1.8 mA. If you desire more precision, use a 1.0 kΩ potentiometer for R_3 and adjust it accordingly.

NI myDAQ video tutorials:

- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`

Further Exploration with NI myDAQ

Resistor R_3 shares the same current as the current source. Study the effect of this resistor on the rest of the circuit.

1. Replace R_3 with a 1.0 kΩ potentiometer.
2. Measure current I_1 and record the range of currents you observe as you adjust the potentiometer over its full range.
3. Repeat for currents I_2 and I_3 and voltage V_1.
4. Which of the four measured values appears to be independent of the value of R_3?
5. Which of the four measured values appears to depend on the value of R_3?
6. Propose an explanation for your observations.

2.2 Equivalent Resistance (2-4)

Find the equivalent resistance between the following terminal pairs under the stated conditions:

1. A-B with the other terminals unconnected,
2. A-D with the other terminals unconnected,
3. B-C with a wire connecting terminals A and D, and
4. A-D with a wire connecting terminals B and C.

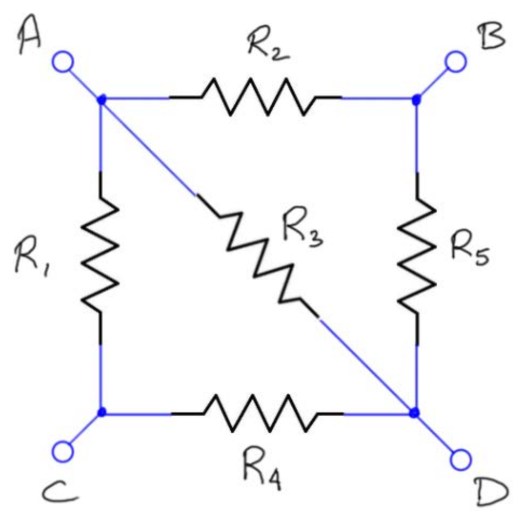


Figure 2.2: Circuit for Problem 2.2

Use these component values:

- $R_1 = 10\ \text{k}\Omega$
- $R_2 = 33\ \text{k}\Omega$
- $R_3 = 15\ \text{k}\Omega$
- $R_4 = 47\ \text{k}\Omega$
- $R_5 = 22\ \text{k}\Omega$

NI Multisim Measurements

Enter the circuit of Figure 2.2 on the facing page into NI Multisim and use the multimeter to measure each of the four resistances under the stated conditions.

- Place components from the "Virtual Components" palette.
- Place a Simulate → Instruments → Multimeter and choose the ohmmeter setting ("Ω" button).
- Place a ground symbol and attach it to one of the multimeter terminals.

NI Multisim video tutorials:

- Find commonly-used circuit components:
  `http://youtu.be/G6ZJ8C0ja9Q`
- Measure resistance with an ohmmeter:
  `http://youtu.be/3G5V0Hxjkbg`

NI myDAQ Measurements

Build the circuit of Figure 2.2 on the preceding page. Use the myDAQ DMM (digital multimeter) as an ohmmeter to measure each of the four resistances under the stated conditions.

- Measure and record the resistance of each resistor individually; do this before you connect the resistors together.
- Place the resistors to match the resistor orientations shown in Figure 2.2 on the facing page.
- The circuit need not connect to the myDAQ analog ground AGND terminal; only Multisim requires the ground connection.

NI myDAQ video tutorials:

- DMM ohmmeter:
  `http://decibel.ni.com/content/docs/DOC-12938`

Further Exploration with NI myDAQ

Ohm's Law states that a resistor creates a proportional relationship between its voltage and current $v = iR$ where the resistance R is the proportionality factor. Setting the resistor voltage v to a known value and measuring the resulting current with an ammeter provides another way to measure resistance. Apply this method to measure each of the four resistances and compare with your previous results.

1. Apply the NI myDAQ 5-volt source to the terminals A and D. Use the 5V and DGND (digital ground) terminals, with 5V connected to terminal A and DGND to terminal D.

2. Use the DMM voltmeter to measure the voltage v as it appears at the resistor network, and then record this value. Expect the voltage to be slightly less than 5.0 volts, and also expect that it will vary somewhat from one circuit connection to the next.

3. Use the DMM ammeter to measure the current i flowing into terminal A; record this value, too.

4. Calculate the effective resistance R of the resistor network from your two measurements, and then compare this value to your other measurements.

5. Repeat for the remaining three resistance measurements.

2.3 Current and Voltage Dividers (2-4)

Apply the concepts of voltage dividers, current dividers, and equivalent resistance to find the currents I_1 to I_3 and the voltages V_1 to V_3.

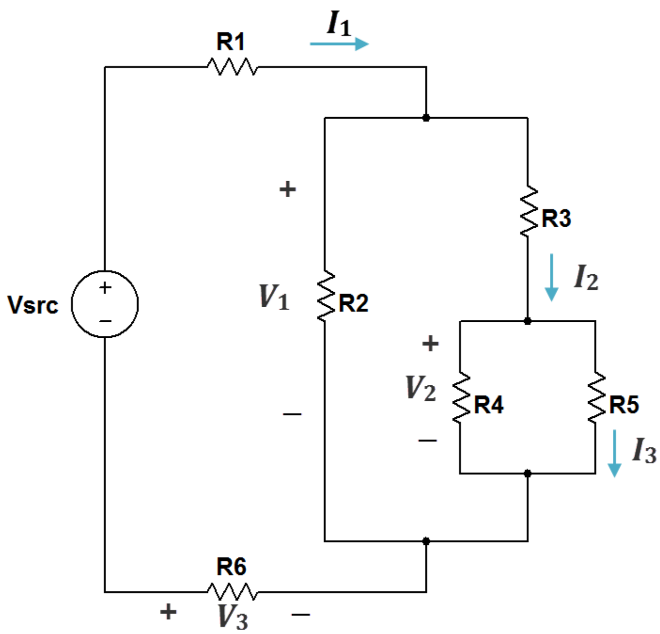


Figure 2.3: Circuit for Problem 2.3

Use these component values:

- $V_{\text{SRC}} = 12$ V
- $R_1 = 1.0\,\text{k}\Omega$, $R_2 = 10\,\text{k}\Omega$, $R_3 = 1.5\,\text{k}\Omega$, $R_4 = 2.2\,\text{k}\Omega$, $R_5 = 4.7\,\text{k}\Omega$, and $R_6 = 3.3\,\text{k}\Omega$

NI Multisim Measurements

Enter the circuit of Figure 2.3 into NI Multisim. Use measurement probes to measure each current; use voltmeter indicators to measure each voltage.

- Place components from the "Virtual Components" palette.
- Place a ground symbol and attach it to the negative terminal of the voltage source.

- Place a Simulate → Instruments → Measurement Probe for each current.
- Place a voltmeter indicator to display each voltage (see video tutorial for details).

NI Multisim video tutorials:

- Find commonly-used circuit components:
  `http://youtu.be/G6ZJ8C0ja9Q`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`
- Measure DC voltage with a voltmeter indicator:
  `http://youtu.be/8h2SAZ9gkBA`

NI myDAQ Measurements

Build the circuit of Figure 2.3 on the preceding page. Use the myDAQ DMM (digital multimeter) as a voltmeter to measure each of the three voltages; use the DMM as an ammeter to measure each of the three currents.

- Measure and record the resistance of each resistor individually; do this before you connect the resistors together.
- Place the resistors to match the resistor orientations shown in Figure 2.3 on the previous page.
- Implement the voltage source V_{SRC} according to the circuit diagram of Figure B.2 on page 164.
- Measure V_{SRC} with the myDAQ DMM voltmeter and adjust the potentiometer to set the voltage as close to 12.0 volts as possible.

NI myDAQ video tutorials:

- DMM ohmmeter:
  `http://decibel.ni.com/content/docs/DOC-12938`
- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`

2.4 Wye-Delta Transformation (2-5)

1. Find the currents I_1 and I_2.
2. Determine the power delivered by each of the two voltage sources.

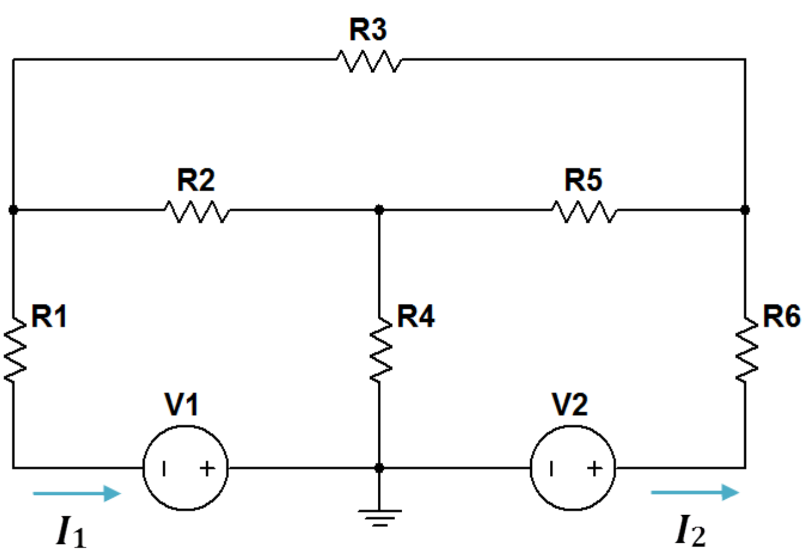


Figure 2.4: Circuit for Problem 2.4

Use these component values:

- $V_1 = 15$ V and $V_2 = 15$ V
- $R_1 = 3.3$ kΩ, $R_2 = 1.5$ kΩ, $R_3 = 4.7$ kΩ, $R_4 = 5.6$ kΩ, $R_5 = 1.0$ kΩ, and $R_6 = 2.2$ kΩ

NI Multisim Measurements

Enter the circuit of Figure 2.4 into NI Multisim. Use measurement probes to measure each current, and use the wattmeter to measure the power associated with each voltage source.

- Place components from the "Virtual Components" palette.

- Place a ground symbol and attach it to the negative terminal of the voltage source.
- Place a Simulate → Instruments → Measurement Probe for each current.
- Place a Simulate → Instruments → Wattmeter for each voltage source, taking care to wire the wattmeters according to the passive sign convention.

NI Multisim video tutorials:

- Find commonly-used circuit components:
  `http://youtu.be/G6ZJ8C0ja9Q`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`
- Measure DC power with a wattmeter:
  `http://youtu.be/-axVClpMpiU`

NI myDAQ Measurements

Build the circuit of Figure 2.4 on the preceding page. Use the myDAQ DMM (digital multimeter) as an ammeter to measure each of the two currents; use the DMM as a voltmeter to measure each of the two voltage source values.

- Measure and record the resistance of each resistor individually; do this before you connect the resistors together.
- Place the resistors to match the resistor orientations shown in Figure 2.4 on the previous page.
- Use the myDAQ -15V power supply connection for the left voltage source and the +15V power supply connection for the right voltage source; connect AGND (Analog Ground) to the node identified by the ground symbol.
- Measure the actual values of V_1 and V_2 when connected to the circuit; expect them to be slightly less than 15 volts.
- Remember to insert the DMM ammeter in series between the voltage source and the resistor.

- Determine power as the product of measured voltage and measured current.

NI myDAQ video tutorials:

- DMM ohmmeter:
  `http://decibel.ni.com/content/docs/DOC-12938`
- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`

Chapter 3

Analysis Techniques

3.1 Node-Voltage Method (3-1)

Apply the node-voltage method to determine the node voltages V_1 to V_4 for the circuit of Figure 3.1 on the following page. From these results determine which resistor dissipates the most power and which resistor dissipates the least power, and report these two values of power.

Use these component values:

- $I_{\text{src1}} = 3.79$ mA and $I_{\text{src2}} = 1.84$ mA
- $V_{\text{src}} = 4.00$ V
- $R_1 = 3.3$ kΩ, $R_2 = 2.2$ kΩ, $R_3 = 1.0$ kΩ, and $R_4 = 4.7$ kΩ

NI Multisim Measurements

Enter the circuit of Figure 3.1 on the next page into NI Multisim. Use DC operating point analysis to determine the four node voltages and the power dissipated by each resistor.

- Display the net names and rename them to match the four node voltages V_1 to V_4; use only the numbers for the net names.
- Set up a Simulate → Analyses → DC Operating Point analysis to display the four node voltages and the power associated with each resistor.

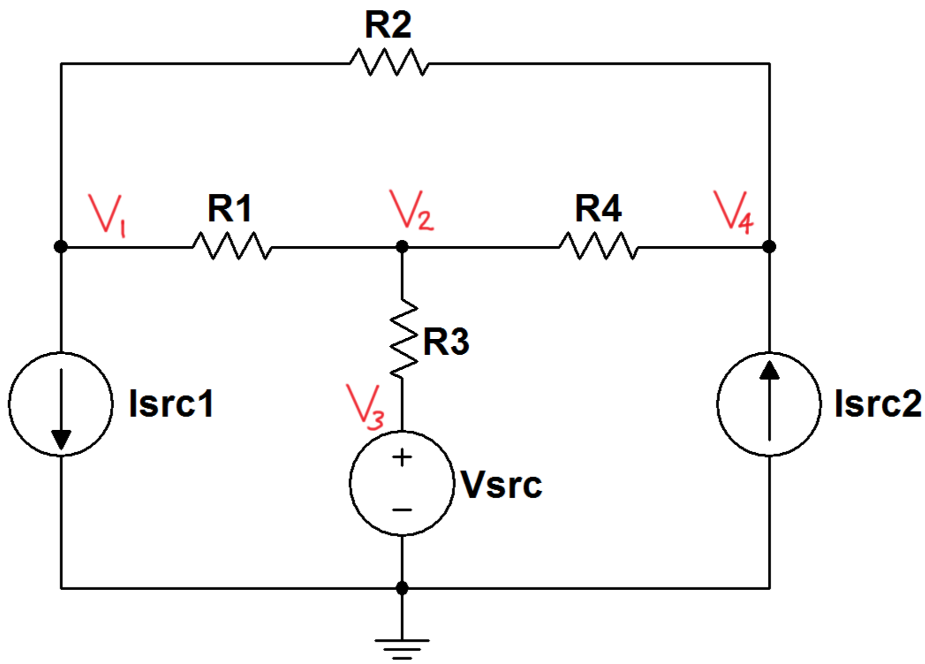


Figure 3.1: Circuit for Problem 3.1

NI Multisim video tutorials:

- Display and change net names:
  `http://youtu.be/0iZ-ph9pJjE`
- Find node voltages with DC Operating Point analysis:
  `http://youtu.be/gXBCqP17AZs`
- Find resistor power with DC Operating Point analysis:
  `http://youtu.be/NxXmVDW9spo`

NI myDAQ Measurements

Build the circuit of Figure 3.1. Use the myDAQ DMM (digital multimeter) as a voltmeter to measure each of the four node voltages.

- Implement the voltage source V_{SRC} according to the circuit diagram of Figure B.2 on page 164.

- Measure V_{SRC} with the myDAQ DMM voltmeter and adjust the potentiometer to set the voltage as close to 4.00 volts as possible. Record the actual voltage you measured.

- Implement the current source I_{src1} according to the circuit diagram of Figure B.5 on page 167. Use a 330 Ω resistor for the adjustment resistor R.

- Measure I_{src1} with the myDAQ DMM ammeter and confirm that the current is close to 3.79 mA. Record the actual current you measured.

- Implement the current source I_{src2} according to the circuit diagram of Figure B.4 on page 166. Use a 680 Ω resistor for the adjustment resistor R.

- Measure I_{src2} with the myDAQ DMM ammeter and confirm that the current is close to 1.84 mA. Record the actual current you measured.

NI myDAQ video tutorials:

- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`

- Measure node voltage:
  `http://decibel.ni.com/content/docs/DOC-12947`

- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`

Further Exploration with NI myDAQ

As you are by now aware, the analytical solution and the simulation results always agree very well, largely because you can enter exact component values into the simulator. However, the physical circuit component values do not match the nominal values exactly: the 5%-tolerance resistors can vary ±5% from the nominal value represented by the color-coded bands, and the "$1250/R$ mA" formula for the LM317-based current source is an approximation.

Explore what happens when you recalculate the analytical solution using measured component values.

1. Recalculate the four node voltages using the *nominal* resistor values and the *measured* values for I_{src1}, I_{src2}, and V_{SRC}. Create a data table to compare these values with your analytical results in terms of difference and relative difference.

2. Measure and record the five resistances R_1 to R_4.

3. Recalculate the four node voltages using the *measured* resistor values and the measured values for I_{src1}, I_{src2}, and V_{SRC}. Create a data table to compare these values with your analytical results in terms of difference and relative difference.

4. Summarize your results: What level of agreement did you achieve between the analytical solution and the physical measurements?

3.2 Mesh-Current Method (3-2)

Apply the mesh-current method to determine the mesh currents I_1 to I_4. From these results determine the voltage across the current source V_1.

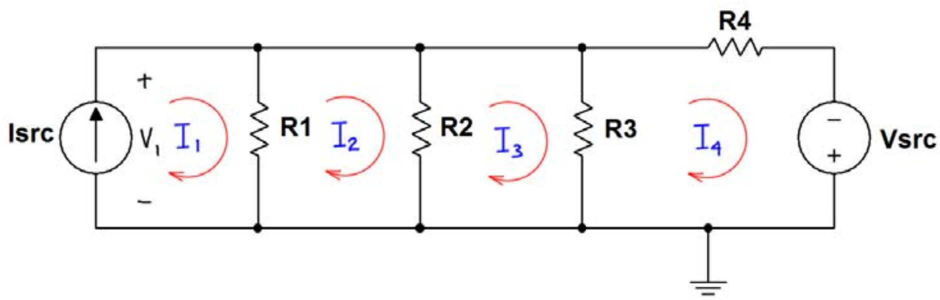


Figure 3.2: Circuit for Problem 3.2

Use these component values:

- $I_{\text{SRC}} = 12.5$ mA
- $V_{\text{SRC}} = 15$ V
- $R_1 = 5.6$ kΩ, $R_2 = 2.2$ kΩ, $R_3 = 3.3$ kΩ, and $R_4 = 4.7$ kΩ

NI Multisim Measurements

Enter the circuit of Figure 3.2 into NI Multisim. Use "Measurement Probes" and interactive simulation to measure the four mesh currents. Use the multimeter or a measurement probe to display the voltage across the current source.

- Place the measurement probe on wires that carry only a single mesh current; remember that many of the resistors carry two mesh currents.

NI Multisim video tutorials:

- Measure DC mesh current with a measurement probe:
  `http://youtu.be/lK0LcTNroXI`
- Measure DC node voltage with a measurement probe:
  `http://youtu.be/svNGHA2-uK4`
- Measure DC voltage with a voltmeter:
  `http://youtu.be/XLyslyikUws`

NI myDAQ Measurements

Build the circuit of Figure 3.2 on the previous page. Use the myDAQ DMM (digital multimeter) as an ammeter to measure each of the four mesh currents; use the DMM voltmeter to measure the voltage across the current source.

- Place the resistors to match the resistor orientations shown in Figure 3.2 on the preceding page. Use 1-inch jumper wires to establish the top connections between the resistors to facilitate measurement of the mesh currents.
- Implement the voltage source V_{SRC} with the NI myDAQ -15V power supply.
- Implement the current source I_{SRC} according to the circuit diagram of Figure B.4 on page 166. Use a 100 Ω resistor for the adjustment resistor R.
- Measure I_{SRC} with the myDAQ DMM ammeter and confirm that the current is close to 12.5 mA. If you desire more precision, use a 1.0 kΩ potentiometer for the adjustment resistor.

NI myDAQ video tutorials:

- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`

Further Exploration with NI myDAQ

Some types of digital-to-analog converters require binary-weighted currents that can be selectively summed together. With a slight modification to your existing circuit topology you can redesign it to produce mesh currents that meet your own specifications such as those required by the digital-to-analog converter.

1. Consider the modified circuit of Figure 3.3. Apply mesh-current analysis to write a set of equations in terms of the indicated currents and resistor values.
2. Choose resistor values that will establish the binary-weighted current values $I_2 = I_{SRC}/2$, $I_3 = I_{SRC}/4$, $I_4 = I_{SRC}/8$, and $I_5 = I_{SRC}/16$, *and* that will limit the current source voltage V_1 to 5 volts or less.
3. *Note:* The standard parts list of Appendix A on page 159 includes resistors that are close to the calculated values you need.
4. Build the circuit and measure all five mesh currents I_1 to I_5.
5. Measure the current source voltage V_1.
6. Evaluate your results to determine how well the circuit produces the desired binary-weighted currents.

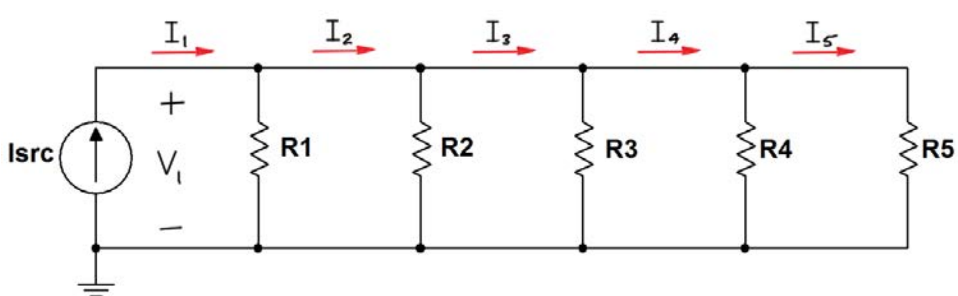


Figure 3.3: Modified circuit for Problem 3.2 to produce binary-weighted currents.

3.3 Superposition (3-4)

1. Apply the superposition method to determine the current I_A and the voltage V_B, i.e., find the current I_{A1} due to the current source I_1 acting alone, the current I_{A2} due to the voltage source V_2 acting alone, and the current I_{A3} due to the voltage source V_3 acting alone, and then evaluate the sum $I_A = I_{A1} + I_{A3} + I_{A3}$. Make use of current dividers and voltage dividers as much as possible.

2. Use the superposition method to determine the voltage V_B.

3. Apply the node-voltage method to find I_A and V_B, and then compare these results to those of the superposition method.

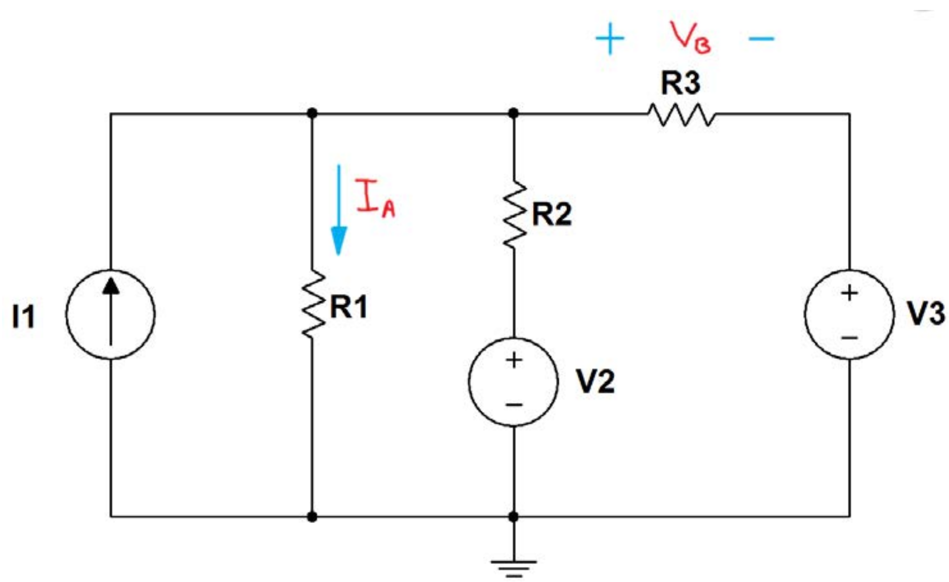


Figure 3.4: Circuit for Problem 3.3

Use these component values:

- $I_1 = 1.84$ mA
- $V_2 = 3.0$ V and $V_3 = 4.9$ V
- $R_1 = 1.0$ kΩ, $R_2 = 2.2$ kΩ, and $R_3 = 4.7$ kΩ

NI Multisim Measurements

Enter the circuit of Figure 3.4 on the facing page into NI Multisim. Use interactive analysis and the voltmeter and ammeter indicators.

- Place the `AMMETER_V` and `VOLTMETER_H` components to display the current I_A and the voltage V_B.
- Set the active source to its intended value, and then set all of the other sources to zero. After stopping the simulator, press Ctrl+Z ("undo") two times to return the sources to their original values.
- Repeat to determine the responses due to each source acting alone.

NI Multisim video tutorials:

- Measure DC voltage with a voltmeter indicator:
  `http://youtu.be/8h2SAZ9gkBA`
- Measure DC current with an ammeter indicator:
  `http://youtu.be/8P4oFw6sIzQ`

NI myDAQ Measurements

1. Build the circuit of Figure 3.4 on the preceding page. Use the myDAQ DMM to measure I_A and V_B when all sources are active.
2. Measure I_{A1} to I_{A3} by activating only one source at a time. Disable the other sources by disconnecting the current source (replace it by an open circuit) and disconnecting the voltage source and replacing it with a jumper wire (short circuit).
3. Repeat for V_{B1} to V_{B3}.
4. Add your results together for each source acting alone, and then compare this result to your original measurement when all sources were active.

Additional helpful tips:

- Implement the voltage source V_2 according to the circuit diagram of Figure B.2 on page 164.

- Measure V_2 with the myDAQ DMM voltmeter and adjust the potentiometer to set the voltage as close to 3.00 volts as possible. Record the actual voltage you measured.
- Implement the voltage source V_3 with the NI myDAQ 5V power supply. Connect the myDAQ digital ground DGND terminal to the analog ground AGND at your breadboard.
- Measure V_3 with the myDAQ DMM voltmeter when the circuit is connected. Expect to find this value slightly less than 5.0 volts. Record the actual voltage you measured.
- Implement the current source I_1 according to the circuit diagram of Figure B.4. Use a 680 Ω resistor for the adjustment resistor R.
- Measure I_1 with the myDAQ DMM ammeter and confirm that the current is close to 1.84 mA. Record the actual current you measured.

NI myDAQ video tutorials:

- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`

3.4 Thévenin Equivalents, Maximum Power Transfer (3-5, 3-6)

1. Find the Thévenin equivalent of the circuit of Figure 3.5 at terminals (a,b) as seen by the load resistance R_L.

2. Determine the open-circuit voltage V_{OC} that appears at terminals (a,b).

3. Determine the short-circuit current I_{SC} that flows through a wire connecting terminals (a,b) together.

4. Determine the maximum power P_{Lmax} that could be delivered by this circuit.

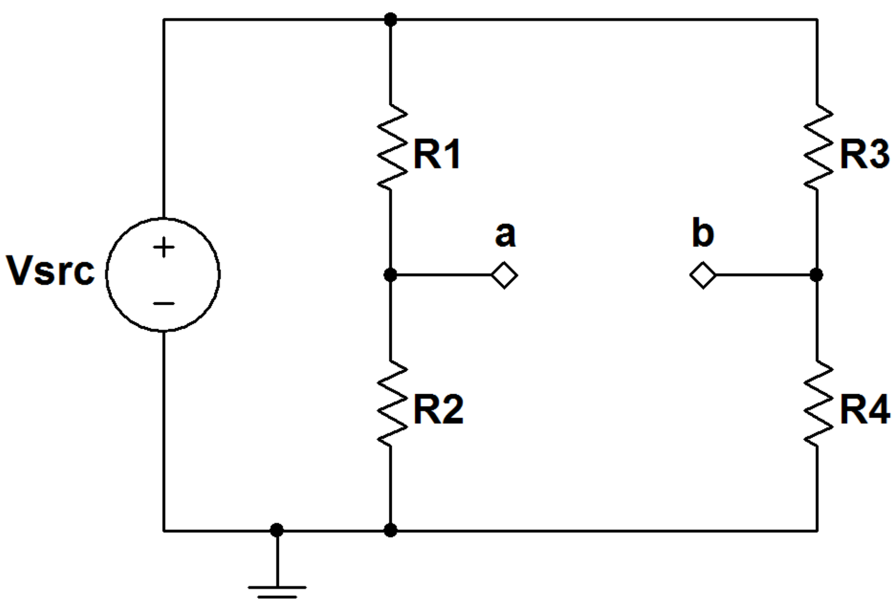


Figure 3.5: Circuit for Problem 3.4

Use these component values:

- $V_{SRC} = 10$ V
- $R_1 = 680\ \Omega$, $R_2 = 3.3\ \text{k}\Omega$, $R_3 = 4.7\ \text{k}\Omega$, and $R_4 = 1.0\ \text{k}\Omega$

NI Multisim Measurements

1. Enter the circuit of Figure 3.5 on the preceding page into NI Multisim. Connect a resistor R_{L} as a load between terminals (a, b).

2. Use interactive analysis and measurement probes to determine the open-circuit voltage.

3. Use interactive analysis and measurement probes to determine the short-circuit current.

4. Run a parameter sweep to plot the load resistance power as a function of load resistance connected between terminals (a, b). Use a plot cursor to determine the value of maximum power.

These tips provide more detail about the Multisim techniques for this problem:

- Place a measurement probe on terminal b to display the load current.

- Place a measurement probe on terminal a referenced to the probe you placed on terminal b to display the voltage across the load.

- Set the load resistance to a small yet finite value such as 0.1 Ω. Run the interactive simulator to determine the short-circuit current.

- Set the load resistance to a large yet finite value such as 100 MΩ; enter this value as `100MEG` rather than `100M` because ”m” means ”milli” regardless of case. Run the simulator to determine the open-circuit voltage.

- Set up a Simulate → Analyses → Parameter Sweep to plot `P(RL)` over the range 1 Ω to 10 kΩ. Choose a linear plot type, select “DC Operating Point” for “Analysis to Sweep,” and plot 100 evenly-spaced points to create a smooth curve.

- Use the plot cursors to find the maximum value of the load power. Compare this value to the maximum power you calculated analytically.

NI Multisim video tutorials:

- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`
- Measure DC voltage with a referenced measurement probe:
  `http://youtu.be/xKEQ3EXEaP8`
- Use a Parameter Sweep analysis to plot resistor power as a function of resistance:
  `http://youtu.be/3k2g9Penuag`
- Find the maximum value of trace in Grapher View:
  `http://youtu.be/MzYK60mfh2Y`

NI myDAQ Measurements

1. Build the circuit of Figure 3.5 on page 35. Calculate the Thévenin equivalent circuit from the measurements taken in the next two parts.

2. Recall that the DMM voltmeter has very high resistance and thus appears as an open circuit. Connect the voltmeter between terminals (a, b) to measure the open-circuit voltage.

3. Also recall that the DMM ammeter has very low resistance and thus appears as a short circuit. Connect the ammeter between terminals (a, b) to measure the short-circuit current.

4. Connect the variable load circuit shown in Figure 3.6 on the following page between terminals (a, b) and connect the myDAQ analog input channels to measure the overall load voltage and the voltage that appears across the shunt resistor; this latter voltage is proportional to the load current. Run the LabVIEW VI "VIPR.vi" (described below) to display the load's voltage, current, power, and resistance. First sweep the potentiometer throughout its full range to get a sense of the overall behavior, and then collect and tabulate at least 10 measurements of load power and load resistance; adjust the potentiometer to take measurements in 1 mW steps. Also record the maximum power and associated load resistance. Finally, plot the load power as a function of load resistance.

LabVIEW "VIPR.vi" details:

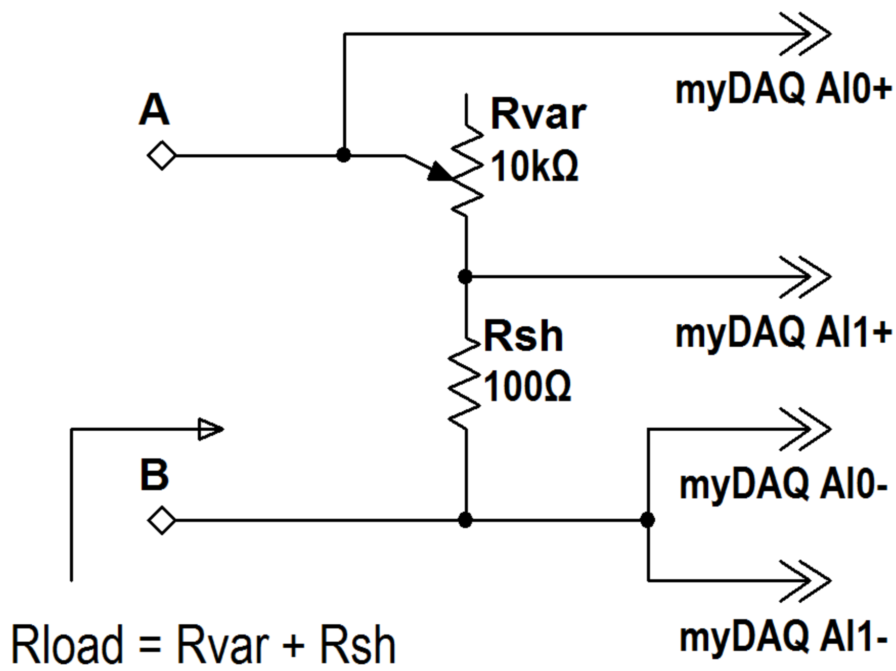


Figure 3.6: Variable load with potentiometer (variable resistor) R_{var} and shunt resistor R_{sh}. The total load resistance is $R_{\text{var}}+R_{\text{sh}}$. NI myDAQ Analog Input 0 (AI0) monitors the overall load voltage between terminals A-B and Analog Input 1 (AI1) monitors the voltage across the shunt resistor; the load current is the shunt resistor voltage divided by R_{sh}.

- The LabVIEW VI "VIPR.vi" measures the overall load voltage on analog input channel 0 (AI0+ and AI0-) and the shunt resistor voltage on analog input channel 1 (AI1+ and AI1-). Enter the measured shunt resistance for best accuracy. "VIPR.vi" calculates the load current as the voltage on AI1 divided by the entered shunt resistance value, the load power as the product of load voltage and current, and the load resistance as the load voltage divided by the current.

- The measured current value can become somewhat noisy, and "VIPR.vi" applies a noise filter to improve your ability to read the display. The noise filter calculates the average value of all of the measurements accumulated since the last time the measured voltage changed by at least 0.01 volts. Disable the noise filter, if desired.

- "VIPR.vi" is linked at the bottom of `http://decibel.ni.com/content/docs/DOC-16389`. Download this source file and double-click it to open in LabVIEW; click the "Run" button to start the VI.

NI myDAQ video tutorials:

- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`

Further Exploration with NI myDAQ

Try this simple yet effective technique to directly measure Thévenin resistance:

1. Measure the open-circuit voltage at terminals (a, b),
2. Connect a variable resistor as the load (10 kΩ potentiometer works well for this circuit),
3. Monitor the load voltage and adjust the potentiometer until the voltage is exactly *one half* of the open-circuit voltage,
4. Disconnect the potentiometer from the circuit, and
5. Measure the potentiometer resistance with an ohmmeter; this value is the Thévenin resistance.

Apply this method to the circuit of this problem and compare your results to your other measurements of Thévenin resistance.

Explain why this method works. HINT: Consider a Thévenin equivalent circuit connected to a load resistor and recall what you know about voltage dividers.

Chapter 4

Operational Amplifiers

4.1 Ideal Op-Amp Model (4-3)

1. Determine a general expression for v_{out} in terms of the resistor values and i_{S} for the circuit of Figure 4.1 on the next page.

2. Find v_{out} for these specific component values: $R_1 = 3.3$ kΩ, $R_2 = 4.7$ kΩ, $R_3 = 1.0$ kΩ, and i_{S} = 1.84 mA.

3. Determine the range of R_2 for which $-11 \leq v_{out} \leq +11$ volts.

NI Multisim Measurements

1. Enter the circuit of Figure 4.1 on the following page into NI Multisim. Use the virtual three-terminal op amp model.

2. Measure v_{out} for the given set of component values.

3. Plot v_{out} as a function of R_2 with a Simulate → Analyses → Parameter Sweep analysis of the "DC Operating Point" type. Increase the number of points as needed to ensure an adequate number of measurements to characterized the op amp output voltage in the region of the specified voltage limits.

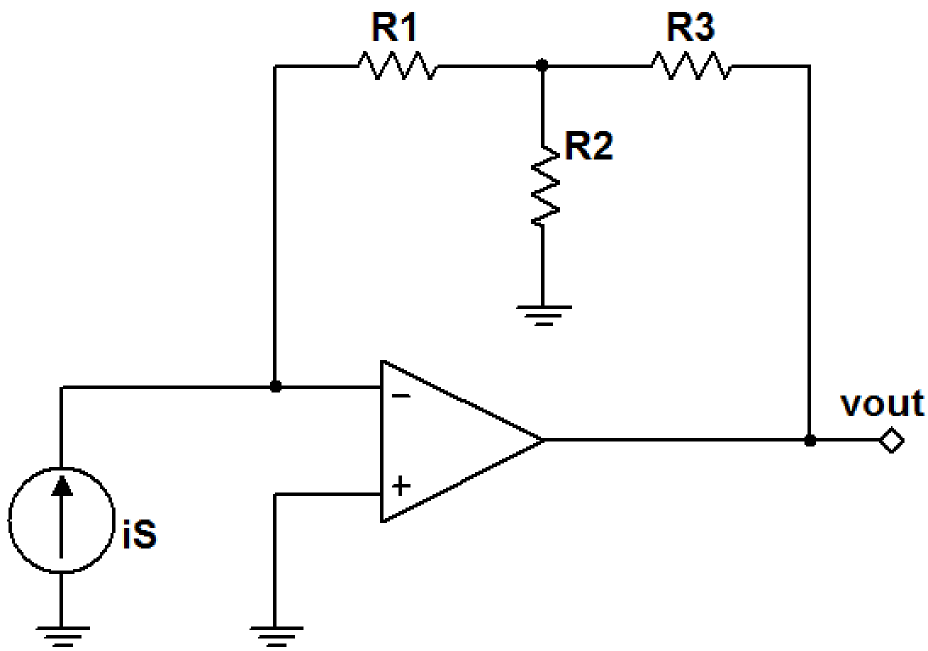


Figure 4.1: Circuit for Problem 4.1

NI Multisim video tutorials:

- Use a Parameter Sweep analysis to plot resistor power as a function of resistance:
  `http://youtu.be/3k2g9Penuag`
- Measure DC node voltage with a measurement probe:
  `http://youtu.be/svNGHA2-uK4`

NI myDAQ Measurements

1. Build the circuit of Figure 4.1 with the given component values. Implement the current source I_{src1} according to the circuit diagram of Figure B.4 on page 166. Use a 680 Ω resistor for the adjustment resistor R. When complete, measure and record the source current i_S with the DMM ammeter and confirm that it is close to 1.84 mA.

2. Measure the value of the v_{out} with the DMM voltmeter.

3. Replace R_2 with a 10 kΩ potentiometer. Monitor v_{out} and adjust the potentiometer until the voltage reaches the specified limit. Disconnect the potentiometer from the circuit and then measure its resistance with the DMM ohmmeter.

Additional helpful tips for this section:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 170 for either of the two available op amps in the package. You may also use an equivalent dual-supply op amp.
- Power the op amp with myDAQ +15V to V_{CC+} and -15V to V_{CC-}. Use AGND for the circuit ground.

NI myDAQ video tutorials:

- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`

4.2 Noninverting Amplifier (4-3)

The circuit in Figure 4.2 uses a potentiometer whose total resistance is R_1. The movable stylus on terminal 2 creates two variable resistors: βR_1 between terminals 1–2 and $(1-\beta)R_1$ between terminals 2–3. The movable stylus varies β over the range $0 \leq \beta \leq 1$.

1. Obtain an expression for $G = v_o/v_s$ in terms of β.
2. Calculate the amplifier gain for $\beta = 0.0$, $\beta = 0.5$, and $\beta = 1.0$ with component values $R_1 = 10\ \text{k}\Omega$ and $R_2 = 1.5\ \text{k}\Omega$.
3. Let v_s be a 100-Hz sinusoidal signal with a 1-volt peak value. Plot v_o and v_s to scale for $\beta = 0.0$, $\beta = 0.5$, and $\beta = 1.0$.

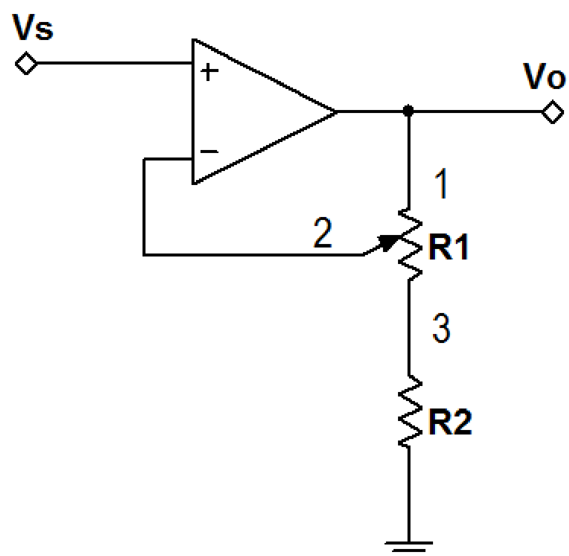


Figure 4.2: Circuit for Problem 4.2

NI Multisim Measurements

1. Enter the circuit of Figure 4.2 into NI Multisim. Use these specific components: `OPAMP_3T_VIRTUAL`, `AC_VOLTAGE`, and virtual linear

potentiometer; see the video tutorial below to learn how to search for parts by name. Set the AC voltage source frequency to 100 Hz.

2. Observe v_s and v_o with the oscilloscope. Vary the potentiometer value over its full range of 0 to 100%, and then use the oscilloscope cursors to measure the circuit gain for $\beta = 0.0$, $\beta = 0.5$, and $\beta = 1.0$.

3. Print screen shots of the oscilloscope for $\beta = 0.0$, $\beta = 0.5$, and $\beta = 1.0$. Use the same Channel A and Channel B vertical scale (volts per division) for all three screen shots.

Additional helpful tips:

- Use the Simulate $\rightarrow$ Instruments $\rightarrow$ Oscilloscope with v_{s} on Channel A and v_{o} on Channel B. Run an interactive simulation until several cycles of oscillation appear on the oscilloscope display.

- Use the cursors to measure the peak values of the input and output signals, and then calculate the amplifier gain as the output value divided by the input value.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`

- AC (sinusoidal) voltage source:
  `http://youtu.be/CXbuz7MVLSs`

- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`

- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`

NI myDAQ Measurements

1. Build the circuit of Figure 4.2 on the facing page with the given component values. Use the following myDAQ signal connections:

   - AO0 (Analog Output 0) for v_{s},
   - AI0 (Analog Input 0) to display v_{s}; connect AI0+ to the source voltage and AI0- to ground, and

- AI1 (Analog Input 1) to display v_o; connect AI1+ to the output voltage and AI1- to ground.

Create the 100-Hz sinusoidal waveform with the NI ELVISmx Function Generator.

2. Observe v_s and v_o with the NI ELVISmx Oscilloscope. Vary the potentiometer value over its full range of 0 to 100%, and then use the oscilloscope cursors to measure the circuit gain for $\beta = 0.0$, $\beta = 0.5$, and $\beta = 1.0$.

3. Print screen shots of the oscilloscope for $\beta = 0.0$, $\beta = 0.5$, and $\beta = 1.0$. Use the same Channel 0 and Channel 1 vertical scale (volts per division) for all three screen shots.

Additional helpful tips:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 170 for either of the two available op amps in the package. You may also use an equivalent dual-supply op amp.
- Power the op amp with myDAQ +15V to V_{CC+} and -15V to V_{CC-}. Use AGND for the circuit ground.
- The potentiometer terminals of Figure 4.2 on page 44 follow the standard pinout used by single-turn trim potentiometers. If your potentiometer does not label the pins, the stylus pin (Pin 2) is normally placed between Pins 1 and 3.

NI myDAQ video tutorials:

- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`
- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`

Further Exploration with NI myDAQ

Signal amplifiers apply a gain $G \geq 1$ to increase the amplitude of weak signals, thereby making the signal information easier to use elsewhere in the system. Use a voltage divider circuit (known as an *attenuator* in this context) to create a "weak" signal from a portable audio player or computer audio output and then investigate how well the amplifier you built in this problem can restore the original signal strength.

1. Add the two-resistor voltage divider circuit to your amplifier as shown in Figure 4.3 on the next page.

2. Connect one plug of the 3.2 mm stereo cable supplied with your myDAQ kit to your audio player. Connect the other plug to the attenuator input v_m as shown in Figure 4.3 on the following page to apply the left channel of the stereo audio signal to the attenuator input.

3. Play some music and observe the signal v_s with the oscilloscope. Confirm that signal is indeed "weak" – its amplitude should be well under one volt peak.

4. Observe the amplifier output signal v_o with the oscilloscope. Confirm that you can adjust the circuit gain to strengthen the music signal's amplitude.

5. Connect your earphones to the circuit output and listen as you vary the circuit gain.

   IMPORTANT – PROTECT YOUR HEARING!
   Do **NOT** disturb your circuit connections while you are wearing earphones. Accidently shorting together circuit connections can produce a very loud and unexpected noise. Alternatively, use a speaker to listen to the amplifier output or hold the earphones some distance from your ears.

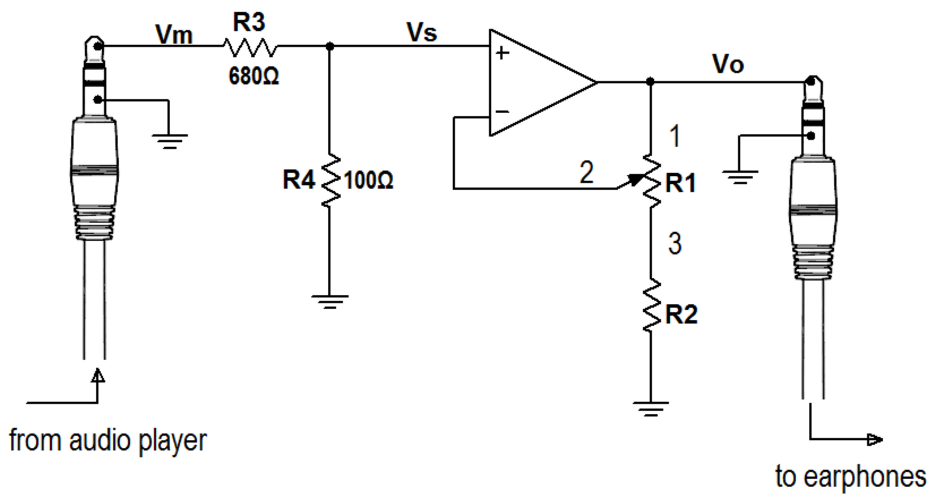


Figure 4.3: Circuit for Problem 4.2 with voltage-divider attenuator and audio signal connections. The music signal is v_{m}, the attenuated signal to be amplified is v_{s}, and the amplified signal is v_{o}.

4.3 Summing Amplifier (4-5)

1. Design an op amp summing circuit that performs the operation $v_o = -(2.14v_1 + 1.00v_2 + 0.47v3)$. Use not more than four standard-value resistors with values between 10 kΩ and 100 kΩ. Refer to the resistor parts list in Appendix A on page 159.

2. Draw the output waveform v_o for the input waveforms v_1 and v_2 shown in Figure 4.4 and v_3 = 4.7 volts.

3. State the minimum and maximum values of v_o.

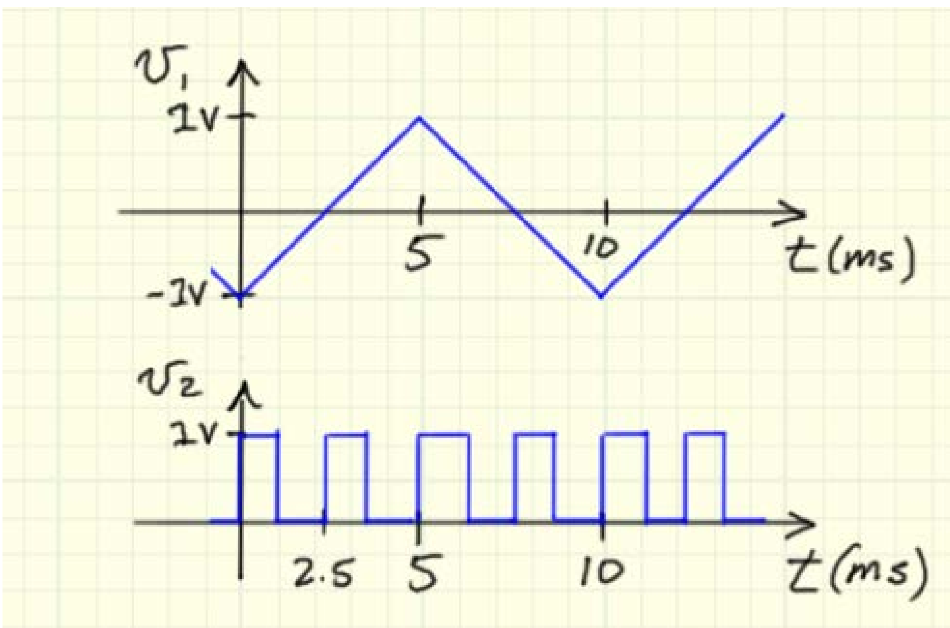


Figure 4.4: Input waveforms for Problem 4.3

NI Multisim Measurements

1. Enter the op amp summing circuit that you designed earlier. Use the following components and instruments:

    - Virtual three-terminal op amp model `OPAMP_3T_VIRTUAL`

- Piecewise-linear voltage source PIECEWISE_LINEAR_VOLTAGE for v_1
- Pulse voltage source PULSE_VOLTAGE for v_2
- Four-channel oscilloscope

2. Plot v_o and the three inputs v_1 to v_3 with the four-channel oscilloscope.

3. Use the oscilloscope display cursors to identify the minimum and maximum values of v_o.

Additional Multisim tips for this problem:

- Specify the PWL voltage source waveform by entering endpoints of straight lines as time-voltage pairs. The triangle waveform of Figure 4.4 on the previous page requires only three entries to specify a complete period. Select "Repeat data during simulation" to create a periodic triangle waveform.

- Three fields need to be adjusted for the pulse voltage source to make it match the required square waveform shape: "Initial Value," "Pulse Width," and "Period."

NI Multisim video tutorials:

- Basic operation of the four-channel oscilloscope:
  http://youtu.be/iUqs_c1Bc4Y
- Piecewise linear (PWL) voltage source:
  http://youtu.be/YYU5WuyebD0
- Pulse voltage source:
  http://youtu.be/RdgxVfr28C8
- Find components by name:
  http://youtu.be/5wlFweh4n-c

NI myDAQ Measurements

1. Build the op amp summing circuit that you designed earlier. Use the following myDAQ signal connections:

   - AO0 (Analog Output 0) for v_1,

- AO1 (Analog Output 1) for v_2,
- AI0 (Analog Input 0) to display either v_1 or v_2; connect AI0+ to the input voltage of interest and AI0- to ground,
- AI1 (Analog Input 1) to display v_o; connect AI1+ to the output voltage and AI1- to ground,

Create the triangle and square waveforms with the NI ELVISmx Arbitrary Waveform Generator; use 50 kS/s as the sampling rate.

IMPORTANT: The two waveform files must of the same length (10 ms).

2. Plot v_o and v_1 with the NI ELVISmx Oscilloscope. Repeat with v_2.
3. Use the oscilloscope display cursors to identify the minimum and maximum values of v_o.

Additional tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C on page 169. Follow the pinout diagram of Figure C.1 on page 170 for either of the two available op amps in the package. You may also use an equivalent dual-supply op amp.
- Power the op amp with myDAQ +15V to V_{CC+} and -15V to V_{CC-}. Use AGND for the circuit ground.
- Implement the constant voltage source v_3 according to the circuit diagram of Figure B.2 on page 164. Adjust the potentiometer until the measured voltage is as close to 4.70 volts as possible.

NI myDAQ video tutorials:

- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

Further Exploration with NI myDAQ

Investigate the effect of the gain constants for waveform inputs v_1 and v_2. You can quickly and easily vary a resistor value by placing another resistor in parallel with it, thereby reducing the effective resistance. Place a 10 kΩ in parallel with the source resistor associated with waveform v_1 and observe the impact on the output voltage waveform. Plot the new output waveform, summarize the difference from the original waveform, and explain why reducing the resistor value causes this change in appearance.

Repeat the experiment with the source resistor associated with waveform v_2.

4.4 Signal Processing Circuits (4-8)

1. Design a two-stage signal processor to serve as a "distortion box" for an electric guitar. The first-stage amplifier applies a variable gain magnitude in the range 13.3 to 23.3 while the second-stage amplifier attenuates the signal by 13.3, i.e., the second-stage amplifier has a fixed gain of 1/13.3. Note that when the first-stage amplifier gain is 13.3 the overall distortion box gain is unity. The distortion effect relies on intentionally driving the first-stage amplifier into saturation (also called "clipping") when its gain is higher than 13.3.

   Use a 10 kΩ potentiometer and standard-value resistors in the range 1.0 kΩ to 100 kΩ; see the resistor parts list in Appendix A on page 159. You may combine two standard-value resistors in series to achieve the required amplifier gains.

2. Derive a general formula for *percent clipping* of a unit-amplitude sinusoidal test signal; this is the percent of time during one period in which the signal is clipped. The formula includes the peak sinusoidal voltage V_P that would appear at the output of the first-stage amplifier with saturation ignored and the actual maximum value V_S due to saturation.

3. Apply your general formula to calculate percent clipping of a 1-volt peak amplitude sinusoidal signal for the potentiometer dial in three positions: fully counter-clockwise (no distortion), midscale (moderate distortion), and fully clockwise (maximum distortion). Assume the op amp outputs saturate at ±13.5 volts.

4. Apply a 1-volt peak amplitude sinusoidal signal with 100-Hz frequency to the distortion box input and plot its output for the potentiometer dial in the same three positions as above. State the maximum and minimum values of the distortion box output.

NI Multisim Measurements

1. Enter your design for the distortion box into NI Multisim. Use the virtual five-terminal op amp model for both stages. Connect the power supply terminals ±13.5 volts. Apply the AC (sinusoidal) voltage source as the signal input; configure the source for 1-volt peak amplitude and 100 Hz frequency.

2. Observe the distortion box input and output signals with the oscilloscope and vary the potentiometer value over its full range of 0 to 100%, and then use the oscilloscope cursors to measure the percent clipping for the potentiometer settings 0%, 50%, and 100%.

3. Print screenshots of the oscilloscope display to show the distortion box input and output signals for the three potentiometer settings in the previous step.

4. Measure the maximum and minimum values of the distortion box output.

Additional helpful tips:

- Use these specific components: `OPAMP_5T_VIRTUAL`, `AC_VOLTAGE`, and virtual linear potentiometer.

- Remember that the five-terminal op amp symbol when initially placed has its positive power supply connection on top; applying a vertical flip to the symbol also flips the positive power supply connection to the bottom.

- Place the "CMOS Supply (VDD)" as the op amp positive power supply connection and "CMOS Supply (VSS)" as the negative power supply connection.

- Use the basic two-channel oscilloscope with the distortion box signal input on Channel A and its output on Channel B. Run an interactive simulation until several cycles of oscillation appear on the oscilloscope display.

- Take cursor measurements to determine the time duration of clipping for a half-cycle of the sinusoidal signal. Divide this time by the duration of the entire half-cycle and multiply by 100%.

NI Multisim video tutorials:

- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`
- AC (sinusoidal) voltage source:
  `http://youtu.be/CXbuz7MVLSs`
- VDD and VSS power supply voltages:
  `http://youtu.be/XrPVLgYsDdY`

NI myDAQ Measurements

1. Build your distortion box circuit and use the following myDAQ signal connections:

   - AI0 (Analog Input 0) to display the input signal; connect AI0+ to the source voltage and AI0- to ground,
   - AI1 (Analog Input 1) to display the output signal; connect AI1+ to the output voltage and AI1- to ground,

   Create the 100-Hz sinusoidal waveform with the NI ELVISmx Function Generator.

2. Observe the distortion box input and output signals with the NI ELVISmx Oscilloscope and vary the potentiometer over its full range. Use the oscilloscope cursors to measure the percent clipping with the potentiometer dial in three positions: fully counter-clockwise (no distortion), midscale (moderate distortion), and fully clockwise (maximum distortion).

3. Print screenshots of the oscilloscope display to show the distortion box input and output signals for the three potentiometer settings in the previous step.

4. Measure the maximum and minimum values of the distortion box output.

Additional helpful tips:

- Use the Texas Instruments TL072 op amp described in Appendix C on page 169. Follow the pinout diagram of Figure C.1 on page 170 for the two available op amps in the package. You may also use a pair of equivalent dual-supply op amp.
- Power the op amp with myDAQ +15V to V_{CC+} and -15V to V_{CC-}. Use AGND for the circuit ground.
- The TL072 op amp and similar devices saturate at approximately 1.5 volts under the supply voltage, consequently the actual saturation levels are about ±13.5 volts. If you use a different type of op amp with *rail-to-rail outputs* then you should expect to see the output saturation levels match the measured values of the myDAQ 15-volt dual power supply.
- Should you need to observe the output of the first-stage amplifier with the oscilloscope for troubleshooting purposes, you *must* consider the ±10 volt input range limitation of the myDAQ analog input channels. This range limit effectively makes you blind to any signal activity between 10 volts and the myDAQ power supply of 15 volts.
- Take cursor measurements to determine the time duration of clipping for a half-cycle of the sinusoidal signal. Divide this time by the duration of the entire half-cycle and multiply by 100%.

NI myDAQ video tutorials:

- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`
- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`

Chapter 5

RC and RL First-Order Circuits

5.1 Capacitors (5-2)

The voltage $v(t)$ across a 10-μF capacitor is given by the waveform shown in Figure 5.1.

1. Determine the equation for the capacitor current $i(t)$ and plot it over the time 0 to 50 ms.

2. Calculate the values of capacitor current at times 0, 25, and 30 ms.

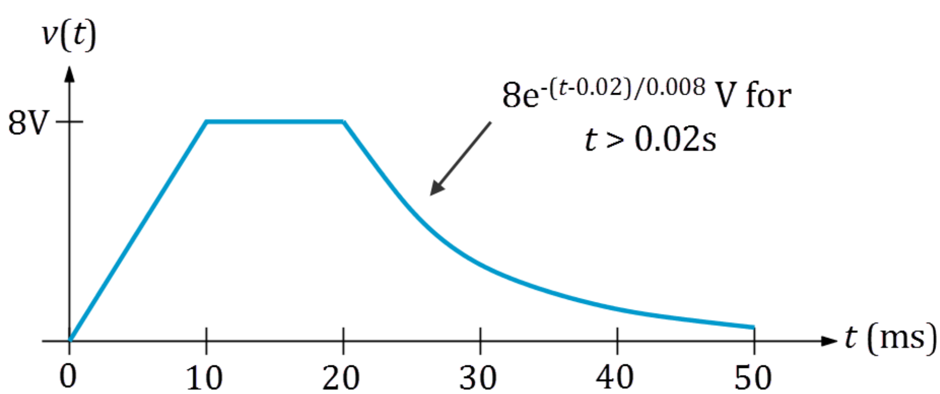


Figure 5.1: Voltage waveform for Problem 5.1

NI Multisim Measurements

Oscilloscopes display a time-varying voltage as a function of time. The current through a component such as the capacitor in this problem can also be displayed on an oscilloscope with a small-valued "shunt resistor" placed in series with the component. The shunt resistor produces a proportional voltage according to Ohm's Law $v = Ri$ where the resistance R serves as the proportionality constant. A trade-off exists here: a small shunt resistance minimizes disruption to the surrounding circuit, but a large shunt resistance maximizes the available signal to the oscilloscope.

1. Enter a circuit that contains the following components:
   - 10-μF capacitor and 10-Ω shunt resistor connected in series
   - ABM (Analog Behavioral Modeling) voltage source (`ABM_VOLTAGE`) connected across the capacitor-resistor combination; set up the voltage value to match the waveform of Figure 5.1 on the previous page.
   - Two-channel oscilloscope showing the capacitor voltage on Channel A and the shunt resistor voltage on Channel B.

   Run interactive simulation, adjusting the oscilloscope settings to display the capacitor voltage and current with each waveform filling a reasonable amount of the available display.

2. Use the oscilloscope display cursors to measure the capacitor current at times 0, 25, and 30 ms. Divide the cursor measurement by the shunt resistor value.

Additional Multisim tips for this problem:

- Build the ABM voltage source "Voltage Value" string by combining the following functions:
  - `u(TIME)` – Step function $u(t)$
  - `uramp(TIME)` – Ramp function $r(t)$
  - `exp(TIME)` – Exponential function e^t

  For example, the string `800*(uramp(TIME) - uramp(TIME-0.01))` implements the first 20 milliseconds of the capacitor voltage waveform.

- Place a DC voltage source someplace on the schematic sheet to enable interactive simulation. Do *not* connect the DC source to the capacitor circuit itself.

NI Multisim video tutorials:

- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`
- ABM (Analog Behavioral Model) voltage source:
  `http://youtu.be/8pPynWRwhO4`
- Distinguish oscilloscope traces by color:
  `http://youtu.be/bICbjggcTiQ`

NI myDAQ Measurements

The myDAQ analog outputs AO0 and AO1 cannot source more than 2 mA and still maintain the expected voltage output. Use an op amp voltage follower (see Ulaby Section 4-7) to create a "strengthened" copy of the myDAQ analog output.

1. Construct a circuit similar to the Multisim circuit you created earlier, i.e., place a 10-ohm shunt resistor in series with the capacitor, and connect the capacitor-resistor combination between the voltage follower output and ground.

   IMPORTANT: Electrolytic capacitors are *polarized*; ensure that the positive-labeled terminal connects to the op amp output. Alternatively, if the capacitor is marked with a negative-labeled terminal, connect this terminal to the shunt resistor.

   Establish the following myDAQ connections:

   - AO0 (Analog Output 0) to the voltage follower input,
   - AI0 (Analog Input 0) to display the capacitor voltage; connect AI0+ to the positive capacitor terminal and AI0- to the negative capacitor terminal,
   - AI1 (Analog Input 1) to display the shunt resistor voltage; connect AI1- to ground.

Create the capacitor voltage waveform of Figure 5.1 on page 57 with the NI ELVISmx Arbitrary Waveform Generator; use 50 kS/s as the sampling rate.

Adjust the NI ELVISmx Oscilloscope settings to display the capacitor voltage and current with each waveform filling a reasonable amount of the available display. Use a combination of edge triggering on Channel 0 and the "Horizontal Position" control to place the upper left corner of the voltage waveform at time 10 ms.

2. Use the oscilloscope display cursors to measure the capacitor current at times 0, 25, and 30 ms. Divide the cursor measurement by the shunt resistor value. Improve your measurement accuracy by using the measured shunt resistance obtained by the myDAQ DMM ohmmeter.

Additional tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C on page 169 for the voltage follower. Follow the pinout diagram of Figure C.1 on page 170 for either of the two available op amps in the package. You may also use an equivalent dual-supply op amp.
- Power the op amp with myDAQ +15V to V_{CC+} and -15V to V_{CC-}. Use AGND for the circuit ground.
- Use Cursor 1 to take measurements on the shunt resistor voltage. Leave Cursor 2 at time zero to make the "dT" indicator show time directly.
- The shunt resistor voltage signal is relatively low amplitude. Expect the waveform to jump vertically somewhat. Simply click the oscilloscope "Stop" button to freeze the display.
- Create three waveform segments with the ARB "Waveform Editor," two of length 10 ms and the third of length 30 ms. Choose the "Expression" option for each segment and enter expressions that include the time variable `t` as needed. Set the "X Range From" value of the 30 ms segment to 0.03 seconds and set "Cycles" to 1. You can then enter the exponential function `exp()` as shown in Figure 5.1 on page 57. For convenience, create the complete waveform with a unit amplitude and then adjust the "Gain" value of the Arbitrary Waveform Generator to 8.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`
- Increase current drive of analog output (AO) channels with an op amp voltage follower:
  `http://decibel.ni.com/content/docs/DOC-12665`

5.2 Inductors (5-3)

The voltage $v(t)$ across a 33-mH inductor is given by the sinusoidal pulse waveform shown in Figure 5.2.

1. Determine the equation for the inductor current $i(t)$ and plot it over the time 0 to 0.4 ms. Assume zero initial inductor current.
2. Determine the time at which the inductor current reaches its maximum value.
3. Calculate the total range of inductor current, i.e., the maximum value minus the minimum value.

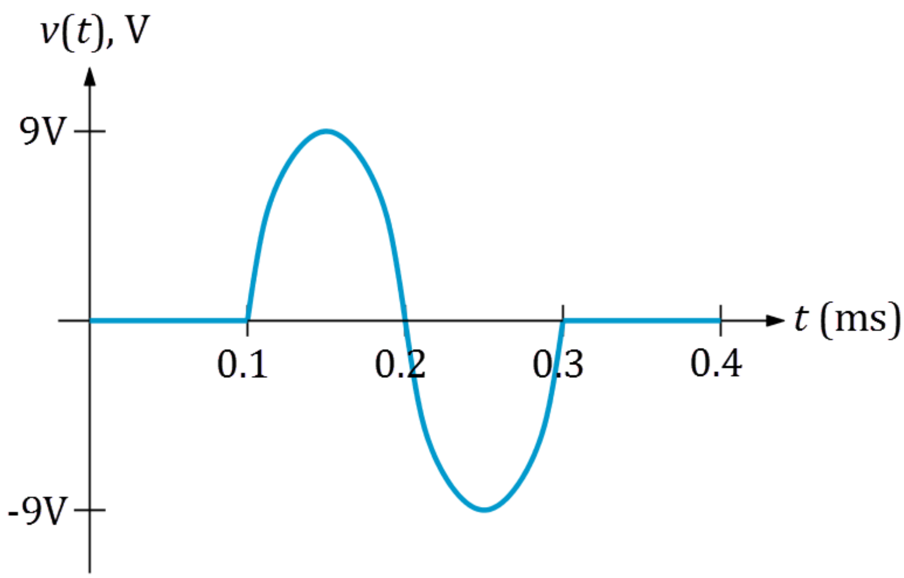


Figure 5.2: Voltage waveform for Problem 5.2

NI Multisim Measurements

Oscilloscopes display a time-varying voltage as a function of time. The current through a component such as the inductor in this problem can also be displayed on an oscilloscope with a small-valued “shunt resistor” placed in series with the component. The shunt resistor produces a proportional voltage according to Ohm's Law $v = Ri$ where the resistance R serves as the proportionality constant. A trade-off exists here: a small shunt resistance minimizes disruption to the surrounding circuit, but a large shunt resistance maximizes the available signal to the oscilloscope.

1. Enter a circuit that contains the following components:

   - 33-mH inductor and 10-Ω shunt resistor connected in series
   - Two AC voltage sources (`AC_VOLTAGE`) connected in series and also connected across the inductor-resistor combination. Flip the orientation of one of the sources so that the series combination forms the *difference* of the two voltages. Set one voltage for a delay of 0.1 ms and the other for a delay of 0.3 ms; in this way only a single cycle of the sinusoid appears across the pair of sources as in the waveform of Figure 5.2 on the preceding page.
   - Two-channel oscilloscope showing the inductor voltage on Channel A and the shunt resistor voltage on Channel B.

   Run interactive simulation, adjusting the oscilloscope settings to display the inductor voltage and current with each waveform filling a reasonable amount of the available display.

2. Use the oscilloscope display cursors to measure the time at which the inductor current reaches its maximum value.

3. Use the oscilloscope cursors to determine the total range of inductor current, i.e., the maximum value minus the minimum value.

NI Multisim video tutorials:

- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`
- AC (sinusoidal) voltage source:
  `http://youtu.be/CXbuz7MVLSs`
- Distinguish oscilloscope traces by color:
  `http://youtu.be/bICbjggcTiQ`

NI myDAQ Measurements

The myDAQ analog outputs AO0 and AO1 cannot source more than 2 mA and still maintain the expected voltage output. Use an op amp voltage follower (see Ulaby Section 4-7) to create a "strengthened" copy of the myDAQ analog output.

1. Construct a circuit similar to the Multisim circuit you created earlier, i.e., place a 10-ohm shunt resistor in series with the inductor, and connect the inductor-resistor combination between the voltage follower output and ground.

   Establish the following myDAQ connections:

   - AO0 (Analog Output 0) to the voltage follower input,
   - AI0 (Analog Input 0) to display the inductor voltage; connect AI0+ to inductor terminal connected to the op amp output and connect AI0- to the other inductor terminal,
   - AI1 (Analog Input 1) to display the shunt resistor voltage; connect AI1- to ground.

   Create the inductor voltage waveform of Figure 5.2 on page 62 with the NI ELVISmx Arbitrary Waveform Generator; use 200 kS/s as the sampling rate.

   Adjust the NI ELVISmx Oscilloscope settings to display the inductor voltage and current with each waveform filling a reasonable amount of the available display. Use a combination of edge triggering on Channel 0 and the "Horizontal Position" control to center the inductor voltage pulse.

2. Use the oscilloscope display cursors to measure the time at which the inductor current reaches its maximum value; use the same time reference as Figure 5.2 on page 62.

3. Use the cursors to determine the total range of inductor current, i.e., the maximum value minus the minimum value. Divide the cursor measurement by the shunt resistor value. Improve your measurement accuracy by using the measured shunt resistance obtained from the myDAQ DMM ohmmeter.

Additional tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C on page 169 for the voltage follower. Follow the pinout diagram of Figure C.1 on page 170 for either of the two available op amps in the package. You may also use an equivalent dual-supply op amp.
- Power the op amp with myDAQ +15V to V_{CC+} and -15V to V_{CC-}. Use AGND for the circuit ground.
- Create three waveform segments with the ARB "Waveform Editor," two of length 0.1 ms on either end and the middle segment of length 0.3 ms. Choose the "Library" option for the middle segment and experiment with parameters until you obtain a single cycle with unit amplitude. For convenience, create the complete waveform with a unit amplitude and then set the "Gain" value of the Arbitrary Waveform Generator to 9.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`
- Increase current drive of analog output (AO) channels with an op amp voltage follower:
  `http://decibel.ni.com/content/docs/DOC-12665`

5.3 Response of the RC Circuit (5-4)

Figure 5.3 shows a resistor-capacitor circuit with a pair of switches and Figure 5.4 on the next page shows the switch opening-closing behavior as a function of time. The initial capacitor value is -9 volts.

1. Determine the equation that describes $v(t)$ over the time range 0 to 50 ms.
2. Plot $v(t)$ over the time range 0 to 50 ms.
3. Determine the values of $v(t)$ at the times 5, 15, 25, 35, and 45 ms.

Use these component values:

- $R_1 = 10$ kΩ, $R_2 = 3.3$ kΩ, and $R_3 = 2.2$ kΩ
- $C = 1.0\ \mu$F
- $V_1 = 9$ V and $V_2 = -15$ V

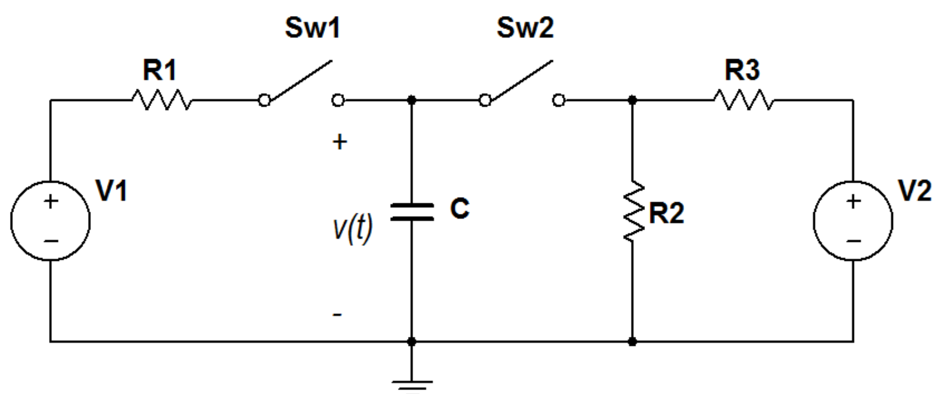


Figure 5.3: Circuit for Problem 5.3

NI Multisim Measurements

1. Enter the circuit of Figure 5.3 using the following components:
   - VOLTAGE_CONTROLLED_SWITCH

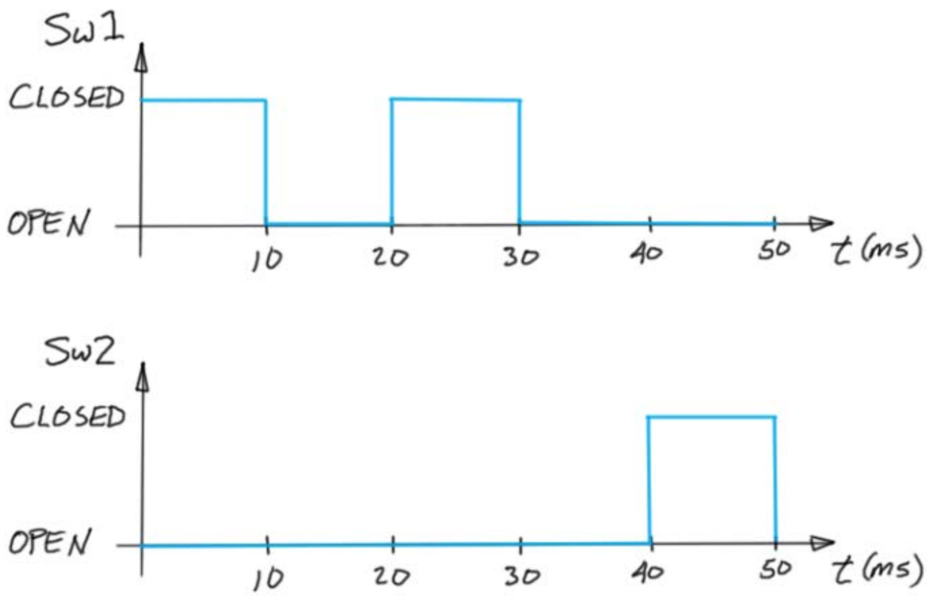


Figure 5.4: Switch positions for Problem 5.3

- `ABM_VOLTAGE` (Analog Behavioral Modeling) voltage source; use step functions (`u(TIME)`) to create the switch control waveforms of Figure 5.4.
- Capacitor with initial value of -9 volts.

2. Name the net that connects the two switches to the capacitor. Set up a Simulate → Analyses → Transient analysis with the end time set to 0.05 seconds and with "Initial Conditions" set to "User-defined." Select the "Output" tab and add the capacitor voltage to the list of analysis variables. Run the simulator to plot $v(t)$.

3. Use the oscilloscope cursor to measure the values of $v(t)$ at the times 5, 15, 25, 35, and 45 ms.

NI Multisim video tutorials:

- Find the maximum value of trace in Grapher View:
  `http://youtu.be/MzYK60mfh2Y`
- Voltage-controlled switch:
  `http://youtu.be/BaEBjhD4TOw`
- ABM (Analog Behavioral Model) voltage source:
  `http://youtu.be/8pPynWRwhO4`

NI myDAQ Measurements

1. Construct the circuit of Figure 5.3 on page 66 using the following components and NI ELVISmx instruments:

   - Two normally-open Switches 1 and 4 contained in the Intersil DG413 quad analog switch described in Appendix D on page 173. Refer to the pinout diagram of Figure D.1 on page 174 and connect power according to the photograph of Figure D.2 on page 175.
   - 9.0 volt source created with the LM317 variable voltage circuit of Figure B.2 on page 164.
   - 1.0 μF electrolytic capacitor. Connect the negative terminal of the capacitor to ground.
   - AO0 (Analog Output 0) to the switch control input of Switch 1.
   - AO1 (Analog Output 1) to the switch control input of Switch 4.
   - AI0 (Analog Input 0) to display the switch control voltage for Switch 1; connect AI0+ to the switch control input and connect AI0- to ground.
   - AI1 (Analog Input 1) to display the capacitor voltage $v(t)$; connect AI1+ to the positive side of the electrolytic capacitor and connect AI1- to ground.
   - Arbitrary Waveform Generator to create the switch control waveforms of Figure 5.4 on the previous page.
   - Oscilloscope to view the Switch 1 control waveform and the capacitor voltage $v(t)$. Adjust the Oscilloscope settings to display the voltage $v(t)$ so that the waveform fills a reasonable amount of the available display. Use a combination of edge triggering and the "Horizontal Position" control. You may find it helpful

to set the "Acquisition Mode" to "Run Once" and then click the "Run" button repeatedly until you capture a good trace.

2. Use the oscilloscope cursor to measure the values of $v(t)$ at the times 5, 5, 25, 35, and 45 ms.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`
- Digital Writer (DigOut):
  `http://decibel.ni.com/content/docs/DOC-12945`

Further Exploration with NI myDAQ

The circuit of Figure 5.3 on page 66 permits the capacitor to be charged to a desired voltage by closing one of the switches that connects the capacitor to a source. After charging, opening both switches should in principle allow the capacitor to maintain its "charge" (or stored energy) indefinitely. However, the physical capacitor contains a nonideal dielectric material between its plates that allows a slow trickle of current that eventually depletes the stored energy. The nonideal dielectric can be modeled as a resistor in parallel with the capacitor plates.

Devise a method to estimate the value of the equivalent resistance that connects the capacitor plates. Consider the half-life measurement technique of Figure E.1 on page 178 to measure the time constant. Connect the switch control inputs to DIO0 and DIO1 and use the NI ELVISmx Digital Writer to manually operate the switches. Use NI ELVISmx Oscilloscope to display the capacitor voltage, taking special care to enable only *one* active oscilloscope channel. Enabling both oscilloscope channels greatly reduces the effective input resistance of the myDAQ analog inputs due to the rapid switching between these channels to a common analog-to-digital converter.

5.4 Response of the RL Circuit (5-5)

The circuit of Figure 5.5 on the facing page demonstrates how an inductor can produce a high-voltage pulse across a load resistance R_{load} that is considerably higher than the circuit's power supply V_{batt}, a 1.5-volt "AA" battery. High-voltage pulses drive photo flash bulbs, strobe lights, and cardiac defibrillators, as examples.

R_{s} models the finite resistance of an electronic analog switch and R_{w} models the finite winding resistance of the inductor.

1. Determine the load voltage v after the switch has been closed for a long time.

2. Determine the equation that describes $v(t)$ after the switch opens at time $t = 0$.

3. Determine the magnitude of the peak value of $v(t)$. How many times larger is this value compared to the battery voltage V_{batt}?

4. State the value of the circuit time constant τ with the switch open. Plot $v(t)$ over the time range $-\tau \leq t \leq 5\tau$.

Use these component values:

- $R_{\text{s}} = 16\ \Omega$, $R_{\text{w}} = 90\ \Omega$, and $R_{\text{load}} = 680\ \Omega$
- $L = 33$ mH
- $V_{\text{batt}} = 1.5$ V

NI Multisim Measurements

1. Enter the circuit of Figure 5.5 on the next page. Use the interactive switch `SPST` (single pole, single throw) and a measurement probe to determine v with the switch closed for a long time.

2. Connect the oscilloscope to monitor the voltage $v(t)$. Run interactive simulation, adjusting the oscilloscope settings to make the waveform fill a reasonable amount of the available display in both the vertical and horizontal directions. Use edge triggering and the "Normal" triggering mode to capture the transient when the switch opens. You may wish to decrease the time step size of interactive simulation to achieve higher resolution; see the tutorial video linked at the end of this section.

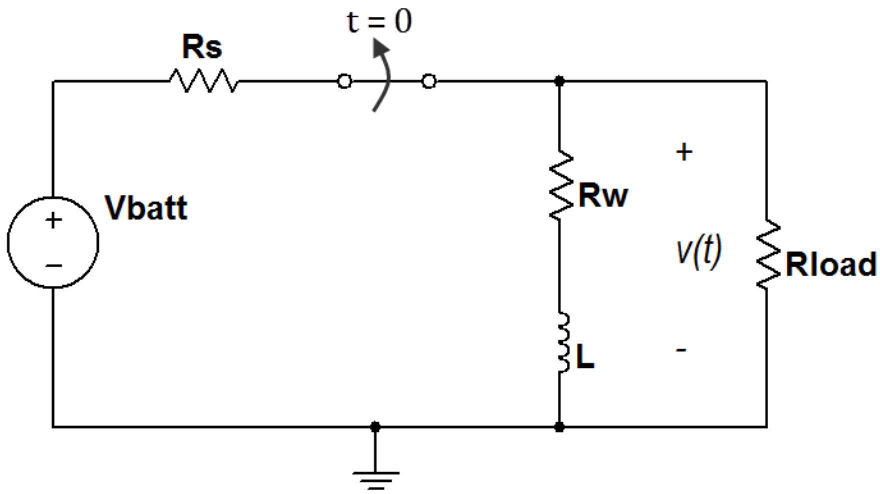


Figure 5.5: Circuit for Problem 5.4

3. Use the oscilloscope cursor to measure the magnitude of the peak value of $v(t)$.

4. Measure the time constant using the half-life technique described in Figure E.1 on page 177.

NI Multisim video tutorials:

- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Stabilize the oscilloscope display with edge triggering:
  `http://youtu.be/d69zYYSEG7E`
- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`
- Find components by name:
  `http://youtu.be/5wlFweh4n-c`

NI myDAQ Measurements

1. Construct the circuit of Figure 5.5 on the previous page using the normally-closed Switch 2 contained in the Intersil DG413 quad analog switch described in Appendix D on page 173. Refer to the pinout diagram of Figure D.1 on page 174 and connect power according to the photograph of Figure D.2 on page 175. Do not place actual resistors for R_s and R_w because these simply model the finite resistance of the analog switch and inductor winding resistance. Create the 1.5 volt source with the LM317 variable voltage circuit of Figure B.2 on page 164 and connect it to the DG413 "Source (Input)" terminal; connect the "Drain (Output)" terminal to the inductor.

   Establish the following myDAQ signal connections to the DG413:

   - DIO0 (Digital Input/Output 0) to the "Logic Control" (switch control) input for Switch 2,
   - AI0 (Analog Input 0) to display the switch control voltage; connect AI0+ to the switch control input and connect AI0- to ground,
   - AI1 (Analog Input 1) to display the voltage $v(t)$; connect AI1- to ground.

   Use the NI ELVISmx Digital Writer ("DigOut" on the NI ELVISmx Instrument Launcher) to operate DIO0 as an output. Toggle the button for Line 0 to operate the analog switch. Use the NI ELVISmx DMM voltmeter to measure v when the switch is closed.

2. Change the switch control voltage to AO0, Analog Output 0. Create the switch control voltage with the NI ELVISmx Function Generator. Choose the squarewave shape and adjust the amplitude and offset to make the squarewave swing between 0 and 5 volts. Observe this waveform on the oscilloscope to confirm your correct setup before you connect it to the analog switch.

   Adjust the NI ELVISmx Oscilloscope settings to display the voltage $v(t)$ so that the waveform fills a reasonable amount of the available display. Use a combination of edge triggering and the "Horizontal Position" control. You may find it helpful to set the "Acquisition Mode" to "Run Once" and then click the "Run" button repeatedly until you capture a good trace. Alternatively, try increasing the squarewave frequency to keep the oscilloscope from timing out; a squarewave frequency of about $(5\tau)^{-1}$ Hz allows the voltage transient to reach its final value before the switch closes again.

3. Use the oscilloscope display cursors to measure the magnitude of the peak value of $v(t)$.

4. Measure the time constant using the half-life technique described in Figure E.1 on page 177.

NI myDAQ video tutorials:

- Digital Writer (DigOut):
  `http://decibel.ni.com/content/docs/DOC-12945`
- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

Further Exploration with NI myDAQ

The switch model resistance R_{s} and the inductor winding resistance R_{w} values used in the circuit of Figure 5.5 on page 71 were based on measurements taken with actual equipment, but may not necessarily match the values for your devices.

Measure the on-resistance of your analog switch and also measure the resistance of your inductor. Recalculate your theoretical time constant value using your measurements. Report the degree to which you see closer agreement between your theoretical and measured values for the time constant τ.

Chapter 6

RLC Circuits

6.1 Initial and Final Conditions (6-1)

The SPST switch in the circuit of Figure 6.1 on the following page opens at $t = 0$ after it had been closed for a long time. Draw the circuit configurations that appropriately represent the state of the circuit at $t = 0^-$, $t = 0$, and $t = \infty$ and use them to determine:

1. $v_C(0)$, $i_C(0)$, and $v_C(\infty)$, and
2. $i_L(0)$, $v_L(0)$, and $i_L(\infty)$.

Use these component values:

- $R_1 = 680\ \Omega$, $R_2 = 100\ \Omega$, and $R_3 = 100\ \Omega$
- $R_{sw} = 10\ \Omega$ and $R_w = 10\ \Omega$
- $L = 3.3$ mH
- $C = 0.1\ \mu$F
- $V_s = 4.7$ V

NI Multisim Measurements

Enter the circuit of Figure 6.1 on the next page using the `SPST` switch for interactive simulation. Select Simulate → Interactive Simulation Settings and set "Maximum time step (TMAX)" to `1e-006` to obtain the needed resolution for this circuit.

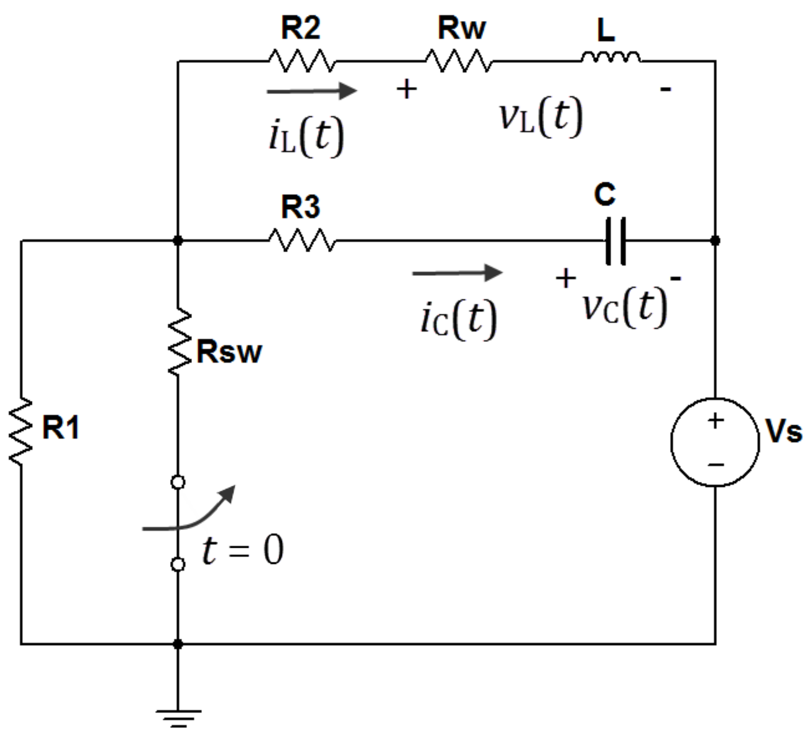


Figure 6.1: Circuit for Problem 6.1

1. Connect the two-channel oscilloscope to plot $v_C(t)$ and the voltage across resistor R_3; divide by the value of R_3 to obtain the capacitor current $i_C(t)$. Start interactive simulation and adjust the oscilloscope settings to clearly show the two waveforms when the switch opens; operate the switch with the space bar. Choose two different colors for the oscilloscope traces to make them easy to identify. Take cursor measurements to determine $v_C(0)$ and $i_C(0)$ just before the switch opens. Take another cursor measurement to determine $v_C(\infty)$. Remember that "$t = \infty$" means the circuit has settled to it new steady-state value.

2. Reconnect the two-channel oscilloscope to plot $v_L(t)$ and the voltage

across resistor R_2; divide by the value of R_2 to obtain the inductor current $i_\mathrm{L}(t)$. Repeat the techniques from the previous step to display the two waveforms. Take cursor measurements to determine $i_\mathrm{L}(0)$ and $v_\mathrm{L}(0)$ just before the switch opens. Take another cursor measurement to determine $i_\mathrm{L}(\infty)$.

NI Multisim video tutorials:

- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`
- Distinguish oscilloscope traces by color:
  `http://youtu.be/bICbjggcTiQ`

NI myDAQ Measurements

1. Construct the circuit of Figure 6.1 on the facing page using the following components and NI ELVISmx instruments (do *not* place resistors R_sw and R_w because they simply model the finite resistance of the analog switch and the inductor):

   - Normally-closed Switch 3 contained in the Intersil DG413 quad analog switch described in Appendix D on page 173. Refer to the pinout diagram of Figure D.1 on page 174 and connect power according to the photograph of Figure D.2 on page 175.
   - 4.7 volt source created with 5V and DGND. The loading effect of this circuit reduces the unloaded 5-volt source to about 4.7 volts.
   - AO0 (Analog Output 0) to the switch control input of Switch 2.
   - AI0 (Analog Input 0) to display the voltage across the current-sensing resistor R_2 for inductor current or R_3 for capacitor current.
   - AI1 (Analog Input 1) to display the capacitor voltage $v_\mathrm{C}(t)$ or inductor voltage $v_\mathrm{L}(t)$.
   - Function Generator to create the switch control waveform: choose "Squarewave," set the peak-to-peak amplitude to 5 V, the offset to 2.5 V, and the frequency to 1 kHz.

- Oscilloscope to view the current and voltage waveforms. Adjust the Oscilloscope settings to display the voltage waveforms filling a reasonable amount of the available display. Use a combination of edge triggering and the “Horizontal Position” control. You may find it helpful to set the “Acquisition Mode” to “Run Once” and then click the “Run” button repeatedly until you capture a good trace.

2. Establish oscilloscope connections to display $v_{\mathrm{C}}(t)$ and $i_{\mathrm{C}}(t)$. Take cursor measurements to determine $v_{\mathrm{C}}(0)$ and $i_{\mathrm{C}}(0)$ just before the switch opens, and also measure $v_{\mathrm{C}}(\infty)$.

3. Modify the connections to display $v_{\mathrm{L}}(t)$ and $i_{\mathrm{L}}(t)$. Take cursor measurements to determine $i_{\mathrm{L}}(0)$ and $v_{\mathrm{L}}(0)$ just before the switch opens, and also measure $i_{\mathrm{L}}(\infty)$.

NI myDAQ video tutorials:

- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`

- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

6.2 Natural Response of the Series RLC Circuit (6-3)

The SPST switch in the circuit of Figure 6.2 on the following page opens at $t = 0$ after it had been closed for a long time.

1. Determine $v_{\mathrm{C}}(t)$ for $t \geq 0$.
2. Plot $v_{\mathrm{C}}(t)$ over the time range $0 \leq t \leq 1$ ms with a plotting tool such as MathScript or MATLAB.
3. Determine the following numerical values; use either the equation $v_{\mathrm{C}}(t)$ or take cursor measurements from the plot you created in the previous step:
   - Initial voltage $v_{\mathrm{C}}(0)$,
   - Minimum value of v_{C},
   - Maximum value of v_{C},
   - Damped oscillation frequency $f_{\mathrm{d}} = \omega_{\mathrm{d}}/2\pi$ in Hz, and
   - Damping coefficient α.

Use these component values:

- $R_1 = 220\ \Omega$ and $R_2 = 330\ \Omega$
- $L = 33$ mH and $C = 0.01\ \mu$F
- $V_{\mathrm{src}} = 3.0$ V

NI LabVIEW video tutorials:

- Plot two functions of time: `http://youtu.be/XQlAai1-YVc`
- Take cursor measurements on a plot: `http://youtu.be/bgK1p5O6OXc`

NI Multisim Measurements

1. Enter the circuit of Figure 6.2 on the next page using the `SPST` switch for interactive simulation. Select Simulate → Interactive Simulation Settings and set "Maximum time step (TMAX)" to `1e-007` to obtain the needed resolution for this circuit.

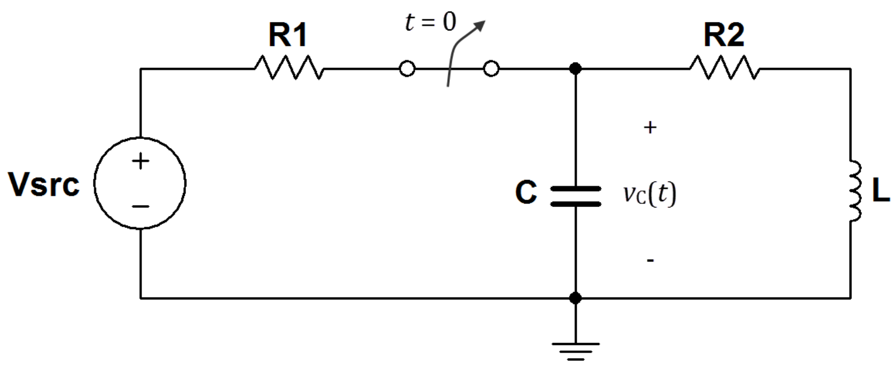


Figure 6.2: Circuit for Problem 6.2

2. Connect the two-channel oscilloscope to plot $v_{\mathrm{C}}(t)$ over the time range $0 \leq t \leq 1$ ms.

3. Determine the following numerical values with cursor measurements:
   - Initial voltage $v_{\mathrm{C}}(0)$ just before the switch opens,
   - Minimum value of v_{C},
   - Maximum value of v_{C},
   - Damped oscillation frequency f_{d} in Hz, and
   - Damping coefficient α.

Additional Multisim tips:

- Measure the damped oscillation frequency by using the cursors to measure the time between an integer number of oscillation cycles; zero crossings are the easiest to identify. Determine the measured oscillation period T and then take the reciprocal of this value for the oscillation frequency.

- Measure the damping coefficient with the following procedure:
  1. Place a cursor at the first peak value after the transient begins; choose either a positive peak or a negative peak,
  2. Place a second cursor at a peak value several cycles after the first peak; choose the same type of peak (positive or negative) as you did in the previous step,

3. Record the voltage of the first peak as V_1,
4. Record the voltage of the second peak as V_2,
5. Measure the time difference between the two cursors and record its value as T_{12}, and
6. Calculate $\alpha = \ln(V_1/V_2)/T_{12}$.

NI Multisim video tutorials:

- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`
- Find the maximum value of trace in Grapher View:
  `http://youtu.be/MzYK60mfh2Y`

NI myDAQ Measurements

1. Construct the circuit of Figure 6.2 on the facing page using the following components and NI ELVISmx instruments (do *not* place resistors R_{sw} and R_{w} because they simply model the finite resistance of the analog switch and the inductor):

   - Normally-open Switch 1 contained in the Intersil DG413 quad analog switch described in Appendix D on page 173. Refer to the pinout diagram of Figure D.1 on page 174 and connect power according to the photograph of Figure D.2 on page 175.
   - 3.0 volt source created with the LM317 variable voltage circuit of Figure B.2 on page 164.
   - AO0 (Analog Output 0) to the switch control input of Switch 1.
   - AI0 (Analog Input 0) to display the switch control voltage for Switch 1; connect AI0+ to the switch control input and connect AI0- to ground.
   - AI1 (Analog Input 1) to display the capacitor voltage $v_{\text{C}}(t)$.
   - Function Generator to create the switch control waveform: choose "Squarewave," set the peak-to-peak amplitude to 5 V and the offset to 2.5 V.

- Oscilloscope to view the switch control and capacitor voltage waveforms.

2. Display the switch control waveform and $v_C(t)$ over the range $0 \leq t \leq 1$ ms. Adjust the oscilloscope settings to display the voltage waveforms filling a reasonable amount of the available display. Use a combination of edge triggering and the "Horizontal Position" control. You may find it helpful to set the "Acquisition Mode" to "Run Once" and then click the "Run" button repeatedly until you capture a good trace. Choose a function generator frequency that allows the natural response to occupy most of the display.

3. Determine the following numerical values with cursor measurements:

   - Initial voltage $v_C(0)$ just before the switch opens,
   - Minimum value of v_C,
   - Maximum value of v_C,
   - Damped oscillation frequency f_d in Hz, and
   - Damping coefficient α.

   NOTE: Do not expect close agreement with your earlier analytical and simulation results. Refer to the "Further Exploration" section below to learn why and the steps you can take to achieve closer agreement.

NI myDAQ video tutorials:

- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

Further Exploration with NI myDAQ

The finite wire resistance of the physical inductor significantly contributes to the total resistance of the series-connected loop. In fact, you should have observed an unusually large mismatch between the physical circuit measurements and the analytical as well as simulated results. Try reducing the 330-ohm resistor value to account for the inductor resistance and obtain closer agreement between the physical circuit and the mathematical models.

Measure and record the resistance of your inductor with the DMM ohmmeter. How does this value compare on a percentage basis with the 330-ohm resistor?

Next, connect a 10K potentiometer in parallel with the 330-ohm resistor. Measure the resistance of this combination in series with the inductor; remember to disconnect the other circuit elements. Adjust the potentiometer until the total measured resistance is 330 ohms. Reconnect your original series RLC circuit including the potentiometer.

Repeat your earlier cursor measurements for initial voltage, minimum and maximum values, damped oscillation frequency, and damping coefficient. Discuss the degree of improved match between the physical circuit and its mathematical model.

6.3 General Solution for Any Second-Order Circuit (6-6)

1. Develop a differential equation for $v_C(t)$ in the circuit of Figure 6.3. Solve it to determine $v_C(t)$ for $t \geq 0$. The component values are V_S = 8 volts, $R_S = 680\ \Omega$, $C = 1.0\ \mu$F, $L = 33$ mH, and $R_W = 90\ \Omega$.

2. Plot $v_C(t)$ from 0 to 5 ms using a tool such as MathScript or MATLAB. Include hardcopy of the script used to create the plot.

3. Determine the following values for $v_C(t)$:

   (a) Maximum value,

   (b) Final value, and

   (c) Damped oscillation frequency $f_d = \omega_d/2\pi$.

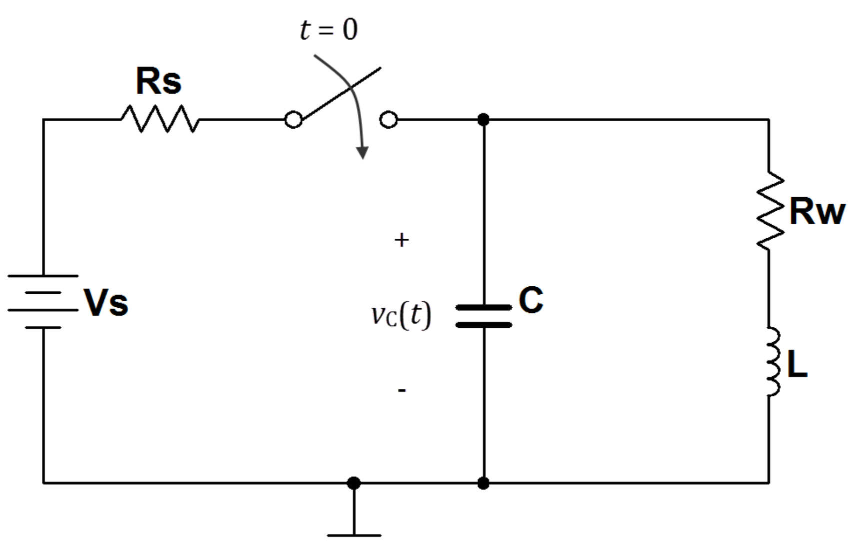


Figure 6.3: Circuit for Problem 6.3

NI LabVIEW video tutorials:

- Plot two functions of time:
  `http://youtu.be/XQlAai1-YVc`
- Take cursor measurements on a plot:
  `http://youtu.be/bgK1p5O6OXc`

NI Multisim Measurements

1. Enter the circuit of Figure 6.3 on the facing page using the same component values listed in the problem statement. Implement the switch with a `VOLTAGE_CONTROLLED_SWITCH` operated by a `PULSE_VOLTAGE` source configured to open the switch at time 1 ms; this delay makes the initial transition easier to see.
2. Plot $v_\mathrm{C}(t)$ from 0 to 5 ms with a Simulate → Analyses → Transient analysis.
3. Use the Grapher View cursors to measure the following values for $v_\mathrm{C}(t)$:
   (a) Maximum value,
   (b) Final value, and
   (c) Damped oscillation frequency $f_\mathrm{d} = \omega_\mathrm{d}/2\pi$.

NI Multisim video tutorials:

- Pulse voltage source:
  `http://youtu.be/RdgxVfr28C8`
- Voltage-controlled switch:
  `http://youtu.be/BaEBjhD4TOw`
- Plot time-domain circuit response with Transient Analysis:
  `http://youtu.be/waKnad_EXkc`

NI myDAQ Measurements

1. Construct the circuit of Figure 6.3 on the preceding page using the following components and NI ELVISmx instruments (do *not* place the resistor R_W as this simply models the finite wire resistance of the inductor):

- Normally-open Switch 1 contained in the Intersil DG413 quad analog switch described in Appendix D on page 173. Refer to the pinout diagram of Figure D.1 on page 174 and connect power according to the photograph of Figure D.2 on page 175.
- 8.0 volt source created with the LM317 variable voltage circuit of Figure B.2 on page 164.
- 1.0 μF electrolytic capacitor. **IMPORTANT**: Observe proper polarity of the capacitor by connecting the negative terminal of the capacitor to ground.
- AO0 (Analog Output 0) to the switch control input of Switch 1.
- AI0 (Analog Input 0) to display the switch control voltage for Switch 1; connect AI0+ to the switch control input and connect AI0- to ground.
- AI1 (Analog Input 1) to display the capacitor voltage $v_C(t)$.
- Function Generator to create the switch control waveform: choose "Squarewave," set the peak-to-peak amplitude to 5 V and the offset to 2.5 V.
- Oscilloscope to view the switch control and capacitor voltage waveforms.

2. Display $v_C(t)$ from 0 to 5 ms.
3. Use the oscilloscope cursor to measure the following values for $v_C(t)$:
   (a) Maximum value,
   (b) Final value, and
   (c) Damped oscillation frequency $f_d = \omega_d/2\pi$.

NI myDAQ video tutorials:

- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

6.4 Two-Capacitor Second-Order Circuit (6-6)

1. Develop a differential equation for each node voltage $v_1(t)$ and $v_2(t)$ in the circuit of Figure 6.4. Solve each equation to determine $v_1(t)$ and $v_2(t)$ for $t \geq 0$. The component values are V_S = 9 volts, $R = 10$ kΩ and $C = 0.1$ μF.

2. Plot $v_1(t)$ and $v_2(t)$ on the same graph from 0 to 10 ms using a tool such as MathScript or MATLAB. Include hardcopy of the script used to create the plot.

3. Determine the time at which each node voltage reaches 50% of its final value.

4. Discuss the difference in behavior of the two waveforms just after time $t = 0$ and propose an explanation for this difference.

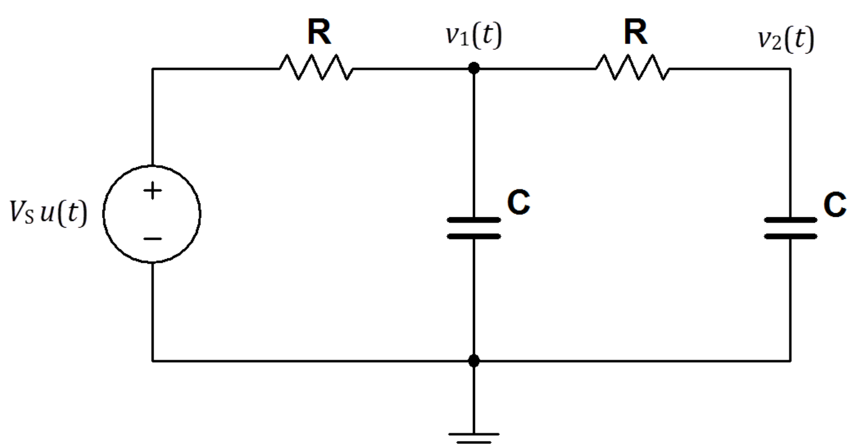


Figure 6.4: Circuit for Problem 6.4

NI LabVIEW video tutorials:

- Plot two functions of time:
  `http://youtu.be/XQlAai1-YVc`

- Take cursor measurements on a plot:
  `http://youtu.be/bgK1p5O6OXc`

NI Multisim Measurements

1. Enter the circuit of Figure 6.4 on the previous page using the same component values listed in the problem statement. Implement the voltage source as a step function using either a `PULSE_VOLTAGE` source or an `ABM_VOLTAGE` source configured to place the step change at 1 ms; this delay makes the initial transition easier to see.

2. Plot $v_1(t)$ and $v_2(t)$ from 0 to 10 ms with a Simulate → Analyses → Transient analysis.

3. Use the Grapher View cursors to determine the time at which each node voltage reaches 50% of its final value.

NI Multisim video tutorials:

- Pulse voltage source:
  `http://youtu.be/RdgxVfr28C8`
- ABM (Analog Behavioral Model) voltage source:
  `http://youtu.be/8pPynWRwhO4`
- Plot time-domain circuit response with Transient Analysis:
  `http://youtu.be/waKnad_EXkc`

NI myDAQ Measurements

1. Construct the circuit of Figure 6.4 on the preceding page using the same component values listed in the problem statement. Implement the voltage source using the NI ELVISmx Function Generator set to squarewave mode. Adjust the function generator settings to match the step-change voltage $V_S u(t)$; choose a frequency that is low enough to allow the NI ELVISmx Oscilloscope to display the 10 ms range without interruption.

2. Display both node voltages $v_1(t)$ and $v_2(t)$ at the same time from 0 to 10 ms.

3. Use the oscilloscope cursors to determine the time at which each node voltage reaches 50% of its final value.

NI myDAQ video tutorials:

- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

Chapter 7

AC Analysis

7.1 Impedance Transformations (7-5)

Determine the equivalent impedance **Z** looking into terminals A-B for the circuit of Figure 7.1 at the following frequencies: 100 Hz, 500 Hz, 1000 Hz, and 2000 Hz. Report your results in polar form.

Use these component values:

- $R_1 = 100\ \Omega$ and $R_2 = 90\ \Omega$
- $C = 1.0\ \mu\text{F}$ and $L = 33\ \text{mH}$

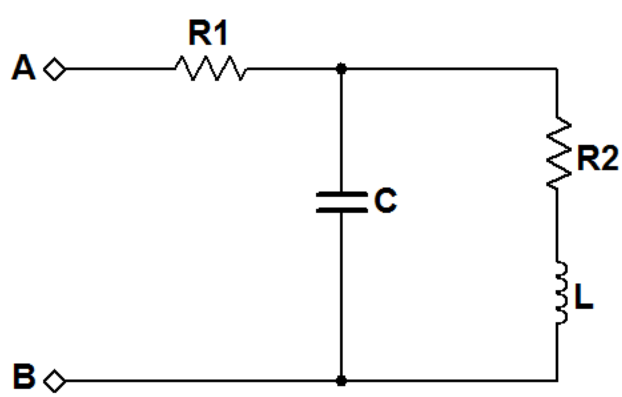


Figure 7.1: Circuit for Problem 7.1

NI Multisim Measurements

Figure 7.2 describes a laboratory technique to measure impedance of the circuit at terminals A-B. The sinusoidal voltage source excites the circuit with a known voltage $v(t)$ and causes the current $i(t)$. The impedance of the circuit at a given operating frequency is $\mathbf{Z} = \mathbf{V}/\mathbf{I}$ where $\mathbf{V}$ and $\mathbf{I}$ are the equivalent phasor representations of the time-domain voltage and current. The magnitude of $\mathbf{V}$ is the same as the amplitude of $v(t)$. Similarly, the magnitude of $\mathbf{I}$ is the same as the amplitude of $i(t)$. The sinusoidal voltage source amplitude can easily be measured with the oscilloscope, but what about the current?

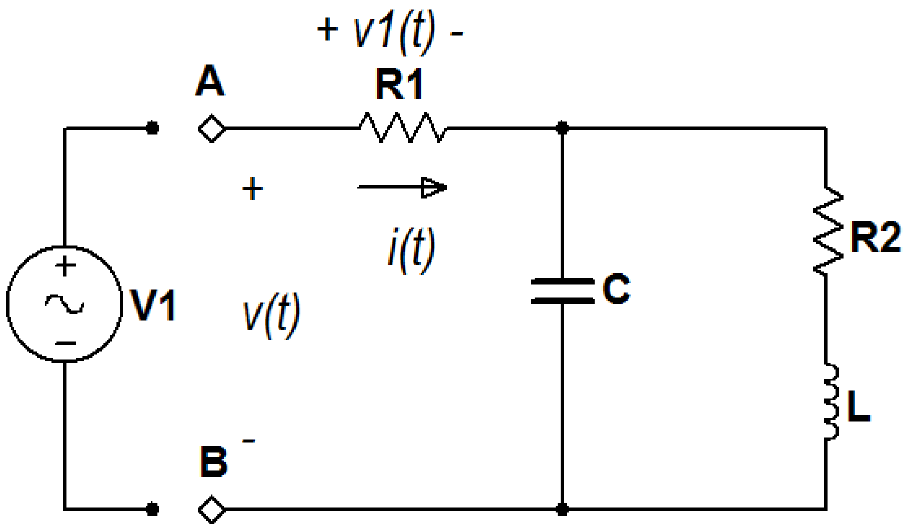


Figure 7.2: Circuit for Problem 7.1

Observe that the current $i(t)$ entering Terminal A also flows through resistor R_1. The voltage $v_1(t)$ that appears across this resistor is directly proportional to the current $i(t)$ due to Ohm's Law, consequently the current can be indirectly measured as $i(t) = v_1(t)/R_1$.

The procedure to measure impedance therefore requires the following steps:

1. Apply a sinusoidal voltage at the desired frequency to establish $v(t)$; typically a unit-amplitude voltage is convenient,

2. Measure the voltage magnitude (also called amplitude and peak value) $V_{\rm M}$ of $v(t)$,

3. Measure the voltage magnitude across the resistor R_1 and divide by the measured resistance R_1 to determine the current magnitude $I_{\rm M}$,

4. Calculate the impedance magnitude as $V_{\rm M}/I_{\rm M}$, and

5. Calculate the impedance phase by measuring the time difference t_D between the two sinusoids; convert this time difference to degrees by multiplying by the sinusoidal frequency in Hz and then multiply by 360 degrees. If the current waveform is delayed compared to the voltage waveform then the phase sign is positive; if advanced, then the phase sign is negative.

Use the above procedure to measure the impedance of the circuit at frequencies 100 Hz, 500 Hz, 1000 Hz, and 2000 Hz. Activate the circuit with the Simulate → Instruments → Function Generator and observe the two voltage signals with the Simulate → Instruments → Oscilloscope .

Additional helpful tips:

- Choose different colors for the traces to make the voltage and current traces easy to identify.

- Increase the time resolution of the simulation to improve the accuracy of your measurements, especially at the two higher frequencies: select Simulate → Interactive Simulation Settings and enter `1e-006` seconds for `Maximumum timestep (TMAX)`.

NI Multisim video tutorials:

- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`

- Distinguish oscilloscope traces by color:
  `http://youtu.be/bICbjggcTiQ`

- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`

- Function generator:
  `http://youtu.be/CeOl6EzD-_c`

NI myDAQ Measurements

The same impedance measurement technique described in the previous Multisim section works well for the physical circuit, too. Activate the circuit with the NI ELVISmx Function Generator on AO0, and place an op amp voltage follower between AO0 and the circuit; the voltage follower is necessary to boost the current drive of the analog output beyond its limit of 2 mA. Do *not* insert a physical resistor for R_2, because this resistor simply models the finite wire resistance of the physical inductor.

Use the oscilloscope to display the voltage $v(t)$ on AI1 and the resistor voltage $v_1(t)$ on AI0. Measure the circuit impedance at the frequencies 100 Hz, 500 Hz, 1000 Hz, and 2000 Hz.

Additional helpful tips:

- Refer to Appendix C on page 169 for details on the TI TL072 dual op amp device.

- The function generator sets it amplitude in terms of "peak-to-peak" voltage; this is twice the amplitude of the sinusoid. Adjust this voltage to yield a unit-amplitude sinusoid.

- The oscilloscope "Display Measurements" panel under the waveform display measures the peak-to-peak voltage of the waveforms. Set the timebase to display at least two cycles to ensure that the measurement is accurate. Display even more cycles to improve the accuracy and stability of the measurement.

- You may also use the cursors to measure the amplitude (peak value).

- Use the oscilloscope cursors to measure the time difference between zero crossings of the sinusoids. For this measurement reduce the timebase value to maximize your ability to accurately measure the time shift of the two sinusoids. Refer to the Multisim section to learn how to properly determine the sign of the impedance phase.

NI myDAQ video tutorials:

- Function Generator (FGEN):
`http://decibel.ni.com/content/docs/DOC-12940`
- Oscilloscope:
`http://decibel.ni.com/content/docs/DOC-12942`
- Increase current drive of analog output (AO) channels with an op amp voltage follower:
`http://decibel.ni.com/content/docs/DOC-12665`

Further Exploration with NI myDAQ

The NI ELVISmx Bode Analyzer instrument provides a quick and effective way to study circuit behavior over a range of frequencies, all within a single measurement step. The Bode Analyzer applies a sinusoidal signal (also called a tone pulse) as the circuit stimulus on AO0, measures the actual circuit stimulus on AI0, and measures the circuit response on AI1. The Bode Analyzer applies a series of tone pulses from low to high frequency and then plots the circuit response – "gain" and "phase" – as a function of frequency. This is an automated form of the method you used earlier in this problem.

The "Gain" display plots the ratio of the circuit response to the stimulus. With the myDAQ connections described in the previous section the "Gain" plot therefore displays the magnitude of the circuit impedance because the current $v(t)$ serves as the "stimulus" and the voltage proportional to $i(t)$ serves as the "response." Simply multiply the "Gain" plot by the value of R_1 to see the impedance in ohms.

Set the following values:

- "Start Frequency" = 10 Hz,
- "Stop Frequency" = 10 kHz,
- "Peak Amplitude" = 1, and
- "Mapping" = Linear.

Try running the Bode Analyzer with its default step size of 5, and then increase the step size until the plot is reasonably smooth and captures the interesting features of the impedance curve.

Use the cursors to read specific values at the frequencies specified earlier, and compare to your previous measurements.

The impedance plot as a function of frequency offers a "birds eye" view of the circuit behavior for a wide range of frequencies. For what frequency range does the circuit appear inductive? Over what range does it appear capacitive? What do you observe about the phase when the impedance reaches its maximum value?

7.2 Equivalent Circuits (7-6)

1. Determine the Thévenin equivalent circuit at the terminal pair A-B for the circuit in Figure 7.3 using the open-circuit/short-circuit method. Show the Thévenin impedance as a resistor in series with a single reactive element (capacitor or inductor) and determine the values of all components in the equivalent circuit. The sinusoidal source $\mathbf{V}_S$ is 3.0 volts amplitude and 500 Hz frequency.

2. Repeat with the source frequency increased to 1100 Hz.

3. Does the circuit seem to "change its personality" with different source frequencies? Explain your answer.

Use these component values:

- $R_1 = 90\ \Omega$ and $R_2 = 100\ \Omega$
- $C = 1.0\ \mu$F and $L = 33$ mH

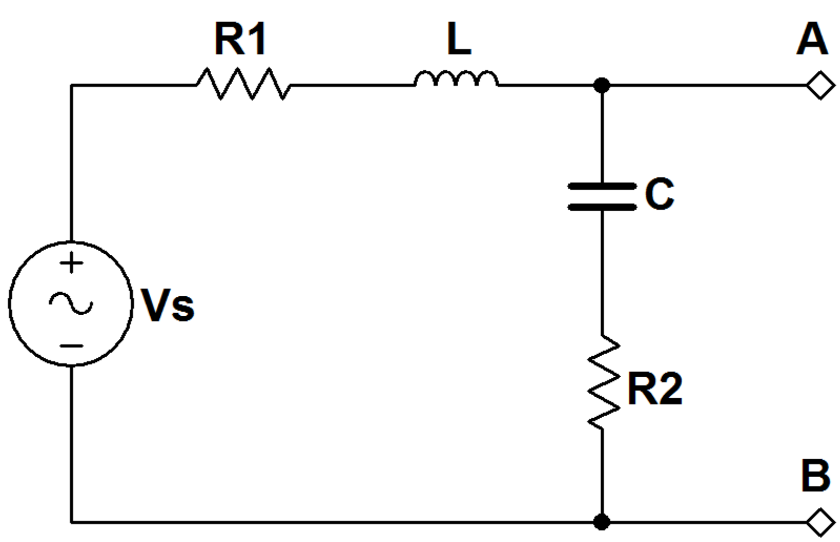


Figure 7.3: Circuit for Problem 7.2

NI Multisim Measurements

1. Enter the circuit of Figure 7.3 on the previous page with an `AC_VOLTAGE` source. Set its "AC Analysis Magnitude" to 3 volts and leave the "AC Analysis Phase" at its default zero value. Run a Simulate → Analyses → Single Frequency AC Analysis to measure the open-circuit voltage magnitude and phase. Next, connect a 0.1 Ω resistor across the terminal pair A-B to approximate a short circuit and then measure the magnitude and phase of the short-circuit current through this small resistor. Calculate the Thévenin voltage and Thévenin impedance from these measurements.

2. Repeat to obtain the open-circuit voltage and short-circuit current at 1100 Hz.

Additional helpful tips:

- Name the net connected to Terminal A to make it easier to find in the AC Analysis "Outputs" tab.

- Choose "Magnitude/Phase" for the "Complex number format" option.

NI Multisim video tutorials:

- Measure phasor voltage with a Single Frequency AC Analysis:
  `http://youtu.be/SwYCsoOwfUs`

- Display and change net names:
  `http://youtu.be/0iZ-ph9pJjE`

NI myDAQ Measurements

1. Construct the circuit of Figure 7.3 on the preceding page; do *not* include resistor R_1 as this device simply models the finite winding resistance of the physical inductor. Build an op amp voltage follower to strengthen the current drive of AO0 and use the voltage follower output as $\mathbf{V}_\mathrm{S}$. Create the sinusoidal voltage with the NI ELVISmx Function Generator. Measure the open-circuit voltage magnitude and phase, taking the source voltage $\mathbf{V}_\mathrm{S}$ as the phase reference. Connect a 10 Ω resistor across the terminal pair A-B to approximate a short circuit. Measure the voltage across this resistor and divide by its measured resistance to obtain the short-circuit current magnitude

and phase. Calculate the Thévenin voltage and Thévenin impedance from these measurements.

2. Repeat to obtain the open-circuit voltage and short-circuit current at 1100 Hz.

Additional helpful tips:

- The function generator sets it amplitude in terms of "peak-to-peak" voltage; this is twice the amplitude of the sinusoid. Adjust this voltage to yield a sinusoid with a 3-volt amplitude.
- Use the NI ELVISmx Oscilloscope to display $\mathbf{V}_{\mathrm{S}}$ on AI0 and the voltage at Terminal A on AI1.
- The oscilloscope "Display Measurements" panel under the waveform display measures the peak-to-peak voltage of the waveforms. Set the timebase to display at least two cycles to ensure that the measurement is accurate. Display even more cycles to improve the accuracy and stability of the measurement.
- You may also use the cursors to measure the amplitude (peak value).
- Refer to Appendix F on page 179 to learn how to measure amplitude and phase of a sinusoidal waveform.

NI myDAQ video tutorials:

- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`
- Increase current drive of analog output (AO) channels with an op amp voltage follower:
  `http://decibel.ni.com/content/docs/DOC-12665`

7.3 Phase-Shift Circuits (7-8)

Figure 7.4 shows a phase-shift circuit based on op amps.

1. Write the general expression for the magnitude of $\mathbf{V}_{\text{out}}$ with frequency taken as a variable. HINT: View the circuit as the cascade of two standard op amp circuits.

2. Write the general expression for the phase of $\mathbf{V}_{\text{out}}$ with frequency taken as a variable.

3. Set C to 0.1 μF and set all resistors to 1.0 kΩ. Determine the frequency in hertz at which $\mathbf{V}_{\text{out}}$ and $\mathbf{V}_{\text{in}}$ share the same magnitude. What is the phase shift at this frequency?

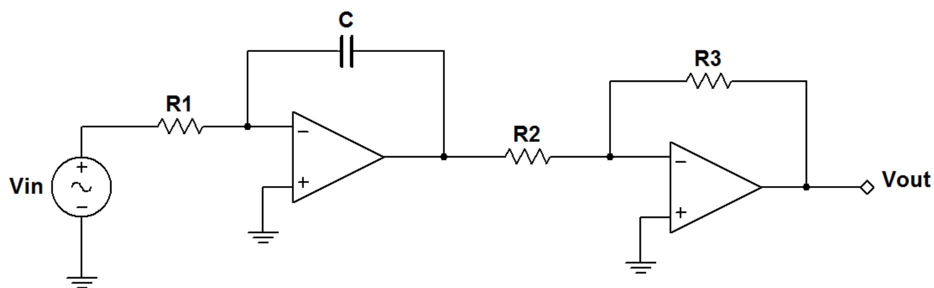


Figure 7.4: Circuit for Problem 7.3

NI Multisim Measurements

Enter the circuit of Figure 7.4 with the component values listed in Part 3 of the problem statement. Use the AC voltage source for $\mathbf{V}_{\text{in}}$ with unit amplitude and frequency set to the value you calculated in Part 3. Use interactive analysis and the oscilloscope to measure the the magnitude and phase of $\mathbf{V}_{\text{out}}$.

Additional helpful tips:

- Use the `AC_VOLTAGE` source. Set the “Voltage (Pk)” to 1 and “Frequency (F)” to the value you found in Part 3.

- **IMPORTANT**: As a practical matter, connect a 100 kΩ resistor in parallel with the capacitor to prevent the output of the first-stage circuit from saturating.
- Connect Simulate → Instruments → Oscilloscope so that you can view $\mathbf{V}_{\text{in}}$ and $\mathbf{V}_{\text{out}}$ at the same time.
- Use oscilloscope cursors to measure magnitude and phase with the technique described in Appendix F on page 179.

NI Multisim video tutorials:

- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`

NI myDAQ Measurements

Construct the circuit of Figure 7.4 on the preceding page with the component values listed in Part 3 of the problem statement. Use the NI ELVISmx Function Generator to create the sinusoidal voltage source on AO0; set its frequency to the value you calculated in Part 3. Use the NI ELVISmx Oscilloscope to measure the node voltages $\mathbf{V}_{\text{in}}$ and $\mathbf{V}_{\text{out}}$.

Additional helpful tips:

- The function generator amplitude control uses "peak-to-peak" units. The amplitude of a sinusoid is one-half of its peak-to-peak value.
- **IMPORTANT**: As a practical matter, connect a 100 kΩ resistor in parallel with the capacitor to prevent the output of the first-stage circuit from saturating.
- Use oscilloscope cursors to measure magnitude a phase with the technique described in Appendix F on page 179.

NI myDAQ video tutorials:

- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

7.4 Phasor-Domain Analysis Techniques (7-9)

The circuit of Figure 7.5 operates at 1 kHz. Determine the node voltages $\mathbf{V}_{\rm A}$ and $\mathbf{V}_{\rm B}$.

Use these component values:

- $R_1 = 4.7$ kΩ, $R_2 = 3.3$ kΩ and $R_3 = 2.2$ kΩ
- $C_1 = 0.047$ μF and $C_2 = 0.1$ μF
- $\mathbf{V}_1 = 9\angle 0°$ V and $\mathbf{V}_2 = 3\angle -90°$ V

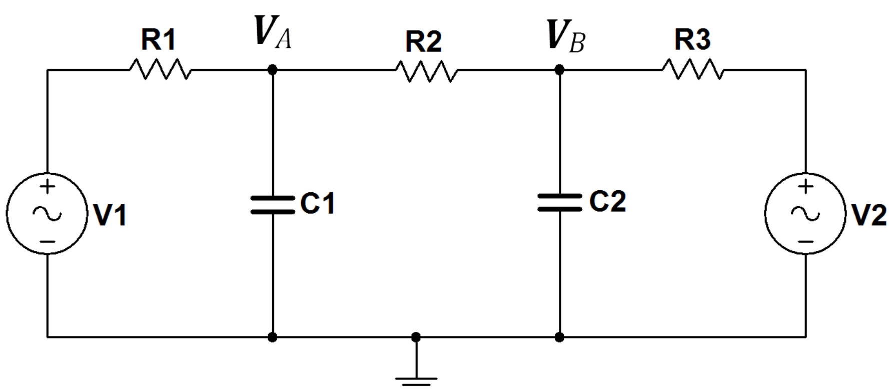


Figure 7.5: Circuit for Problem 7.4

NI Multisim Measurements

Enter the circuit of Figure 7.5 and run an AC analysis to measure the voltage magnitude and phase of $\mathbf{V}_{\rm A}$ and $\mathbf{V}_{\rm B}$.

More specifically:

- Use the AC_VOLTAGE source. Set the "AC Analysis Magnitude" and "AC Analysis Phase" values according to the specified voltage source values.
- Set up an Simulate → Analyses → AC Analysis to sweep the frequency over a range that includes 1 kHz; use a "Linear" sweep type with "Vertical Scale" set to "Linear."

- Use the "Grapher View" cursors to measure the voltage magnitude and phase at 1 kHz.

NI Multisim video tutorials:

- Measure frequency response with AC Analysis:
  `http://youtu.be/tgCPDBtRcso`
- Find the maximum value of trace in Grapher View:
  `http://youtu.be/MzYK60mfh2Y`

NI myDAQ Measurements

Construct the circuit of Figure 7.5 on the preceding page. Use the NI ELVISmx Arbitrary Waveform Generator to create the sinusoidal voltage sources; use AO0 to create $\mathbf{V}_1$ and AO1 to create $\mathbf{V}_2$. Use the NI ELVISmx Oscilloscope to measure the node voltages $\mathbf{V}_A$ and $\mathbf{V}_B$. Take $\mathbf{V}_1$ as the reference voltage when measuring phase.

Additional helpful tips:

- Set the Arbitrary Waveform Generator sampling rate to its maximum value of 200 kHz.
- Create a single period of a unit-amplitude 1 kHz sinusoid for each output channel. Adjust the phase of the AO1 channel to match the phase of $\mathbf{V}_2$.
- Set the "Gain" controls of the Arbitrary Waveform Generator to match the magnitudes of the specified voltage sources.
- Refer to Appendix F to learn how to use the oscilloscope to measure magnitude and phase.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

Chapter 8

AC Power

8.1 Periodic Waveforms (8-1)

Merriam-Webster defines a *rectifier* as a "device for converting alternating current into direct current." A *half-wave* rectifier does so by setting the negative portion of the waveform to zero. A *full-wave* rectifier negates the negative portion of the waveform, effectively reflecting it to its positive equivalent.

1. Plot the half-wave rectifier output and full-wave rectifier output for each of the three standard waveforms shown in Figure 8.1 on page 109.

2. Determine the general expressions for the (a) average value and (b) rms value for each of the six rectified waveforms.

3. Evaluate your expressions for average and rms values for $V_m = 10$ volts and $T = 10$ ms.

NI Multisim Measurements

1. Create the half-wave rectified and full-wave rectified versions of the three standard waveforms of Figure 8.1 on page 109 using the following approach:

   - Create the unrectified original waveform with the Function Generator,

- Connect a wire to the + terminal of the function generator and double-click to terminate the wire in "empty space" without connecting to any other terminal. Display the net names to determine the number of this net. Connect the `Common` terminal to ground.

- Place the `ABM_VOLTAGE` source, ground its negative terminal, and enter the function `positive(v(n))` (where `n` is the net number of the wire connected to the function generator output) to make the ABM source create the half-wave rectified version of the function generator waveform,

- Place another ABM source (also with its negative terminal grounded) with the absolute value function `abs(v(n))` to create the full-wave rectified waveform, and

- Place an `SPDT` switch (single-pole double-throw) to conveniently connect one ABM source or the other to the oscilloscope to display the rectified waveform.

2. Place a Measurement Probe to display the average value and rms value of the rectified waveform. Determine the average and rms values of each of the rectified waveforms.

Additional tips:

- The "Analog Behavior Modeling" (ABM) voltage source "senses" the voltage of net `n` with the function `v(n)`. The connection between the function generator and the ABM source does not appear visually.

- The *Multisim User Manual* describes a wide variety of functions that can be used by the ABM source. Visit `ni.com/manuals` and enter "Multisim User Manual" into the search box. See the "Mathematical Expressions" section of Chapter 7.

- Set up the Simulate → Instruments → Measurement Probe to display `V(dc)` for average value and `V(rms)` for rms value.

NI Multisim video tutorials:

- Display and change net names:
  `http://youtu.be/0iZ-ph9pJjE`
- Measure RMS and average value with a measurement probe:
  `http://youtu.be/OnK-Unld17E`
- Set the digits of precision of a measurement probe:
  `http://youtu.be/GRO60XLgzHg`
- ABM (Analog Behavioral Model) voltage source:
  `http://youtu.be/8pPynWRwhO4`

NI myDAQ Measurements

1. Create the half-wave rectified and full-wave rectified versions of the three standard waveforms of Figure 8.1 on page 109 with the arbitrary waveform generator on AO0; use 200 kS/s for the sampling rate. Display this signal on the oscilloscope using AI0. Extract the average value of the waveform using the RC circuit shown in Figure 8.2 on page 110 and display the output of this circuit on AI1.

2. Read the average value of the rectified waveform using either the cursor value or the "RMS" indicator below the waveform display for AI1. Read the "RMS" indicator for AI0 to measure the rms value of the rectified waveform. Display at least two periods of the waveform to ensure accuracy of the "RMS" indicator.

Additional helpful tips:

- Set both channels of the oscilloscope to the same volts per division. In this way the RC circuit output (average value) overlays the rectified waveform and you can visually see how the average value tends toward the effective center of the waveform.

- **IMPORTANT:** Be sure to observe proper polarity for the electrolytic capacitor. The capacitor may mark the negative lead rather than the positive lead.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

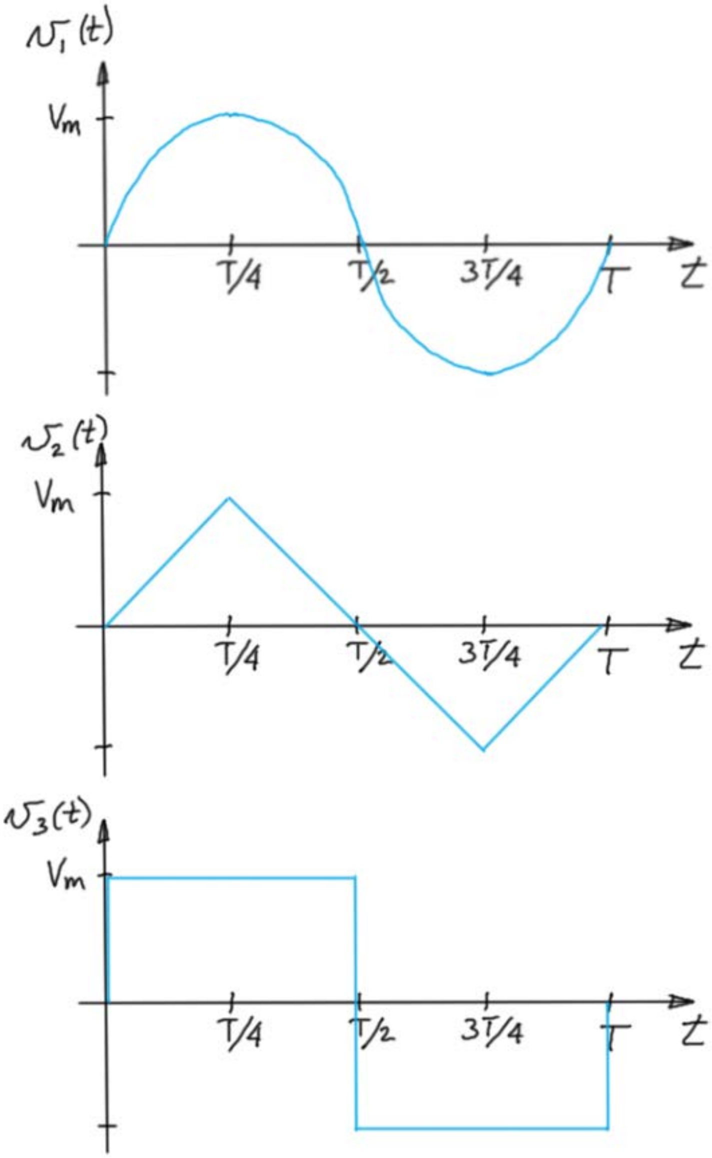


Figure 8.1: Waveforms for Problem 8.1

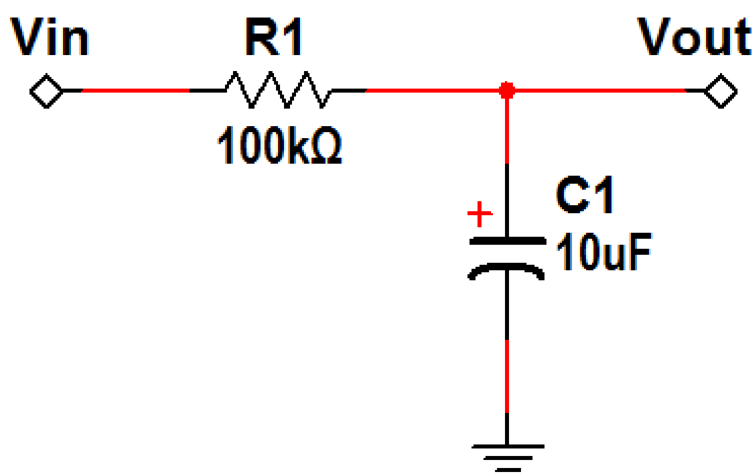


Figure 8.2: RC circuit to extract average value of a waveform.

8.2 Average Power (8-2)

The circuit shown in Figure 8.3 operates in sinusoidal steady state at 1500 Hz. The voltage source amplitude is 3 volts. Find the average power delivered by the voltage source.

Use these component values:

- $R = 100\ \Omega$,
- $C = 1.0\ \mu$F, and $L = 3.3$ mH

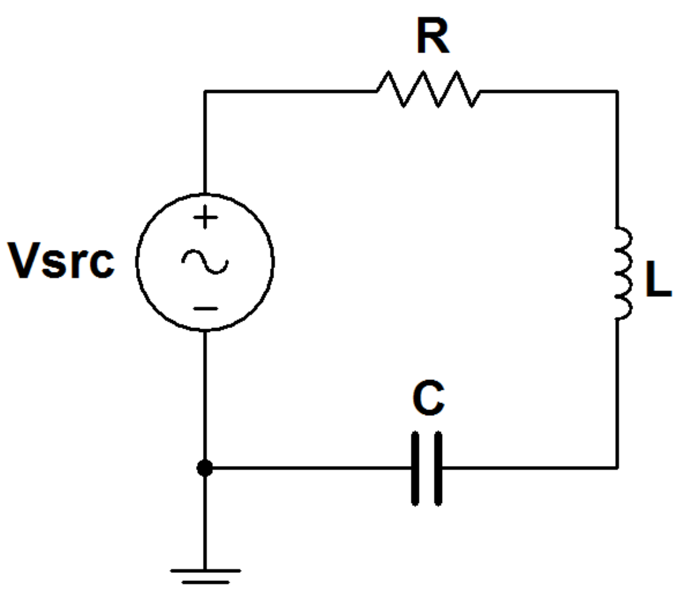


Figure 8.3: Circuit for Problem 8.2

NI Multisim Measurements

Enter the circuit of Figure 8.3 with an `AC_VOLTAGE` source, setting its peak value and frequency according to the values specified in the problem statement. Place a Simulate → Instruments → Wattmeter instrument to measure the voltage source power. The wattmeter displays average power in sinusoidal steady state.

NI Multisim video tutorials:

- Measure average power and power factor with a wattmeter:
  `http://youtu.be/kYliPwbWInc`

NI myDAQ Measurements

Build the circuit of Figure 8.3 on the preceding page. Activate the circuit with the NI ELVISmx Function Generator on AO0, and place an op amp voltage follower between AO0 and the remaining circuit; the voltage follower is necessary to boost the current drive of the analog output beyond its limit of 2 mA.

Use the NI ELVISmx Oscilloscope to display the voltage source on AI0; this voltage serves as the reference sinusoid for phase measurement. Display the voltage across the resistor on AI1, and realize that this voltage is proportional to the voltage source current.

Calculate average power from three measurements: voltage source magnitude, current magnitude, and the phase difference between the source voltage and current.

Additional helpful tips:

- The oscilloscope numerical display shows the RMS voltage of the waveforms. Use the appropriate equation for average power based on these RMS voltages.

- The oscilloscope numerical display also shows the peak-to-peak voltage of the waveforms. One half of this value is the amplitude of a sinusoidal waveform. If you choose this measurement, use the appropriate equation for average power based on amplitudes.

- Refer to Figure F on page 179 to learn how to measure amplitude and phase of a sinusoidal waveform.

NI myDAQ video tutorials:

- Increase current drive of analog output (AO) channels with an op amp voltage follower:
  `http://decibel.ni.com/content/docs/DOC-12665`

Further Exploration with NI myDAQ

The power delivered by the voltage source depends on it frequency:

1. Monitor the voltage source current on the oscilloscope and adjust the voltage source frequency to maximize the current amplitude, thereby maximizing the average source power delivered to the circuit. Record the voltage source frequency $f_{\max}$ at which the source average power is maximized, and then calculate the average power delivered by the source at this frequency.

2. Calculate the impedance of the inductor and capacitor at $f_{\max}$. Explain how these impedance values relate to maximizing the average power delivered by the source.

3. Adjust the voltage source frequency in a range above and below $f_{\max}$ and observe the phase difference between the source voltage and current. Discuss your observations.

8.3 Complex Power (8-3)

The circuit shown in Figure 8.4 operates in sinusoidal steady state at 1000 Hz. The voltage source amplitude is 2.5 volts.

1. Find the complex power in rectangular format for each of the four circuit elements: $\mathbf{S}_{\mathrm{SRC}}$, $\mathbf{S}_{\mathrm{R}}$, $\mathbf{S}_{\mathrm{L}}$, and $\mathbf{S}_{\mathrm{C}}$.
2. Demonstrate conservation of complex power with these four values.

Use these component values:

- $R = 100\ \Omega$, $C = 1.0\ \mu$F, and $L = 3.3$ mH

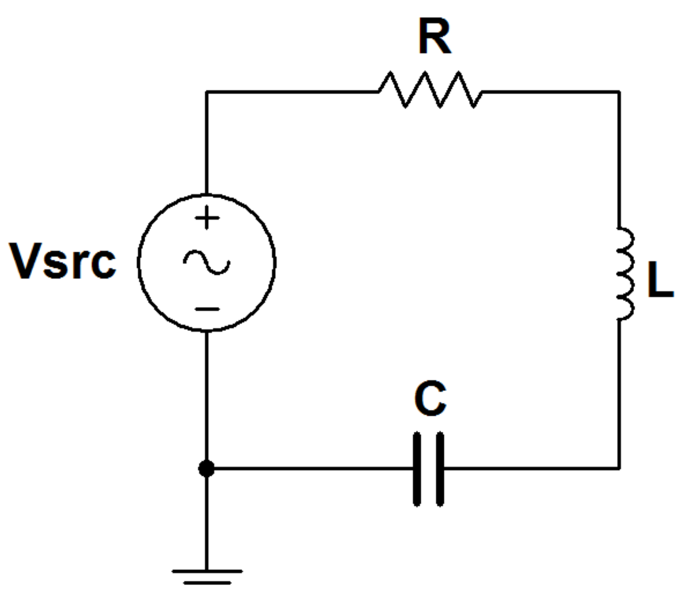


Figure 8.4: Circuit for Problem 8.3

NI Multisim Measurements

Enter the circuit of Figure 8.4 with an `AC_VOLTAGE` source. Set the "AC Analysis Magnitude" parameter to match the amplitude specified in the problem statement.

1. Run a Simulate → Analyses → Single Frequency AC Analysis to determine the complex powers $\mathbf{S}_{\text{SRC}}$, $\mathbf{S}_{\text{R}}$, $\mathbf{S}_{\text{L}}$, and $\mathbf{S}_{\text{C}}$ in rectangular format.

2. Demonstrate conservation of complex power with these four values.

Additional helpful tips:

- The single-frequency AC analysis calls element power "P" (which suggests average power only) but in fact calculates complex power "S."

NI Multisim video tutorials:

- Measure phasor voltage with a Single Frequency AC Analysis: `http://youtu.be/SwYCsoOwfUs`

NI myDAQ Measurements

Build the circuit of Figure 8.4 on the facing page. Activate the circuit with the NI ELVISmx Function Generator on AO0, and place an op amp voltage follower between AO0 and the remaining circuit; the voltage follower is necessary to boost the current drive of the analog output beyond its limit of 2 mA.

Use the NI ELVISmx Oscilloscope to display the device voltage on AI0. Display the voltage across resistor R on AI1, and realize that this voltage is proportional to the current through every device in the circuit.

1. Measure the complex power for each of the four circuit elements using the following procedure:

   - Read the RMS voltage V_{rms} from the oscilloscope numerical display,
   - Read the RMS current I_{rms} from the oscilloscope numerical display (divide the measured voltage by the resistance R),
   - Measure the time shift between the voltage and current sinusoids with the oscilloscope cursors,
   - Convert time shift to phase angle $\phi_Z = \phi_v - \phi_i$ (see Appendix F on page 179 for details),
   - Note whether the current waveform lags or leads the voltage; ϕ_Z is positive when current lags and negative when current leads,

  - Express complex power as $\mathbf{S} = V_{\mathrm{rms}} I_{\mathrm{rms}} \angle \phi_Z$, and
  - Convert $\mathbf{S}$ to rectangular form.

2. Demonstrate conservation of complex power with these four values.

Additional helpful tips:

- Observe passive sign convention for each circuit element when measuring voltage and current: positive current enters the positive polarity terminal.
- Refer to Figure F on page 179 to learn how to measure amplitude and phase of a sinusoidal waveform.

NI myDAQ video tutorials:

- Increase current drive of analog output (AO) channels with an op amp voltage follower: `http://decibel.ni.com/content/docs/DOC-12665`

8.4 The Power Factor (8-4)

The circuit shown in Figure 8.5 on the next page is a "scale model" of two industrial electric motors and a heating unit connected to a manufacturing plant power distribution network. The resistor/inductor combinations R_1-L_1 and R_2-L_2 model the winding resistance and magnetic fields of the motors. Resistor R_3 models the heater coils. C represents the power factor compensation equipment – essentially a capacitor bank with high power capacity.

1. Determine the power factor of the uncompensated load, and draw its power triangle to scale.
2. Determine the value of the compensation capacitor C required to improve the load power factor to 0.90 lagging.
3. Available power factor compensation capacitors include 0.1 μF, 1.0 μF, and 10 μF; the cost of compensation equipment increases with capacitance. Choose the least expensive compensation capacitor closest to C and then determine the power factor and power triangle (also drawn to scale) of the compensated load.

Use these component values:

- $R_1 = 10\ \Omega$, $R_2 = 100\ \Omega$, and $R_3 = 100\ \Omega$
- $L = 3.3$ mH and $L = 33$ mH
- V_{SRC} with 1 volt amplitude and 2500 Hz frequency (actual industrial motors operate at hundreds of volts and 50 Hz to 60 Hz frequency)

NI Multisim Measurements

Enter the circuit of Figure 8.5 on the following page with an `AC_VOLTAGE` source, setting its peak value and frequency according to the values specified in the problem statement. Place a `SPST` switch to conveniently engage or disengage the power factor compensation capacitor. Connect a wattmeter to measure the power factor of the combined load and capacitor.

IMPORTANT: Reduce the interactive simulation maximum time step `TMAX` to `1e-006` seconds; find this parameter at Simulate → Interactive

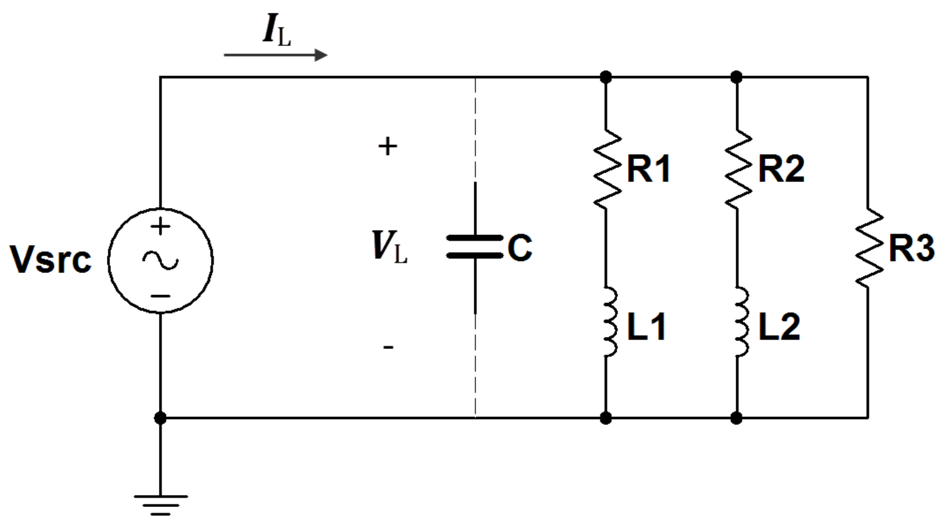


Figure 8.5: Circuit for Problem 8.4

Simulation Settings. The default time step does not produce sufficient sample points per period at the operating frequency in this problem to allow the wattmeter to produce an accurate power factor measurement.

1. Measure the power factor of the uncompensated load (switch open).
2. Close the switch to engage the compensation capacitor.
3. Measure the power factor of the compensated load.

NI Multisim video tutorials:

- Measure average power and power factor with a wattmeter:
  `http://youtu.be/kYliPwbWInc`

NI myDAQ Measurements

Build the circuit of Figure 8.5. Do *not* place resistors R_1 and R_2, as these simply model the finite winding resistances of the two inductors.

Activate the circuit with the NI ELVISmx Function Generator on AO0, and place an op amp voltage follower between AO0 and the remaining circuit; the voltage follower boosts the current drive of the analog output beyond its limit of 2 mA.

Use the NI ELVISmx Oscilloscope to display the load voltage on AI0. Place a 10 Ω shunt resistor between the op amp output and the remaining circuit; display the voltage across this shunt resistor on AI1, and recognize that its voltage is proportional to the load current.

1. Measure the power factor of the uncompensated load.

2. Place your selected compensation capacitor into the circuit.

3. Measure the power factor of the compensated load.

Additional helpful tips:

- Place the myDAQ AI1 connections with proper polarity to measure positive current flowing *into* the load.

- Recall that the power factor is $\cos(\phi_Z)$ where $\phi_Z = \phi_v - \phi_i$, the difference between the load's voltage phase and current phase.

- Refer to Appendix F on page 179 to learn how to measure the phase of a sinusoidal waveform.

NI myDAQ video tutorials:

- Increase current drive of analog output (AO) channels with an op amp voltage follower:
  `http://decibel.ni.com/content/docs/DOC-12665`

Further Exploration with NI myDAQ

Long-haul electrical energy distribution networks rely on *transformers* to boost the voltage to hundreds of thousands of volts while simultaneously reducing the current to very low levels – remember that power is the *product* of voltage and current, hence a given power can be transferred with low voltage and high current or vice versa. Reducing the current reduces the resistive losses in the transmission wires and improves efficiency.

Experience how power factor compensation impacts the amount of current that the utility must supply to the customer's equipment:

1. Measure and record the RMS voltage and current as displayed on the oscilloscope for the uncompensated load.

2. Repeat these measurements for the compensated load.

3. Discuss which values remain similar and which values change significantly. Comment on the value of power factor compensation as far as energy transmission efficiency is concerned.

Chapter 9

Frequency Response of Circuits and Filters

9.1 Scaling (9-2)

Figure 9.1 on the next page shows a prototype bandreject filter with center frequency $\omega_0 = 1$ rad/s. The prototype component values are $R = 1\ \Omega$, $L = 1.817$ H, and $C = 0.5505$ F.

1. Apply magnitude and frequency scaling to the bandreject filter so that $R' = 100\Omega$ and $L' = 33$ mH. Draw the finished circuit diagram.

2. Determine the center frequency in hertz of the scaled bandreject filter.

NI Multisim Measurements

1. Enter the circuit of Figure 9.1 on the following page using the scaled component values calculated earlier. Drive the filter input with an `AC_VOLTAGE` source with "AC Analysis Magnitude" set to 1 V.

2. Plot the frequency response of the filter over the range 100 Hz to 10 kHz with Simulate → Analyses → AC Analysis. Change "Vertical Scale" to "Linear" and increase "Number of points per decade" as needed to plot a smooth curve. Use a cursor to identify the filter's center frequency.

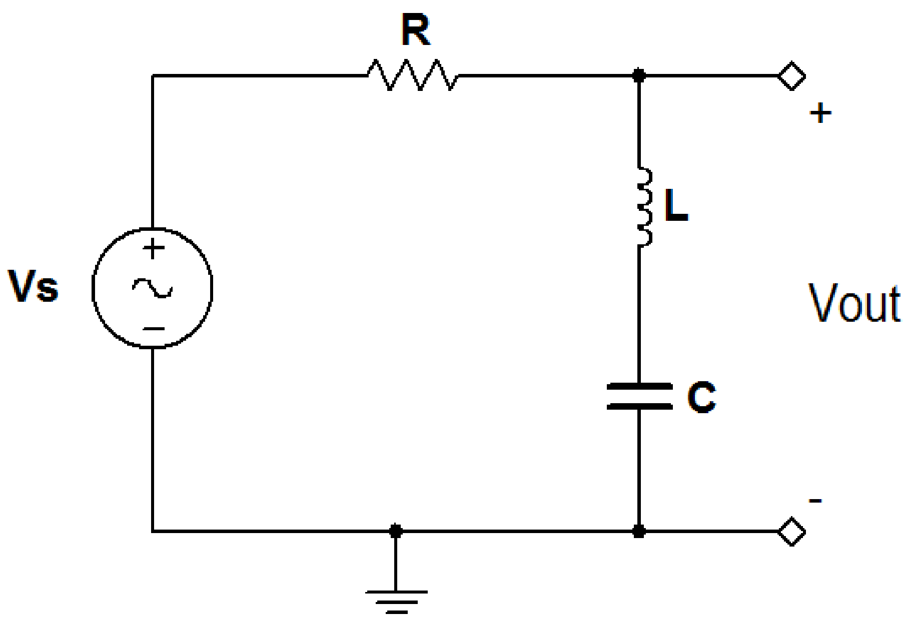


Figure 9.1: Circuit for Problem 9.1

NI Multisim video tutorials:

- Measure frequency response with AC Analysis:
  http://youtu.be/tgCPDBtRcso

NI myDAQ Measurements

1. Build the circuit of Figure 9.1 using the scaled component values calculated earlier. Drive the filter input with AO0 strengthened by an op amp voltage follower. Monitor the filter input with AI0 and the filter output with AI1.

2. Plot the frequency response of the filter over the range 100 Hz to 10 kHz with the ELVISmx Bode Analyzer. Change "Mapping" to "Linear" and increase "Steps" as needed to plot a smooth curve. Use a cursor to identify the filter's center frequency.

NI myDAQ video tutorials:

- Bode Analyzer:
  `http://decibel.ni.com/content/docs/DOC-12943`
- Increase current drive of analog output (AO) channels with an op amp voltage follower:
  `http://decibel.ni.com/content/docs/DOC-12665`

Further Exploration with NI myDAQ

NI Multisim provides a way to compare simulated results and physical measurement results from NI myDAQ on the *same* ELVISmx instrument. Study the video tutorial below to learn how to simultaneously display simulated and measured frequency response on the ELVISmx Bode Analyzer, and then do the following:

1. Plot the frequency response of the simulated and physical circuit of Figure 9.1 on the facing page using the scaled component values calculated earlier. Compare the two plots and discuss their similarities and differences.
2. The simulated circuit is a *model* of the physical circuit and may not capture every phenomenon of the real circuit. Recall that an inductor is formed by hundreds of turns of very fine (small diameter) wire, consequently the small yet finite resistance per unit length adds up to form a significant resistance. Measure the resistance of your inductor with the myDAQ DMM and then place a resistor with this value in series with the ideal inductor in your Multisim circuit. Re-run the simulator. Compare the two plots and discuss the performance of the improved circuit model.

NI Multisim video tutorials:

- Combine Multisim simulation and myDAQ measurements in the same instrument – Bode Analyzer:
  `http://youtu.be/3UmTmUj4h1g`

9.2 Bode Plots (9-3)

1. Determine the voltage transfer function $\mathbf{H}(\omega)$ of the filter circuit shown in Figure 9.2. Write your finished result in standard form suitable for creating a Bode plot.

2. Substitute $\omega = 2\pi f$ to express the voltage transfer function in terms of oscillation frequency f in hertz.

3. Generate Bode magnitude and phase plots for $\mathbf{H}(f)$ using oscillation frequency f as the independent variable. Use the following component values: $R_1 = 3.3$ kΩ, $R_2 = 10$ kΩ, $C_1 = 0.01$ μF, and $C_2 = 0.1$ μF.

4. Determine the following filter circuit properties by inspecting the Bode plot:

   (a) Low-frequency asymptotes for magnitude and phase

   (b) High-frequency asymptotes for magnitude and phase

   (c) Corner frequencies (this filter circuit has two such frequencies)

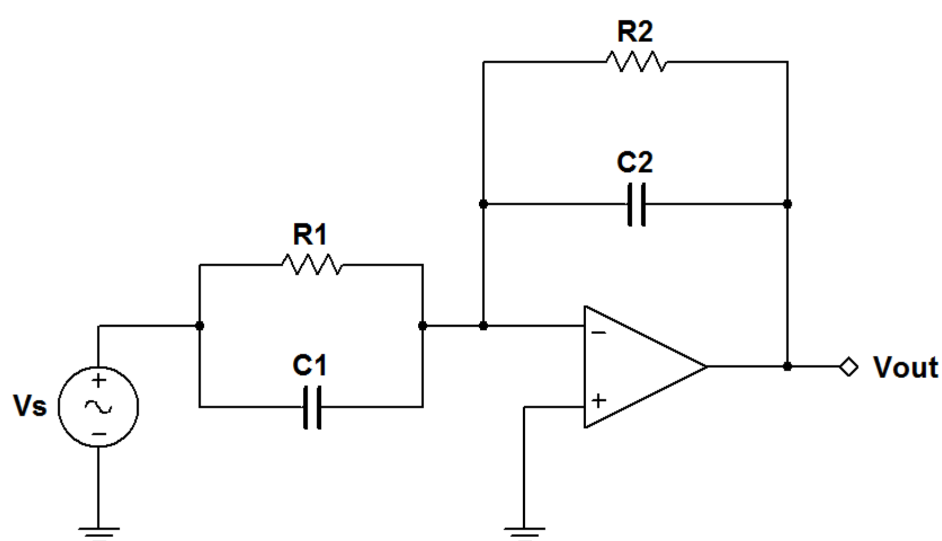


Figure 9.2: Circuit for Problem 9.2

NI Multisim Measurements

1. Enter the filter circuit of Figure 9.2 on the facing page. Drive the filter input with an AC_VOLTAGE source with “AC Analysis Magnitude” set to 1 V. Use the three-terminal virtual op amp model OPAMP_3T_VIRTUAL.

2. Plot the frequency response of the filter over the range 10 Hz to 100 kHz with Simulate → Analyses → AC Analysis. Set “Vertical Scale” to “Decibel” and “Sweep Type” to “Decade” to create a standard Bode plot presentation of frequency response. Increase “Number of points per decade” as needed to plot a smooth curve.

3. Determine the following filter circuit properties by inspecting the frequency response plot with cursors:

   (a) Low-frequency asymptotes for magnitude and phase

   (b) High-frequency asymptotes for magnitude and phase

   (c) Corner frequencies; look for a change of 3 dB in magnitude from an asymptote

NI Multisim video tutorials:

- Measure frequency response with AC Analysis: http://youtu.be/tgCPDBtRcso

NI myDAQ Measurements

1. Build the filter circuit of Figure 9.2 on the preceding page. Drive the filter input with AO0. Monitor the filter input with AI0 and the filter output with AI1.

2. Plot the frequency response of the filter over the range 10 Hz to 10 kHz with the ELVISmx Bode Analyzer; note that this frequency range omits the last decade compared to your analytical and simulation work. Increase “Steps” as needed to plot a smooth curve. **IMPORTANT:** Set “Peak Amplitude” to 1 volt.

3. Determine the following filter circuit properties by inspecting the frequency response plot with cursors:

   (a) Low-frequency asymptotes for magnitude and phase

   (b) High-frequency asymptotes for magnitude and phase

(c) Corner frequencies; look for a change of 3 dB in magnitude from an asymptote

Additional helpful tips:

- The low- and high-frequency phase asymptotes of any filter are *always* integer multiples of 90°. If the phase plot does not seem to flatten out enough, simply estimate the trend to the closest integer multiple of 90°.

NI myDAQ video tutorials:

- Bode Analyzer:
  `http://decibel.ni.com/content/docs/DOC-12943`

Further Exploration with NI myDAQ

Use the technique described in the video tutorial below to simultaneously display the simulated and measured frequency response with the ELVISmx Bode Analyzer. You may need to set "Op-Amp Signal Polarity" to "Inverted" to make the measured phase response overlay the simulated response.

Discuss the level of agreement between the two plots and explain any discrepancies you observe.

NI Multisim video tutorials:

- Combine Multisim simulation and myDAQ measurements in the same instrument – Bode Analyzer:
  `http://youtu.be/3UmTmUj4h1g`

9.3 Filter Order (9-5)

The filter circuit shown in Figure 9.3 uses the component values $R = 1.0\ \text{k}\Omega$ and $C = 1.0\ \mu\text{F}$.

1. Obtain an expression for $\mathbf{H}(\omega) = \mathbf{V}_\text{o}/\mathbf{V}_\text{i}$ in standard form.
2. Substitute $\omega = 2\pi f$ to express $\mathbf{H}(\omega)$ in terms of oscillation frequency f in hertz.
3. Generate spectral plots for the magnitude and phase of $\mathbf{H}(f)$.
4. Determine the cutoff frequency f_c.

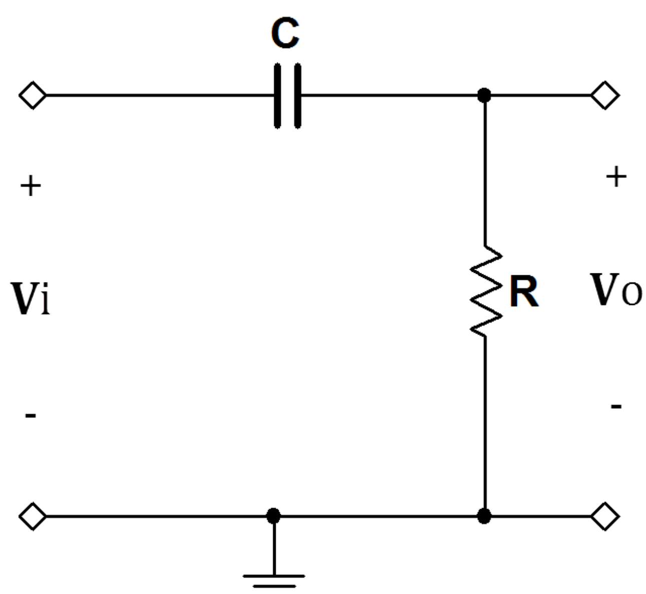


Figure 9.3: Circuit for Problem 9.3

NI Multisim Measurements

1. Enter the circuit of Figure 9.3. Drive the filter input with an `AC_VOLTAGE` source with "AC Analysis Magnitude" set to 1 V.

2. Plot the frequency response of the filter over the range 1 Hz to 10 kHz with Simulate → Analyses → AC Analysis. Set "Vertical Scale" to "Decibel" and "Sweep Type" to "Decade" to create a standard Bode plot presentation of frequency response. Increase "Number of points per decade" as needed to plot a smooth curve.

3. Use cursors to measure the cutoff frequency f_c (look for a change of 3 dB in magnitude).

NI Multisim video tutorials:

- Measure frequency response with AC Analysis:
  `http://youtu.be/tgCPDBtRcso`

NI myDAQ Measurements

1. Build the circuit of Figure 9.3 on the preceding page. Drive the filter input with AO0. Monitor the filter input with AI0 and the filter output with AI1.

2. Plot the frequency response of the filter over the range 1 Hz to 10 kHz with the ELVISmx Bode Analyzer. Increase "Steps" as needed to plot a smooth curve.

3. Use cursors to measure the cutoff frequency f_c (look for a change of 3 dB in magnitude).

NI myDAQ video tutorials:

- Bode Analyzer:
  `http://decibel.ni.com/content/docs/DOC-12943`

Further Exploration with NI myDAQ

Listen to the output of the Bode Analyzer to develop a more intuitive feel for its operation. Connect AO0 AGND to your headphones similar to the connection pictured in Figure 4.3 on page 48; connect the filter output to both the right and left channels for best listening (the middle ring of the audio plug carries the right channel signal). Comment on your listening experience.

IMPORTANT – PROTECT YOUR HEARING!

Do **NOT** disturb your circuit connections while you are wearing earphones. Accidently shorting together circuit connections can produce a very loud and unexpected noise. Alternatively, use a speaker to listen to the amplifier output or hold the earphones some distance from your ears.

9.4 Cascaded Active Filters (9-7)

A telephone line provides sufficient bandwidth (3 kHz) for intelligible voice conversations, but human hearing has a much higher bandwidth, typically 20 Hz to 20,000 Hz.

1. Design an active bandpass filter to mimic the bandwidth of a telephone line subject to the following constraints:

   (a) Cascade a first-order active lowpass filter and a first-order active highpass filter,

   (b) Set the corner frequencies to 300 Hz and 3.0 kHz,

   (c) Set the passband gain to 0 dB,

   (d) Choose resistors in the range 1.0 kΩ to 100 kΩ, and

   (e) Use a total of four fixed-value resistors and two fixed-value capacitors selected from the parts listed in Appendix A on page 159.

   Draw the schematic diagram of your finished design.

2. Predict the performance of your finished design by calculating the following values:

   (a) Low-frequency passband corner in hertz,

   (b) High-frequency passband corner in hertz, and

   (c) Passband gain in decibels.

NI Multisim Measurements

1. Enter your bandpass filter.

2. Plot the frequency response of the filter over the audio frequency range 20 Hz to 20 kHz with Simulate → Analyses → AC Analysis. Set "Vertical Scale" to "Decibel" and "Sweep Type" to "Decade" to create a standard Bode plot presentation of frequency response.

3. Evaluate the performance of your design by measuring the following values:

   (a) Low-frequency passband corner in hertz (move cursor to -3.0 dB and read its frequency),

   (b) High-frequency passband corner in hertz, and

(c) Passband gain in decibels.

NI Multisim video tutorials:

- Measure frequency response with AC Analysis: `http://youtu.be/tgCPDBtRcso`
- Set cursor to a specific value: `http://youtu.be/48sQja58I10`

NI myDAQ Measurements

1. Build your bandpass filter. Drive the filter input with AO0. Monitor the filter input with AI0 and the filter output with AI1.
2. Plot the frequency response of the filter over the audio frequency range 20 Hz to 20 kHz with the ELVISmx Bode Analyzer. Increase "Steps" as needed to plot a smooth curve.
3. Evaluate the performance of your design by measuring the following values with cursors:
   (a) Low-frequency passband corner in hertz (move cursor to -3.0 dB and read its frequency),
   (b) High-frequency passband corner in hertz, and
   (c) Passband gain in decibels.

Additional helpful tips:

- You can obtain more accuracy on the low- and high-frequency corners by sweeping the frequency over a narrow range that brackets the frequency of interest and substantially increasing the number of sweep steps.

NI myDAQ video tutorials:

- Bode Analyzer: `http://decibel.ni.com/content/docs/DOC-12943`

Further Exploration with NI myDAQ

Listen to the audible effects of your telephone line emulator:

1. Remove the AO0 connection,
2. Connect your audio player's output to your filter's input using the 3.5 mm stereo audio cable included with your NI myDAQ product; use test leads to connect the left channel similar to the connection pictured in Figure 4.3 on page 48.
3. Connect your headphones to the filter output; connect the filter output to both the right and left channels for best listening (the middle ring of the audio plug carries the right channel signal),
4. Play music through your filter, and
5. Compare the sound of the music "with filtering" and "without filtering" by temporarily shorting the high-pass filter capacitor and disconnecting the low-pass filter capacitor. Comment on your listening experience.

   IMPORTANT – PROTECT YOUR HEARING!
   Do **NOT** disturb your circuit connections while you are wearing earphones. Accidently shorting together circuit connections can produce a very loud and unexpected noise. Alternatively, use a speaker to listen to the amplifier output or hold the earphones some distance from your ears.

For more sophistication, use the DG413 analog switch to conveniently engage or disengage the telephone line emulator under control of one of the myDAQ DIO digital outputs. Use one normally-open switch and one normally-closed switch for single-line control of the two switches necessary to engage/disengage the filter.

Chapter 10

Laplace Transform Analysis Techniques

10.1 s-Domain Circuit Analysis (10-7)

1. Determine $v(t)$ of the circuit shown in Figure 10.1 on the following page for $t \geq 0$, given that the switch is opened at $t = 0$ after having been closed for a long time. Use the following component values: $V_{\text{src}} = 8$ V, $R_1 = 470\ \Omega$, $R_2 = 100\ \Omega$, $R_{\text{w}} = 90\ \Omega$, $C = 1.0\ \mu$F, and $L = 33$ mH.

2. Plot $v(t)$ from 0 to 5 ms using a tool such as MathScript or MATLAB. Include hardcopy of the script used to create the plot.

3. Determine the following values for $v(t)$:

   (a) Initial value $v(0)$,

   (b) Final value of $v(t)$,

   (c) Minimum value of $v(t)$, and

   (d) Time to reach the minimum value of $v(t)$.

NI LabVIEW video tutorials:

- Plot two functions of time:
  `http://youtu.be/XQlAai1-YVc`

- Take cursor measurements on a plot:
  `http://youtu.be/bgK1p5O6OXc`

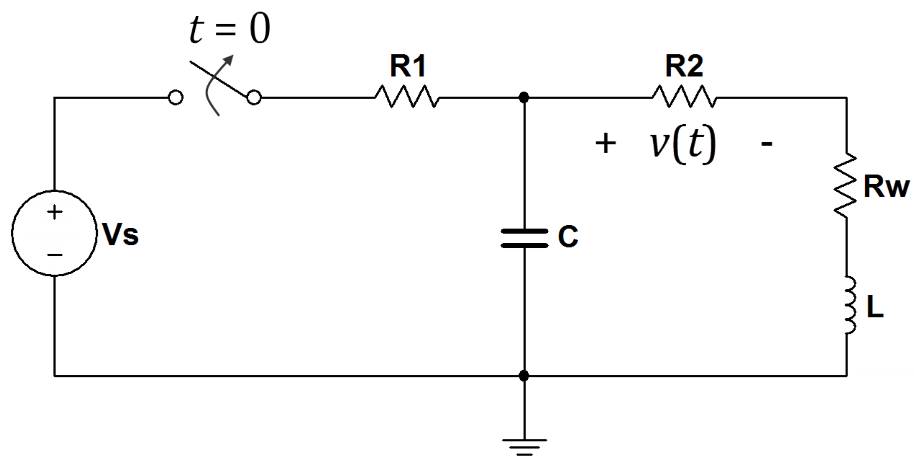


Figure 10.1: Circuit for Problem 10.1

NI Multisim Measurements

1. Enter the circuit of Figure 10.1 using the same component values listed in the problem statement. Implement the switch with a `VOLTAGE_CONTROLLED_SWITCH` operated by a `PULSE_VOLTAGE` source configured to open the switch at time 1 ms; this delay makes the initial transition easier to see.
2. Plot $v(t)$ from 0 to 5 ms with a Simulate → Analyses → Transient analysis.
3. Use the Grapher View cursors to measure the following values for $v(t)$:
   (a) Initial value $v(0)$,
   (b) Final value of $v(t)$,
   (c) Minimum value of $v(t)$, and
   (d) Time to reach the minimum value of $v(t)$.

Helpful tip for this problem:

- Remember that Multisim voltages are all *node* voltages, i.e., a voltage with respect to ground. The voltage $v(t)$ in this problem exists *be-*

tween two nodes, however. Name the nets on either side of R_2 (or display their default net numbers), click "Add expression" in the "Output" tab of the transient analysis setup panel, and enter an expression of the form "`v(pos)-v(neg)`" where `pos` and `neg` denote the two net names that connect to R_2 with positive and negative polarity; the expression forms the mathematical difference between the two node voltages.

NI Multisim video tutorials:

- Pulse voltage source:
  `http://youtu.be/RdgxVfr28C8`
- Voltage-controlled switch:
  `http://youtu.be/BaEBjhD4TOw`
- Plot time-domain circuit response with Transient Analysis:
  `http://youtu.be/waKnad_EXkc`
- Display and change net names:
  `http://youtu.be/0iZ-ph9pJjE`

NI myDAQ Measurements

1. Construct the circuit of Figure 10.1 on the facing page using the following components and NI ELVISmx instruments:

   - One normally-closed Switch 2 contained in the Intersil DG413 quad analog switch described in Appendix D on page 173.
   - 8.0 volt source created with the LM317 variable voltage circuit of Figure B.2 on page 164.
   - 1.0 μF electrolytic capacitor. **IMPORTANT**: Observe proper polarity of the capacitor by connecting the negative terminal of the capacitor to ground.
   - AIO0 (Analog Output 0) to the "Logic Control" (switch control) input for Switch 2.
   - AI0 (Analog Input 0) to display the switch control voltage for Switch 2; connect AI0+ to the switch control input and connect AI0- to ground.
   - AI1 (Analog Input 1) to display $v(t)$.

- Function Generator to create the switch control waveform; set the frequency to 100 Hz and adjust the amplitude and offset to place the control waveform between 0 and 5 volts.
- Oscilloscope to view the Switch 2 control waveform and the voltage $v_\mathrm{t}(t)$.

2. Display $v(t)$ from 0 to 5 ms.
3. Use the oscilloscope cursor to measure the following values for $v(t)$:
   (a) Initial value $v(0)$,
   (b) Final value of $v(t)$,
   (c) Minimum value of $v(t)$, and
   (d) Time to reach the minimum value of $v(t)$.

NI myDAQ video tutorials:

- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

Further Exploration with NI myDAQ

Switching circuits such as the one in this problem generally demand high current from the power supply for brief periods of time. These high-current pulses can cause *spikes* on the supply line that could disrupt proper operation of other connected devices such as digital microcontrollers. Connecting a capacitor between the power supply rail and ground provides a local supply of temporary current for the switching circuit to stabilize the power supply rail for other devices.

1. Observe the V_SRC rail created by the LM317 on the oscilloscope; it should still be set to 8.0 V. Estimate the magnitude of the voltage spike and express its value as a percentage of 8.0 V.
2. Continue to observe the power supply rail as you connect a 10 μF capacitor between the V_SRC rail and ground; place the capacitor in close proximity to the switching circuit and remember to observe its polarity. Discuss the improvement in the stability of the power supply rail.

10.2 Step Response (10-8)

1. Determine the transfer function $\mathbf{H}(\mathbf{s}) = \mathbf{V}_\mathrm{o}(\mathbf{s})/\mathbf{V}_\mathrm{s}(\mathbf{s})$ of the circuit shown in Figure 10.2 on the next page. Write the transfer function in simplified standard form with symbolic values.

2. Determine the output response $v_\mathrm{o}(t)$ to the input $v_\mathrm{s}(t) = 4u(t)$ by working in the Laplace domain. Assume the capacitor is initially discharged.

3. Plot $v_\mathrm{s}(t)$ and $v_\mathrm{o}(t)$ on the same graph from 0 to 5 ms using a tool such as MathScript or MATLAB for $R = 5.6$ kΩ and $C = 0.1$ μF. Include hardcopy of the script used to create the plot.

4. Determine the following values for $v_\mathrm{o}(t)$:

   (a) Initial value $v_\mathrm{o}(0^+)$,

   (b) Time to reach 50% of the initial value, and

   (c) Final value.

NI Multisim Measurements

1. Enter the circuit of Figure 10.2 on the following page using the same component values listed in the problem statement. Drive the circuit input with a PULSE_VOLTAGE source configured to produce $v_\mathrm{s}(t) = 4u(t)$. Delay the pulse by 1 ms to make the initial step transition visible.

2. Plot $v_\mathrm{s}(t)$ and $v_\mathrm{o}(t)$ on the same graph from 0 to 5 ms with a Simulate → Analyses → Transient analysis.

3. Use the Grapher View cursors to measure the following values for $v_\mathrm{o}(t)$:

   (a) Initial value $v_\mathrm{o}(0^+)$,

   (b) Time to reach 50% of the initial value, and

   (c) Final value.

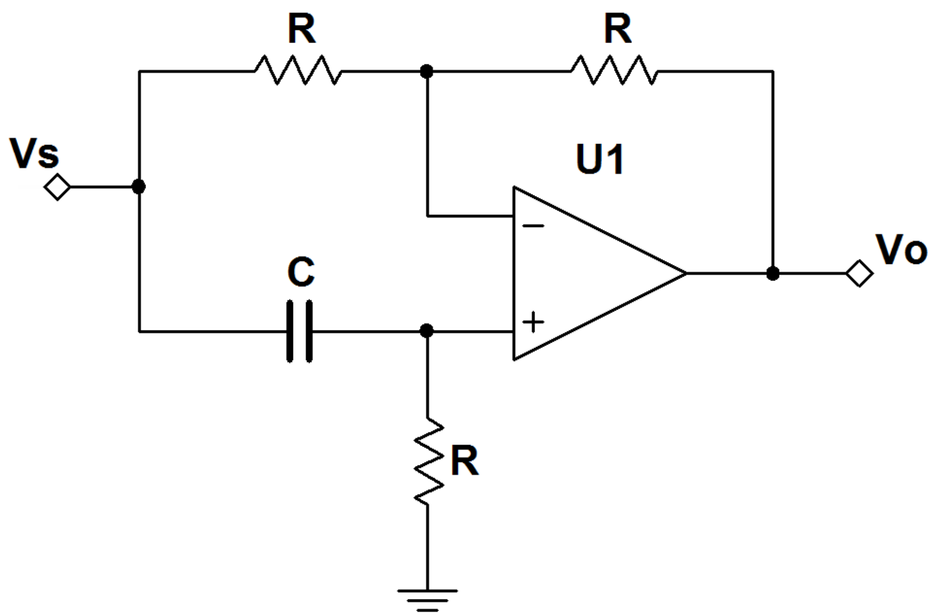


Figure 10.2: Circuit for Problem 10.2

NI Multisim video tutorials:

- Pulse voltage source:
  http://youtu.be/RdgxVfr28C8
- Plot time-domain circuit response with Transient Analysis:
  http://youtu.be/waKnad_EXkc

NI myDAQ Measurements

1. Build the circuit of Figure 10.2 using the same component values listed in the problem statement. Drive the circuit input with AO0 and use the ELVISmx Function Generator to produce a zero-to-four volt step transition with a period of 10 ms. Monitor the input voltage $v_s(t)$ with AI0 and the output voltage $v_o(t)$ with AI1.
2. Display $v_s(t)$ and $v_o(t)$ on the ELVISmx Oscilloscope.
3. Use the oscilloscope cursors to measure the following:

(a) Initial value $v_o(0^+)$,

(b) Time to reach 50% of the initial value, and

(c) Final value.

NI myDAQ video tutorials:

- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

Further Exploration with NI myDAQ

The circuit in this problem represents one implementation of an *all-pass filter*. Set up the ELVISmx Bode Analyzer to measure the frequency response of the circuit; the necessary myDAQ connections should already be in place. Set up the Bode analyzer controls as follows:

- Start frequency = 10 Hz
- Stop frequency = 10 kHz
- Steps = 10 per decade
- Mapping = linear

After running the frequency sweep set the "Gain" axis range to a minimum of zero and a maximum of 2; double-click the numerical values at the top and bottom of the axis display to set these values.

Study the response and then discuss the following questions:

1. Why is the circuit called an "all-pass" filter?
2. What is the general behavior of the phase response? More specifically, what are the maximum and minimum values of phase shift?
3. Use the cursor to measure the frequency at the midpoint between the maximum and minimum phase shift values. Compare this frequency to the critical frequencies in the transfer function $\mathbf{H}(\mathbf{s})$ you derived in the analytical section. HINT: Remember to account for angular frequency versus oscillation frequency.

10.3 Transfer Function and Impulse Response (10-8)

Figure 10.3 shows a passive highpass filter circuit. After you determine its behavior (transfer function), design an input waveform to achieve a specified output waveform shape; do the first three parts of this problem symbolically to maintain generality:

1. Determine the transfer function $\mathbf{H}(\mathbf{s}) = \mathbf{Y}(\mathbf{s})/\mathbf{X}(\mathbf{s})$ of the circuit.

2. Determine and sketch the impulse response $h(t)$ of the circuit.

3. Create an input waveform $x(t)$ that will cause the output waveform $y(t)$ to be a unit step function $u(t)$. HINT: Work first in terms of $\mathbf{H}(\mathbf{s})$, $\mathbf{Y}(\mathbf{s})$, and $\mathbf{X}(\mathbf{s})$.

4. Plot $x(t)$ and $y(t)$ on the same graph from 0 to 100 ms using a tool such as MathScript or MATLAB for $R = 10$ kΩ and $C = 1.0$ μF. Include hardcopy of the script used to create the plot.

5. Evaluate $x(t)$ and $y(t)$ at $t = 50$ ms.

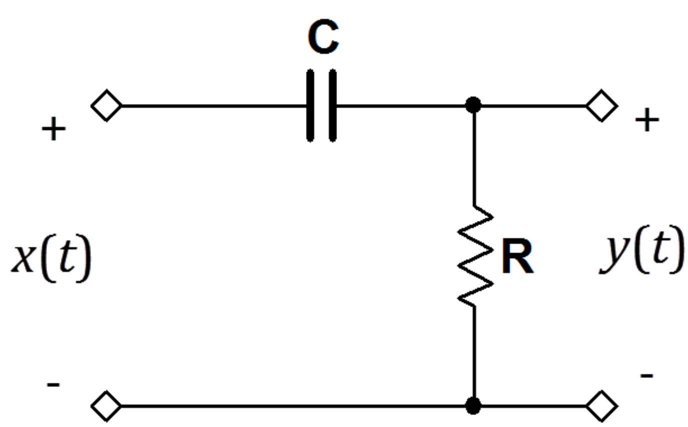


Figure 10.3: Circuit for Problem 10.3

NI Multisim Measurements

1. Enter the circuit of Figure 10.3 on the preceding page using the same component values listed in the analytical section. Drive the circuit input with an `ABM_VOLTAGE` source.

2. Plot $x(t)$ and $y(t)$ on the same graph from 0 to 100 ms with a Simulate → Analyses → Transient analysis.

3. Use the Grapher View cursors to measure the voltages $x(t)$ and $y(t)$ at $t = 50$ ms.

Additional Multisim tips for this problem:

- Build the ABM voltage source "Voltage Value" string by combining one or more of the following functions:
  - `TIME` – Time function t
  - `u(TIME)` – Unit step function $u(t)$
  - `uramp(TIME)` – Unit ramp function $r(t)$
  - Standard math operators: `+`, `−`, `*`, and `/`

NI Multisim video tutorials:

- ABM (Analog Behavioral Model) voltage source:
  http://youtu.be/8pPynWRwhO4

- Plot time-domain circuit response with Transient Analysis:
  http://youtu.be/waKnad_EXkc

NI myDAQ Measurements

1. Build the circuit of Figure 10.3 on the facing page using the same component values listed in the analytical section. Drive the circuit input with AO0 and use the ELVISmx Arbitrary Function generator to produce the waveform $x(t)$ with a period of 100 ms. Set the voltage to zero before the end of the period to allow the circuit sufficient time to return to its zero-output state.

2. Display $x(t)$ and $y(t)$ on the ELVISmx Oscilloscope.

3. Use the oscilloscope cursors to measure the voltages $x(t)$ and $y(t)$ at $t = 50$ ms.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

Further Exploration with NI myDAQ

The impulse function $\delta(t)$ is physically unrealizable as an input voltage waveform, however, it can be approximated by a large-valued short-duration rectangular pulse. Use the ELVISmx Arbitrary Waveform Generator to create a 10-volt pulse with very short duration compared to the 100 ms period. View this input waveform and the circuit's output (an approximation to the impulse response) on the oscilloscope. Discuss the similarity and difference between what you observe and what you calculated earlier as the circuit's $h(t)$.

10.4 Convolution Integral (10-9)

Figure 10.4 shows a passive lowpass filter circuit and its input voltage waveform $x(t)$.

1. Determine the impulse response $h(t)$ of the circuit; work this and the next part with symbolic values.

2. Determine the output of the circuit $y(t) = x(t) * h(t)$ by integrating the convolution analytically, i.e., use Method 2 of Example 10-15 in your text. Carry out this step using symbolic values.

3. Plot $x(t)$ and $y(t)$ on the same graph from 0 to 100 ms using a tool such as MathScript or MATLAB for the following values: $R = 10$ kΩ, $C = 1.0$ μF, $A = 5$ volts, and $t_0 = 50$ ms. Include hardcopy of the script used to create the plot.

4. Evaluate $y(t)$ at the following times: 25 ms, 50 ms, and 60 ms.

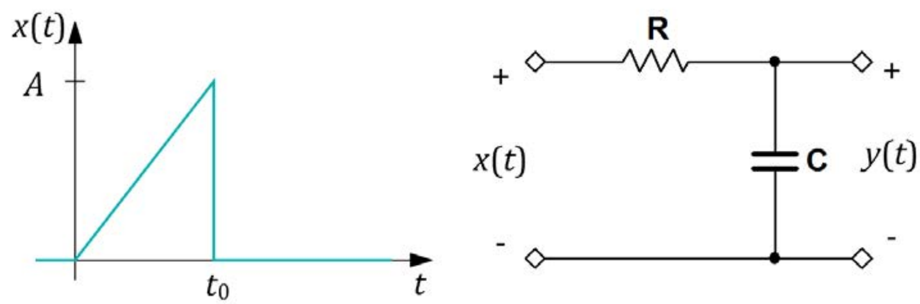


Figure 10.4: Circuit and input voltage waveform for Problem 10.4

NI Multisim Measurements

1. Enter the circuit of Figure 10.4 using the same component values listed in the problem statement. Drive the circuit input with an `ABM_VOLTAGE` source configured to produce the waveform $x(t)$ with $A = 5$ volts and $t_0 = 50$ ms. Alternatively, use a `PIECEWISE_LINEAR_VOLTAGE` source.

2. Plot $x(t)$ and $y(t)$ on the same graph from 0 to 100 ms with a Simulate → Analyses → Transient analysis. Adjust the maximum time step setting to plot at least 100 time points.

3. Use the Grapher View cursors to measure the output voltage $y(t)$ at the times 25, 50, and 60 ms.

Additional Multisim tips for this problem:

- Build the ABM voltage source "Voltage Value" string by combining the following functions:
  - `TIME` – Time function t
  - `u(TIME)` – Step function $u(t)$
  - Standard math operators: `+`, `−`, `*`, and `/`

NI Multisim video tutorials:

- ABM (Analog Behavioral Model) voltage source:
  `http://youtu.be/8pPynWRwhO4`
- Piecewise linear (PWL) voltage source:
  `http://youtu.be/YYU5WuyebD0`
- Plot time-domain circuit response with Transient Analysis:
  `http://youtu.be/waKnad_EXkc`

NI myDAQ Measurements

1. Build the circuit of Figure 10.4 on the previous page using the same component values listed in the problem statement. Drive the circuit input with AO0 and use the ELVISmx Arbitrary Waveform Generator to produce the waveform shown in the same figure with $A = 5$ volts, and $t_0 = 50$ ms. Set the period of the waveform to 100 ms.

2. Display $x(t)$ and $y(t)$ on the ELVISmx Oscilloscope.

3. Use the oscilloscope cursors to measure the output voltage $y(t)$ at the times 25, 50, and 60 ms.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`

Chapter 11

Fourier Analysis Techniques

11.1 Fourier Series Representation (11-2)

Consider the voltage waveform $v(t)$ shown in Figure 11.1 on the following page.

1. Determine if the waveform has dc, even, or odd symmetry.

2. Obtain its cosine/sine Fourier series representation.

3. Convert the representation to amplitude format and plot the amplitude line spectrum for n = 0 to 5 using $A = 10$ volts and $T = 4$ ms.

NI Multisim Measurements

1. Create the voltage waveform $v(t)$ of Figure 11.1 on the next page with a PIECEWISE_LINEAR_VOLTAGE source. Use the same amplitude and period as in the problem statement.

2. Plot and tabulate the amplitude line spectrum of $v(t)$ with a Simulate → Analyses → Fourier Analysis :

   (a) Set the "Frequency Resolution (fundamental frequency)" parameter to match the fundamental frequency f_0 of the voltage waveform $v(t)$.

   (b) Leave the remaining parameters at their default settings.

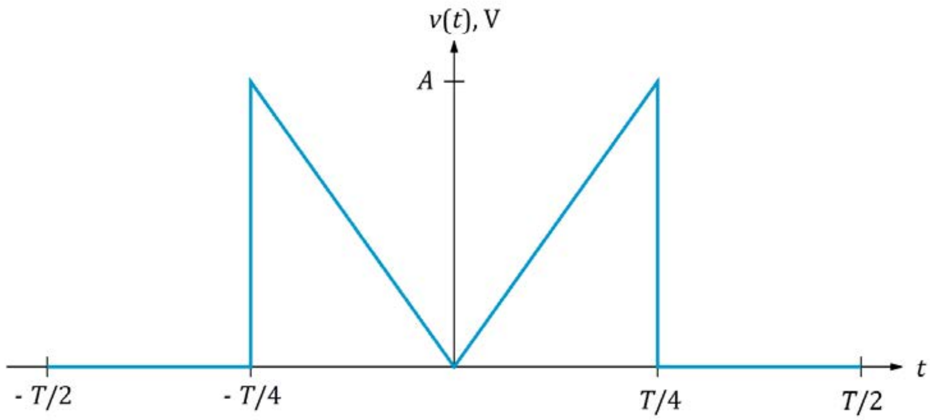


Figure 11.1: Voltage waveform for Problem 11.1

NI Multisim video tutorials:

- Find commonly-used circuit components:
  `http://youtu.be/G6ZJ8C0ja9Q`
- Piecewise linear (PWL) voltage source:
  `http://youtu.be/YYU5WuyebD0`

NI myDAQ Measurements

1. Connect myDAQ Analog Output 0 to Analog Input 0, i.e., AO0 to AI0+ and AGND to AI0-.
2. Create the voltage waveform $v(t)$ with the ELVISmx Arbitrary Waveform Generator using the same amplitude and period as in the problem statement. Set the sampling frequency to 200 kS/s.
3. Plot the power spectrum of $v(t)$ on the ELVISmx Dynamic Signal Analyzer (DSA). *Carefully* adjust the panel controls to match the following settings:
   - Input Settings:
     (a) Source Channel = `AI0`
     (b) Voltage Range = `+/-10V`
   - FFT Settings:

(a) Frequency Span = `10000`
(b) Resolution (lines) = `400`
(c) Window = `None`

- Averaging:

  (a) Mode = `RMS`
  (b) Weighting = `Exponential`
  (c) Number of Averages = `5`

- Frequency Display:

  (a) Units = `Linear`
  (b) Mode = `Peak`

- Scale Settings:

  (a) Scale = `Auto`

- Cursor Settings:

  (a) Cursors On = enabled
  (b) Cursor Select = `C1`

4. Measure the amplitude spectrum for $n = 0$ to 5 using Cursor 1; take the square root of the displayed cursor value "`dVpk^2`" to obtain the voltage amplitude. **IMPORTANT**: Position Cursor 2 *between* a pair of spectral lines to set its measured value to zero; the value displayed as `dVpk^2` is the *difference* between the two cursors and you want Cursor 2 to serve as the zero reference.

Additional helpful tips:

- Use the cursor position buttons to make fine adjustments in the vicinity of a spectral line; these are the pair of gray diamonds at the bottom center of the DSA.

- Double-click the upper limit value on the horizontal frequency axis and select a lower value to zoom in on the lower-frequency spectral lines. Do *not* change the "Frequency Span" value for this purpose because this changes the measurement itself.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`

Further Exploration with NI myDAQ

The ELVISmx Digital Signal Analyzer (DSA) represents a sophisticated instrument that performs a wide variety of frequency-domain measurements. Experiment with the settings and discuss your findings:

- Averaging:
  1. Choose Mode = `Peak Hold` and note that the "Restart" button lower down becomes active. Also try Mode = `None`.
  2. Number of Averages: Try different values including 0.
- Frequency Display:
  1. Choose Units = `dB`; what advantage do you see in a logarithmic display compared to a linear display?

The "FFT Settings" control the *Fast Fourier Transform* computation that serves as the heart of the DSA. These critical settings must be carefully selected to obtain correct amplitude spectrum measurements of periodic signals. First learn how the DSA takes a measurement and then experiment with the settings in a moment.

The DSA repetitively captures a snapshot of the input signal with duration "Resolution (lines)" (R) divided by "Frequency Span" (f_{span}); this time-domain record appears below the frequency display. Take a moment to calculate this time duration from your current DSA FFT settings and confirm that the value does indeed match the upper limit of the time-domain plot.

When measuring a periodic signal the captured time-domain signal *must* contain an *integer multiple* of periods, consequently R/f_{span} divided by the signal period T must be an integer N. Since the periodic signal frequency f_0 is $1/T$, the frequency span may be readily calculated as

$$f_{\text{span}} = \frac{Rf_0}{N}, \tag{11.1}$$

where f_{span} is the frequency span in Hz, R is the resolution in "lines" (sample points), f_0 is the fundamental frequency of the periodic input signal in Hz, and N is the number of periods captured. $N = 10$ cycles provides a good starting point for most measurements.

Now, return the DSA settings to match those of your earlier work in the NI myDAQ section of this problem. Calculate the value of N. Also calculate the values of N that result from choosing the other available values for resolution R (the DSA offers a total of five resolutions). Change the DSA FFT resolution to each of the other available values, and note the effects on the frequency spectrum display and on the time-domain display. In particular, note the degree to which the amplitude line spectral values change.

Return the resolution to $R = 400$ lines. Calculate the frequency span f_{span} for $N = 10.5$, i.e., for a time-domain record that contains ten periods with a half-period tacked onto the end. Enter this value into the DSA and note the degree to which the amplitude line spectral values change.

11.2 Circuit Applications (11-3)

The sawtooth voltage waveform $v_s(t)$ shown in Figure 11.2 with $A = 5$ V and $T = 2$ ms serves as the input to the circuit of Figure 10.2 on page 138.

1. Determine the Fourier series representation of $v_o(t)$.
2. Plot $v_o(t)$ and $v_s(t)$ with MathScript or MATLAB as follows:
   (a) Time $0 \leq t \leq 5$ ms,
   (b) Sum of $n_{\max} = 100$ terms, and
   (c) Circuit components $R = 5.6$ kΩ and $C = 0.1$ μF.

   Use sufficient time resolution to display Gibbs phenomenon ringing.
3. Measure the maximum value of $v_o(t)$ from the plot, and also measure the first time at which the maximum value occurs after $t = 0$.

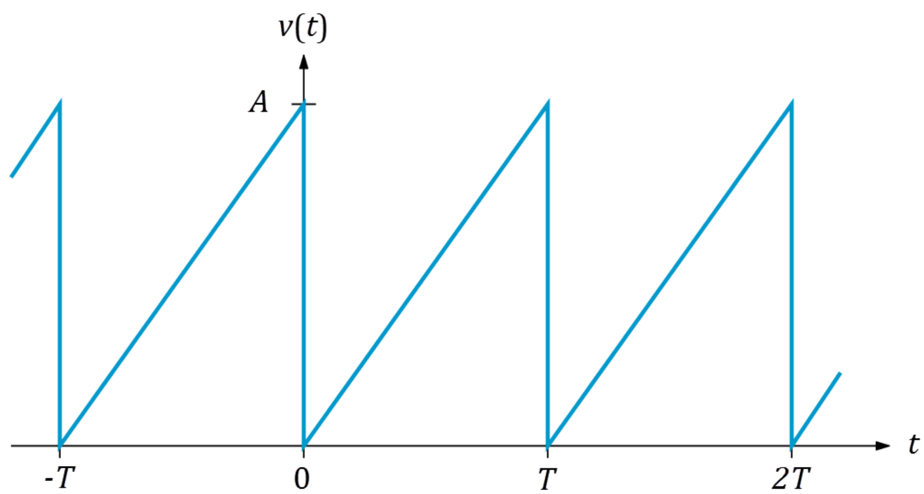


Figure 11.2: Voltage waveform for Problem 11.2

NI LabVIEW video tutorials:

- Plot two functions of time:
  `http://youtu.be/XQlAai1-YVc`
- Take cursor measurements on a plot:
  `http://youtu.be/bgK1p5O6OXc`

NI Multisim Measurements

1. Enter the circuit of Figure 10.2 on page 138 using the same component values listed in the problem statement. Drive the circuit input with a `TRIANGULAR_VOLTAGE` source configured with the amplitude and period specified in the problem statement.
2. Plot $v_\mathrm{o}(t)$ and $v_\mathrm{s}(t)$ using Simulate → Analyses → Transient analysis. Extend the plot time to 7 ms to allow the output to reach AC steady-state. Increase the minimum number of time points as needed to produce a smooth plot.
3. Measure the maximum value of $v_\mathrm{o}(t)$ with the Grapher View cursors, and also measure the first time at which the maximum value occurs after $t = 0$; ignore the transient start-up behavior during the first period.

NI Multisim video tutorials:

- Find commonly-used circuit components:
  `http://youtu.be/G6ZJ8C0ja9Q`
- Plot time-domain circuit response with Transient Analysis:
  `http://youtu.be/waKnad_EXkc`

NI myDAQ Measurements

1. Build the circuit of Figure 10.2 on page 138 using the same component values listed in the problem statement. Drive the circuit input with AO0 and use the ELVISmx Arbitrary Waveform Generator to produce the sawtooth waveform of Figure 11.2 on the facing page with the amplitude and period specified in the problem statement. Monitor the input voltage $v_\mathrm{s}(t)$ with AI0 and the output voltage $v_\mathrm{o}(t)$ with AI1.
2. Display $v_\mathrm{s}(t)$ and $v_\mathrm{o}(t)$ on the ELVISmx Oscilloscope.
3. Use the oscilloscope cursors to measure the maximum value of $v_\mathrm{o}(t)$ and the first time at which the maximum value occurs after $t = 0$.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`

11.3 Fourier Transform (11-5)

1. Determine the Fourier transform of the rectangular pulse $f(t)$ shown in Figure 11.3.

2. Plot the amplitude spectrum $|\mathbf{F}(\omega)|$ with MathScript or MATLAB as follows:

   (a) Frequency $0 \leq f \leq 4000$ Hz (remember to convert angular frequency ω to oscillation frequency f),

   (b) $A = 10$, and

   (c) τ = 1, 2, and 4 ms (create three distinct plots).

3. Determine the frequency at which the first null occurs in each of the three plots.

4. Discuss the relationship between the rectangular pulse width and the width of the main lobe of the amplitude spectrum.

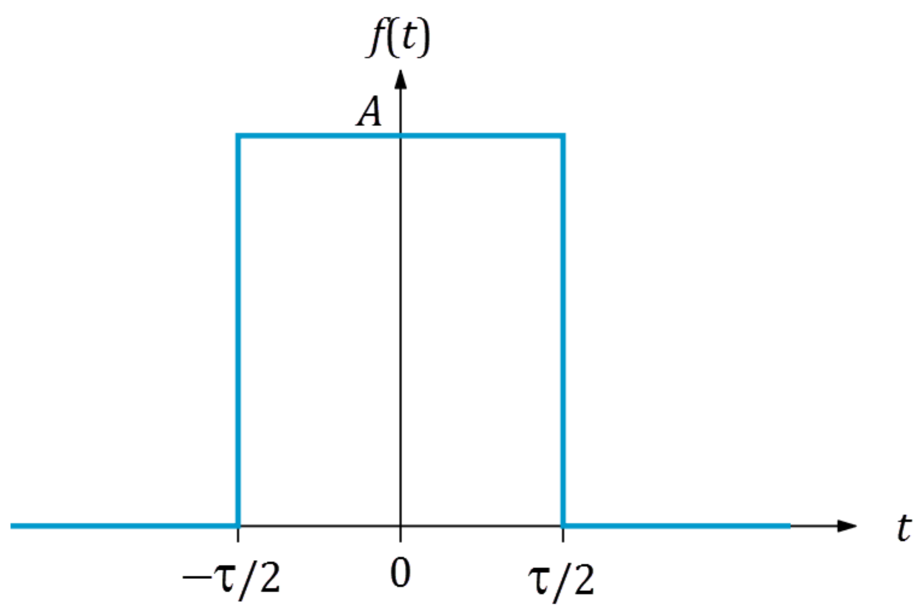


Figure 11.3: Rectangular pulse waveform for Problem 11.3

NI Multisim Measurements

1. Create a circuit with a PULSE_VOLTAGE source. Set the "Period" to 100 ms; set the remaining parameters as needed to create the pulse shown in Figure 11.3 on the facing page with $A = 10$. Note that the pulse must shift right to begin at $t = 0$; this shift does not affect the amplitude spectrum.

2. Plot the amplitude line spectrum of $f(t)$ with a Simulate → Analyses → Fourier Analysis for τ = 1, 2, and 4 ms (create three plots). Set the following parameter values:

   - "Frequency Resolution (fundamental frequency)" = 10 Hz,
   - "Number of harmonics" = 400,
   - "Display" = "Graph," and
   - "Vertical scale" = "Linear."

3. Determine the frequency at which the first null occurs in each of the three plots.

NI Multisim video tutorials:

- Find commonly-used circuit components:
  http://youtu.be/G6ZJ8C0ja9Q

NI myDAQ Measurements

1. Set up your myDAQ and ELVISmx instruments as follows:

   (a) Connect myDAQ Analog Output 0 to Analog Input 0, i.e., AO0 to AI0+ and AGND to AI0-.

   (b) Create the rectangular pulse $f(t)$ with the ELVISmx Function Generator in squarewave mode. Set the frequency to 10 Hz. Adjust the amplitude and DC offset controls to match the pulse waveform shown in Figure 11.3 on the preceding page with $A = 10$. Control the pulse width with the "Duty Cycle" control.

2. Plot the power spectrum of $f(t)$ on the ELVISmx Dynamic Signal Analyzer (DSA) for τ = 1, 2, and 4 ms (create three plots). Adjust the panel controls to match the following settings:

   - FFT Settings:

   (a) Frequency Span = `4000`
   (b) Resolution (lines) = `400`
   (c) Window = `None`

- Averaging:
   (a) Mode = `None`
- Frequency Display:
   (a) Units = `Linear`
   (b) Mode = `Peak`
- Scale Settings:
   (a) Scale = `Auto`

3. Determine the frequency at which the first null occurs in each of the three plots.

Further Exploration with NI myDAQ

1. Frequency spectrum plots normally possess much higher *dynamic range* than their corresponding time-domain plots. Review the DSA plots you created for the widest rectangular plot ($\tau = 4$ ms), especially the side lobe amplitudes beyond the first null frequency. Note how their values appear quite small compared to the amplitude of the main lobe. Now set the "Frequency Display" units to "dB" (decibels); you can stabilize the display by setting the "Scale Settings" from "Auto" to "Manual." Discuss the merits of a logarithmic display scale compared to a linear display scale.

2. To further experience the advantages of a logarithmic display, repeat the experiment with a single sinusoidal component. Set the function generator to sinusoidal mode at 500 Hz and remove the DC offset. Display the spectrum with "Linear" units and then with "dB" units. Stabilize and improve the measurement by setting the "Averaging" mode to "RMS." Discuss your observations.

11.4 Circuit Analysis with Fourier Transform (11-8)

The circuit of Figure 11.4 is excited by the double-pulse waveform shown in the same figure.

1. Derive the expression for $v_o(t)$ using Fourier analysis.
2. Plot $v_s(t)$ and $v_o(t)$ on the same graph over the time span $0 \leq t \leq 5$ ms with MathScript or MATLAB for the following values: $A = 5$ volt, $T = 1$ ms, $R = 5.6$ kΩ, and $C = 0.1$ μF.
3. Determine the value of $v_o(t)$ at times $t = 2$ ms and $t = 3$ ms.

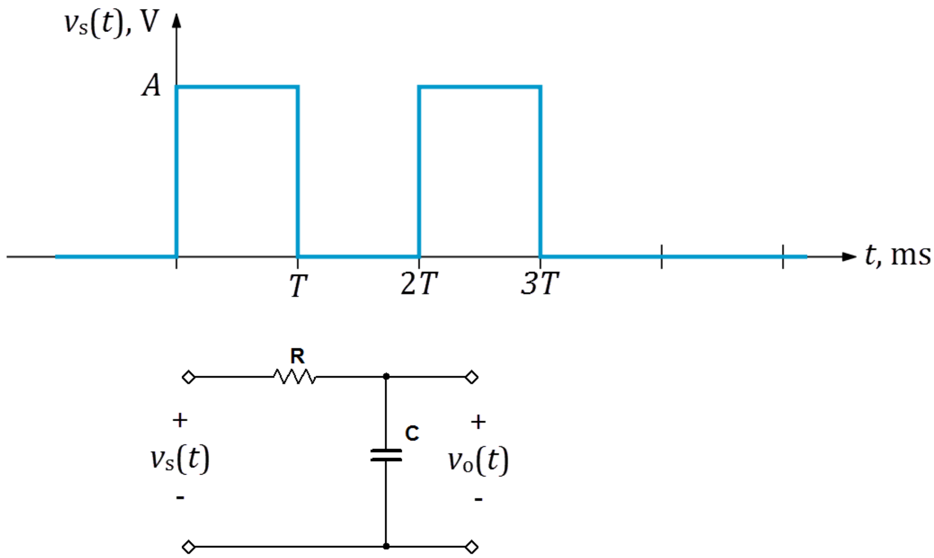


Figure 11.4: Double-pulse waveform and circuit for Problem 11.4

NI Multisim Measurements

1. Enter the circuit of Figure 11.4 using an `ABM_VOLTAGE` source and the component values specified in the problem statement. Build the ABM voltage source "Voltage Value" string with multiple step functions `u(TIME)`. Add, subtract, and delay the step functions as needed to create the double-pulse waveform of Figure 11.4 with the parameters specified in the problem statement.

2. Set up a Simulate → Analyses → Transient to plot $v_s(t)$ and $v_o(t)$ over the time span $0 \leq t \leq 5$ ms. Choose "Set to zero" for the "Initial Conditions" parameter to properly model a discharged capacitor before the first pulse occurs. Increase the minimum time step as necessary to obtain a smooth plot at discontinuities.

3. Use the Grapher View cursors to measure the value of $v_o(t)$ at times $t = 2$ ms and $t = 3$ ms.

NI Multisim video tutorials:

- Plot time-domain circuit response with Transient Analysis:
  http://youtu.be/waKnad_EXkc
- ABM (Analog Behavioral Model) voltage source:
  http://youtu.be/8pPynWRwhO4

NI myDAQ Measurements

1. Build the circuit of Figure 11.4 on the previous page using the component values specified in the problem statement. Drive the circuit with the ELVISmx Arbitrary Waveform Generator on AO0. Create the double-pulse waveform of Figure 11.4 on the preceding page with the parameters specified in the problem statement. Monitor $v_s(t)$ on AI0 and $v_o(t)$ on AI1.

2. Display $v_s(t)$ and $v_o(t)$ on the ELVISmx Oscilloscope. Adjust the settings for a time span of 5 ms and an appropriate scale (volts per division) to clearly see the output trace behavior. Adjust the triggering level and horizontal position to place the leading edge of the first pulse at the far left of the display. You will likely need to use the "Run Once" acquisition mode to obtain a stable display; click "Run" repeatedly until you obtain a satisfactory display.

3. Use the oscilloscope cursors to measure the value of $v_o(t)$ at times $t = 2$ ms and $t = 3$ ms.

NI myDAQ video tutorials:

- Arbitrary Waveform Generator (ARB):
  http://decibel.ni.com/content/docs/DOC-12941

Appendix A

Parts List

Resistors

The following resistors are standard-value 5% tolerance 1/4 watt carbon film devices. All listed resistors are available in resistor kits from Digi-Key, Jameco, and RadioShack:

Resistor Kit Description	Supplier	Part #
365 pcs, 5 ea of 1.0Ω to 1.0MΩ	Digi-Key	RS125-ND
540 pcs, 30 values, 10Ω to 10MΩ	Jameco	103166
500 pcs, 64 values, 1.0Ω to 10MΩ	RadioShack	271-312

See *Resistor Color Codes* at `http://www.allaboutcircuits.com/vol_5/chpt_2/1.html` to learn how to read the color bands on carbon film resistors.

Qty	Value (Ω)	Color Code
1	10	Brown - Black - Black
2	100	Brown - Black - Brown
1	330	Orange - Orange - Brown
1	470	Yellow - Violet - Brown
1	680	Blue - Gray - Brown

Qty	Value (kΩ)	Color Code
4	1.0	Brown - Black - Red
1	1.5	Brown - Green - Red
1	2.2	Red - Red - Red
2	3.3	Orange - Orange - Red
1	4.7	Yellow - Violet - Red
4	5.6	Green - Blue - Red
2	10	Brown - Black - Orange
1	15	Brown - Green - Orange
1	22	Red - Red - Orange
1	33	Orange - Orange - Orange
2	47	Yellow - Violet - Orange
1	100	Brown - Black - Yellow

Potentiometers

The following potentiometers (variable resistors) are $3/8$-inch square single-turn trimming style devices with $1/2$-watt power rating.

Qty	Description	Digi-Key #
1	100Ω trimpot (Bourns 3386P-1-501LF)	3386P-101LF-ND
1	1K trimpot (Bourns 3386P-1-103LF)	3386P-102LF-ND
2	10K trimpot (Bourns 3386P-1-103LF)	3386P-103LF-ND

Capacitors

Qty	Value (μF)	Type
1	0.047	Ceramic
1	0.01	Ceramic
2	0.1	Ceramic
1	1	Electrolytic
1	10	Electrolytic

Inductors

Qty	Description	Digi-Key #
1	3.3-mH inductor (Murata Power Solutions 22R335C)	811-1295-ND
1	33-mH inductor (Murata Power Solutions 22R336C)	811-1294-ND

Active Devices and Integrated Circuits

NOTE: Texas Instruments offers free samples. Go to http://www.ti.com and click "Sample & Buy" to get started.

Qty	Description	Digi-Key #
3	LM317L voltage regulator, 100mA (Texas Instruments)	296-17221-1-ND
1	TL072CP dual op amp (Texas Instruments)	296-1775-5-ND
1	DG413DJZ quad analog switch (Intersil)	DG413DJZ-ND

Breadboard

Circuit Specialists part number WB-102, http://www.circuitspecialists.com/prod.itml/icOid/6885

Jumper Wire Kit

Circuit Specialists part number WK-1 (350 pieces, pre-formed, 22 AWG), http://www.circuitspecialists.com/prod.itml/icOid/6920

Circuit Specialists part number MJW-70B (140 pieces, pre-formed, 22 AWG), http://www.circuitspecialists.com/prod.itml/icOid/7590

Test Leads

Alligator clip style, cut in half with tinned ends.

Circuit Specialists part number M000F0003, http://www.circuitspecialists.com/prod.itml/icOid/7682

Appendix B

LM317 Voltage and Current Sources

The Texas Instruments LM317 adjustable voltage regulator is a flexible device that when combined with suitable external resistors and the NI myDAQ power supply can serve as the basis for a fixed or variable voltage source and a fixed or variable current source. Figure B.1 shows the LM317 package terminals as well as its schematic symbol. The LM317 sources current up to 1.5 amps, while the LM317L sources up to 100 mA. See the datasheets available at `http://www.ti.com`; enter "lm317" in the "Search by Part Number" field.

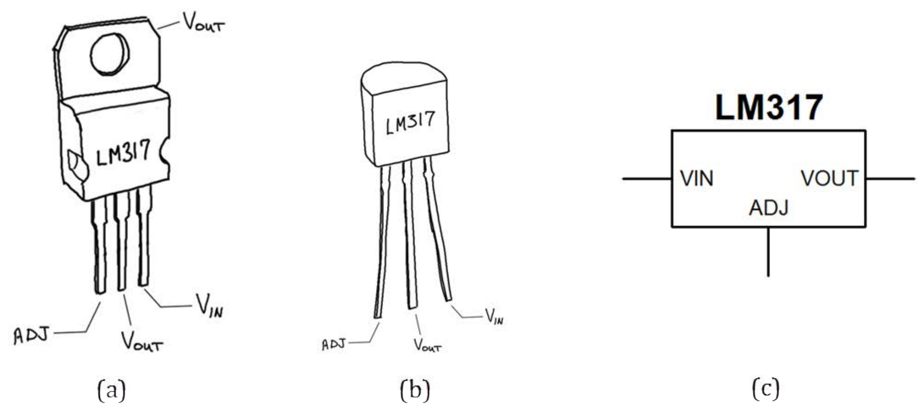


Figure B.1: LM317 adjustable voltage regulator: (a) package and terminals for 1.5-amp device (TO-220 package), (b) package and terminals for 100-mA device (TO-92 package), and (c) schematic symbol.

B.1 Variable Voltage Source

The circuit shown in Figure B.2 produces a variable voltage in the range 1.5 V to 13.5 V from the NI myDAQ +15V power supply. Figure B.3 on the next page shows the recommended breadboard layout for this circuit. Use bare-wire loops to facilitate easy connections with test leads to the NI myDAQ ±15-volt dual power supply. The horizontal voltage "rails" follow the top-to-bottom order of high to low voltage: +15 volts, variable voltage, ground, and -15 volts. Build this circuit on the left edge of your breadboard and leave it in place for all of your circuits projects.

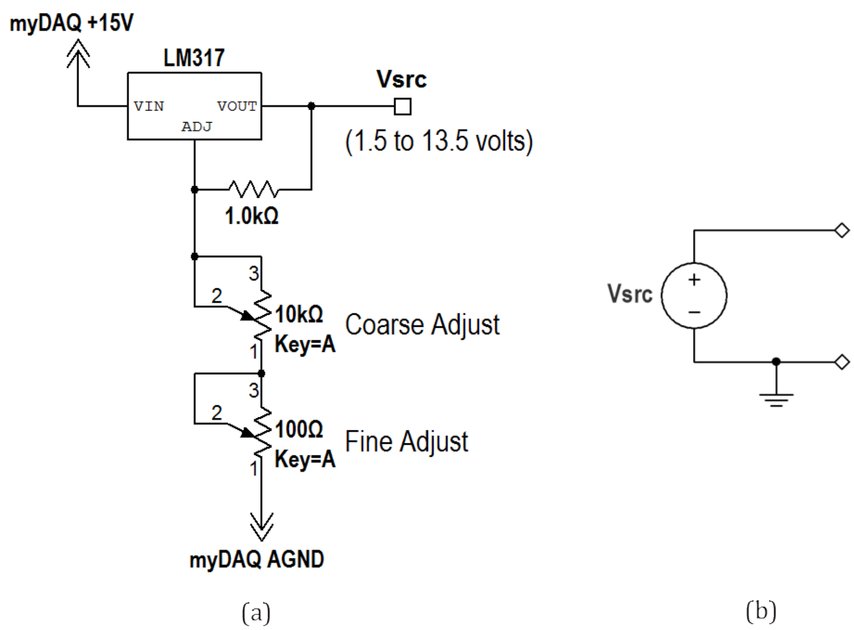
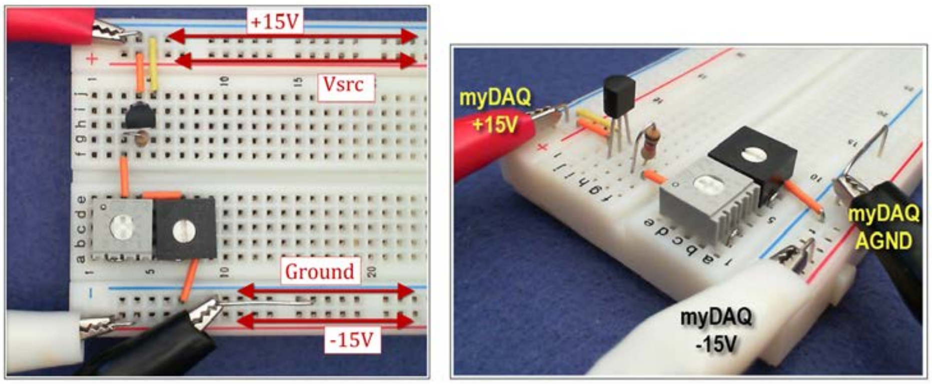


Figure B.2: LM317 as a variable voltage source: (a) schematic diagram and (b) equivalent circuit.

B.2 Current Source

The circuit shown in Figure B.4 on page 166 produces a current whose value is approximately $1250/R$ mA. This circuit configuration "sources" current



(a) (b)

Figure B.3: Recommended breadboard layout for the LM317-based variable voltage source: (a) top view of breadboard showing the component layout and voltage rail order, and (b) side view showing test lead connections between NI myDAQ and wire loops on the breadboard.

from the NI myDAQ +15V power supply and effectively operates as a current source with one terminal permanently attached to the NI myDAQ analog ground AGND.

The current source will operate as expected for circuits powered by the NI myDAQ ± 15 V dual power supply provided the following conditions hold:

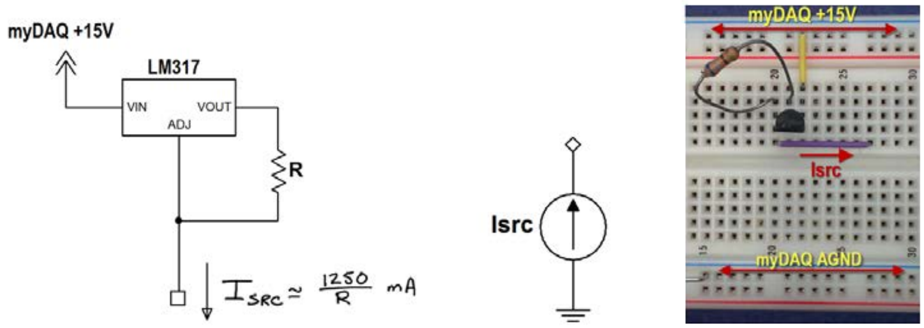
1. The requested current does not exceed the 30 mA current limit of the NI myDAQ +15V power supply,

2. The voltage of the ungrounded current source terminal does not rise higher than 13.5 V above ground, and

3. The current set resistor R does not exceed approximately 1.2 kΩ (the minimum current I_{SRC} is approximately 1 mA).

Figure B.5 on page 167 illustrates a similar current source that "sinks" current to the NI myDAQ -15 V power supply. The current source will operate as expected for circuits powered by the NI myDAQ ± 15 V dual power supply provided the following conditions hold:

1. The requested current does not exceed the 30 mA current limit of the NI myDAQ -15 V power supply,

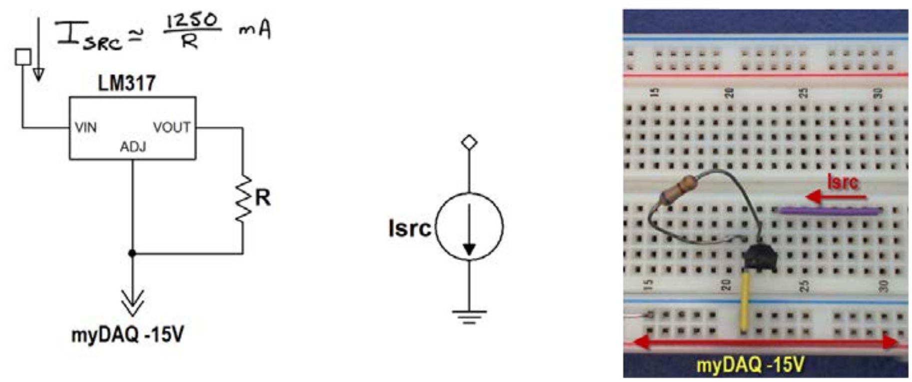


(a) (b) (c)

Figure B.4: LM317 adjustable voltage regulator as a grounded current source sourcing current from the NI myDAQ +15V power supply: (a) schematic diagram, (b) equivalent circuit model, and (c) recommended layout with the standard breadboard layout of Figure B.3 on the preceding page.

2. The voltage of the ungrounded current source terminal does not fall lower than 13.5 V below ground, and

3. The current set resistor R does not exceed approximately 1.2 kΩ (the minimum current I_{SRC} is approximately 1 mA).



(a) (b) (c)

Figure B.5: LM317 adjustable voltage regulator as a grounded current source sinking current to the NI myDAQ -15V power supply: (a) schematic diagram, (b) equivalent circuit model, and (c) recommended layout with the standard breadboard layout of Figure B.3 on page 165.

Appendix C

TL072 Operational Amplifier

The Texas Instruments TL072 dual operational amplifier ("op amp") provides two op amp devices in a single 8-pin package. For more details see the datasheet available at `http://www.ti.com`; enter "tl072" in the "Search by Part Number" field.

Figure C.1 on the following page shows the pinout diagram for the TL072. Note the requirement for a dual power supply; the NI myDAQ ± 15 V supply serves this purpose. Also note that the op amp device itself does not have a ground terminal. Instead the myDAQ AGND (analog ground) establishes the ground reference.

Figure C.2 on page 171 shows the TL072 placed on the standard breadboard layout described in Figure B.3 on page 165, connected to power, and ready for additional circuitry.

NI Multisim provides a circuit model for the TL072: place the `TL072ACP` device.

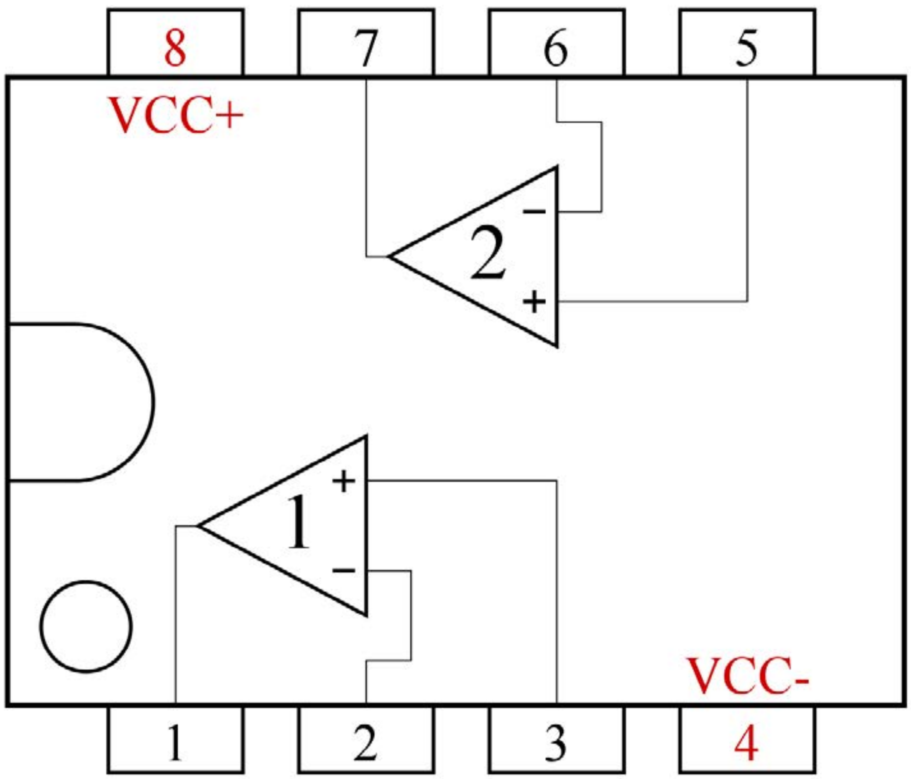


Figure C.1: Texas Instruments TL072 dual op amp pinout diagram (top view). The plastic package uses either a U-shaped cutout to indicate the left side or an indented circle to indicate pin 1.

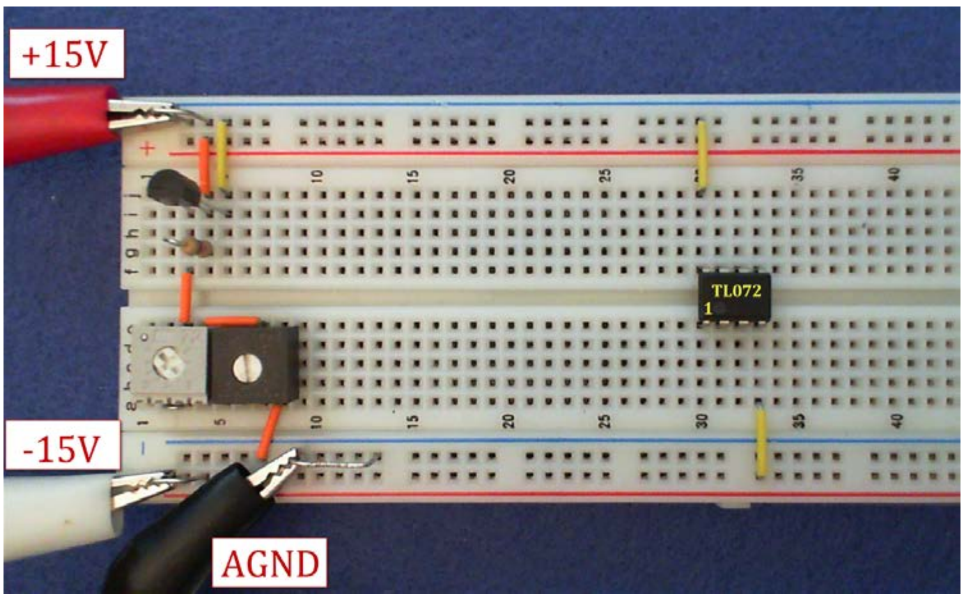


Figure C.2: Texas Instruments TL072 dual op amp placed on the standard breadboard layout, connected to power, and ready for additional circuitry.

Appendix D

DG413 Quad Analog Switch

The Intersil DG413 provides four analog switches; refer to Figure D.1 on the next page for the pinout diagram. Each switch is bidirectional and operates just like a physical SPDT switch but under the control of a digital (two-level) control signal. Two of the switches are normally-open and the other two switches are normally-closed. For more details see the datasheet available at `http://www.intersil.com`; enter "dg413" in the search box.

When connected and powered as shown in Figure D.2 on page 175 the switches operate as expected over the full ± 15 V range under the control of the myDAQ digital outputs DIO0 to DIO7. The analog outputs AO0 and AO1 may also serve as control signals provided they produce a voltage of either 0 V (inactive switch state) or 5 V (active switch state).

NI Multisim provides a circuit model for the DG413: place the `ADG413BN` device.

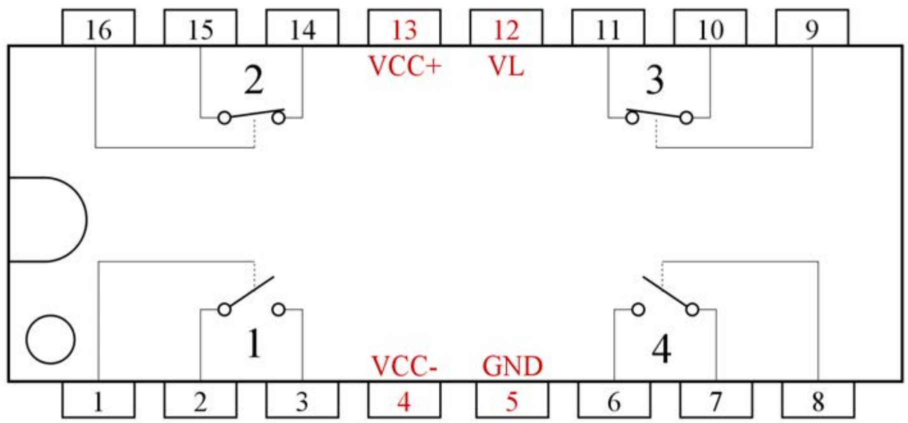


Figure D.1: Intersil DG413 quad analog switch pinout diagram (top view).Two of the switches are normally-open and the other two switches are normally-closed. The switch positions indicate the normal (inactive) state with the switch control voltage at low level. Switches 1 and 4 are normally-open (NO) and Switches 2 and 3 are normally-closed (NC).

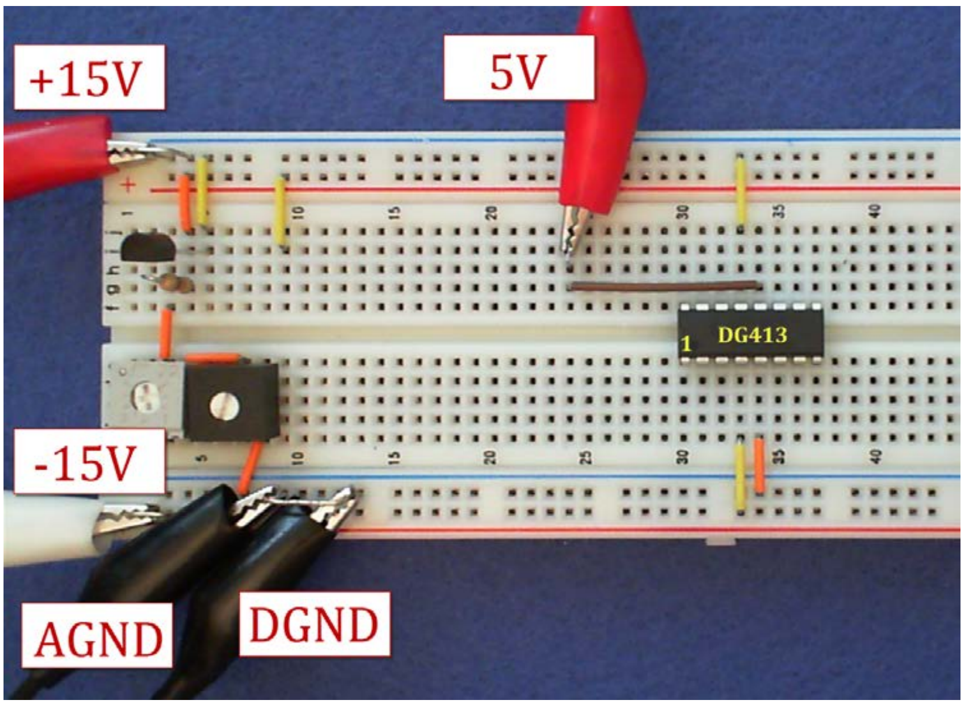


Figure D.2: Intersil DG413 quad analog switch placed on the standard breadboard layout, connected to power, and ready for additional circuitry.

Appendix E

Transient Response Measurement Techniques

E.1 Time Constant

The general first-order circuit response takes the form

$$x(t) = x(\infty) + [x(0) - x(\infty)]e^{-t/\tau}, t \geq 0, \tag{E.1}$$

where $x(0)$ is the initial value at the start of the transient, $x(\infty)$ is the final value, and τ is the time constant. Figure E.1 on the next page plots this equation for the case of a final value lower than the initial value. This figure also shows the *half-life* time T_{HL}, defined as the time interval from the onset of the transient at time $t = 0$ to the time at which $x(t)$ reaches the midpoint of the initial and final values. The half-life is easy to measure on an oscilloscope from which the time constant follows as

$$\tau = \frac{T_{\mathrm{HL}}}{\ln 2}. \tag{E.2}$$

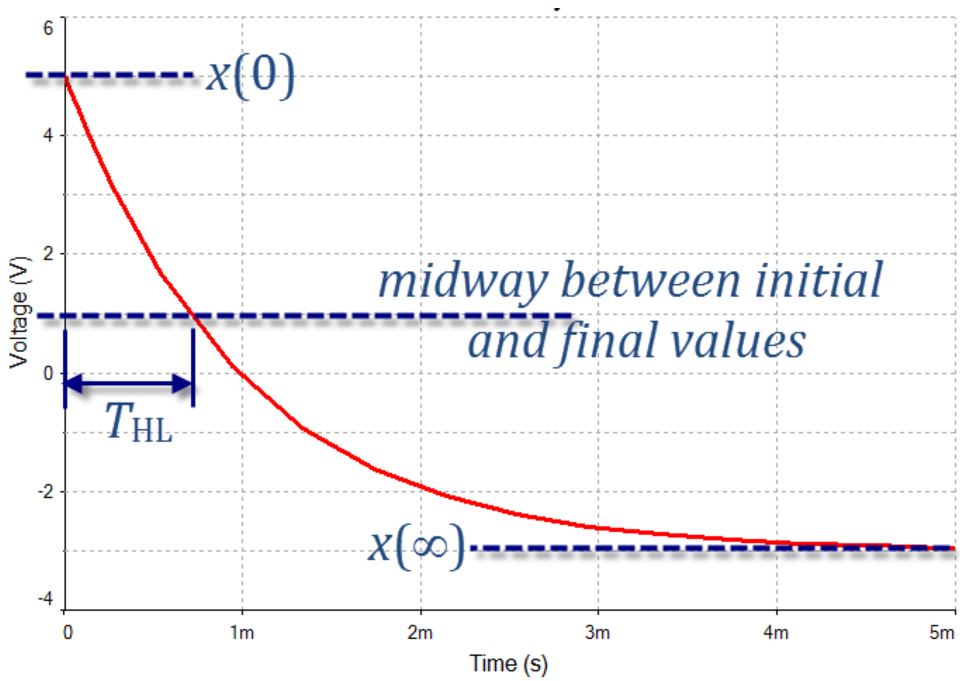


Figure E.1: General first-order circuit response showing initial value, final value, and half-life time.

Appendix F

Sinusoid Measurement Techniques

F.1 Amplitude and Phase Measurements

Figure F.1 shows a pair of sinusoidal signals as displayed on an oscilloscope. The "Reference" sinusoid serves as the phase reference for the "Signal of Interest" and has a phase of zero degrees. Both sinusoids oscillate at the same frequency f_0. Three measurements suffice to determine the phasor representation of each sinusoid as follows:

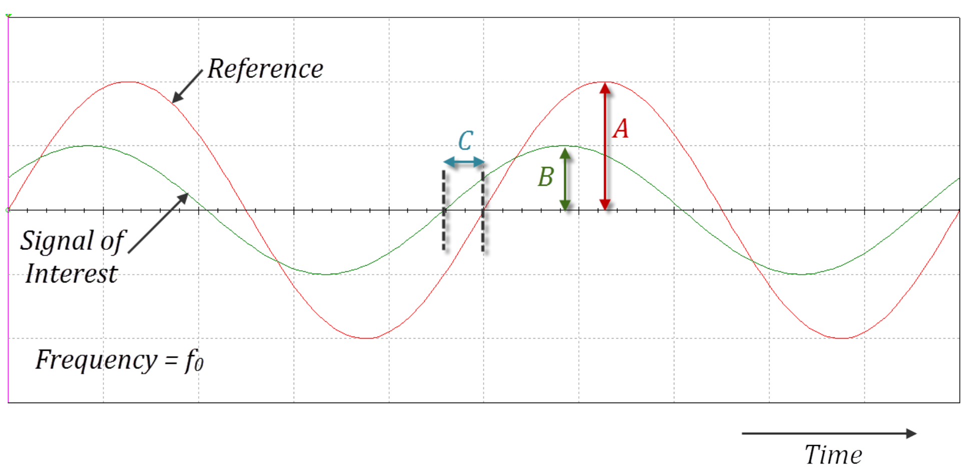


Figure F.1: A pair of sinusoidal signals at the same frequency, one as the reference and the other as the signal of interest. A and B indicate amplitude measurements and C indicates the time shift measurement for phase.

1. The reference sinusoid is $A\angle 0^\circ$,

2. The phase shift θ in degrees is $\pm\theta = C \times f_0 \times 360^\circ$ where C is the absolute value of the time shift between the two sinusoids in seconds and f_0 is sinusoidal frequency in Hz; choose positive sign when the signal of interest *leads* the reference (its zero crossing occurs before the reference as pictured in Figure F.1 on the previous page) and negative sign otherwise, and

3. The signal of interest is $B\angle\theta^\circ$.

Consider the NI ELVISmx Oscilloscope display of Figure F.2 on the facing page as an example of this measurement technique. Note that the "Volts per Division" scales have been adjusted to make both sinusoids fill as much of the screen as possible. The green trace on Channel 0 serves as the reference while the blue trace on Channel 1 is the signal of interest. The cursors have been adjusted to measure the time shift between the two signals. Note that the numerical display under the traces provides all necessary measurements:

1. The reference sinusoid amplitude is 1.696 volts divided by 2 (to convert from peak-to-peak) or 848 mV with a phase of zero degrees,

2. The phase shift is -65 μs $\times$ 2.502 kHz $\times$ 360° $= -59^\circ$; the phase is negative because the signal of interest (blue trace) lags the reference, i.e., the zero crossing occurs after the zero crossing of the reference, and

3. The amplitude of the signal of interest is 427 mV divided by 2 or 214 mV.

From these measurements the phasor form of the reference is $848\angle 0^\circ$ mV and the phasor form of the signal of interest is $214\angle -59^\circ$ mV.

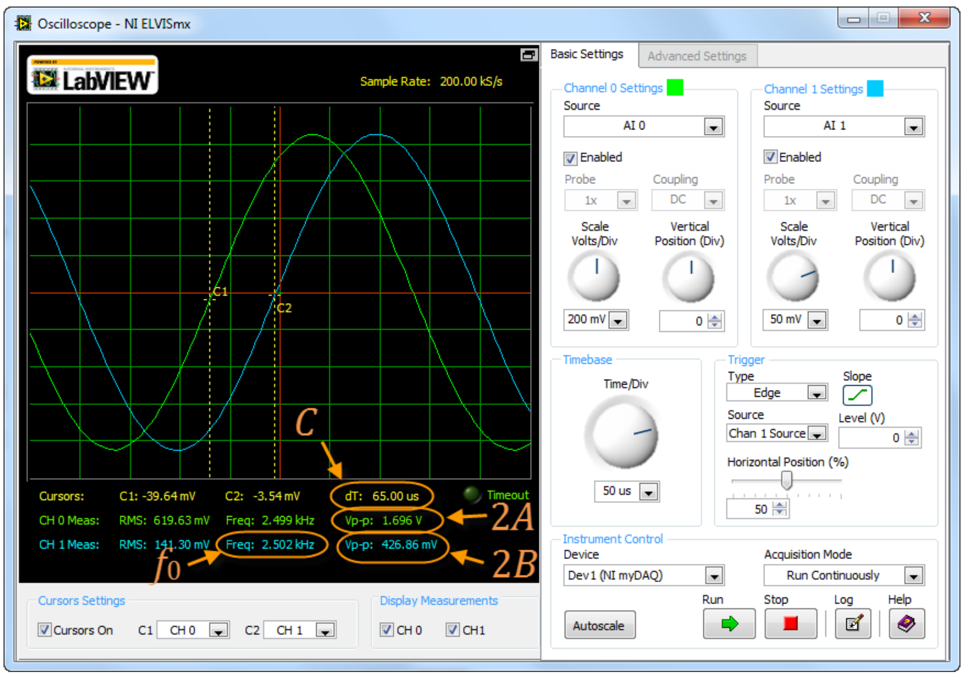


Figure F.2: A pair of sinusoidal signals at the same frequency measured by the NI ELVISmx Oscilloscope. After adjusting the cursors to measure time shift the numerical display provides all required information in the numerical display area.

Appendix G

Video Links

NI LabVIEW MathScript Video Tutorials

- Plot two functions of time:
  `http://youtu.be/XQlAai1-YVc`
- Take cursor measurements on a plot:
  `http://youtu.be/bgK1p5O6OXc`

NI Multisim & NI myDAQ Video Tutorials

- Compare simulated and physical DMM measurements:
  `http://youtu.be/MZiZ_C-ngkY`

NI Multisim Video Tutorials

Place components:

- Find commonly-used circuit components:
  `http://youtu.be/G6ZJ8C0ja9Q`
- Find components by name:
  `http://youtu.be/5wlFweh4n-c`

Sources:

- Function generator:
  `http://youtu.be/CeOl6EzD-_c`

- AC (sinusoidal) voltage source:
  `http://youtu.be/CXbuz7MVLSs`

- ABM (Analog Behavioral Model) voltage source:
  `http://youtu.be/8pPynWRwhO4`

- Pulse voltage source:
  `http://youtu.be/RdgxVfr28C8`

- Piecewise linear (PWL) voltage source:
  `http://youtu.be/YYU5WuyebD0`

- VDD and VSS power supply voltages:
  `http://youtu.be/XrPVLgYsDdY`

Measure DC current:

- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`

- Measure DC mesh current with a measurement probe:
  `http://youtu.be/lK0LcTNroXI`

- Measure DC current with an ammeter indicator:
  `http://youtu.be/8P4oFw6sIzQ`

Measure DC voltage:

- Measure DC voltage with a voltmeter:
  `http://youtu.be/XLyslyikUws`

- Measure DC voltage with a referenced measurement probe:
  `http://youtu.be/xKEQ3EXEaP8`

- Measure DC voltage with a voltmeter indicator:
  `http://youtu.be/8h2SAZ9gkBA`

- Set the digits of precision of a measurement probe:
  `http://youtu.be/GRO60XLgzHg`

Measure DC node voltage:

- Measure DC node voltage with a measurement probe:
  http://youtu.be/svNGHA2-uK4

- Find node voltages with DC Operating Point analysis:
  http://youtu.be/gXBCqP17AZs

Measure DC power:

- Measure DC power with a wattmeter:
  http://youtu.be/-axVClpMpiU

- Find resistor power with DC Operating Point analysis:
  http://youtu.be/NxXmVDW9spo

- Use a Parameter Sweep analysis to plot resistor power as a function of resistance:
  http://youtu.be/3k2g9Penuag

Measure resistance:

- Measure resistance with an ohmmeter:
  http://youtu.be/3G5V0Hxjkbg

Measure RMS and average value:

- Measure RMS and average value with a measurement probe:
  http://youtu.be/OnK-Unld17E

Measure AC phasor voltage:

- Measure phasor voltage with a Single Frequency AC Analysis:
  http://youtu.be/SwYCsoOwfUs

Measure frequency response:

- Measure frequency response with AC Analysis:
  http://youtu.be/tgCPDBtRcso

Measure AC power:

- Measure average power and power factor with a wattmeter:
  http://youtu.be/kYliPwbWInc

Net names:

- Display and change net names:
  http://youtu.be/0iZ-ph9pJjE

Grapher View and oscilloscope cursor measurements:

- Find the maximum value of trace in Grapher View:
  http://youtu.be/MzYK60mfh2Y
- Set cursor to a specific value:
  http://youtu.be/48sQja58I10

Oscilloscope:

- Basic operation of the two-channel oscilloscope:
  http://youtu.be/qnRK6QyqjvQ
- Waveform cursor measurements with the two-channel oscilloscope:
  http://youtu.be/snBRFq1Y1q4
- Distinguish oscilloscope traces by color:
  http://youtu.be/bICbjggcTiQ
- Stabilize the oscilloscope display with edge triggering:
  http://youtu.be/d69zYYSEG7E
- Basic operation of the four-channel oscilloscope:
  http://youtu.be/iUqs_c1Bc4Y

Transient response:

- Plot time-domain circuit response with Transient Analysis:
  http://youtu.be/waKnad_EXkc
- Voltage-controlled switch:
  http://youtu.be/BaEBjhD4TOw

Combined Multisim / myDAQ measurements:

- Combine Multisim simulation and myDAQ measurements in the same instrument – Bode Analyzer:
  `http://youtu.be/3UmTmUj4h1g`

NI myDAQ Video Tutorials

See *Electrical Circuits with NI myDAQ* for more video tutorials and projects:
`http://decibel.ni.com/content/docs/DOC-12654`

NI ELVISmx Instruments for NI myDAQ:

- DMM ohmmeter:
  `http://decibel.ni.com/content/docs/DOC-12938`
- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`
- Function Generator (FGEN):
  `http://decibel.ni.com/content/docs/DOC-12940`
- Arbitrary Waveform Generator (ARB):
  `http://decibel.ni.com/content/docs/DOC-12941`
- Oscilloscope:
  `http://decibel.ni.com/content/docs/DOC-12942`
- Bode Analyzer:
  `http://decibel.ni.com/content/docs/DOC-12943`
- Digital Reader (DigIn):
  `http://decibel.ni.com/content/docs/DOC-12944`
- Digital Writer (DigOut):
  `http://decibel.ni.com/content/docs/DOC-12945`

Measurement techniques:

- Measure current with a shunt resistor and DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12946`
- Measure node voltage:
  `http://decibel.ni.com/content/docs/DOC-12947`
- Increase current drive of analog output (AO) channels with an op amp voltage follower:
  `http://decibel.ni.com/content/docs/DOC-12665`