PROBLEMS & EXPLORATIONS IN MICROELECTRONICS WITH NI myDAQ AND MULTISIM

Ed Doering

Do Engineering

nts PRESS

Contents

0 Introduction **7**

0.1 Resources . . . 8
0.2 Textbook Cross Reference . . . 9
0.3 Goals for Student Deliverables . . . 9
0.4 Student Deliverables Checklist . . . 13
0.5 Acknowledgements . . . 14

1 Diode Circuits **15**

1.1 Ideal Diodes . . . 15
1.2 Temperature Sensor . . . 19
1.3 Graphical Analysis . . . 22
1.4 Series-Connected Diodes . . . 26
1.5 Zener Regulator . . . 30
1.6 Half-Wave Rectifier . . . 33
1.7 Another Half-Wave Rectifier . . . 38
1.8 Full-Wave Rectifier . . . 39
1.9 Peak Rectifier . . . 43
1.10 Limiter . . . 47

2 Operational Amplifier Circuits **51**

2.1 Comparator . . . 51
2.2 Inverting Amplifier . . . 55
2.3 Summing Amplifier . . . 59
2.4 Noninverting Amplifier . . . 63
2.5 Difference Amplifier . . . 67
2.6 Instrumentation Amplifier . . . 70
2.7 Integrator . . . 74
2.8 Differentiator . . . 78
2.9 Precision Rectifier . . . 83

2.10 Schmitt Trigger 87
2.11 Squarewave Oscillator 91

3 Bipolar Junction Transistor (BJT) Circuits 95
3.1 Logic Inverter 95
3.2 Current Source 100
3.3 Fixed Bias 104
3.4 Fixed Bias, PNP 108
3.5 Voltage-Divider Bias 112
3.6 Darlington-Pair Emitter Follower 115
3.7 Common-Emitter Amplifier 119
3.8 Current Mirror 123
3.9 Differential Pair 126
3.10 Class B Push-Pull Amplifier 129
3.11 Voltage-to-Current Converter 132

4 MOSFET Circuits 135
4.1 Voltage-to-Current Converter 135
4.2 Triode Operation 139
4.3 Switch 142
4.4 Feedback Bias 146
4.5 Voltage-Divider Bias 150
4.6 Common-Source Amplifier 154
4.7 Common-Drain Amplifier 158
4.8 Logic Inverters: RTL and CMOS 162

A Parts List 167

B LM317 Voltage and Current Sources 171
B.1 Variable Voltage Source 172
B.2 Current Source 172

C TL072 Operational Amplifier 177

D Diodes 181
D.1 1N4148 Switching Diode 181
D.2 Light-Emitting Diode (LED) 182
D.3 1N5232B 5.6 V Zener Diode 183

E Transistors 185
E.1 2N2222A NPN BJT 185
E.2 2N3904 NPN BJT 186
E.3 2N3906 PNP BJT 187
E.4 CA3083 NPN BJT Array 188
E.5 ZVN3306A N-Channel Enhancement MOSFET 190
E.6 ZVP3306A P-Channel Enhancement MOSFET 192

F Video Links 195

Chapter 0

Introduction

Practical experience remains a critical component of engineering education to reinforce fundamental objectives and concepts. Circuit simulators provide one way for electrical engineering students to gain hands-on experience with circuit behavior, especially real-time interactive simulators such as National Instruments (NI) Multisim. However, nothing replaces hands-on experience with real hardware. A traditional electronics laboratory helps students learn how to use instrumentation, take measurements, and observe circuits in action. However, this experience traditionally occurs only at a specific place and time. Recent advances in computer-based instrumentation coupled with low-cost USB-based data acquisition devices such as the NI myDAQ enable student learning to extend beyond the traditional laboratory setting to a students preferred environment and pace of learning.

Faculty wishing to incorporate portable instrumentation into their analog electronics courses will find in this text forty end-of-chapter-style problems covering basic concepts in diodes, operational amplifiers, bipolar junction transistors (BJTs), and enhancement-mode MOSFETs. Each problem guides students through a three-way solution process of analysis, simulation, and measurement. Solving a given problem three ways empowers students to triangulate on the correct result by mutual reinforcement while simultaneously improving their analytical and laboratory skills. Short video tutorials integrate closely with the problems to demonstrate relevant software and measurement techniques. Pilot tests of the three-way approach confirm that students who worked to harmonize each part of the their three-way solution elevated their confidence in their own analytical and lab skills.

Nearly every 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 and simulated-to-measurement results indicates the degree to which the student has achieved a correct solution. Students also begin to develop an appreciation for the differences that can emerge among the three methods due to analysis approximations, simulation model mismatch, and temperature effects.

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

0.1 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 173 should be constructed at the beginning of the term and left in place for subsequent circuits.

- Appendix C provides a pinout diagram and data sheet links for the Texas Instruments TL072 dual operational amplifier used in many of the circuits.

- Appendix D provides diagrams for the required diodes and links to vendor data sheets.

- Appendix E provides pinout diagrams for the required BJT and MOSFET transistors and links to vendor data sheets. This appendix also includes SPICE models and links to video tutorials detailing how to add several components to the NI Multisim "User Database" that are not included with version 11.0.

- Appendix F lists all of the available video links.

0.2 Textbook Cross Reference

Table 1 on the next page provides a cross reference between the forty problems of this text and corresponding sections in the following textbooks:

- Jaeger, Richard C. and Travis N. Blalock. *Microelectronic Circuit Design*, 4th ed., McGraw-Hill, 2011, ISBN 978-0-07-338045-2.
- Rashid, Muhammad H. *Microelectronic Circuits Analysis and Design*, 2nd ed., Cengage Learning, 2011, ISBN 978-0-495-66772-8.
- Razavi, Behzad. *Fundamentals of Microelectronics*, Wiley, 2008, ISBN 978-0-471-47846-1.
- Sedra, Adel S. and Kenneth C. Smith. *Microelectronic Circuits*, 6th ed., Oxford University Press, 2010, ISBN 978-0-19-532303-0.

0.3 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 on page 11 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

Table 1: Problems cross referenced to related sections of popular textbooks.

Problem	Jaeger/Blalock	Rashid	Razavi	Sedra/Smith
1.1 – Ideal Diodes	3.11	4.2	3.3	4.1
1.2 – Temperature Sensor	3.5	4.4.3	3.4	4.2
1.3 – Graphical Analysis	3.10	4.5.1		4.3
1.4 – Series-Connected Diodes	3.11	4.2	3.4	4.3.8
1.5 – Zener Regulator	3.12	4.7	3.5.2	4.4
1.6 – Half-Wave Rectifier	3.13	5.2.1	3.5.1	4.5.1
1.7 – Another Half-Wave Rectifier	3.13	5.2.1	3.5.1	4.5.1
1.8 – Full-Wave Rectifier	3.15	5.2.3	3.5.1	4.5.2
1.9 – Peak Rectifier	3.13	5.4	3.5.1	4.5.4
1.10 – Limiter		5.5	3.5.3	4.6.1
2.1 – Comparator	12.9	3.2	8.1	2.1
2.2 – Inverting Amplifier	10.9.1	3.4.2	8.2.2	2.2
2.3 – Summing Amplifier	10.9.5	3.5.6	8.2.4	2.2.4
2.4 – Noninverting Amplifier	10.9.3	3.4.1	8.2.1	2.3
2.5 – Difference Amplifier	10.9.6	3.5.3		2.4
2.6 – Instrumentation Amplifier	12.2	3.5.4		2.4.2
2.7 – Integrator	10.10.7	3.5.1	8.2.3	2.5.2
2.8 – Differentiator	10.10.8	3.5.2	8.2.3	2.5.3
2.9 – Precision Rectifier	12.8	16.2.3	8.3.1	4.5.5
2.10 – Schmitt Trigger	12.9	16.5.2		17.4
2.11 – Squarewave Oscillator	12.9.2	16.6		17.5
3.1 – Logic Inverter	9.10	8.6	4.5	6.2
3.2 – Current Source	15.4	8.7	5.2.1	6.3
3.3 – Fixed Bias	5.11	8.7	5.2.1	6.3
3.4 – Fixed Bias, PNP	5.11	8.7	5.2.5	6.3
3.5 – Voltage-Divider Bias	5.11	8.7	5.2.3	6.3
3.6 – Darlington-Pair Emitter Follower	5.7, 14.3	8.9, 8.12	5.2.3	6.6.6, 7.6.2
3.7 – Common-Emitter Amplifier	14.2	8.8	5.3.1	6.6.3
3.8 – Current Mirror	16.2	9.6.2, 9.6.6	9.2	7.5.2
3.9 – Differential Pair	15.1	9.7	10.2	8.3
3.10 – Class B Push-Pull Amplifier	15.3	11.5, 11.6	13.3-4	11.3-4
3.11 – Voltage-to-Current Converter		3.5.10	13.3	11.3
4.1 – Voltage-to-Current Converter	4.2	7.3	6.2	5.1
4.2 – Triode Operation	4.2	7.3	6.2	5.1
4.3 – Switch	4.2	7.6	6.2	5.3
4.4 – Feedback Bias	4.9	7.7	7.1	5.3, 5.7.3
4.5 – Voltage-Divider Bias	4.9	7.7	7.1	5.3, 5.7.2
4.6 – Common-Source Amplifier	14.2	7.8	7.2	5.8.2
4.7 – Common-Drain Amplifier	14.3	7.9	7.4	5.8.5
4.8 – Logic Inverters: RTL and CMOS	7.2	15.7	15.2	13.2

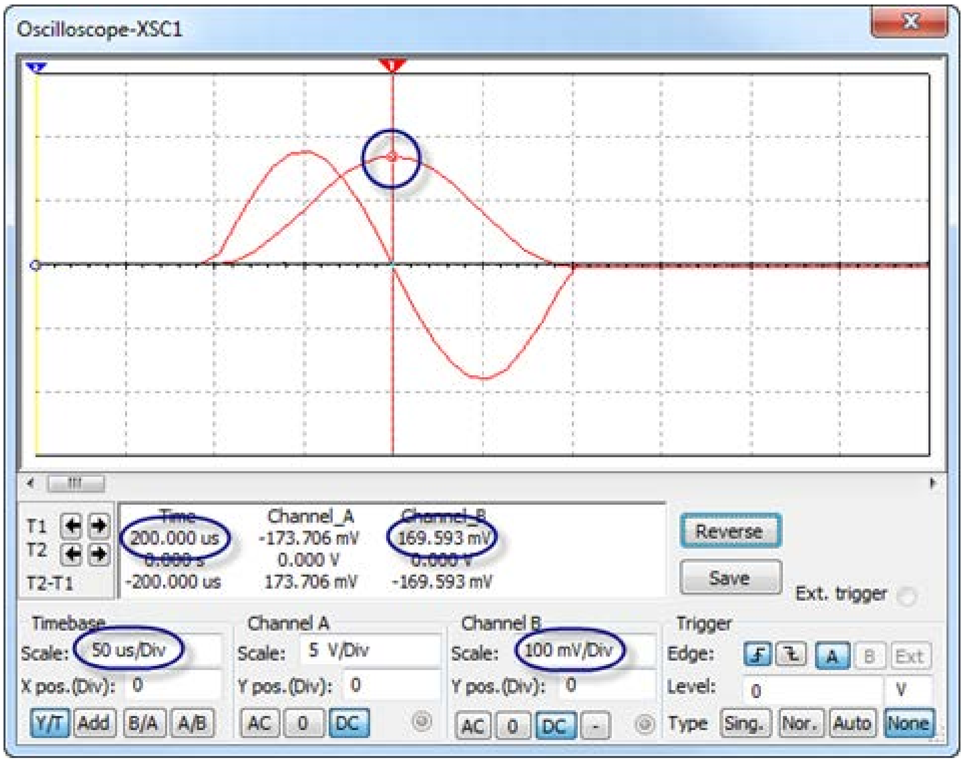


Figure 1: NI Multisim screenshot showing proper markings to indicate control settings adjusted away from default values as well as regions where measurement was obtained.

including units, and write any additional calculations that were necessary to reach an end result for simulation.

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 (analysis, simulation, and measurement) and tabulate three percentage differences for each requested numerical quantity:

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

Consider setting up a spreadsheet template to conveniently tabulate the measurements and calculate the percentage differences.

0.4 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

0.5 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,
- Mark Walters (National Instruments) for his helpful support of the NI Multisim product, and
- Tina Hudson (Rose-Hulman Institute of Technology) for her thoughtful suggestions and insights regarding analog electronics.

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

doering@rose-hulman.edu

Chapter 1

Diode Circuits

1.1 Ideal Diodes

Consider the diodes in the circuit of Figure 1.1 to be ideal.

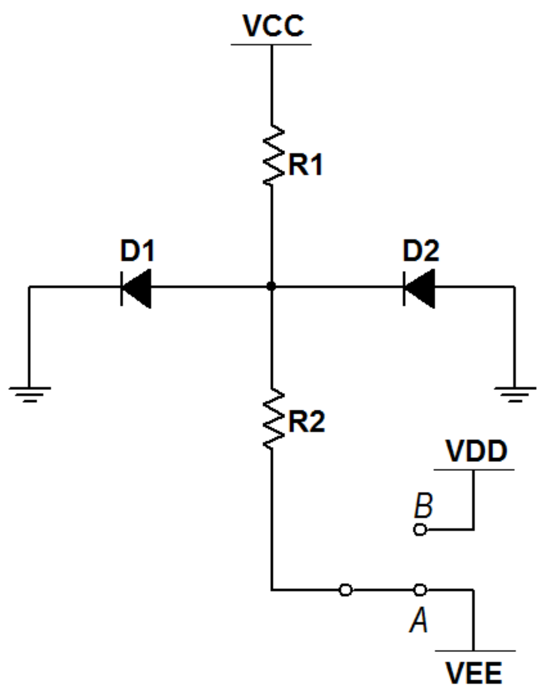


Figure 1.1: Circuit for Problem 1.1

Analysis

NOTE: A three-way numerical comparison is not required for this problem.

1. Determine the state of each diode (on or off) when the switch is in position "A."

2. Repeat when the switch in position "B."

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{DD}	+5 volts
V_{EE}	-15 volts
D_1 and D_2	ideal diode
R_1	2.2 kΩ
R_2	1.0 kΩ

Simulation with NI Multisim

Use the DIODE_VIRTUAL device for the diodes in the circuit of Figure 1.1 on the previous page; use the SPDT device for the switch. Place the VCC power supply and set its value to $+15$ V; similarly place VEE and VDD and set their values according the table in the problem statement.

1. Use interactive simulation to determine the state of each diode (on or off) when the switch is in position "A." Use a Simulate → Instruments → Measurement Probe to display the current entering the anode of each diode, and recall that zero or negligible current indicates the diode is off.

2. Repeat when the switch in position "B."

Additional Multisim tips for this problem:

- The triangle designates the anode side of the diode symbol. Think of the triangle as an arrow that signifies the preferred direction of current flow.

NI Multisim video tutorials:

- Find commonly-used circuit components:
  `http://youtu.be/G6ZJ8C0ja9Q`
- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- VCC and VEE power supply voltages:
  `http://youtu.be/XkZTwKD-WjE`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`

Measurement with NI myDAQ

Construct the circuit of Figure 1.1 on page 15 using two 1N4148 diodes and resistors with values that match the problem statement. The switch can simply be a direct connection to either -15V or 5V.

1. Determine the state of each diode (on or off) when the switch is in position "A." Use the ELVISmx DMM voltmeter to measure the voltage across each diode (red probe on the anode, black probe on the cathode), and recall that a positive voltage indicates the diode is on.

2. Repeat when the switch in position "B."

Additional myDAQ tips for this problem:

- Power the circuit with myDAQ +15V for V_{CC}, -15V for V_{EE}, and 5V for V_{DD}; use AGND for circuit ground.

- Use the 1N4148 diode described in Appendix D. Follow the diagram of Figure D.1 on page 181 to determine the anode and cathode connections; the diode's cathode marked by a black band corresponds to the bar on the diode symbol.

NI myDAQ video tutorials:

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

Further Exploration with NI myDAQ

Replace the 1N4148 diodes with two light-emitting diodes, or LEDs. Refer to the diagram of Figure D.2 on page 182 to determine the anode and cathode connections of an LED.

Visually observe the “on” and “off” states of the LEDs for the switch in each position. Compare your observation to the results of your circuit based on the 1N4148 diodes.

1.2 Temperature Sensor

A diode's forward voltage drop decreases by approximately 2 mV for each 1°C increase in its temperature, assuming constant diode current. The circuit of Figure 1.2 applies this property as the basis of a simple *temperature sensor*. The diodes must be of the same type, ideally from the same lot. Both diodes are forward-biased using the same resistor value to establish the same current. Diode D_{sensor} serves as the temperature sensor while diode D_{ref} serves as the reference – it is maintained at a constant temperature, with room temperature (25°C) being convenient. The difference in diode drops V_o is consequently proportional to temperature.

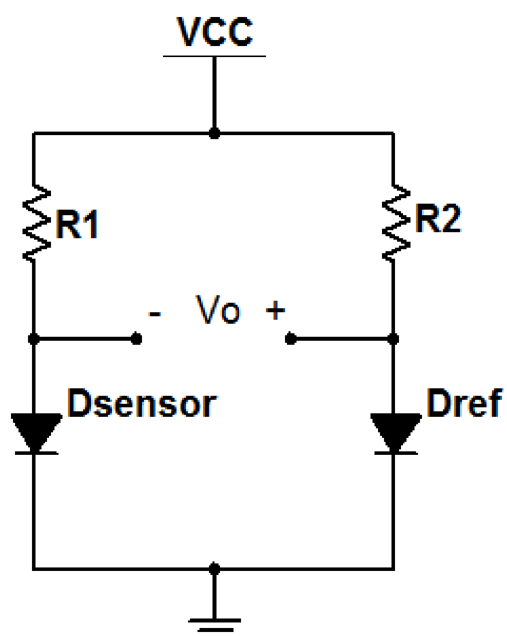


Figure 1.2: Circuit for Problem 1.2

Analysis

1. Write an equation for V_o as a function of temperature.
2. Plot V_o as a function of temperature over the range 0°C to 100°C.

3. Determine V_o at the following temperatures:
   (a) Freezing point of water
   (b) Room temperature (25°C)
   (c) Body temperature (98.6° F)
   (d) Boiling point of water
4. Determine the sensitivity of the temperature sensor output V_o in millivolts per °C.

Use the following circuit components:

Component	Value
D_{sensor} and D_{ref}	standard p-n junction diodes
R_1 and R_2	4.7K kΩ
V_{CC}	15 V

Simulation with NI Multisim

Enter the circuit of Figure 1.2 on the previous page using the `1N4148` diode model. Place the `VCC` power supply and set its value to +15 V.

1. Set up a Simulate → Analyses → Parameter Sweep to sweep the *temperature* of diode D_{sensor} over the range 0°C to 100°C with a 1°C increment. Select "DC Operating Point" for the "Analysis to sweep" option. Plot V_o as the difference of the two diode voltages.
2. Take cursor measurements to determine V_o at the four temperatures listed in the problem statement.
3. Determine the sensitivity of the temperature sensor output V_o in millivolts per °C.

Additional Multisim tips for this problem:

- Do *not* use the "Temperature Sweep" analysis because this sweeps the temperature of every device in the circuit.
- Name the nets connected to the diode anodes to make them easy to find in the "Parameter Sweep" dialog box. Alternatively, connect a Simulate → Instruments → Measurement Probe to each anode.

- Click “Add Expression” in the “Output” tab to form an expression that is the difference of the two diode voltages.

NI Multisim video tutorials:

- Find components by name:
  http://youtu.be/5wlFweh4n-c
- Waveform cursor measurements with the two-channel oscilloscope:
  http://youtu.be/snBRFq1Y1q4
- Display and change net names:
  http://youtu.be/0iZ-ph9pJjE
- Vary the temperature of one component with a Parameter Sweep:
  http://youtu.be/OfXXPmpdlEU

Measurement with NI myDAQ

Construct the circuit of Figure 1.2 on page 19 using two 1N4148 diodes. Connect AI0+ to the positive terminal of V_o and connect AI0- to the negative terminal of V_o. Use the ELVISmx Oscilloscope to monitor the value of V_o using the RMS indicator under the main display. Set the “Volts/Div” scale to its most sensitive value (10 mV) and ensure that Channel 1 is disabled.

1. Allow both diodes to reach the same temperature, i.e., $T_{\text{sensor}} = T_{\text{ref}}$. Measure and record the voltage offset as V_{offset}; subtract this offset voltage from your subsequent measurements.
2. Heat the sensor diode by squeezing it between your fingers. Wait for the voltage to stabilize, subtract V_{offset}, and then record this value as the “body temperature” voltage. You may also blow through a straw to direct warm air at the sensor diode.
3. If available, wrap crushed ice in a thin plastic bag and use it to chill the sensor diode. Again, wait for the voltage to stabilize, subtract V_{offset}, and then record its value as the “freezing point of water” voltage.
4. Determine the sensitivity of the temperature sensor output V_o in millivolts per °C.

Additional myDAQ tips for this problem:

- Use the 1N4148 diode described in Appendix D. Follow the diagram of Figure D.1 on page 181 to determine the anode and cathode connections; the diode's cathode marked by a black band corresponds to the bar on the diode symbol.

NI myDAQ video tutorials:

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

1.3 Graphical Analysis

The diode in the circuit of Figure 1.3 is forward-biased and modelled by the simplified Shockley equation $I_D = I_S e^{V_D/nV_T}$ where I_S is the scale current, n is the ideality factor, and V_T is the thermal voltage.

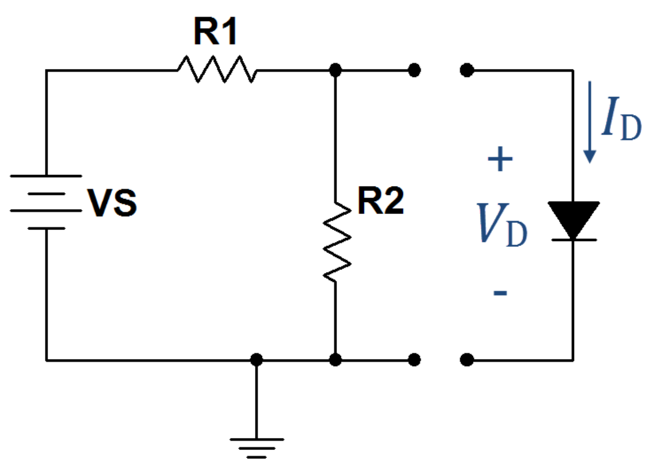


Figure 1.3: Circuit for Problem 1.3

Analysis

1. Plot the diode current-voltage characteristic over the domain $0.0 \leq V_D \leq 1.0$ V and range $0.0 \leq I_D \leq 10.0$ mA using a tool such as LabVIEW MathScript or MATLAB. Use a voltage step size of 0.01 V and calculate values only out to V_D=0.8 V. Use the `axis` function to set the indicated plot axis limits; in MathScript simply double-click the upper axis limit of the plot and type in the required value.

2. Determine the diode current I_D for the diode voltage $V_D = 0.60$ volts.

3. Determine the Thévenin equivalent of the circuit connected to the diode, and then use this information to draw the load line on the plot you created in the previous step.

4. Determine the diode's Q-point, i.e., its operating point I_{DQ} and V_{DQ} at the intersection of the diode characteristic and the load line.

Use the following circuit components:

Component	Value
V_S	1.70 V
diode	I_S = 9.076 nA, n=2, and V_T=25.84 mV
R_1	330 Ω
R_2	470 Ω

Simulation with NI Multisim

1. Plot the current-voltage characteristic of the `1N4148` diode model over the domain $0.0 \leq V_D \leq 1.0$ V and range $0.0 \leq I_D \leq 10.0$ mA using a Simulate → Analyses → DC Sweep with a 0.01 V step. Limit the sweep voltage itself to 0.8 V, and then double-click the upper axis limit for both current and voltage to set the required values.

   Connect a DC voltage source directly to the diode to establish the voltage V_D. Plot the expression `-I(V1)`, i.e., the voltage source current that is the same as the diode current except for direction.

2. Connect the remaining circuit of Figure 1.3 on the facing page to the diode and run a DC operating point analysis to determine the diode's Q-point I_{DQ} and V_{DQ}.

NI Multisim video tutorials:

- Find commonly-used circuit components:
  `http://youtu.be/G6ZJ8C0ja9Q`
- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Find node voltages with DC Operating Point analysis:
  `http://youtu.be/gXBCqP17AZs`
- Plot DC circuit response with DC Sweep:
  `http://youtu.be/vcYuCt9QjdI`

Measurement with NI myDAQ

1. Download and open the LabVIEW VI `Diode Curve Tracer.vi` available at `https://decibel.ni.com/content/docs/DOC-20392`. Construct the circuit shown in the "Connection Diagram" tab; you will need an op amp such as the TI TL072 (see Appendix C) or similar dual-supply device and a 100-ohm resistor. Refer to the "Help" tab for basic instructions and the "Plotting Tips" tab to learn how to work with the finished plot.

   Plot the current-voltage characteristic of the `1N4148` diode over the domain $0.0 \leq V_D \leq 1.0$ V and range $0.0 \leq I_D \leq 10.0$ mA using a 0.01 V step. Limit the sweep voltage itself to 0.8 V, and then double-click the upper axis limit for both current and voltage to set the required values.

2. Connect the remaining circuit of Figure 1.3 on page 22 to the diode. Use the variable voltage source of Figure B.2 on page 172 adjusted to the value of V_S stated in the problem statement. Use the NI myDAQ multimeter to measure the diode's Q-point I_{DQ} and V_{DQ}.

Additional myDAQ tips for this problem:

- Use the 1N4148 diode described in Appendix D. Follow the diagram of Figure D.1 on page 181 to determine the anode and cathode connections; the diode's cathode marked by a black band corresponds to the bar on the diode symbol.

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

Measure the diode operating point with as many 1N4148 diodes as you can find; ask to borrow some from a friend to get even more samples. Calculate the mean and coefficient of variation (CV) of your measurements, with CV defined as the standard deviation divided by the mean expressed as a percentage. Discuss the amount of variation you observed, a measure of what semiconductor engineers call *process variation*.

1.4 Series-Connected Diodes

Battery-powered appliances often require multiple power supply voltages for their various subsystems. Many digital integrated circuits require 3.3 V supplies, for example, yet other analog circuits such as audio amplifiers normally require higher voltages. The circuit of Figure 1.4 shows a 6 V DC battery (which can be created by four standard 1.5 V batteries connected in series) and four series-connected diodes that provide a reduced voltage to power the digital integrated circuits that must operate as close to 3.3 V as possible.

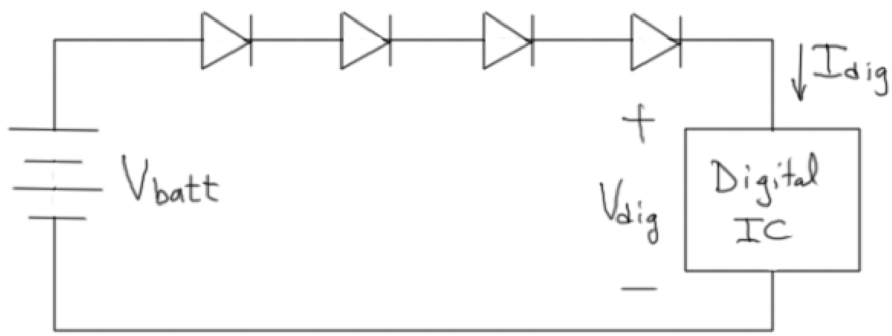


Figure 1.4: Circuit for Problem 1.4

Analysis

1. Determine the actual power supply voltage V_{dig} for three scenarios of digital device operation:

   (a) Moderate current required, $I_{dig} = 10$ mA,

   (b) Low current required, $I_{dig} = 1$ mA, and

   (c) No current required.

2. Determine the percentage difference of each of the three power supply voltages from the nominal 3.3 V voltage expected by the digital device.

Use the following circuit components:

Component	Value
$D_1 - D_4$	0.7 volt forward-biased voltage drop, ideal diode otherwise
V_{batt}	6.0 V

Simulation with NI Multisim

Enter the circuit of Figure 1.4 on the facing page using the `1N4148` diode model. Use a resistor R_{dig} to model the digital device.

1. Determine the actual power supply voltage V_{dig} for three scenarios of digital device operation:

   (a) Moderate current required, $R_{\text{dig}} = 330\ \Omega$,

   (b) Low current required, $R_{\text{dig}} = 3.3\ \text{k}\Omega$,

   (c) No current required, $R_{\text{dig}} =$ open circuit.

   Note that these resistor values approximate the device current values used in the problem statement.

2. Determine the percentage difference of each of the three power supply voltages from the nominal 3.3 V voltage expected by the digital device.

Additional Multisim tips for this problem:

- Use a voltage probe or multimeter to measure the voltage V_{dig}.
- Completely remove the resistor for the "no current required" measurement.

NI Multisim video tutorials:

- Measure DC node voltage with a measurement probe:
  `http://youtu.be/svNGHA2-uK4`
- Set the digits of precision of a measurement probe:
  `http://youtu.be/GRO60XLgzHg`

Measurement with NI myDAQ

Construct the circuit of Figure 1.4 on page 26 using 1N4148 diodes. Create the battery voltage source V_{batt} with the LM317 variable voltage circuit of Figure B.2 on page 172. Use the ELVISmx Digital Multimeter (DMM) to measure V_{batt} and adjust the variable source to set the voltage as close as possible to the required battery voltage listed in the problem statement.

1. Determine the actual power supply voltage V_{dig} for three scenarios of digital device operation:

   (a) Moderate current required, R_{dig} = 330 Ω,

   (b) Low current required, R_{dig} = 3.3 kΩ,

   (c) No current required, R_{dig} = open circuit.

   Note that these resistor values approximate the device current values used in the problem statement.

2. Determine the percentage difference of each of the three power supply voltages from the nominal 3.3 V voltage expected by the digital device.

Additional myDAQ tips for this problem:

- Use the 1N4148 diode described in Appendix D. Follow the diagram of Figure D.1 on page 181 to determine the anode and cathode connections; the diode's cathode marked by a black band corresponds to the bar on the diode symbol.

NI myDAQ video tutorials:

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

Further Exploration with NI myDAQ

Recall the *voltage divider* circuit that you studied in your linear circuits course. A two-resistor voltage divider can be designed to produce 3.3 V from the 6.0 V battery, as well. Explore how well this technique works compared to the series-connected diodes approach.

Create a two-resistor voltage divider from a 10 kΩ potentiometer: connect the outer potentiometer terminals across the "battery" and use the center terminal (movable stylus) as V_{dig}. Adjust the stylus to set V_{dig} to 3.3 V under open-circuit conditions.

Evaluate the performance of the voltage divider under the same three conditions of moderate, low, and no current. Compare and contrast the voltage-divider method and the series-connected diodes method as two possible ways to create a power supply for the digital circuit.

1.5 Zener Regulator

A *voltage regulator* seeks to maintain a constant voltage at a load regardless of changes in the load resistance. For example, the "load" could be a microprocessor-based system that requires a constant supply voltage even as its demand for current varies with processing activity.

The zener-diode regulator of Figure 1.5 offers a simple way to maintain the load voltage V_L at the same value as the reverse breakdown voltage of the zener diode, provided that the load resistance R_L remains sufficiently high. The voltage source V_S and resistor R_S model the Thévenin resistance of a circuit that has converted a high-voltage source such as the 120-VAC mains power to an intermediate DC voltage source.

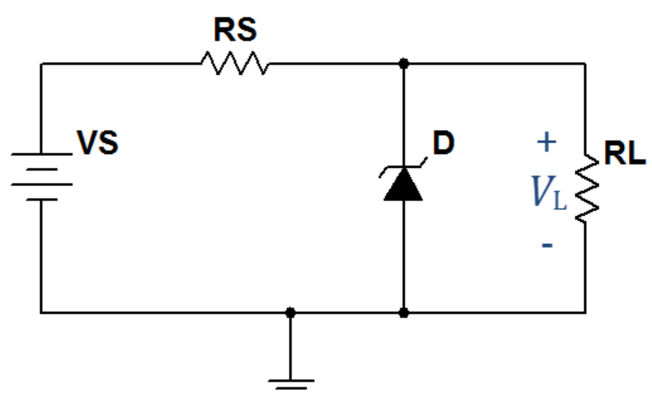


Figure 1.5: Circuit for Problem 1.5

Analysis

1. Determine the load voltage V_L for R_L equal to:
   (a) Open circuit,
   (b) 10 kΩ,
   (c) 1 kΩ, and
   (d) 100 Ω.
2. Determine the minimum value of R_L for which V_L remains within 10% of the zener voltage V_Z.

Use the following circuit components:

Component	Value
D	ideal zener diode with reverse breakdown voltage V_Z = 5.6 volts
R_S	$R = 1\ \text{k}\Omega$
V_S	15 V

Simulation with NI Multisim

Enter the circuit of Figure 1.5 on the facing page using the `ZENER_VIRTUAL` zener diode model. Double-click the diode symbol and set the "breakdown voltage" parameter to match V_Z stated in the problem statement. Use a `POTENTIOMETER` (variable resistor) as the load R_L.

1. Use interactive simulation and a Simulate → Instruments → Measurement Probe to measure the load voltage V_L for R_L equal to:
   (a) Open circuit (disconnect the potentiometer),
   (b) 10 kΩ,
   (c) 1 kΩ, and
   (d) 100 Ω.
2. Determine the minimum value of R_L for which V_L remains within 10% of the zener voltage V_Z. Reduce the step size of the potentiometer to allow the finer control of the resistance in the critical region.

Additional Multisim tips for this problem:

- Connect to the middle terminal and one end terminal of the potentiometer.
- Double-click the potentiometer and change the "Increment" field to 1% or smaller to gain finer control of its resistance.
- Change the potentiometer value either by moving the slider under the percent indicator or by pressing "A" (increment) or "Shift+A" (decrement).

NI Multisim video tutorials:

- Place and operate linear potentiometer:
  `http://youtu.be/oazwGLzWvhs`

Measurement with NI myDAQ

Construct the circuit of Figure 1.5 on page 30 using the 1N5232B 5.6-volt zener diode. Use 15V and AGND to establish the DC supply.

1. Monitor the load voltage V_L with the ELVISmx DMM voltmeter to measure V_L for R_L equal to:
   (a) Open circuit,
   (b) 10 kΩ,
   (c) 1 kΩ, and
   (d) 100 Ω.
2. Replace the load R_L with a 1K potentiometer, and adjust the potentiometer to determine the minimum value of R_L for which V_L remains within 10% of the zener voltage V_Z.

Additional myDAQ tips for this problem:

- Use the 1N5232B diode described in Appendix D. Follow the diagram of Figure D.3 on page 183 to determine the anode and cathode connections; the diode's cathode marked by a black band corresponds to the bar on the diode symbol.

NI myDAQ video tutorials:

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

Further Exploration with NI myDAQ

Investigate the current-voltage characteristic curve for the zener diode using the same technique described in Problem 1.3 on page 22. Be sure to extend the horizontal voltage range to include the 5.6-volt breakdown voltage.

Discuss your results, in particular the ways in which the zener diode is similar to and different from a conventional diode.

1.6 Half-Wave Rectifier

Figure 1.6 shows a diode half-wave rectifier connected to a load resistance.

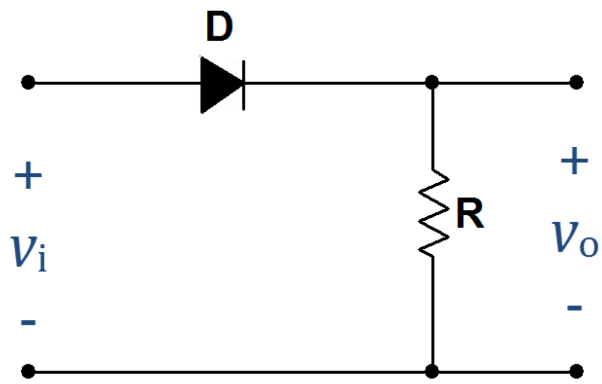


Figure 1.6: Circuit for Problem 1.6

Analysis

1. Plot two periods of $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$ for each of the three input waveforms shown in Figure 1.7 on page 37.
2. Determine the following numerical descriptors for each type of waveform output $v_\mathrm{o}(t)$:
   (a) Peak value, and
   (b) Effective DC value, also known as RMS value. NOTE: These calculations are challenging for the sine wave and triangle wave and are therefore optional.
3. Determine the typical forward-biased diode voltage drop at the peak output voltage.

Use the following circuit components:

Component	Value
D	0.7 volt forward-biased voltage drop, ideal diode otherwise
R	10 kΩ
$v_\mathrm{i}(t)$	V_M = 4 V, 100 Hz frequency ($T = 0.01$ ms)

Simulation with NI Multisim

Enter the circuit of Figure 1.6 on the previous page using the `1N4148` diode model. Use the Simulate → Instruments → Function Generator and Simulate → Instruments → Oscilloscope instruments. Use a Simulate → Instruments → Measurement Probe configured to display RMS voltage to measure the effective DC value of the output $v_{\mathrm{o}}(t)$.

1. Plot two periods of $v_{\mathrm{i}}(t)$ and $v_{\mathrm{o}}(t)$ for each of the three input waveforms shown in Figure 1.7 on page 37.

2. Determine the following numerical descriptors for each type of waveform output $v_{\mathrm{o}}(t)$:

   (a) Peak value (use a cursor measurement), and

   (b) Effective DC value, also known as RMS value.

3. Determine the typical forward-biased diode voltage drop at the peak output voltage.

Additional Multisim tips for this problem:

- Set distinct colors for the waveform traces to easily distinguish the input and output waveforms.

- Refer to the tutorial videos below to learn how to take cursor measurements from the oscilloscope.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Measure RMS and average value with a measurement probe:
  `http://youtu.be/OnK-Unld17E`
- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Find the maximum value of a trace in Grapher View:
  `http://youtu.be/MzYK60mfh2Y`
- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`
- Distinguish oscilloscope traces by color:
  `http://youtu.be/bICbjggcTiQ`

Measurement with NI myDAQ

Construct the circuit of Figure 1.6 on page 33 using a 1N4148 diode. Use the ELVISmx Function Generator to create $v_\mathrm{i}(t)$ and the ELVISmx Oscilloscope to display both $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$.

1. Plot two periods of $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$ for each of the three input waveforms shown in Figure 1.7 on page 37.
2. Determine the following numerical descriptors for each type of waveform output $v_\mathrm{o}(t)$:
   (a) Peak value (use a cursor measurement), and
   (b) Effective DC value, also known as RMS value; read this value in the measurements display area under the waveform display.
3. Determine the typical forward-biased diode voltage drop at the peak output voltage.

Additional myDAQ tips for this problem:

- Use the 1N4148 diode described in Appendix D. Follow the diagram of Figure D.1 on page 181 to determine the anode and cathode connections; the diode's cathode marked by a black band corresponds to the bar on the diode symbol.

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

Replace the 1N4148 diode with a light-emitting diode, or LED. Follow the diagram of Figure D.2 on page 182 to determine the anode and cathode connections. You may need to darken the room lights or shade the LED with your hand.

1. How does the waveform for $v_o(t)$ compare to your earlier results with the 1N4148 diode? By how many times does the forward-bias voltage drop increase?
2. Experiment with the three different waveform shapes while the function generator remains set to 100 Hz, paying particular attention to the brightness of the LED. Discuss your observations of waveform shape and brightness and relate these observations to your measured effective DC values for each waveform shape.
3. Reduce the function generator frequency, and experiment with values as low at 0.2 Hz (one cycle every five seconds). Discuss the behavior of the LED optical intensity for each of the three waveform shapes when the function generator frequency is 1 Hz or less.
4. At what function generator frequency does the flashing LED begin to appear as a constant intensity?

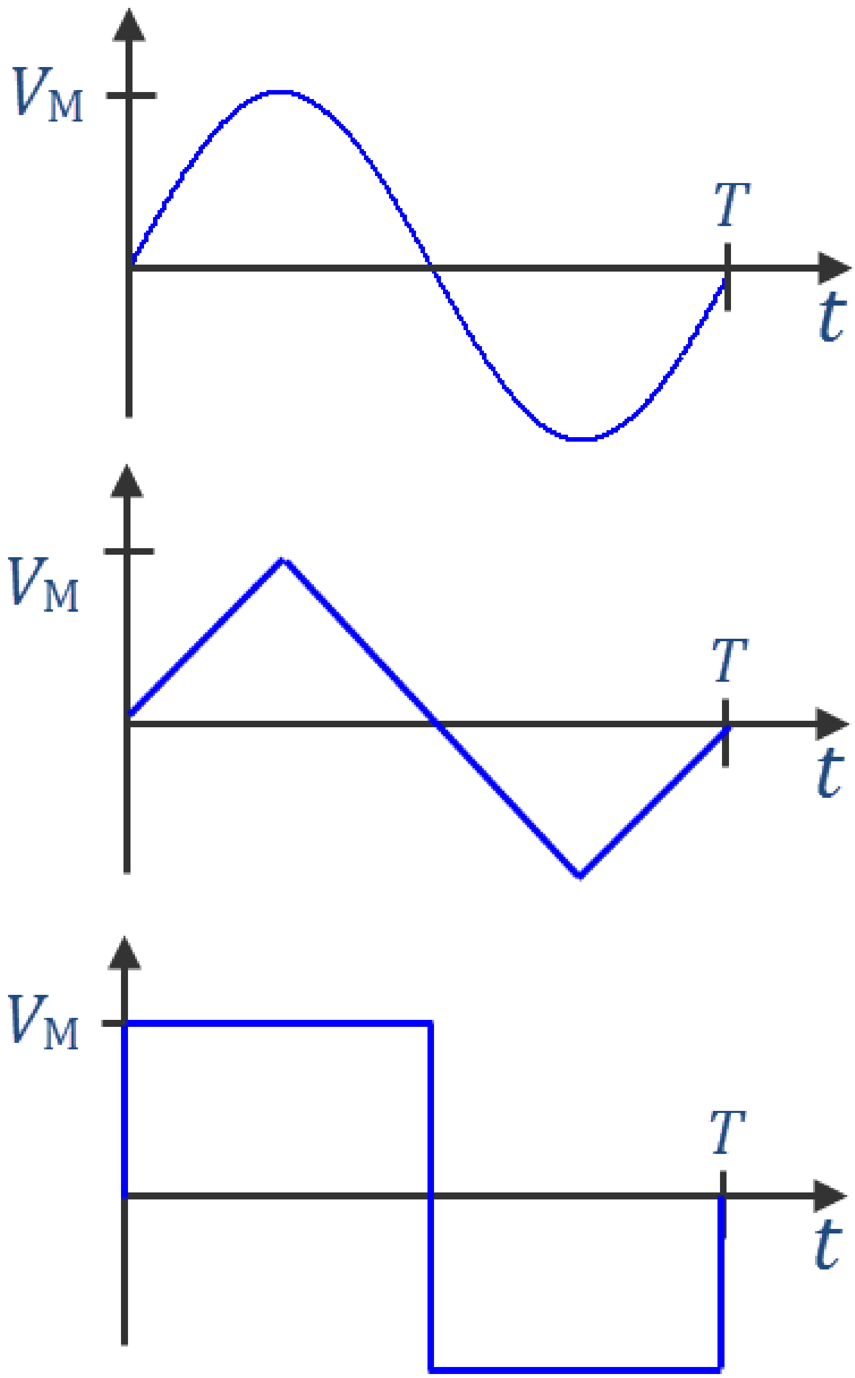


Figure 1.7: Input waveforms for Problem 1.6 on page 33, Problem 1.8 on page 39, and Problem 2.9 on page 83

1.7 Another Half-Wave Rectifier

Consider the diode half-wave rectifier circuit of Figure 1.6 on page 33. Reverse the diode polarity and work Problem 1.6 on page 33 again with V_{M} increased to 5 volts, and interpret all references to "peak value" as peak *absolute* value.

Further Exploration with NI myDAQ

Replace the 1N4148 diode with a light-emitting diode, or LED. Follow the diagram of Figure D.2 on page 182 to determine the anode and cathode connections. You may need to darken the room lights or shade the LED with your hand.

Set the function generator to sinewave mode and observe $v_{\mathrm{i}}(t)$ and $v_{\mathrm{o}}(t)$ with the oscilloscope as you connect a 1 kΩ resistor in parallel with the existing resistor, and then discuss the following points:

1. How does the parallel resistor affect the combined load seen by the diode?

2. What happens to the LED intensity? What does this suggest about the new diode current?

3. Observe the "clipped" input waveform $v_{\mathrm{i}}(t)$, a consequence of the limited current available on the myDAQ analog output AO. Based on your oscilloscope measurement, describe a method to calculate the maximum possible current that the myDAQ can supply on the analog output.

1.8 Full-Wave Rectifier

Figure 1.8 shows a diode full-wave rectifier connected to a load resistance.

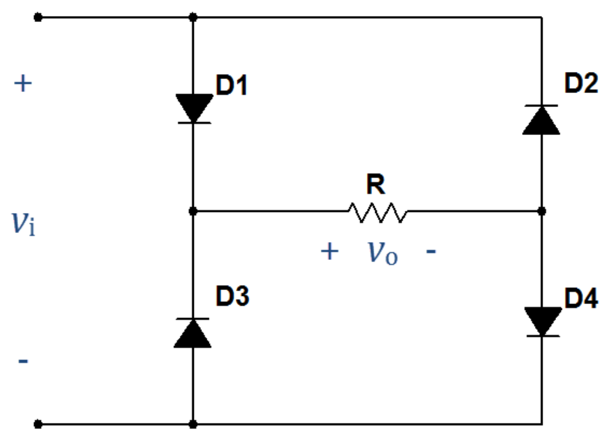


Figure 1.8: Circuit for Problem 1.8

Analysis

1. Copy the circuit of Figure 1.8 and sketch the flow of positive current throughout the entire circuit for $v_\mathrm{i} > 0$. Repeat for $v_\mathrm{i} < 0$.

2. Plot two periods of $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$ for each of the three input waveforms shown in Figure 1.7 on page 37.

3. Determine the following numerical descriptors for each type of waveform output $v_\mathrm{o}(t)$:

   (a) Peak value, and

   (b) Effective DC value, also known as RMS value. NOTE: These calculations are challenging for the sine wave and triangle wave and are therefore optional.

4. Determine the typical forward-biased diode voltage drop at the peak output voltage.

Use the following circuit components:

Component	Value
D_1–D_4	0.7 volt forward-biased voltage drop, ideal diode otherwise
R	2.2 kΩ
$v_\mathrm{i}(t)$	V_M = 5 V, 100 Hz frequency ($T = 0.01$ ms)

Simulation with NI Multisim

Enter the circuit of Figure 1.8 on the preceding page using the `1N4148` diode model. Use the Simulate → Instruments → Function Generator and Simulate → Instruments → Oscilloscope instruments. Use a Simulate → Instruments → Measurement Probe configured to display RMS voltage to measure the effective DC value of the output $v_\mathrm{o}(t)$.

1. Plot two periods of $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$ for each of the three input waveforms shown in Figure 1.7 on page 37.

2. Determine the following numerical descriptors for each type of waveform output $v_\mathrm{o}(t)$:

   (a) Peak value (use a cursor measurement), and

   (b) Effective DC value, also known as RMS value.

3. Determine the typical forward-biased diode voltage drop at the peak output voltage.

Additional Multisim tips for this problem:

- Set distinct colors for the waveform traces to easily distinguish the input and output waveforms.

- Remember that the voltage probe reports its voltage measurements with respect to ground. Because the load resistor is ungrounded, connect one probe to the positive terminal of v_o and another probe to the associated negative terminal, and then reference the positive probe to the negative probe.

NI Multisim video tutorials:

- Find components by name:
  http://youtu.be/5wlFweh4n-c
- Measure RMS and average value with a measurement probe:
  http://youtu.be/OnK-Unld17E
- Basic operation of the two-channel oscilloscope:
  http://youtu.be/qnRK6QyqjvQ
- Find the maximum value of a trace in Grapher View:
  http://youtu.be/MzYK60mfh2Y
- Waveform cursor measurements with the two-channel oscilloscope:
  http://youtu.be/snBRFq1Y1q4
- Distinguish oscilloscope traces by color:
  http://youtu.be/bICbjggcTiQ
- Measure DC voltage with a referenced measurement probe:
  http://youtu.be/xKEQ3EXEaP8

Measurement with NI myDAQ

Construct the circuit of Figure 1.8 on page 39 using a 1N4148 diode. Use the ELVISmx Function Generator to create $v_\mathrm{i}(t)$ and the ELVISmx Oscilloscope to display both $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$.

1. Plot two periods of $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$ for each of the three input waveforms shown in Figure 1.7 on page 37.
2. Determine the following numerical descriptors for each type of waveform output $v_\mathrm{o}(t)$:
   (a) Peak value (use a cursor measurement), and
   (b) Effective DC value, also known as RMS value; read this value in the measurements display area under the waveform display.
3. Determine the typical forward-biased diode voltage drop at the peak output voltage.

Additional myDAQ tips for this problem:

- Use the 1N4148 diode described in Appendix D. Follow the diagram of Figure D.1 on page 181 to determine the anode and cathode connections; the diode's cathode marked by a black band corresponds to the bar on the diode symbol.

- Place the diodes to physically match their positions in the schematic diagram.

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

Replace the 1N4148 diodes with light-emitting diodes (LEDs). Refer to the diagram of Figure D.2 on page 182 to determine the anode and cathode connections of an LED. Place the diodes to physically match their positions in the schematic diagram for best results.

1. How does the waveform for $v_{\mathrm{o}}(t)$ compare to your earlier results with the 1N4148 diode? By how many times does the forward-bias voltage drop increase?

2. Experiment with the three different waveform shapes while the function generator remains set to 100 Hz, paying particular attention to the brightness of the LEDs. Discuss your observations of waveform shape and brightness and relate these observations to your measured effective DC values for each waveform shape.

3. Reduce the function generator frequency, and experiment with values as low at 0.2 Hz (one cycle every five seconds). Discuss the behavior of the LED optical intensity for each of the three waveform shapes when the function generator frequency is 1 Hz or less.

4. At what function generator frequency do the flashing LEDs begin to appear as a constant intensity?

1.9 Peak Rectifier

Figure 1.9 shows a *peak rectifier*, also known as a *peak detector*, a circuit whose output tracks the most-recent maximum value of the input signal.

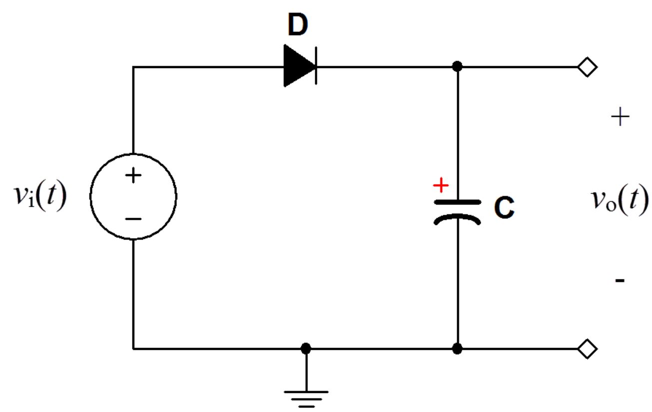


Figure 1.9: Circuit for Problem 1.9

Analysis

1. Plot the input and output voltage waveforms $v_i(t)$ and $v_o(t)$ as well as the capacitor current $i_C(t)$ for the input waveform shown in Figure 1.10 on the next page. Assume the capacitor is initially discharged.

2. Determine the following numerical descriptors for $v_o(t)$ and $i_C(t)$:

   (a) Voltage values of $v_o(t)$ at times $t =$250, 650, and 950 ms.

   (b) Peak capacitor current.

3. Discuss the relationship between the plots of the capacitor current $i_C(t)$ and the output voltage $v_o(t)$.

Use the following circuit components:

Component	Value
D	0.7 volt forward-biased voltage drop, ideal diode otherwise
C	1.0 μF

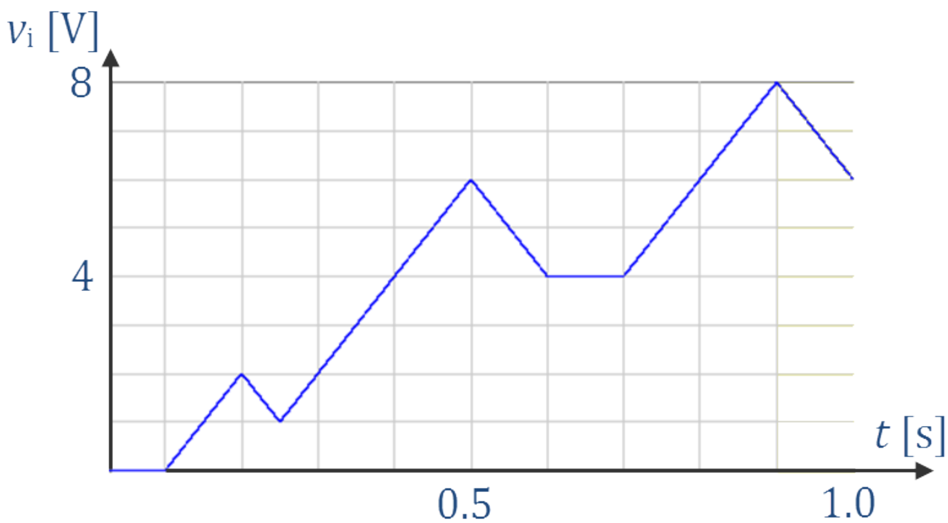


Figure 1.10: Input waveform for Problem 1.9 on the preceding page

Simulation with NI Multisim

Enter the circuit of Figure 1.9 on the preceding page using the `1N4148` diode model. Use the `PIECEWISE_LINEAR_VOLTAGE` source to create the input waveform pictured in Figure 1.10.

1. Set up a Simulate → Analyses → Transient analysis to plot the input and output voltage waveforms $v_i(t)$ and $v_o(t)$.
2. Set up a second transient analysis to plot the input voltage waveform and the capacitor current $i_C(t)$; use a second axis on the right side of the plot for current.
3. Determine the following numerical descriptors for $v_o(t)$ and $i_C(t)$:
   (a) Voltage values of $v_o(t)$ at times t =250, 650, and 950 ms.
   (b) Peak capacitor current.

Additional Multisim tips for this problem:

- Set distinct colors for the waveform traces to easily distinguish the input and output waveforms.

- Refer to the tutorial videos below to learn how to take cursor measurements from the oscilloscope.

NI Multisim video tutorials:

- Piecewise linear (PWL) voltage source:
  http://youtu.be/YYU5WuyebD0
- Plot time-domain circuit response with Transient Analysis:
  http://youtu.be/waKnad_EXkc
- Plot second variable on its own axis in Grapher View:
  http://youtu.be/rULFKRTphcI

Measurement with NI myDAQ

Construct the circuit of Figure 1.9 on page 43 using a 1N4148 diode and an electrolytic capacitor; be sure to observe proper polarity for the capacitor. Use the ELVISmx Arbitrary Waveform Generator to create $v_\mathrm{i}(t)$ and the ELVISmx Oscilloscope to display both $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$.

1. Use the ELVISmx Oscilloscope to display $v_\mathrm{i}(t)$ on AI0 and $v_\mathrm{o}(t)$ on AI1. Carefully follow this procedure to achieve a stable and correct display:
   (a) Select "Edge" triggering and trigger on the input waveform (Channel 0), use falling-edge (negative edge) triggering, set the trigger level to 0.5 volts, and move the horizontal position to the far right (100%),
   (b) Set both the oscilloscope and arbitrary waveform generator to "Run Once" acquisition mode,
   (c) Temporarily short the capacitor with a wire to reset its voltage to zero, and
   (d) Click "Run" on the oscilloscope and then "Run" on the arbitrary waveform generator as quickly as possible thereafter.
2. Insert a 1 kΩ shunt resistor in series with the capacitor. Continue to display $v_\mathrm{i}(t)$ on AO0 and then display the voltage across the shunt on AO1; note that the shunt carries the same current as the capacitor, hence $i_\mathrm{C}(t)$ is proportional to the shunt resistor voltage waveform. Repeat the procedure from the previous step to plot capacitor current;

adjust the scale ("Volts/Div" setting) until the current waveform is clearly visible.

3. Determine the following numerical descriptors for $v_o(t)$ and $i_C(t)$:
   (a) Voltage values of $v_o(t)$ at times t =250, 650, and 950 ms.
   (b) Peak capacitor current.

Additional myDAQ tips for this problem:

- Use the 1N4148 diode described in Appendix D. Follow the diagram of Figure D.1 on page 181 to determine the anode and cathode connections; the diode's cathode marked by a black band corresponds to the bar on the diode symbol.
- Remember to divide by the shunt resistor value to read capacitor current from the oscilloscope.

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`

1.10 Limiter

Digital inputs normally include *diode limiters* to ensure that input signals that stray beyond the power supply limits do not damage the sensitive input stages. Figure 1.11 shows such a limiter circuit constructed from a pair of diodes and a resistor.

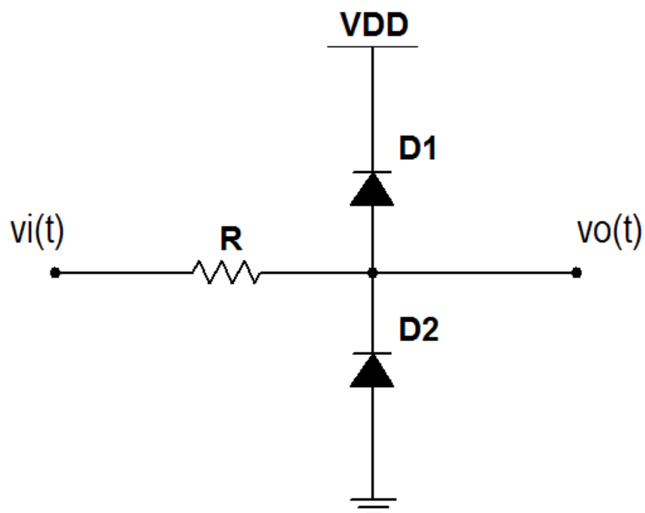


Figure 1.11: Circuit for Problem 1.10

Analysis

1. Plot the input and output voltage waveforms $v_{\mathrm{i}}(t)$ and $v_{\mathrm{o}}(t)$ for a 100-Hz triangle wave with 5 volts peak amplitude.

2. Determine the following numerical descriptors for $v_{\mathrm{o}}(t)$:

   (a) Maximum value,

   (b) Minimum value, and

   (c) Percentage of period in which the input waveform is limited.

Use the following circuit components:

Component	Value
V_{DD}	3.3 V
D_1, D_2	0.7 volt forward-biased voltage drop, ideal diode otherwise
R	$R = 10\ \text{k}\Omega$

Simulation with NI Multisim

Enter the circuit of Figure 1.11 on the preceding page using the `1N4148` diode model. Use the Simulate → Instruments → Function Generator to create the input waveform described earlier and attach the Simulate → Instruments → Oscilloscope to view the input and output waveforms.

1. Set up a Simulate → Run analysis to plot the input and output voltage waveforms $v_i(t)$ and $v_o(t)$.
2. Determine the following numerical descriptors for $v_o(t)$:
   (a) Maximum value,
   (b) Minimum value, and
   (c) Percentage of period in which the input waveform is limited.

Additional Multisim tips for this problem:

- Set distinct colors for the waveform traces to easily distinguish the input and output waveforms.
- Refer to the tutorial videos below to learn how to take cursor measurements from the oscilloscope.

NI Multisim video tutorials:

- Function generator:
  `http://youtu.be/CeOl6EzD-_c`
- 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`
- VDD and VSS power supply voltages:
  `http://youtu.be/XrPVLgYsDdY`

Measurement with NI myDAQ

Construct the circuit of Figure 1.11 on page 47 using 1N4148 diodes. Use the variable voltage source of Figure B.2 on page 172 adjusted to the value of V_{DD} stated in the problem statement. Use the ELVISmx Function Generator to create $v_{\text{i}}(t)$ on AO0 and the ELVISmx Oscilloscope to display both $v_{\text{i}}(t)$ and $v_{\text{o}}(t)$.

1. Use the ELVISmx Oscilloscope to display $v_{\text{i}}(t)$ on AI0 and $v_{\text{o}}(t)$ on AI1.
2. Determine the following numerical descriptors for $v_{\text{o}}(t)$:
   (a) Maximum value,
   (b) Minimum value, and
   (c) Percentage of period in which the input waveform is limited.

Additional myDAQ tips for this problem:

- Use the 1N4148 diode described in Appendix D. Follow the diagram of Figure D.1 on page 181 to determine the anode and cathode connections; the diode's cathode marked by a black band corresponds to the bar on the diode symbol.

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

1. Predict what you would see with the other function generator waveforms (sine and square), and then apply them to your circuit.
2. Normally digital inputs have a very high input resistance which is equivalent to having negligible input current. But is this always the case, even when the input signal is substantially higher in voltage than the power supply limit?

   Return the function generator to triangle wave output. Move the AI1 channel to monitor the voltage across the resistor, placing AI1+ on the left side and AI1- on the right side of the resistor. Recognize that this voltage is proportional to the resistor current, which is the same thing as digital input current for this circuit. Study this waveform for the triangle wave input and discuss your findings. What is the maximum absolute input current?

Chapter 2

Operational Amplifier Circuits

2.1 Comparator

Figure 2.1 on the following page shows a *comparator* constructed from an operational amplifier. Note that no feedback path exists from the op amp output back to any of its inputs, therefore this is an example of an *open-loop* circuit. The comparator serves as a basic component of analog-to-digital converters by comparing the continuous analog signal to a voltage threshold to produce a high-level voltage when the signal exceeds the threshold and a low-level voltage otherwise. The LED indicator circuit serves the dual role of showing the output state of the comparator and limiting the output voltage to a range that can be measured by the NI myDAQ analog output.

Analysis

NOTE: A three-way numerical comparison is not required for this problem.

1. Let $v_\mathrm{i}(t)$ be a 5-volt peak triangle wave at 10 Hz. Plot $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$. HINTS: Determine the voltage across R_1 using properties of ideal op amps, and account for saturation of the op amp (the TL072 saturates at approximately ±13.5 V, or 1.5 V inside each power supply rail).

2. Superimpose a 1 kHz sinewave with 1 volt amplitude on the triangle wave to model unwanted high-frequency noise on the input signal. Sketch $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$.

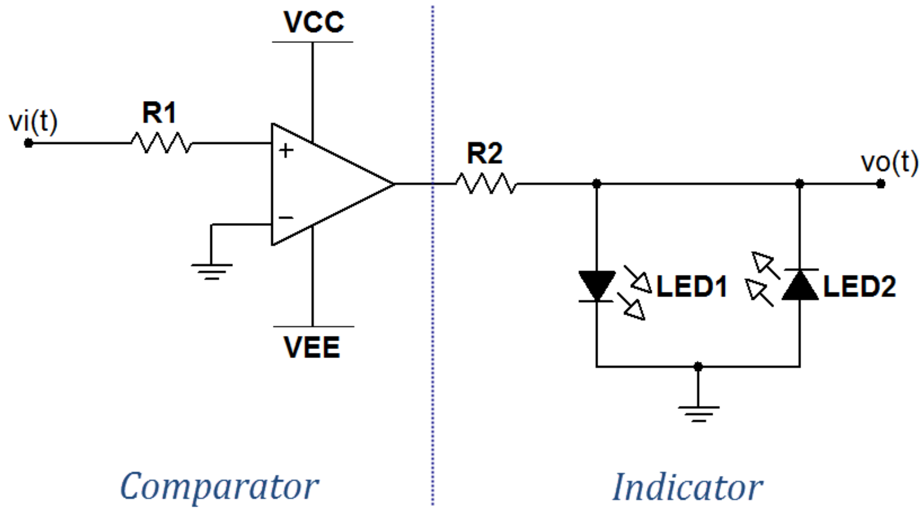


Figure 2.1: Circuit for Problem 2.1 on the previous page

3. Discuss the performance of this comparator for real analog signals that always have some degree of noise, especially when the comparator output is connected directly to digital circuits.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	-15 volts
LEDs	2 V forward drop
resistors	$R_1 = 10$ kΩ and $R_2 = 1$ kΩ

Simulation with NI Multisim

Enter the circuit of Figure 2.1 using the `TL072CP` op amp model. Place the `VCC` and `VEE` power supplies and set their values to $+15$ V and -15 V. Use `LED_green` and `LED_red` for the LEDs.

1. Use the Simulate → Instruments → Function Generator to create the triangle wave input signal described in the problem statement, and

use the Simulate → Instruments → Oscilloscope to plot the input and output signals $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$. Use distinct colors for the input and output traces. Run an interactive simulation to view the input and output waveforms.

2. Place a second function generator configured to create the sinewave described in the problem statement, and then connect the two function generators in series. Plot the input and output signals $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$.

Additional Multisim tips for this problem:

- Ground the center terminal of the triangle wave function generator and leave the "–" terminal unconnected.
- Use the "+" terminal and center terminal of the sinewave function generator.

NI Multisim video tutorials:

- Function generator:
  `http://youtu.be/Ce0l6EzD-_c`
- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Distinguish oscilloscope traces by color:
  `http://youtu.be/bICbjggcTiQ`

Measurement with NI myDAQ

1. Build the circuit of Figure 2.1 on the preceding page. Use AO0 to create v_i. Set up the NI ELVISmx Function Generator to create the triangle waveform described in the problem statement. Use the NI ELVISmx Oscilloscope to display $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$.

2. Set up the NI ELVISmx Arbitrary Waveform Generator (ARB) to produce the triangle waveform and superimposed sinewave:

   - Set the sample rate to 200 kS/s.
   - Create a single segment of length 100 ms.

- Create *two* components under this segment, one for the triangle wave and another for the sine wave. Ensure that "Function" is set to "+" to add the components together.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 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 for V_{CC} and -15V for V_{EE}; use AGND for circuit ground.

NI myDAQ video tutorials:

- 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`

Further Exploration with NI myDAQ

1. Change the sinewave to Gaussian noise (level = 1.00) in the arbitrary waveform generator to get a more realistic view of high-frequency random noise.

2. The frequencies used above were selected to easily visualize the signals on the oscilloscope. Reduce the frequencies to make the comparator behavior visible on the LEDs; try reducing the arbitrary waveform generator sampling rate to 5.0 kS/s.

2.2 Inverting Amplifier

Figure 2.2 illustrates an *inverting amplifier* constructed from an operational amplifier. The voltage gain of the inverting amplifier is $v_o/v_i = -R_2/R_1$.

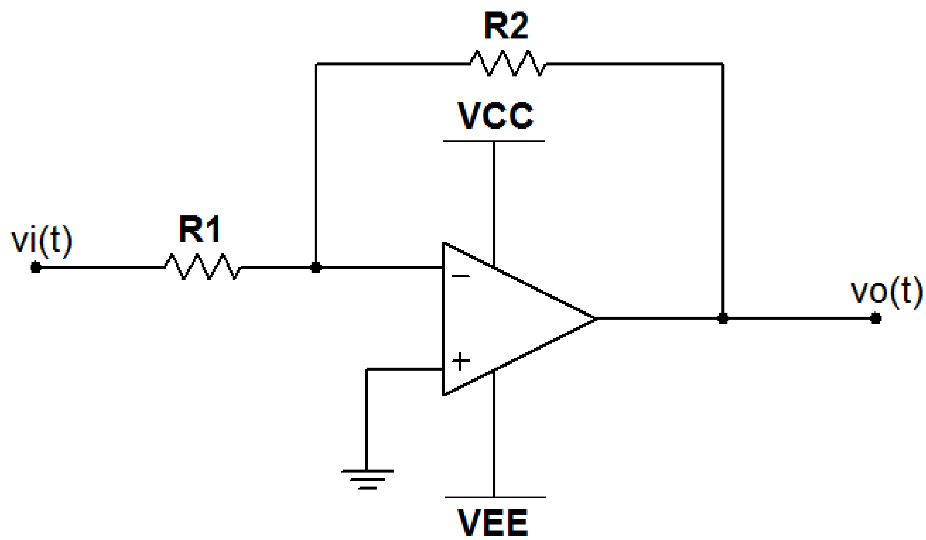


Figure 2.2: Circuit for Problem 2.2

Analysis

1. Sketch $v_i(t)$ and $v_o(t)$ for $R_1 = 10\ \text{k}\Omega$, $R_2 = 10\ \text{k}\Omega$, and with a 2-volt peak-to-peak triangle wave at 100 Hz as $v_i(t)$. What is the maximum value of $v_o(t)$?

2. The output is to be limited to 6 volts peak. What is the maximum peak voltage of the input when $R_1 = 2.2\ \text{k}\Omega$ and $R_2 = 47\ \text{k}\Omega$?

3. The amplifier of Part (1) is *cascaded* with the amplifier of Part (2), i.e., the output of the first amplifier feeds the input of the second to make a two-stage amplifier. Sketch the output of the second stage when the input to the first stage is a 200-mV peak square wave at 100 Hz. What is the maximum value of $v_o(t)$?

4. Is the two-stage amplifier of Part (3) still an inverting amplifier? Explain your answer.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	−15 volts

Simulation with NI Multisim

Enter the circuit of Figure 2.2 on the previous page using the `TL072CP` op amp model. Place both "A" and "B" devices (the TL072CP contains two opamps in a single package), reserving the "B" device for the last part of the problem. Place the `VCC` and `VEE` power supplies and set their values to $+15$ V and -15 V.

1. Use the Simulate → Instruments → Function Generator to create the triangle wave input signal described in the problem statement, and use the Simulate → Instruments → Oscilloscope to plot the input and output signals $v_i(t)$ and $v_o(t)$ for $R_1 = 10$ kΩ and $R_2 = 10$ kΩ. Use distinct colors for the input and output traces. Run an interactive simulation to view the input and output waveforms. Use the oscilloscope cursor to determine the maximum value of $v_o(t)$.

2. The output is to be limited to 6 volts peak when $R_1 = 2.2$ kΩ and $R_2 = 47$ kΩ. Adjust the maximum peak voltage of the input triangle waveform until the output peak voltage is 6 volts. Report this maximum peak input voltage.

3. The amplifier of Part (1) is *cascaded* with the amplifier of Part (2), i.e., the output of the first amplifier feeds the input of the second to make a two-stage amplifier. Plot the output of the second stage when the input to the first stage is a 200-mV peak square wave at 100 Hz; use the same volts-per-division setting on the oscilloscope. Use the oscilloscope cursor to determine the maximum value of $v_o(t)$.

Additional Multisim tips for this problem:

- Ground the center terminal of the function generator and leave the "–" terminal unconnected.

- Note that connecting the power supplies to pins 8 and 4 of the "A" device causes "X"s to appear at the same pins on the "B" device because both devices share the same power supply connections.

- The inverting amplifier is most commonly drawn with the op amp "–" terminal on top as shown in Figure 2.2 on page 55. However, when first placed the TL072CP symbol has the "+" terminal on top. While it is possible to flip the orientation of the symbol (select the device and type Alt+Y), this causes the power supply pins to flip, as well. Normally the positive supply is drawn to connect to the top of the op amp. Use whichever op amp orientation you prefer, but ensure that the signal connections are correct.

NI Multisim video tutorials:

- Place dual op amp second device:
  `http://youtu.be/-QDFEf-KdEw`

- VCC and VEE power supply voltages:
  `http://youtu.be/XkZTwKD-WjE`

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

- 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`

Measurement with NI myDAQ

1. Build the circuit of Figure 2.2 on page 55 with $R_1 = 10\ \mathrm{k}\Omega$ and $R_2 = 10\ \mathrm{k}\Omega$, and establish connections for the function generator and oscilloscope as follows:

   - Select the "A" device of the TL072 op amp,
   - Use the ELVISmx Function Generator on AO0 to create the triangle wave input signal described in the problem statement,

- Use the ELVISmx Oscilloscope to plot the input $v_{\mathrm{i}}(t)$ on AI0 and the output $v_{\mathrm{o}}(t)$ on AI1; use the same volts-per-division setting for both channels.

Use the oscilloscope cursor to determine the maximum value of $v_{\mathrm{o}}(t)$.

2. The output is to be limited to 6 volts peak when $R_1 = 2.2\ \mathrm{k}\Omega$ and $R_2 = 47\ \mathrm{k}\Omega$. Construct this amplifier using the "B" device of the TL072. Adjust the maximum peak voltage of the input triangle waveform until the output peak voltage is 6 volts. Report this maximum peak input voltage.

3. The amplifier of Part (1) is *cascaded* with the amplifier of Part (2), i.e., the output of the first amplifier feeds the input of the second to make a two-stage amplifier. Plot the output of the second stage when the input to the first stage is a 200-mV peak square wave at 100 Hz; use the same volts-per-division setting on the oscilloscope. Use the oscilloscope cursor to determine the maximum value of $v_{\mathrm{o}}(t)$.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 for both 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 for V_{CC} and -15V for V_{EE}; use AGND for circuit ground.

NI myDAQ video tutorials:

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

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

2.3 Summing Amplifier

Figure 2.3 shows a three-channel *summing amplifier* constructed from an operational amplifier. The number of channels may be extended by adding additional resistors connected to the inverting input of the op amp.

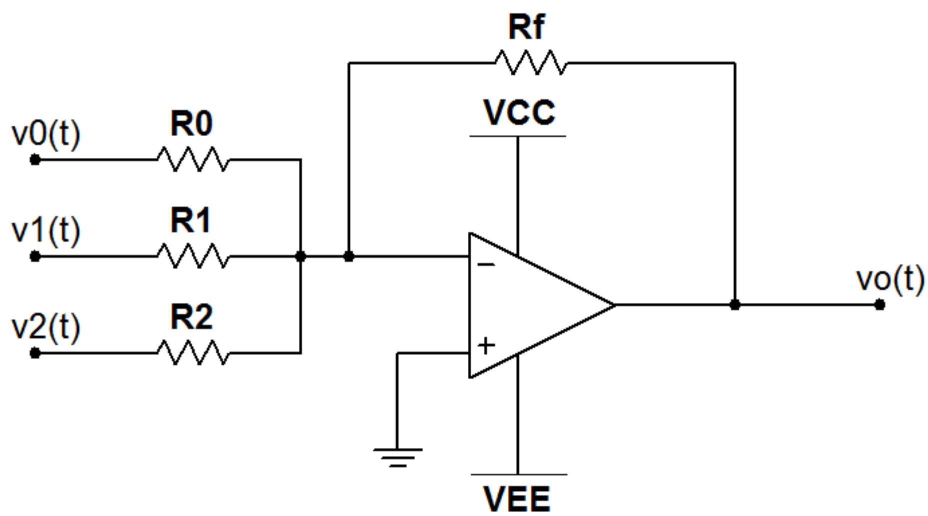


Figure 2.3: Circuit for Problem 2.3

Analysis

1. Derive the general equation for the output voltage v_o as a function of the three input voltages v_0, v_1, and v_2 and generic resistor values. HINT: Apply the *superposition* circuit analysis method.

2. Suppose the input voltages are binary (two-level) signals, taking only the values 0 V and V_{max}. Tabulate all eight possible values of v_o using the specific resistor values listed below.

3. Plot $v_o(t)$ for the input voltages following the binary sequence $(v_2 v_1 v_0)$ = 000, 001, 010, 011, 100, 101, 110, and 111 where “0” denotes zero volts and “1” denotes V_{max}.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	-15 volts
resistors	$R_0 = 10$ kΩ, $R_1 = 4.7$ kΩ, $R_2 = 2.2$ kΩ, and $R_f = 3.3$ kΩ
V_{max}	3.3 volts

Simulation with NI Multisim

Enter the circuit of Figure 2.3 on the previous page using the `TL072CP` op amp model. Place the `VCC` and `VEE` power supplies and set their values to $+15$ V and -15 V.

1. Use the Simulate → Instruments → Word Generator to create the binary sequence. Configure the Word Generator as follows:
   - Double-click the Word Generator.
   - Set "Controls" to "Cycle," set "Trigger" to "Internal," set "Frequency" to 100 Hz, and set "Display" to "Binary."
   - Click "Set...", select "Up counter" as the preset pattern, set the buffer size to 8, and choose 0 for the initial pattern. Click "Accept."
   - Right-click the top row and choose "Set Cursor" to position the cursor at the beginning of the sequence.
2. Place three ABM sources to translate the Word Generator outputs to the correct voltage ranges as follows:
   - Connect wire stubs to the Word Generator outputs 0, 1, and 2. Name these wires `wg0`, `wg1`, and `wg2`.
   - Configure each ABM source value with a math expression of the form `if(v(wg`N`)>2.5,3.3,0)` where N is either 0, 1, or 2. This expression translates the Word Generator voltage signal to a voltage that is either 0 V or V_{max}.
   - Connect the three ABM sources to the inputs to the summing amplifier.

   Next, use the Simulate → Instruments → Oscilloscope to plot $v_o(t)$ and $v_0(t)$. Use distinct colors for the input and output traces. Run an interactive simulation to view the input and output waveforms.

3. Use the oscilloscope cursor to measure the eight possible values of v_o.

Additional Multisim tips for this problem:

- Flip the Word Generator symbol horizontally (select the symbol and type Alt+X) to make the needed signals emerge from the right side.
- The Word Generator produces a low-level voltage of 0.5 V and a high-level voltage of 4.5 V; these values cannot be changed. The ABM sources serve as threshold devices that set their voltages to 0 V when the Word Generator signal is below the 2.5 V threshold and to 3.3 V when the Word Generator signal is above the threshold.
- The Word Generator pattern cursor stops when the simulation stops and continues from this point when the simulator is restarted. To produce a consistent trace on the oscilloscope for repeated simulations, right-click the top row of the Word Generator display and choose "Set Cursor" to position the cursor at the beginning of the sequence.

NI Multisim video tutorials:

- Create digital sequences with Word Generator:
  `http://youtu.be/vp1l-TeRB3s`
- ABM (Analog Behavioral Model) voltage source:
  `http://youtu.be/8pPynWRwhO4`
- 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`

Measurement with NI myDAQ

1. Build the circuit of Figure 2.3 on page 59. Drive the inputs from the digital I/O lines DIO0, DIO1, and DIO2. Monitor the summing amplifier output with the NI ELVISmx Oscilloscope.

2. Launch and run the NI ELVISmx Digital Writer (DigOut). Click the line buttons to step through all eight possible combinations of lines 0 to 2 and record the RMS measurement indicator value for each combination.

3. Change “Pattern” from “Manual” to “Ramp (0-15)” to produce the binary sequence described in the problem statement, and display $v_o(t)$.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 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 for V_{CC} and -15V for V_{EE}; use AGND for circuit ground.

NI myDAQ video tutorials:

- Digital Writer (DigOut):
  `http://decibel.ni.com/content/docs/DOC-12945`

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

Further Exploration with NI myDAQ

The summing amplifier of this problem is an example of a *digital-to-analog converter* or “D/A.” Display the analog voltage for the other available digital sequences of the Digital Writer: “Alternating 1/0’s” and “Walking 1’s.”

2.4 Noninverting Amplifier

Figure 2.4 illustrates a *noninverting amplifier* constructed from an operational amplifier. The voltage gain of the noninverting amplifier is $v_\mathrm{o}/v_\mathrm{i} = 1 + (R_2/R_1)$.

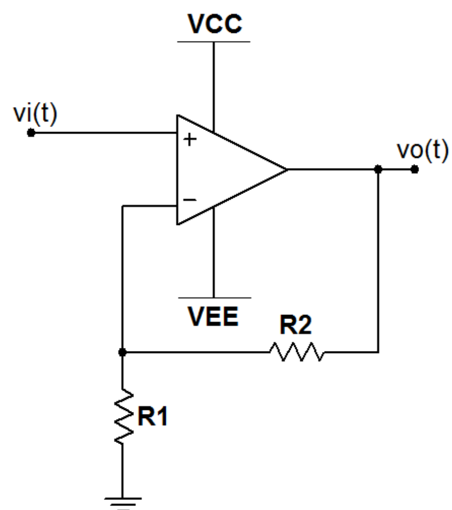


Figure 2.4: Circuit for Problem 2.4

Analysis

1. Sketch $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$ for $R_1 = 10\ \mathrm{k}\Omega$, $R_2 = 10\ \mathrm{k}\Omega$, and with a 2-volt peak-to-peak triangle wave at 100 Hz as $v_\mathrm{i}(t)$. What is the maximum value of $v_\mathrm{o}(t)$?

2. The output is to be limited to 6 volts peak. What is the maximum peak voltage of the input when $R_1 = 2.2\ \mathrm{k}\Omega$ and $R_2 = 47\ \mathrm{k}\Omega$?

3. The amplifier of Part (1) is *cascaded* with the amplifier of Part (2), i.e., the output of the first amplifier feeds the input of the second to make a two-stage amplifier. Sketch the output of the second stage when the input to the first stage is a 200-mV peak square wave at 100 Hz. What is the maximum value of $v_\mathrm{o}(t)$?

4. Compare and contrast your results for this problem with those of Problem 2.2 on page 55.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	−15 volts

Simulation with NI Multisim

Enter the circuit of Figure 2.4 on the previous page using the `TL072CP` op amp model. Place both "A" and "B" devices (the TL072CP contains two opamps in a single package), reserving the "B" device for the last part of the problem. Place the `VCC` and `VEE` power supplies and set their values to $+15$ V and -15 V.

1. Use the Simulate → Instruments → Function Generator to create the triangle wave input signal described in the problem statement, and use the Simulate → Instruments → Oscilloscope to plot the input and output signals $v_i(t)$ and $v_o(t)$ for $R_1 = 10$ kΩ and $R_2 = 10$ kΩ. Use distinct colors for the input and output traces. Run an interactive simulation to view the input and output waveforms. Use the oscilloscope cursor to determine the maximum value of $v_o(t)$.

2. The output is to be limited to 6 volts peak when $R_1 = 2.2$ kΩ and $R_2 = 47$ kΩ. Adjust the maximum peak voltage of the input triangle waveform until the output peak voltage is 6 volts. Report this maximum peak input voltage.

3. The amplifier of Part (1) is *cascaded* with the amplifier of Part (2), i.e., the output of the first amplifier feeds the input of the second to make a two-stage amplifier. Plot the output of the second stage when the input to the first stage is a 200-mV peak square wave at 100 Hz; use the same volts-per-division setting on the oscilloscope. Use the oscilloscope cursor to determine the maximum value of $v_o(t)$.

Additional Multisim tips for this problem:

- Ground the center terminal of the function generator and leave the "–" terminal unconnected.

- Note that connecting the power supplies to pins 8 and 4 of the "A" device causes "X"s to appear at the same pins on the "B" device because both devices share the same power supply connections.

NI Multisim video tutorials:

- Place dual op amp second device:
  `http://youtu.be/-QDFEf-KdEw`
- VCC and VEE power supply voltages:
  `http://youtu.be/XkZTwKD-WjE`
- Function generator:
  `http://youtu.be/CeOl6EzD-_c`
- 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`

Measurement with NI myDAQ

1. Build the circuit of Figure 2.4 on page 63 with $R_1 = 10\ \text{k}\Omega$ and $R_2 = 10\ \text{k}\Omega$, and establish connections for the function generator and oscilloscope as follows:

   - Select the "A" device of the TL072 op amp,
   - Use the ELVISmx Function Generator on AO0 to create the triangle wave input signal described in the problem statement,
   - Use the ELVISmx Oscilloscope to plot the input $v_\text{i}(t)$ on AI0 and the output $v_\text{o}(t)$ on AI1; use the same volts-per-division setting for both channels.

   Use the oscilloscope cursor to determine the maximum value of $v_\text{o}(t)$.

2. The output is to be limited to 6 volts peak when $R_1 = 2.2\ \text{k}\Omega$ and $R_2 = 47\ \text{k}\Omega$. Construct this amplifier using the "B" device of the TL072. Adjust the maximum peak voltage of the input triangle waveform

until the output peak voltage is 6 volts. Report this maximum peak input voltage.

3. The amplifier of Part (1) is *cascaded* with the amplifier of Part (2), i.e., the output of the first amplifier feeds the input of the second to make a two-stage amplifier. Plot the output of the second stage when the input to the first stage is a 200-mV peak square wave at 100 Hz; use the same volts-per-division setting on the oscilloscope. Use the oscilloscope cursor to determine the maximum value of $v_{\mathrm{o}}(t)$.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 for both 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 for V_{CC} and -15V for V_{EE}; use AGND for circuit ground.

NI myDAQ video tutorials:

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

2.5 Difference Amplifier

Figure 2.5 shows a *difference amplifier*, a circuit that amplifies the difference between its two input signals.

NOTE: A three-way numerical comparison is not required for this problem.

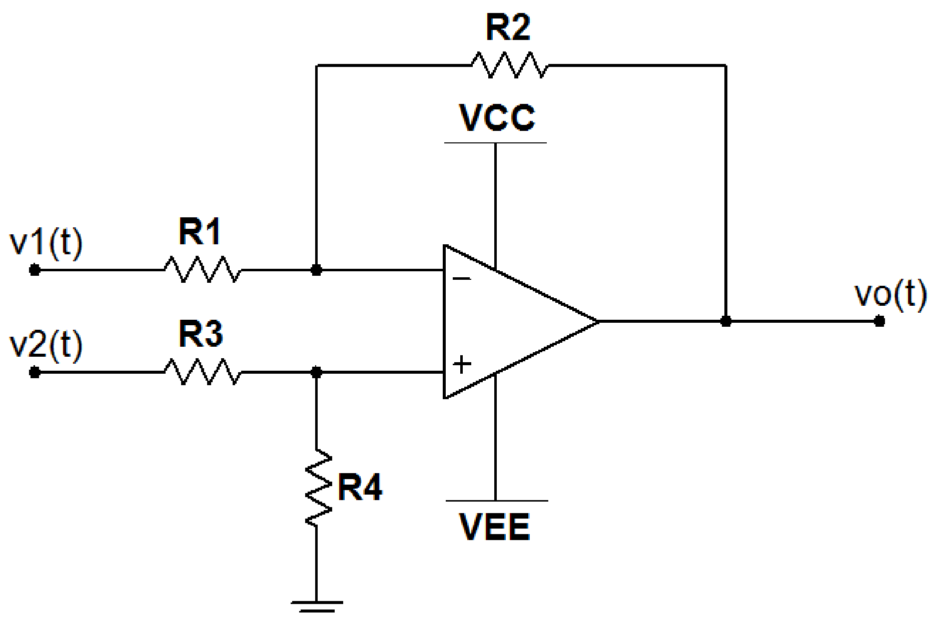


Figure 2.5: Circuit for Problem 2.5

Analysis

1. Derive the equation for v_o in terms of generic resistor values and the input voltages v_1 and v_2. HINT: Apply the superposition circuit analysis technique.

2. Derive the equation for v_o for the special case of $R_3 = R_1$ and $R_4 = R_2$.

3. Plot $v_o(t)$ for the specific inputs and resistor values listed in the table below.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	-15 volts
R_1 = R_3	10 kΩ
R_2 = R_4	100 kΩ
$v_1(t)$	Squarewave, 500 Hz, 0.1 V amplitude, zero DC offset
$v_2(t)$	Sinewave, 100 Hz, 0.5 V amplitude, zero DC offset

Simulation with NI Multisim

Enter the circuit of Figure 2.5 on the preceding page using the `TL072CP` op amp model. Place the `VCC` and `VEE` power supplies and set their values to $+15$ V and -15 V.

1. Use a pair of Simulate → Instruments → Function Generator devices to create the input signals $v_1(t)$ and $v_2(t)$ described in the problem statement. Use the Simulate → Instruments → Four Channel Oscilloscope to plot the two input signals as well as the output $v_{\mathrm{o}}(t)$. Adjust the oscilloscope "Y position" setting for each channel to create three non-overlapping traces; use the same "Scale" setting for the two input signals. Use a distinct color for each trace. Run an interactive simulation to view the signals, and then take cursor measurements to determine the values of $v_{\mathrm{o}}(t)$ at the times $t = 2.5$, 6.5, and 8.5 ms.

NI Multisim video tutorials:

- Function generator:
  `http://youtu.be/CeOl6EzD-_c`
- Basic operation of the four-channel oscilloscope:
  `http://youtu.be/iUqs_c1Bc4Y`
- Distinguish oscilloscope traces by color:
  `http://youtu.be/bICbjggcTiQ`

Measurement with NI myDAQ

1. Build the circuit of Figure 2.5 on the previous page. Use the NI ELVISmx Arbitrary Waveform Generator (ARB) to create the input signals $v_1(t)$ and $v_2(t)$ described in the problem statement. Use AO0 for $v_1(t)$ and

AO1 for $v_2(t)$. Use the NI ELVISmx Oscilloscope to plot $v_1(t)$ and $v_\text{o}(t)$ and then $v_1(t)$ and $v_\text{o}(t)$.

2. Comment on the degree to which your measurement results agree with your simulation results.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 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 for V_CC and -15V for V_EE; use AGND for circuit ground.

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`

2.6 Instrumentation Amplifier

Figure 2.6 on the next page shows an *instrumentation amplifier*, a circuit similar in purpose to the difference amplifier of Figure 2.5 on page 67, but with improved performance in terms of input resistance, differential gain, and common-mode rejection ratio.

The instrumentation amplifier operates as follows:

$$v_o = (1 + 2\frac{R_1}{R_G})(\frac{R_3}{R_2})(v_+ - v_-) \tag{2.1}$$

Note that the instrumentation amplifier gain can be adjusted by all four resistor values, however, it is common to make the single adjustment to R_G, hence "G" stands for gain.
Additional definitions used in this problem:

- Common-mode input voltage: v_{iCM} = $(v_+ + v_-)/2$
- Differential input voltage: v_{iD} = $v_+ - v_-$
- Common-mode gain: $A_{CM} = \frac{v_o}{v_{iCM}}$ for v_{iD} = 0
- Differential gain: $A_D = \frac{v_o}{v_{iD}}$ for v_{iCM} = 0
- Common-mode rejection ratio (CMRR) in decibels: $20\log_{10}\frac{|A_D|}{|A_{CM}|}$

Analysis

Assume ideal op amp behavior for this section.

1. Determine the output offset voltage when v_+ = v_- = 0. Subtract this voltage from all subsequent output measurements.
2. Determine the common-mode gain A_{CM} for a 10 V common-mode input voltage.
3. Determine the differential gain A_D.
4. Calculate the common-mode rejection ratio in dB.

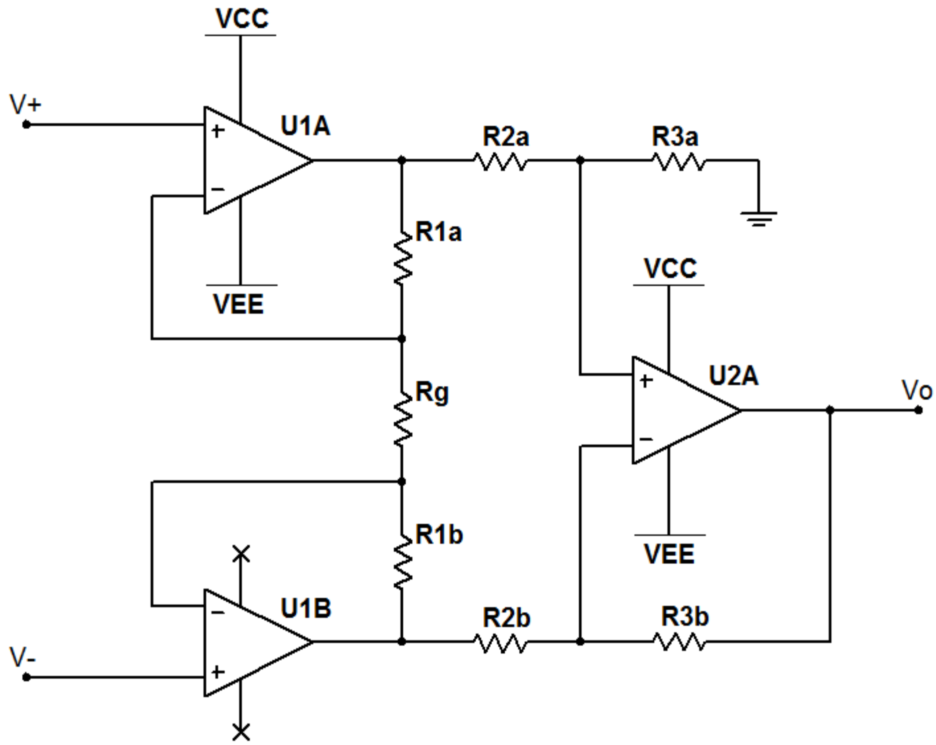


Figure 2.6: Circuit for Problem 2.6 on the preceding page

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	−15 volts
R_G	1 kΩ
R_1	1 kΩ
R_2	2.2 kΩ
R_3	100 kΩ

Simulation with NI Multisim

Enter the circuit of Figure 2.6 using the `TL072CP` op amp model. Place the `VCC` and `VEE` power supplies and set their values to +15 V and −15 V.

1. Use a single DC voltage source `DC_POWER` to drive both inputs of the instrumentation amplifier. Use a Simulate → Instruments → Measurement Probe to monitor the amplifier output, and set its display precision to show four digits. Set the voltage source to zero volts. Run an interactive simulation to determine the offset voltage. Subtract this voltage from all subsequent output measurements.

2. Set the DC voltage source to 10 volts. Re-run the simulation to determine the common-mode output voltage. Use this information to calculate the common-mode gain A_{CM}.

3. Place a second DC voltage, connecting one source to v_+ and the second to v_-. Set v_+ to 15 mV and v_- to -15 mV. Observe the output voltage, and then calculate the differential gain A_{D}.

4. Calculate the common-mode rejection ratio in dB.

NI Multisim video tutorials:

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

Measurement with NI myDAQ

Build the circuit of Figure 2.6 on the previous page. Use the ELVISmx Oscilloscope to monitor the value of v_{o} using the RMS indicator under the main display. Set the "Volts/Div" scale to its most sensitive value (10 mV) and ensure that Channel 1 is disabled.

1. Connect AO0 to both v_+ and v_-. Use the NI ELVISmx Arbitrary Waveform Generator (ARB) to create a constant 10 V DC level on AO0. The sampling rate need not be high; 1.0 kS/s is sufficient. Set the gain to zero and measure the instrumentation amplifier's offset voltage. Subtract this voltage from all subsequent output measurements.

2. Return the ARB gain to 1.00 (which applies 10 volts to both inputs) and measure the common-mode output voltage. Use this information to calculate the common-mode gain A_{CM}.

3. Disconnect v_- from AO0 and instead connect it to AO1. Create another waveform for the ARB, this time at a constant 1 mV DC level.

Use this waveform for both analog analog outputs. Set the ARB gain to $+15.00$ for AO0 and to -15.00 for AO1. Observe the output voltage, and then calculate the differential gain A_{D}.

Additional myDAQ tips for this problem:

- Use two Texas Instruments TL072 op amps described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 for both 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 for V_{CC} and -15V for V_{EE}; use AGND for circuit ground.

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 instrumentation amplifier provides an excellent way to measure and amplify small differential voltages associated with sensors with minimal disruption to the sensor itself. Now that you have built and tested your instrumentation amplifier, use it to amplify the diode temperature sensor of Problem 1.2 on page 19. Follow the procedure in the "NI myDAQ Measurements" section for that problem, changing the oscilloscope volts-per-division setting to a suitable value. You could also use the DMM voltmeter to monitor the amplifier output. Replace resistor R_1 with a 10 kΩ potentiometer and adjust its value to "null" the instrumentation amplifier's offset voltage. If you have previously completed Problem 1.2 on page 19, then compare and contrast your earlier results with those from your instrumentation amplifier.

You may enjoy a visual indicator of the temperature, too. Use the dual LED indicator of Figure 2.12 on page 87 using $R_3 = 1$ kΩ. One LED will light as the temperature sensor gets cooler, and the other will light as the sensor gets warmer.

2.7 Integrator

Figure 2.7 shows an *integrator*, a circuit that performs real-time integration of the input signal. Integrators play an important role in analog control systems and other applications.

The integrator operates as follows:

$$v_o(t) = -\frac{1}{RC}\int_{t_0}^{t} v_i(x)\,\mathrm{d}x + v_o(t_0) \tag{2.2}$$

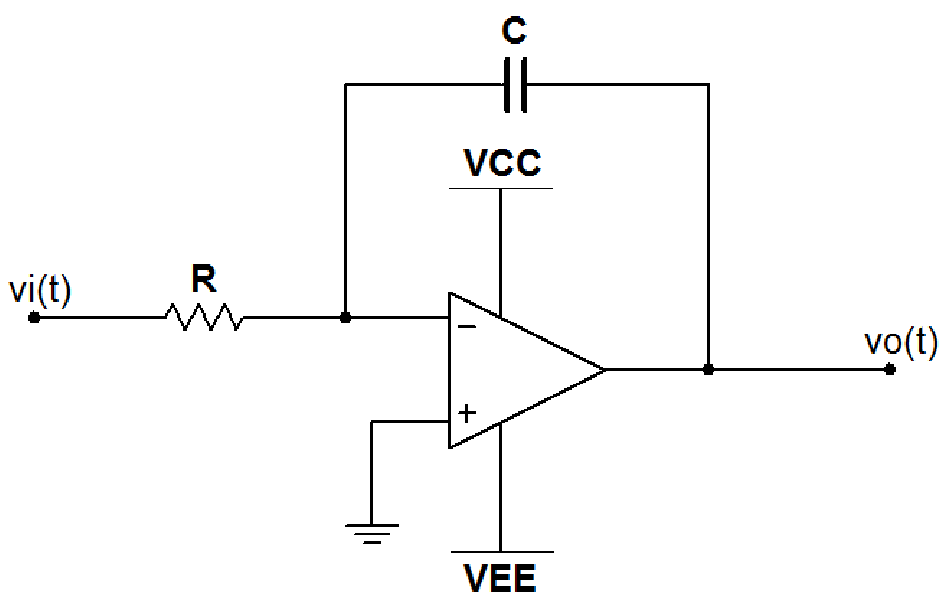


Figure 2.7: Circuit for Problem 2.7

Analysis

1. Let $v_i(t)$ be the pulse waveform of Figure 2.8 on the next page. Plot $v_o(t)$ for $t = 0$ to $t = 10$ ms for an initial integrator output of zero volts, and determine $v_o(t)$ at the times $t = 2.5$, 4.5, and 9.0 ms.

2. Plot $v_o(t)$ when $v_i(t)$ is a square wave with 0.5 V amplitude (1 V peak to peak) at 50, 100, and 200 Hz. Determine the peak-to-peak voltage of $v_o(t)$ for each frequency.

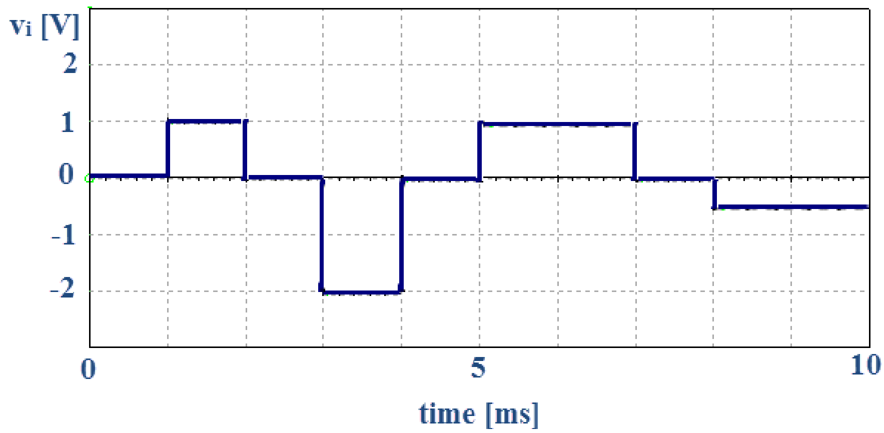


Figure 2.8: Input waveform for Problem 2.7 on the facing page

3. Consider your results from the previous step, and explain the relationship between the frequency of the input and the amplitude of the output.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	−15 volts
R	10 kΩ
C	0.1 μF

Simulation with NI Multisim

Enter the circuit of Figure 2.7 on the facing page using the `TL072CP` op amp model. Place the `VCC` and `VEE` power supplies and set their values to +15 V and −15 V.

IMPORTANT: Select Simulate → Interactive Simulation Settings and choose "Set to zero" for the "Initial conditions" field to ensure that the integrator output begins at zero.

1. Use the `PIECEWISE_LINEAR_VOLTAGE` source to create the pulse waveform shown in Figure 2.8, and use the Simulate → Instruments →

Oscilloscope to plot the signals $v_i(t)$ and $v_o(t)$. Use a distinct color for each trace. Run an interactive simulation to view the signals, and then take cursor measurements to determine the values of $v_o(t)$ at the times $t = 2.5$, 4.5, and 9.0 ms.

2. Place a Simulate → Instruments → Function Generator configured to create the square wave described in the problem statement. Plot the signals $v_i(t)$ and $v_o(t)$ for the frequencies 50, 100, and 200 Hz (create one plot for each frequency), and then take cursor measurements to determine the peak-to-peak voltages of each output. Use the *same* oscilloscope settings for all three waveform plots.

Additional Multisim tips for this problem:

- Consider creating the piecewise linear voltage source data points in a text file and then importing that file.

NI Multisim video tutorials:

- Piecewise linear (PWL) voltage source:
  `http://youtu.be/YYU5WuyebD0`
- Function generator:
  `http://youtu.be/CeOl6EzD-_c`
- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Distinguish oscilloscope traces by color:
  `http://youtu.be/bICbjggcTiQ`

Measurement with NI myDAQ

Build the circuit of Figure 2.7 on page 74. IMPORTANT: Connect a 1.0 MΩ resistor in parallel with the capacitor. The "Further Exploration" section below examines the reason for this resistor.

1. Use the NI ELVISmx Arbitrary Waveform Generator (ARB) to create the pulse waveform shown in Figure 2.8 on the previous page, and use the NI ELVISmx Oscilloscope to plot the signals $v_i(t)$ and $v_o(t)$. Take cursor measurements to determine the values of $v_o(t)$ at the times $t = 2.5$, 4.5, and 9.0 ms. Expect some amount of DC offset

in the output signal, therefore measure the voltage at time $t = 0.5$ ms and subtract this value from each measurement.

2. Set up the NI ELVISmx Function Generator to produce the square wave. Plot the signals $v_{\rm i}(t)$ and $v_{\rm o}(t)$ for the frequencies 50, 100, and 200 Hz (create one plot for each frequency), and then read the peak-to-peak voltage indicator under the main display. Use the *same* oscilloscope settings for all three waveform plots.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 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 for $V_{\rm CC}$ and -15V for $V_{\rm EE}$; use AGND for circuit ground.
- Use edge triggering and adjust the horizontal position to make the leading edge of the pulse waveform appear at the far left side of the oscilloscope display.

NI myDAQ video tutorials:

- 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`

Further Exploration with NI myDAQ

What purpose does the 1.0 MΩ resistor serve? To find out, observe $v_{\rm o}(t)$, disconnect the resistor, and then connect it again. Summarize the behavior you observed, and then propose an explanation.

2.8 Differentiator

Figure 2.9 shows a *differentiator*, a circuit that evaluates in real time the derivative (slope) of the input signal. Differentiators can detect abrupt changes or transitions in a signal.

The differentiator operates as follows:

$$v_{\mathrm{o}}(t) = -RC\frac{dv_{\mathrm{o}}(t)}{dt} \tag{2.3}$$

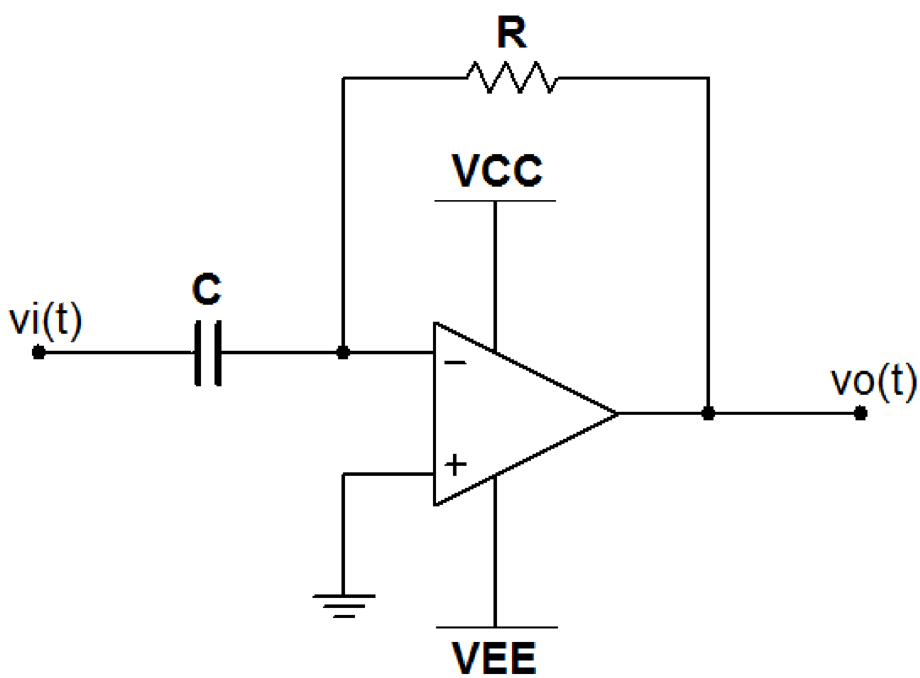


Figure 2.9: Circuit for Problem 2.8

Analysis

1. Let $v_{\mathrm{i}}(t)$ be the ramp waveform of Figure 2.10 on the facing page. Plot $v_{\mathrm{o}}(t)$ for $t = 0$ to $t = 10$ ms and determine $v_{\mathrm{o}}(t)$ at the times $t = 3.5$, 6.5, and 8.5 ms.

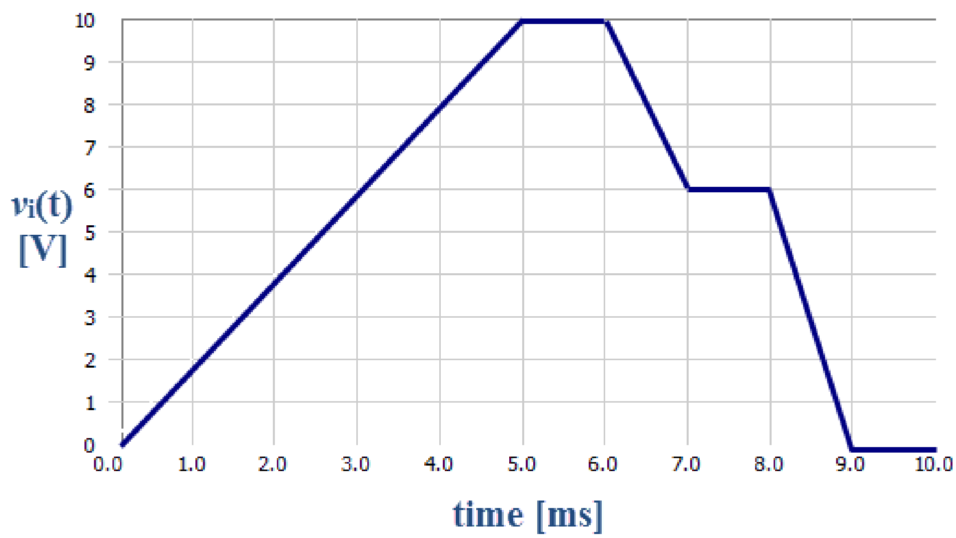


Figure 2.10: Input waveform for Problem 2.8 on the preceding page

2. Plot $v_o(t)$ when $v_i(t)$ is a sinewave with 4 V amplitude (8 V peak to peak) at 50, 100, and 200 Hz. Determine the peak-to-peak voltage of $v_o(t)$ for each frequency.

3. Consider your results from the previous step, and explain the relationship between the frequency of the input and the amplitude of the output.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	-15 volts
R	10 kΩ
C	0.1 μF

Simulation with NI Multisim

Enter the circuit of Figure 2.9 on the preceding page using the `TL072CP` op amp model. Place the `VCC` and `VEE` power supplies and set their values to $+15$ V and -15 V.

IMPORTANT: Insert a 100 Ω resistor in series with the capacitor to ensure that the simulator converges properly.

1. Use the `PIECEWISE_LINEAR_VOLTAGE` source to create the ramp waveform shown in Figure 2.10 on the preceding page, and use the Simulate → Instruments → Oscilloscope to plot the signals $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$. Use a distinct color for each trace. Run an interactive simulation to view the signals, and then take cursor measurements to determine the values of $v_\mathrm{o}(t)$ at the times $t =$ 2.5, 6.5, and 8.5 ms.

2. Place a Simulate → Instruments → Function Generator configured to create the sinewave described in the problem statement. Plot the signals $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$ for the frequencies 50, 100, and 200 Hz (create one plot for each frequency), and then take cursor measurements to determine the peak-to-peak voltages of each output. Use the *same* oscilloscope settings for all three waveform plots.

Additional Multisim tips for this problem:

- Consider creating the piecewise linear voltage source data points in a text file and then importing that file.

NI Multisim video tutorials:

- Piecewise linear (PWL) voltage source:
  `http://youtu.be/YYU5WuyebD0`
- Function generator:
  `http://youtu.be/CeOl6EzD-_c`
- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Distinguish oscilloscope traces by color:
  `http://youtu.be/bICbjggcTiQ`

Measurement with NI myDAQ

Build the circuit of Figure 2.9 on page 78. IMPORTANT: Connect a 100 Ω resistor in series with the capacitor. The "Further Exploration" section below examines the reason for this resistor.

1. Use the NI ELVISmx Arbitrary Waveform Generator (ARB) to create the ramp waveform shown in Figure 2.10 on page 79, and use the NI ELVISmx Oscilloscope to plot the signals $v_{\mathrm{i}}(t)$ and $v_{\mathrm{o}}(t)$. Take cursor measurements to determine the values of $v_{\mathrm{o}}(t)$ at the times t = 3.5, 6.5, and 8.5 ms.

2. Set up the NI ELVISmx Function Generator to produce the sinewave. Plot the signals $v_{\mathrm{i}}(t)$ and $v_{\mathrm{o}}(t)$ for the frequencies 50, 100, and 200 Hz (create one plot for each frequency), and then read the peak-to-peak voltage indicator under the main display. Use the *same* oscilloscope settings for all three waveform plots.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 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 for V_{CC} and -15V for V_{EE}; use AGND for circuit ground.

- Use edge triggering and adjust the horizontal position to make the leading edge of the ramp waveform appear at the far left side of the oscilloscope display.

NI myDAQ video tutorials:

- 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`

Further Exploration with NI myDAQ

1. What purpose does the 100-ohm resistor serve? To find out, observe $v_{\mathrm{o}}(t)$, short the resistor with a jumper wire, and then remove the jumper. Summarize the behavior you observed, and then propose an explanation.

2. The differentiator is useful as a *transition detector*. To see why, apply a 100-Hz square wave to the differentiator input. Discuss your observations of the output. Use the oscilloscope "Vertical Position" controls to vertically offset the input and output waveforms to make them easier to distinguish.

3. Differentiators are one of those circuits that require additional components to work properly in the "real world." You have already seen how the small series resistor with the capacitor improves performance. Differentiators are also notorious "noise amplifiers" because they are sensitive to *changes* in the signal. To experience this effect and learn how to overcome it, do the following:

   (a) Use the NI ELVISmx Arbitrary Waveform Generator to create a 1 V amplitude 100 Hz square wave with additive Gaussian noise at level 0.1: set the "Sample Rate" to 200 and the "Units" to "kHz," create a single 10 ms segment, and then create *two* components, one for the square wave and the other for the noise. Ensure that "Function" is set to "+" so that the two components add together.

   (b) View the differentiator output. Describe how well you can view the transitions in the square wave.

   (c) Place a 0.01 μF capacitor in parallel with the feedback resistor R. Describe how well you can view the transitions in the square wave.

2.9 Precision Rectifier

Figure 2.11 illustrates a *precision rectifier*, a circuit similar in purpose to the half-wave rectifier of Problem 1.6 on page 33 but with improved performance.

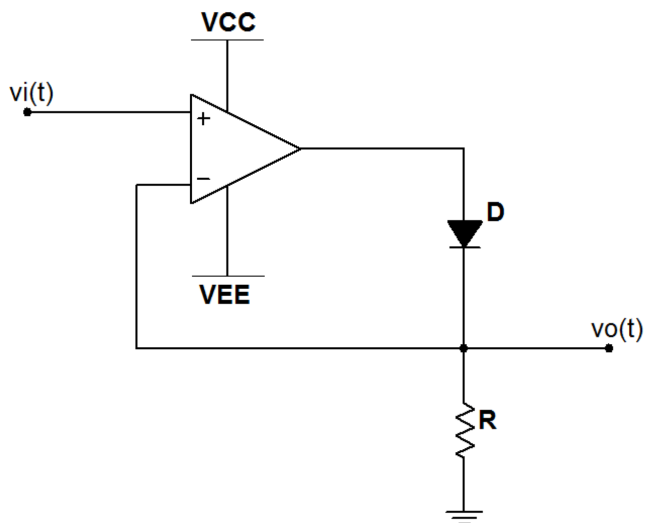


Figure 2.11: Circuit for Problem 2.9

Analysis

1. Determine an equation for v_o when $v_i > 0$ and when $v_i < 0$.
2. Plot two periods of $v_i(t)$ and $v_o(t)$ for each of the three input waveforms shown in Figure 1.7 on page 37.
3. Determine the following numerical descriptors for each type of waveform output $v_o(t)$:
   (a) Peak value, and
   (b) Effective DC value, also known as RMS value. HINT: Refer to `http://en.wikipedia.org/wiki/Root_mean_square` for the RMS values of the original waveforms in terms of the peak value V_M.

4. Compare and contrast your results for this problem with those of Problem 1.6 on page 33.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	-15 volts
R	1 kΩ
D	0.7 volt forward-biased voltage drop, ideal diode otherwise
$v_{\mathrm{i}}(t)$	V_{M} = 4 V, 100 Hz frequency ($T = 0.01$ ms)

Simulation with NI Multisim

Enter the circuit of Figure 2.11 on the previous page using the `TL072CP` op amp model. Place the `VCC` and `VEE` power supplies and set their values to $+15$ V and -15 V. Use the Simulate → Instruments → Function Generator and Simulate → Instruments → Oscilloscope instruments. Use a Simulate → Instruments → Measurement Probe configured to display RMS voltage to measure the effective DC value of the output $v_{\mathrm{o}}(t)$.

1. Plot two periods of $v_{\mathrm{i}}(t)$ and $v_{\mathrm{o}}(t)$ for each of the three input waveforms shown in Figure 1.7 on page 37.

2. Determine the following numerical descriptors for each type of waveform output $v_{\mathrm{o}}(t)$:

   (a) Peak value (use a cursor measurement), and

   (b) Effective DC value, also known as RMS value.

Additional Multisim tips for this problem:

- Ground the center terminal of the function generator and leave the "–" terminal unconnected.
- Set distinct colors for the waveform traces to easily distinguish the input and output waveforms.
- Refer to the tutorial videos below to learn how to take cursor measurements from the oscilloscope.

NI Multisim video tutorials:

- Function generator:
  http://youtu.be/CeOl6EzD-_c
- 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
- Measure RMS and average value with a measurement probe:
  http://youtu.be/OnK-Unld17E
- Find the maximum value of a trace in Grapher View:
  http://youtu.be/MzYK60mfh2Y

Measurement with NI myDAQ

Construct the circuit of Figure 1.6 on page 33 using a TL072 op amp and a 1N4148 diode. Use the ELVISmx Function Generator to create $v_\mathrm{i}(t)$ and the ELVISmx Oscilloscope to display both $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$.

1. Plot two periods of $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$ for each of the three input waveforms shown in Figure 1.7 on page 37.
2. Determine the following numerical descriptors for each type of waveform output $v_\mathrm{o}(t)$:
   (a) Peak value (use a cursor measurement), and
   (b) Effective DC value, also known as RMS value; read this value in the measurements display area under the waveform display.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 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 for V_{CC} and -15V for V_{EE}; use AGND for circuit ground.
- Adjust the "Vertical Position" controls for each oscilloscope channel to place the input waveform in the top half of the display and the output waveform in the bottom half; you will likely not be able to distinguish the output waveform otherwise.

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

Replace the 1N4148 diode with a light-emitting diode, or LED. Follow the diagram of Figure D.2 on page 182 to determine the anode and cathode connections.

1. How does the waveform for $v_{\mathrm{o}}(t)$ compare to your earlier results with the 1N4148 diode? Recalling that the LED has approximately three times higher voltage drop than the 1N4148 diode, propose an explanation for your observations.
2. Reduce the input amplitude to 1 Vpp on the function generator. Experiment with the three different waveform shapes while the function generator remains set to 100 Hz, paying particular attention to the brightness of the LED. Discuss your observations of waveform shape and brightness and relate these observations to your measured effective DC values for each waveform shape.
3. Reduce the function generator frequency, and experiment with values as low at 0.2 Hz (one cycle every five seconds). Discuss the behavior of the LED optical intensity for each of the three waveform shapes when the function generator frequency is 1 Hz or less.
4. At what function generator frequency does the flashing LED begin to appear as a constant intensity?

2.10 Schmitt Trigger

NOTE: Problem 2.1 on page 51 should be worked before attempting this problem.

Figure 2.12 shows a *Schmitt trigger comparator*, a circuit that introduces *hysteresis* to the comparison threshold. Hysteresis, in essence, produces two distinct threshold voltages; the output state of the op amp selects the active threshold voltage, consequently the threshold depends on the time history of the input.

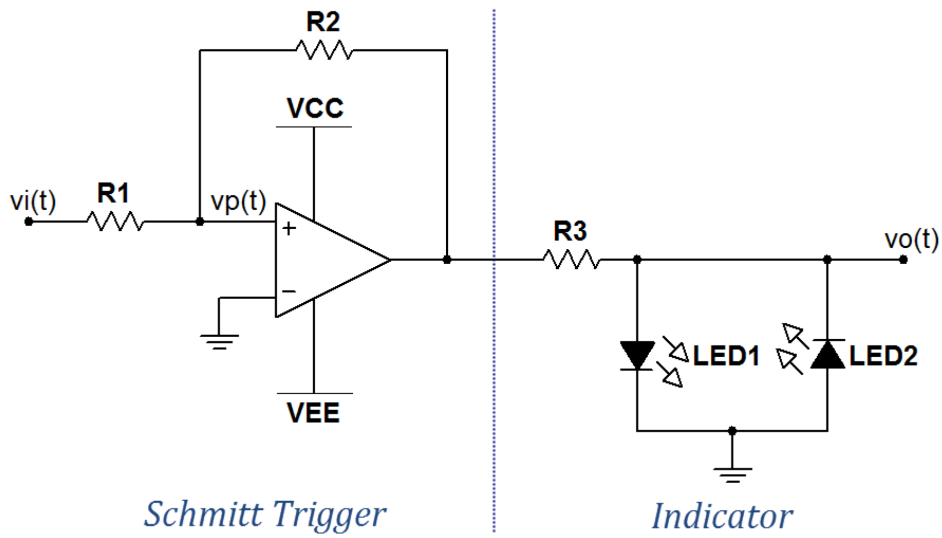


Figure 2.12: Circuit for Problem 2.10

Analysis

1. Determine the two threshold voltages V_{LH} (as the input v_{i} crosses the threshold from low to high) and V_{HL} (as the input crosses the threshold from high to low). The TL072 op amp saturates at approximately 1.5 V inside each power supply voltage, i.e., the op amp output voltage is limited to ± 13.5 V.

2. Let $v_{\mathrm{i}}(t)$ be a 5-volt peak triangle wave at 10 Hz. Plot $v_{\mathrm{i}}(t)$, $v_{\mathrm{p}}(t)$, and $v_{\mathrm{o}}(t)$.

3. Superimpose a 1 kHz sinewave with 1 volt amplitude on the triangle wave to model unwanted high-frequency noise on the input signal. Sketch $v_i(t)$, $v_p(t)$, and $v_o(t)$.

4. Compare and contrast the performance of this Schmitt comparator with that of the basic comparator of Problem 2.1 on page 51 for real analog signals that always have some degree of noise, especially when the comparator output is connected directly to digital circuits.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	-15 volts
LEDs	2 V forward drop
resistors	$R_1 = 10$ kΩ, $R_2 = 100$ kΩ, and $R_3 = 1$ kΩ

Simulation with NI Multisim

Enter the circuit of Figure 2.12 on the previous page using the `TL072CP` op amp model. Place the `VCC` and `VEE` power supplies and set their values to $+15$ V and -15 V. Use `LED_green` and `LED_red` for the LEDs.

1. Use the Simulate → Instruments → Function Generator to create the triangle wave input signal described in the problem statement, and use the Simulate → Instruments → Four Channel Oscilloscope to plot the signals $v_i(t)$, $v_p(t)$, and $v_o(t)$. Use a distinct color for each trace. Run an interactive simulation to view the signals, and then take cursor measurements to determine the two threshold voltages V_{LH} and V_{HL}.

2. Place a second function generator configured to create the sinewave described in the problem statement, and then connect the two function generators in series. Display the signals $v_i(t)$, $v_p(t)$, and $v_o(t)$.

Additional Multisim tips for this problem:

- Ground the center terminal of the triangle wave function generator and leave the "–" terminal unconnected.
- Use the "+" terminal and center terminal of the sinewave function generator.

NI Multisim video tutorials:

- Function generator:
  http://youtu.be/CeOl6EzD-_c
- Basic operation of the four-channel oscilloscope:
  http://youtu.be/iUqs_c1Bc4Y
- Distinguish oscilloscope traces by color:
  http://youtu.be/bICbjggcTiQ

Measurement with NI myDAQ

1. Build the circuit of Figure 2.12 on page 87. Use AO0 to create v_i. Set up the NI ELVISmx Function Generator to create the triangle waveform described in the problem statement. Use the NI ELVISmx Oscilloscope to display $v_\mathrm{i}(t)$ and $v_\mathrm{o}(t)$. Take cursor measurements to determine the two threshold voltages V_LH and V_HL.

2. Set up the NI ELVISmx Arbitrary Waveform Generator (ARB) to produce the triangle waveform and superimposed sinewave:

   - Set the sample rate to 200 kS/s.
   - Create a single segment of length 100 ms.
   - Create *two* components under this segment, one for the triangle wave and another for the sine wave. Ensure that “Function” is set to “+” to add the components together.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 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 for V_CC and -15V for V_EE; use AGND for circuit ground.

NI myDAQ video tutorials:

- 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`

Further Exploration with NI myDAQ

1. Change the sinewave to Gaussian noise (level = 1.00) in the arbitrary waveform generator to get a more realistic view of high-frequency random noise.

   (a) Describe the behavior of the Schmitt trigger with this higher level of random noise.

   (b) Reduce the feedback resistor R_2 to 22 kΩ. Calculate the new values of V_{LH} and V_{HL} and relate these values to the new behavior of the Schmitt trigger.

2. The frequencies used above were selected to easily visualize the signals on the oscilloscope. Reduce the frequencies to make the comparator behavior visible on the LEDs; try reducing the arbitrary waveform generator sampling rate to 5.0 kS/s.

2.11 Squarewave Oscillator

An operational amplifier serves as the basis of the *square wave oscillator* pictured in Figure 2.13. The oscillation period is $T = 2RC\ln(1 + 2(R_1/R_2))$.

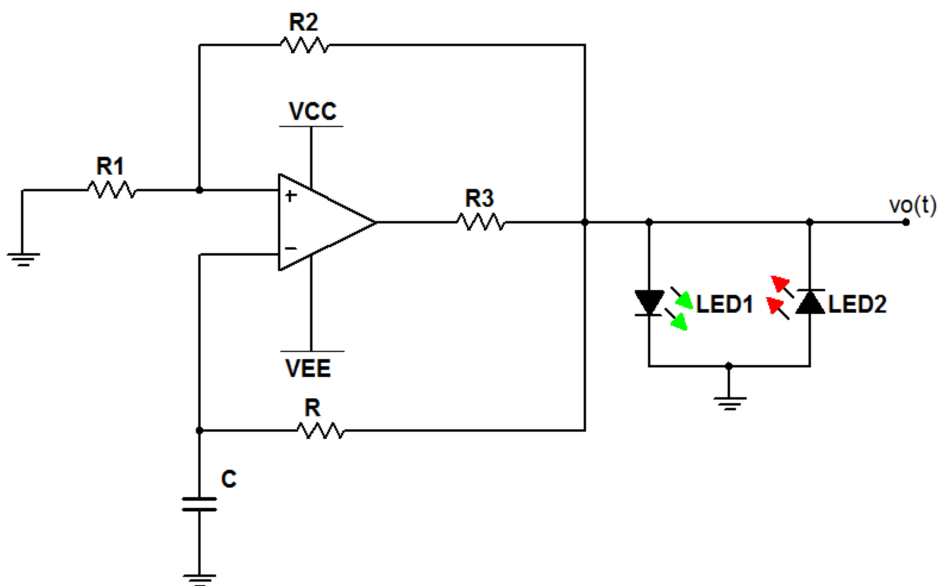


Figure 2.13: Circuit for Problem 2.11

Analysis

1. Determine the frequency of oscillation in hertz using the component values listed below.

2. Determine values for R and C to set the oscillation frequency within $\pm 5\%$ of 1 Hz. Refer to the parts list of Appendix A on page 167 for available values of resistors and capacitors. Determine the actual oscillation frequency using your selected values.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	−15 volts
LEDs	2 V forward drop
R_1	10 kΩ
R_2	10 kΩ
R_3	1 kΩ
R	33 kΩ
C	0.1 μF

Simulation with NI Multisim

Enter the circuit of Figure 2.13 on the previous page using the `TL072CP` op amp model. Place the `VCC` and `VEE` power supplies and set their values to $+15$ V and -15 V. Use `LED_green` and `LED_red` for the LEDs.

1. View the oscillator output with Simulate → Instruments → Oscilloscope to plot $v_o(t)$. Take cursor measurements to determine the period of the square wave, and then calculate the frequency.

2. Update the values of R and C with your calculated values, re-run the simulator, and measure the oscillation frequency. See the tips section below to learn how to adjust the simulator settings for a low oscillation frequency.

Additional Multisim tips for this problem:

- Increase the accuracy of your period measurement by measuring the time of N cycles and then dividing by N.

- Make the following adjustments to allow the simulation to run in a reasonable amount of time:

  – Double-click the capacitor, enable "Initial conditions," and enter 0 volts,

  – Select "Simulate — Interactive Simulation Settings,"

  – Change "Initial conditions" to "User-defined," and

  – Set "Maximum time step" to 0.01 seconds.

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`

Measurement with NI myDAQ

1. Build the circuit of Figure 2.13 on page 91. Use the NI ELVISmx Oscilloscope to display $v_o(t)$. Read the frequency indicator underneath the main display; ensure that you display at least two full periods.
2. Replace R and C with your selected components and measure the oscillation frequency.

Additional myDAQ tips for this problem:

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 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 for V_{CC} and -15V for V_{EE}; use AGND for circuit ground.

NI myDAQ video tutorials:

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

Further Exploration with NI myDAQ

1. Develop insight into how this oscillator works: display the two op amp input signals on the oscilloscope (each with respect to ground), study their behavior, and then develop an explanation of the principle of operation of this square wave oscillator.
2. Measure and make note of the actual forward voltage drop of your LEDs. How close is the value to that obtained by simulation?

Chapter 3

Bipolar Junction Transistor (BJT) Circuits

3.1 Logic Inverter

A BJT serves as the heart of a *logic inverter* in Figure 3.1, a basic component of digital logic circuits. In this application the transistor is driven either to saturation (the "ON" state) or to cutoff (the "OFF" state).

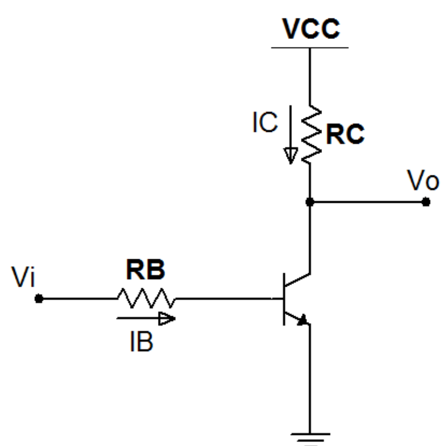


Figure 3.1: Circuit for Problem 3.1

Analysis

1. Determine the value of R_B to establish a "forced beta" (defined as β_forced = I_Csat / I_B) of 5 when the input is 3.3 V. Use this resistor value for the remainder of the problem.

2. Determine the following values when V_i is 3.3 V: V_o, I_C, I_B, R_in (the effective input resistance of the inverter), and β_forced.

3. Plot $v_\mathrm{o}(t)$ when $v_\mathrm{i}(t)$ is a 3.3 V peak-to-peak 100-Hz square wave with zero minimum value.

Use the following circuit components:

Component	Value
V_CC	15 volts
BJT	$\beta = 200$, V_BE = 0.7 volts, V_CEsat = 0.1 volts
R_C	3.3 kΩ

Simulation with NI Multisim

Enter the circuit of Figure 3.1 on the preceding page using the `2N2222A` transistor model. Set R_B to the value you calculated in the analysis section.

1. Determine the following values when V_i is 3.3 V: V_o, I_C, I_B, R_in, and β_forced.

2. Plot $v_\mathrm{o}(t)$ when $v_\mathrm{i}(t)$ is a 3.3 V peak-to-peak 100-Hz square wave with zero minimum value.

Additional Multisim tips for this problem:

- Use a DC source as the input when measuring currents and voltages.
- Use the Function Generator as the 100-Hz square wave source. Recognize that "Vp" signifies "peak" voltage which is the same as amplitude.
- Adjust the "Y position" of the oscilloscope traces to vertically separate them for easier comparison.
- Measure R_in as the input voltage divided by the input current.

NI Multisim video tutorials:

- Find components by name:
  http://youtu.be/5wlFweh4n-c
- Measure DC node voltage with a measurement probe:
  http://youtu.be/svNGHA2-uK4
- Measure DC current with a measurement probe:
  http://youtu.be/uZ56byigymI
- Basic operation of the two-channel oscilloscope:
  http://youtu.be/qnRK6QyqjvQ
- Distinguish oscilloscope traces by color:
  http://youtu.be/bICbjggcTiQ
- Function generator:
  http://youtu.be/CeOl6EzD-_c

Measurement with NI myDAQ

Build the circuit of Figure 3.1 on page 95 using the 2N2222A transistor. Use a 10 kΩ potentiometer for R_{B}; adjust its value to that which you calculated in the analysis section.

1. Measure the following values when V_{i} is 3.3 V: V_{o}, I_{C}, I_{B}, R_{in}, and β_{forced}.
2. Plot $v_{\mathrm{o}}(t)$ when $v_{\mathrm{i}}(t)$ is a 3.3 V peak-to-peak 100-Hz square wave with zero minimum value.

Additional myDAQ tips for this problem:

- Use the 2N2222A BJT transistor described in Appendix E. Follow the pinout diagram of Figure E.1 on page 185 to determine the base, collector, and emitter connections.
- Power the circuit with myDAQ 5V for V_{CC} and AGND for the circuit ground.
- Create the 3.3 V constant voltage using either of these techniques:
  - Digital output DIO0 operated by the NI ELVISmx Digital Writer; you may continue to use AGND because the digital ground DGND connects internally to AGND, or

  - Analog output AO0 operated by the NI ELVISmx Function Generator with a zero-volt amplitude signal and a 3.3 V DC offset.

- Use the NI ELVISmx DMM as a voltmeter and ammeter as required. Do NOT attempt to measure R_{in} with the ohmmeter! Remember that the ohmmeter cannot be used on an active circuit. Instead, measure R_{in} as the input voltage divided by the input current.

- Use the Function Generator to create the 100-Hz square wave (be sure to adjust the DC offset to achieve zero minimum value) and view $v_{\text{o}}(t)$ and $v_{\text{i}}(t)$.

NI myDAQ video tutorials:

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

Digital circuits operate on the assumption that signals exist in one of only two states, high or low. What happens if the input signal does not obey this assumption, as could be the case when the input is derived from some external (uncontrolled) signal? Change the input to a 100-Hz triangle wave (keep the same DC offset you used earlier) and plot $v_{\text{o}}(t)$ and $v_{\text{i}}(t)$. Respond to the following points:

1. Describe the general characteristics of the output waveform $v_{\text{o}}(t)$.

2. What is the logic inverter's switching threshold voltage, i.e., at what value of v_{i} does the output v_{o} change states?

3. Predict how subsequent circuits designed on the two-level assumption will react to $v_{\mathrm{o}}(t)$.

3.2 Current Source

Figure 3.2 shows an NPN transistor connected to establish a constant current in the load resistor independent of transistor beta.

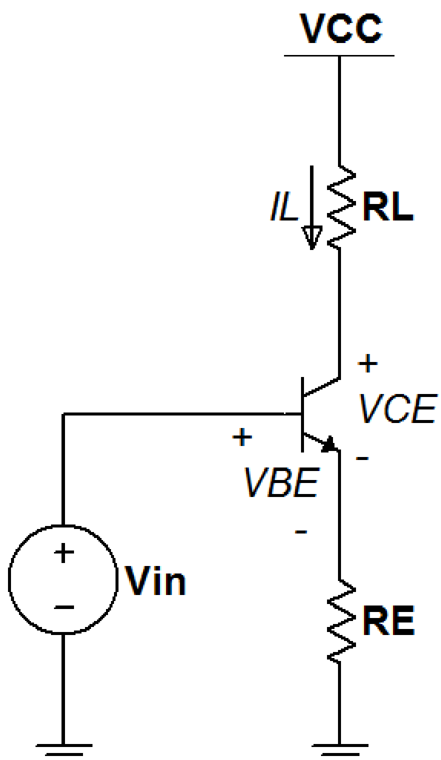


Figure 3.2: Circuit for Problem 3.2

Analysis

1. Derive an equation for the load current I_L as a function of the emitter resistor value R_E and input voltage V_{in} assuming that β is a high value.

2. Use the values of V_{in} and R_E listed in the table below for this and subsequent parts. What is the highest load resistance R_L for which

the equation you derived in the previous step is valid? HINT: Look for the transistor's transition between active and saturation regions, taking V_{CE} = 100 mV as the boundary between the two regions.

3. Calculate I_L for R_L = 0 Ω and 680 Ω.

4. Plot I_L as a function of R_L over the range 0 Ω to 5 kΩ.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
BJT	$\beta = 200$ and V_{BE} = 0.7 volts (active region)
V_{in}	3.0 V
R_E	330 Ω

Simulation with NI Multisim

Enter the circuit of Figure 3.2 on the preceding page using the `2N2222A` transistor model. Set V_{in} = 3 V.

1. Set up a Simulate → Analyses → Parameter Sweep to plot I_L as a function of R_L over the range 0 Ω to 5 kΩ.

2. Take cursor measurements to determine I_L for R_L = 0 Ω and 680 Ω.

3. Identify the highest load resistance R_L for which the current I_L remains constant.

NI Multisim video tutorials:

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

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

Measurement with NI myDAQ

Build the circuit of Figure 3.2 on the facing page using the 2N2222A transistor. Set V_{in} = 3 V and monitor I_L with the DMM ammeter.

1. Measure I_L for R_L = 0 Ω and 680 Ω.

2. Measure I_L for these additional values of R_L: 100, 1000, 2200, 3300, and 4700 ohms. Plot I_L as a function of R_L at these and the previous two values.

3. Identify the highest load resistance R_L for which the current I_L remains constant by following this procedure:

    - Use a 10 kΩ potentiometer connected as a variable resistor for R_L,
    - Start with a low resistance, and then slowly increase the resistance until the current abruptly decreases, and
    - Remove the potentiometer and measure its resistance (remember that you cannot use the ohmmeter while the potentiometer is connected to the circuit).

Additional myDAQ tips for this problem:

- Use the 2N2222A BJT transistor described in Appendix E. Follow the pinout diagram of Figure E.1 on page 185 to determine the base, collector, and emitter connections.

- Power the circuit with myDAQ +15V for V_{CC} and AGND for the circuit ground.

- Implement the voltage source V_{in} with the adjustable DC voltage source circuit of Figure B.2 on page 172.

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

1. This current source uses *negative feedback* to stabilize the operating point of the circuit, and provides a degree of immunity to the value of transistor beta. Try this experiment: replace the load resistor R_L with the DMM ammeter, effectively creating a low-valued load resistance. Record the current I_L. Next, replace the transistor with another of

the same type, and then record I_L again. Repeat for as many transistors as you have available. Also, try using other NPN transistors that you have on hand such as the 2N3904 (remember to observe proper pinout, though, the 2N3904 is different than the 2N2222A). Report the maximum and minimum values you observed, the average value, and the percentage range of variation $(I_{L_{max}} - I_{L_{min}})/I_{L_{avg}}) \times 100\%$.

2. The input voltage V_{in} controls the load current in a proportional fashion, making this circuit an implementation of the *voltage-controlled current source* you recall studying in your circuits course. Place an LED as the load device, and leave the DMM ammeter in place to measure I_L. Connect V_{in} to AO0, and use the NI ELVISmx Function Generator to control the input voltage. Set the amplitude to zero and vary the DC offset. Confirm that you can easily set the LED current to any desired value. Consider using the NI ELVISmx Oscilloscope to monitor AO0 as confirmation that you are controlling V_{in} as you expect.

   (a) At what value of V_{in} is the LED current 1.00 mA?

   (b) At what value of V_{in} does the LED just begin to light? What is the LED current at this value?

3. Extend the previous activity by setting up the function generator as a triangle wave to slowly (and linearly) vary the LED current between its "just visible" level and 1.00 mA. Report your function generator settings for amplitude, DC offset, and frequency. Does the LED brightness appear to be proportional to its current? Explain your findings.

3.3 Fixed Bias

Figure 3.3 shows a simple *fixed bias* circuit for a BJT.

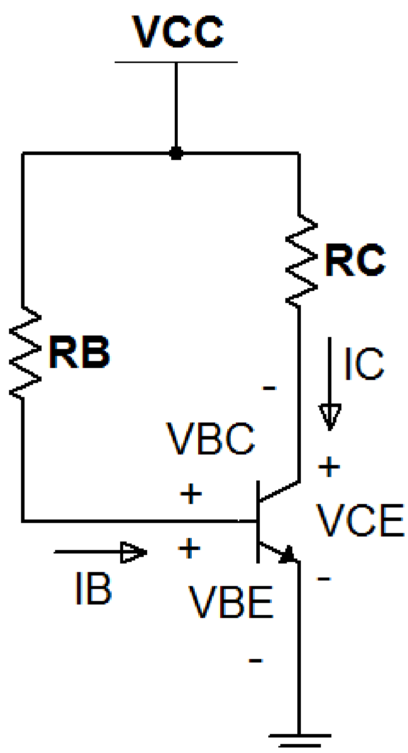


Figure 3.3: Circuit for Problem 3.3

Analysis

1. Find the Q-point (operating point) values for base current I_{B}, collector current I_{C}, collector-emitter voltage V_{CE}, and base-collector voltage V_{BC} for $R_{\mathrm{C}} = 1.0$ kΩ assuming active-region operation. Determine the actual operating region of the transistor by comparing the biases of the base-emitter and base-collector junctions.

2. Repeat for $R_{\mathrm{C}} = 2.2$ kΩ.

3. Repeat for $R_{\mathrm{C}} = 4.7$ kΩ.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
BJT	$\beta = 200$ and V_{BE} = 0.7 volts
R_{B}	470 kΩ

Simulation with NI Multisim

Enter the circuit of Figure 3.3 on the preceding page using the `2N2222A` transistor model. Place the `VCC` power supply and set its value to $+15$ V.

1. Simulate the Q-point (operating point) values for base current I_{B}, collector current I_{C}, collector-emitter voltage V_{CE}, and base-collector voltage V_{BC} for $R_{\mathrm{C}} = 1.0$ kΩ. Also simulate V_{BE} and compare to the assumed value used in the analytical section. State whether the transistor is in active mode or saturated mode, and explain how you know.

2. Repeat for $R_{\mathrm{C}} = 2.2$ kΩ.

3. Repeat for $R_{\mathrm{C}} = 4.7$ kΩ.

Additional Multisim tips for this problem:

- Use either of the following techniques to measure the DC currents and voltages:
  - Interactive analysis with two Simulate → Instruments → Measurement Probe indicators placed at the transistor base and collector and configured to display voltage and current indicators
  - SPICE-style analysis with Simulate → Analyses → DC Operating Point

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Measure DC node voltage with a measurement probe:
  `http://youtu.be/svNGHA2-uK4`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`
- Find node voltages with DC Operating Point analysis:
  `http://youtu.be/gXBCqP17AZs`

Measurement with NI myDAQ

Build the circuit of Figure 3.3 on page 104 using the 2N2222A transistor. Place the `VCC` power supply and set its value to $+15$ V.

1. Measure the Q-point (operating point) values for base current I_{B}, collector current I_{C}, collector-emitter voltage V_{CE}, and base-collector voltage V_{BC} for $R_{\mathrm{C}} = 1.0$ kΩ. Also measure V_{BE} for comparison with the assumed value used in the analytical section.
2. Repeat for $R_{\mathrm{C}} = 2.2$ kΩ.
3. Repeat for $R_{\mathrm{C}} = 4.7$ kΩ.

Additional myDAQ tips for this problem:

- Use the 2N2222A BJT transistor described in Appendix E. Follow the pinout diagram of Figure E.1 on page 185 to determine the base, collector, and emitter connections.
- Power the circuit with myDAQ +15V for V_{CC} and AGND for the circuit ground.
- Carefully measure the resistance of each of the two resistors, and then calculate the associated current from a DMM voltmeter measurement of the voltage across the resistor. This method permits more precision for the base current compared to using the DMM ammeter.

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

The operating point can shift considerably depending on the temperature of the transistor, especially for the fixed-bias circuit of this problem that makes no attempt to stabilize the operating point. To experience this effect, set the collector resistor to 1.0 kΩ and monitor the collector-emitter voltage V_{CE} as you heat the transistor; you can either pinch it between your fingers or blow warm air on it with a straw. What is the maximum voltage change you observed? Express this as an absolute value as well as a percentage change.

3.4 Fixed Bias, PNP

Figure 3.4 shows a simple *fixed bias* circuit similar to that of Problem 3.3 on page 104, but this time with a PNP device.

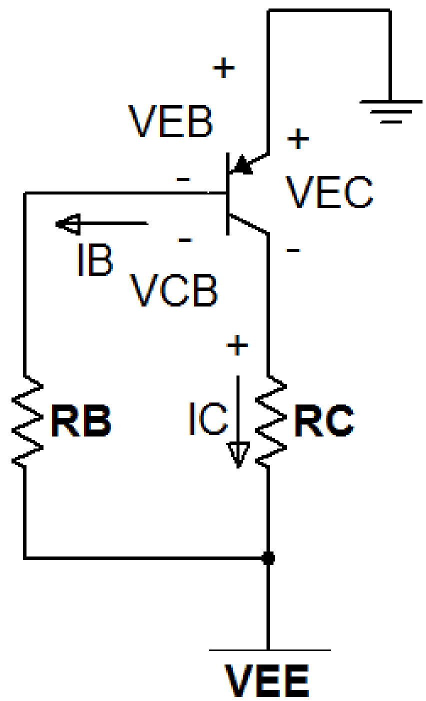


Figure 3.4: Circuit for Problem 3.4

Analysis

1. Find the Q-point (operating point) values for base current I_{B}, collector current I_{C}, emitter-collector voltage V_{EC}, and collector-base voltage V_{CB} for $R_{\mathrm{C}} = 1.0\ \mathrm{k}\Omega$ assuming active-region operation. Determine the actual operating region of the transistor by comparing the biases of the emitter-base and collector-base junctions.

2. Repeat for $R_{\mathrm{C}} = 2.2\ \mathrm{k}\Omega$.

3. Repeat for $R_\mathrm{C} = 4.7$ kΩ.

Use the following circuit components:

Component	Value
V_EE	-15 volts
PNP BJT	$\beta = 200$ and V_BE = 0.7 volts
R_B	470 kΩ

Simulation with NI Multisim

Enter the circuit of Figure 3.4 on the preceding page using the `2N3906` transistor model; do *not* use the similarly-named `2N3906G` model because it has different behavior.

1. Simulate the Q-point (operating point) values for base current I_B, collector current I_C, emitter-collector voltage V_EC, and collector-base voltage V_CB for $R_\mathrm{C} = 1.0$ kΩ. Also simulate V_BE to compare with the assumed value used in the analytical section. State whether the transistor is in active mode or saturated mode, and explain how you know.

2. Repeat for $R_\mathrm{C} = 2.2$ kΩ.

3. Repeat for $R_\mathrm{C} = 4.7$ kΩ.

Additional Multisim tips for this problem:

- Use either of the following techniques to measure the DC currents and voltages:
  - Interactive analysis with two Simulate → Instruments → Measurement Probe indicators placed at the transistor base and collector and configured to display voltage and current indicators
  - SPICE-style analysis with Simulate → Analyses → DC Operating Point

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Measure DC node voltage with a measurement probe:
  `http://youtu.be/svNGHA2-uK4`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`
- Find node voltages with DC Operating Point analysis:
  `http://youtu.be/gXBCqP17AZs`

Measurement with NI myDAQ

Build the circuit of Figure 3.4 on page 108 using the 2N3906 transistor.

1. Measure the Q-point (operating point) values for base current I_{B}, collector current I_{C}, emitter-collector voltage V_{EC}, and collector-base voltage V_{CB} for $R_{\mathrm{C}} = 1.0\ \mathrm{k}\Omega$. Also measure V_{BE} for comparison with the assumed value used in the analytical section.

2. Repeat for $R_{\mathrm{C}} = 2.2\ \mathrm{k}\Omega$.

3. Repeat for $R_{\mathrm{C}} = 4.7\ \mathrm{k}\Omega$.

Additional myDAQ tips for this problem:

- Use the 2N3906 BJT transistor described in Appendix E. Follow the pinout diagram of Figure E.3 on page 187 to determine the base, collector, and emitter connections.

- Power the circuit with myDAQ -15V for V_{EE} and AGND for the circuit ground.

- Carefully measure the resistance of each of the two resistors, and then calculate the associated current from a DMM voltmeter measurement of the voltage across the resistor. This method permits more precision for the base current compared to using the DMM ammeter.

NI myDAQ video tutorials:

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

3.5 Voltage-Divider Bias

Figure 3.5 shows a *voltage-divider bias* circuit for a BJT.

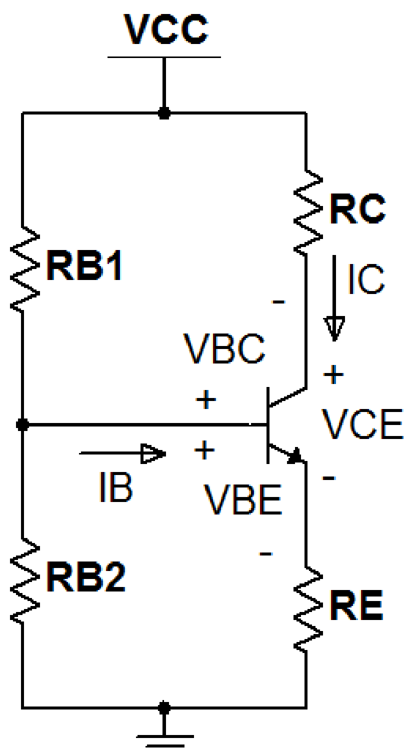


Figure 3.5: Circuit for Problem 3.5

Analysis

Find the Q-point (operating point) values for base current I_B, collector current I_C, collector-emitter voltage V_{CE}, and base-collector voltage V_{BC} assuming active-region operation.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
BJT	$\beta = 200$ and V_{BE} = 0.7 volts
R_{B1}	100 kΩ
R_{B2}	47 kΩ
R_C	1.5 kΩ
R_E	470 Ω

Simulation with NI Multisim

Enter the circuit of Figure 3.5 on the facing page using the `2N2222A` transistor model. Find the Q-point (operating point) values for base current I_B, collector current I_C, collector-emitter voltage V_{CE}, and base-collector voltage V_{BC}. Also find V_{BE} for comparison with the assumed value used in the analytical section.
Additional Multisim tips for this problem:

- Simulate → Instruments → Measurement Probe indicators work well to display currents.
- SPICE-style analysis with Simulate → Analyses → DC Operating Point quickly displays all required values in a single table.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Measure DC node voltage with a measurement probe:
  `http://youtu.be/svNGHA2-uK4`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`
- Find node voltages with DC Operating Point analysis:
  `http://youtu.be/gXBCqP17AZs`

Measurement with NI myDAQ

Build the circuit of Figure 3.5 on the preceding page using the 2N2222A transistor. Find the Q-point (operating point) values for base current I_B,

collector current I_{C}, collector-emitter voltage V_{CE}, and base-collector voltage V_{BC}.
Additional myDAQ tips for this problem:

- Use the 2N2222A BJT transistor described in Appendix E. Follow the pinout diagram of Figure E.1 on page 185 to determine the base, collector, and emitter connections.

- Power the circuit with myDAQ +15V for V_{CC} and AGND for the circuit ground.

- Carefully measure the resistance of resistors R_{C} and R_{E}, and then calculate the associated current from a DMM voltmeter measurement of the voltage across the resistor. Calculate the base current as I_{E} minus I_{C}. This method permits more precision for the base current compared to using the DMM ammeter.

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

In the "Further Exploration" section of Problem 3.3 on page 104 you discovered that the operating point of the fixed-bias circuit changed dramatically in response to elevated temperature of the transistor. The four-resistor bias circuit reduces sensitivity to temperature by introducing *negative feedback* with the emitter resistor that "communicates" between the collector circuit and the base circuit; here's how it works: when external conditions increase the current through R_{E}, its voltage rises which in turn lowers the base-emitter junction voltage, decreasing the base current, which "closes the loop" by acting to reduce the emitter current that was increased in the first place.

Experience the improved operating point stability of the four-resistor bias circuit. Monitor the collector-emitter voltage V_{CE} as you heat the transistor; you can either pinch it between your fingers or blow warm air on it with a straw. What is the maximum voltage change you observed? Express this as an absolute value as well as a percentage change.

3.6 Darlington-Pair Emitter Follower

The *Darlington pair* combines two NPN transistors into a single effective "super beta" transistor. The Darlington pair boosts the current drive ability of the common-collector amplifier (also known as an *emitter follower*) of Figure 3.6.

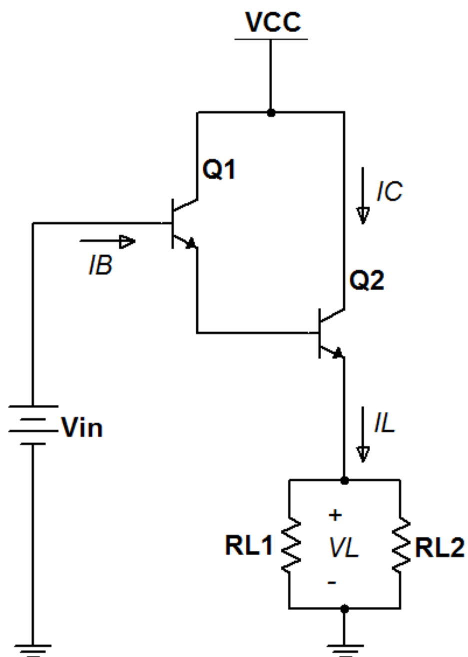


Figure 3.6: Circuit for Problem 3.6

Analysis

1. Calculate the effective $\beta = I_C/I_B$ of the Darlington pair.
2. Determine I_B, I_L, and V_L when V_{in} = 3.3 V.
3. Calculate the effective input resistance of the circuit $R_{in} = V_{in}/I_B$.

Use the following circuit components:

Component	Value
V_{CC}	5 V
BJT	$\beta = 200$ and V_{BE} = 0.7 volts (active region)
resistors	R_{L1} = R_{L1} = 100 Ω

Simulation with NI Multisim

Enter the circuit of Figure 3.6 on the preceding page using the `2N2222A` transistor model. Drive the input with a 3.3 V DC source.

1. Determine the effective $\beta = I_C/I_B$ of the Darlington pair.
2. Determine I_B, I_L, and V_L.
3. Calculate the effective input resistance of the circuit $R_{in} = V_{in}/I_B$.

Additional Multisim tips for this problem:

- Use measurement probes to determine currents and voltages.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Measure DC node voltage with a measurement probe:
  `http://youtu.be/svNGHA2-uK4`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`

Measurement with NI myDAQ

Build the circuit of Figure 3.6 on the previous page using two 2N2222A transistors. Create the 3.3 V constant voltage using digital output DIO0 operated by the NI ELVISmx Digital Writer.

1. Measure the effective $\beta = I_C/I_B$ of the Darlington pair.
2. Measure I_B (see tip below), I_L, and V_L.
3. Calculate the effective input resistance of the circuit $R_{in} = V_{in}/I_B$.

Additional myDAQ tips for this problem:

- Use the 2N2222A BJT transistor described in Appendix E. Follow the pinout diagram of Figure E.1 on page 185 to determine the base, collector, and emitter connections.
- Power the circuit with myDAQ 5V for V_{CC} and DGND for the circuit ground; use either AGND or DGND as the circuit ground because they are connected internally.
- Use the NI ELVISmx DMM as a voltmeter and ammeter as required.
- To measure I_{B}, a current in the sub-microamp range:
  1. Choose a 10 kΩ resistor and carefully measure its resistance with the DMM ohmmeter,
  2. Place the resistor between DIO0 and the transistor base,
  3. Measure the voltage across the resistor, and
  4. Divide the measured voltage by the measured resistance to calculate I_{B}.

NI myDAQ video tutorials:

- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`
- Digital Writer (DigOut):
  `http://decibel.ni.com/content/docs/DOC-12945`

Further Exploration with NI myDAQ

The emitter follower implemented with a Darlington pair provides an excellent way to drive high-current loads such as LED illuminators from digital outputs that have weak current drive. Experience the effectiveness of your emitter follower circuit by comparing the drive strength of the unassisted digital output to that of the strengthened drive of the emitter follower.

1. To begin, disconnect DIO0 from the emitter follower circuit. Connect four LEDs in parallel which in turn are connected through two series

1N4148 diodes to DIO0; refer to Figure 3.7 for the circuit. The diodes mimic the V_{BE} drops of the Darlington pair to make a fair comparison.

2. Toggle the DIO0 line on and off. How well does the output drive the LEDs? Do they get very bright?

3. Remove the diodes, connect the parallel LEDs to the output of your emitter follower (remove the two 100 Ω resistors, too), and re-connect DIO0 to the circuit input.

4. Toggle the DIO0 line on and off – *careful, do NOT look directly into the LEDs!* How well does your circuit drive the LEDs? Compare the "before and after" results of your emitter follower circuit.

5. An LED glows with an intensity directly proportional to current. Measure the total LED current with and without your emitter follower to numerically determine the boost in intensity.

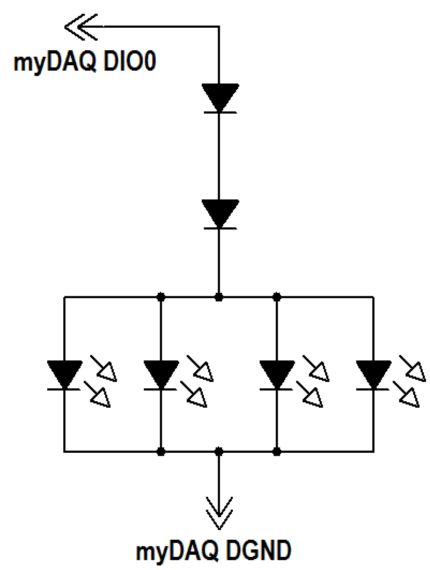


Figure 3.7: Parallel LED load with series diodes for direct connection to DIO0.

3.7 Common-Emitter Amplifier

The *common-emitter amplifier* pictured in Figure 3.8 offers high voltage gain, moderately high input impedance, and moderately low output impedance.

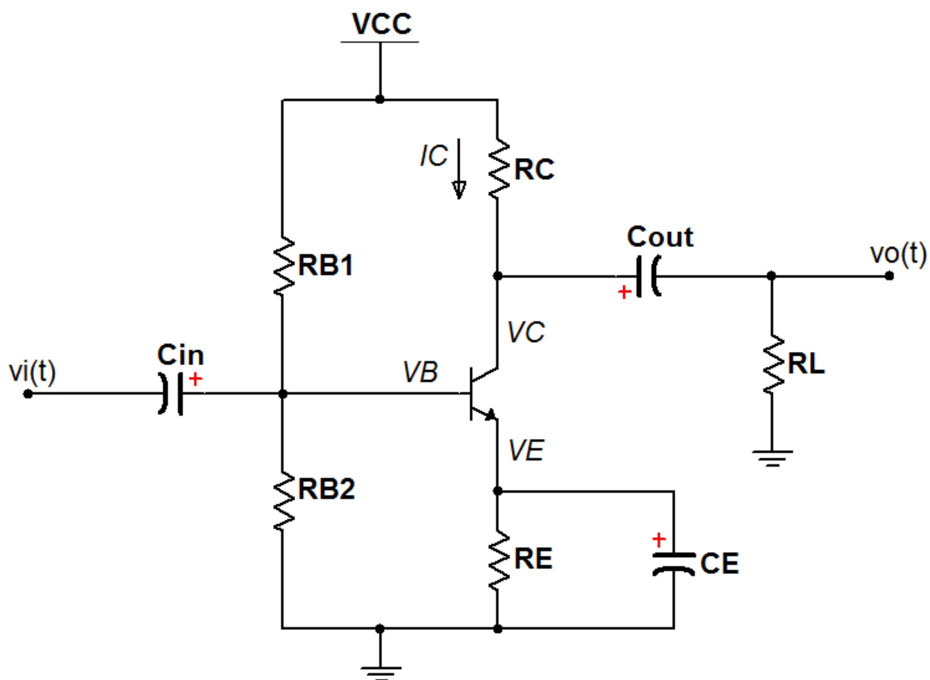


Figure 3.8: Circuit for Problem 3.7

Analysis

1. Determine the Q-point (operating point) values I_{CQ}, V_{CQ}, V_{BQ}, and V_{EQ}.

2. Determine the midband AC small-signal model of the circuit, i.e., with capacitors taken as short circuits.

3. Calculate the open-circuit voltage gain A_{vo} with R_{L} unconnected.

4. Calculate the voltage gain A_{v} with R_{L} connected.

Use the following circuit components:

Component	Value
V_{CC}	9 V
BJT	$\beta = 200$ and V_{BE} = 0.7 volts (active region)
Resistors	$R_{\mathrm{B1}} = 100\ \mathrm{k}\Omega$, $R_{\mathrm{B2}} = 47\ \mathrm{k}\Omega$, $R_{\mathrm{E}} = 330\ \Omega$, $R_{\mathrm{C}} = 680\ \Omega$, and $R_{\mathrm{L}} = 1\ \mathrm{k}\Omega$
Capacitors	$C_{\mathrm{in}} = 10\ \mu\mathrm{F}$, $C_{\mathrm{out}} = 100\ \mu\mathrm{F}$, and $C_{\mathrm{E}} = 470\ \mu\mathrm{F}$

Simulation with NI Multisim

Enter the circuit of Figure 3.8 on the preceding page using the `2N2222A` transistor model. Use the Simulate → Instruments → Function Generator to apply a 20 mV peak-to-peak sinusoid at 4 kHz as the input signal, and use the Simulate → Instruments → Oscilloscope to display the input and output signals.

1. Determine the Q-point (operating point) values I_{CQ}, V_{CQ}, V_{BQ}, and V_{EQ}; temporarily replace the function generator with a ground connection as the input.
2. Determine the open-circuit voltage gain A_{vo} with R_{L} unconnected.
3. Determine the voltage gain A_{v} with R_{L} connected.

Additional Multisim tips for this problem:

- Place the cursors at the maximum and minimum values of the output waveform, and then read the "T2-T1" display as the peak-to-peak value.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Function generator:
  `http://youtu.be/CeOl6EzD-_c`
- Find the maximum value of a trace in Grapher View:
  `http://youtu.be/MzYK60mfh2Y`

Measurement with NI myDAQ

Build the circuit of Figure 3.8 on page 119 using the 2N2222A transistor.

> **IMPORTANT!** *Observe correct polarity on the electrolytic capacitors to ensure safe operation.*

Use the NI ELVISmx Function Generator to apply a 0.02 V peak-to-peak sinusoid as the input signal, and use the ELVISmx Oscilloscope to display the input and output signals.

1. Determine the Q-point (operating point) values I_{CQ}, V_{CQ}, V_{BQ}, and V_{EQ}; temporarily replace the function generator with a ground connection as the input.

2. Determine the open-circuit voltage gain A_{vo} with R_L unconnected; you may also use a high-value resistor such as 100 kΩ or 1 MΩ as R_L to allow the coupling capacitor to charge in a reasonable amount of time.

3. Determine the voltage gain A_v with R_L connected.

Additional myDAQ tips for this problem:

- Use the 2N2222A BJT transistor described in Appendix E. Follow the pinout diagram of Figure E.1 on page 185 to determine the base, collector, and emitter connections.

- Implement the voltage source V_{CC} with the adjustable DC voltage source circuit of Figure B.2 on page 172.

- Use the NI ELVISmx Oscilloscope's peak-to-peak meters to measure the input and output levels for the gain calculation.

- Use the NI ELVISmx Oscilloscope's RMS meters to measure Q-point voltages (remember to set the function generator to zero amplitude).

- Instead of inserting the DMM ammeter to measure I_{CQ} which alters the circuit too much, measure the voltage across R_C and then divide by the measured value of R_C.

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

Study the behavior of your circuit under various operating conditions. Take representative screen shots of the oscilloscope and discuss your findings. Areas to consider include:

- Probe each of the nodes in the circuit, tracing how the input signal “moves” through the amplifier,
- Investigate the triangle and square waveforms,
- Study the effect of increased input amplitude, and
- Observe the impact of reducing the value of the bypass capacitor C_{E} or removing it altogether.

3.8 Current Mirror

Figure 3.9 shows a *current mirror* with base-current compensation provided by transistor Q_{comp}.

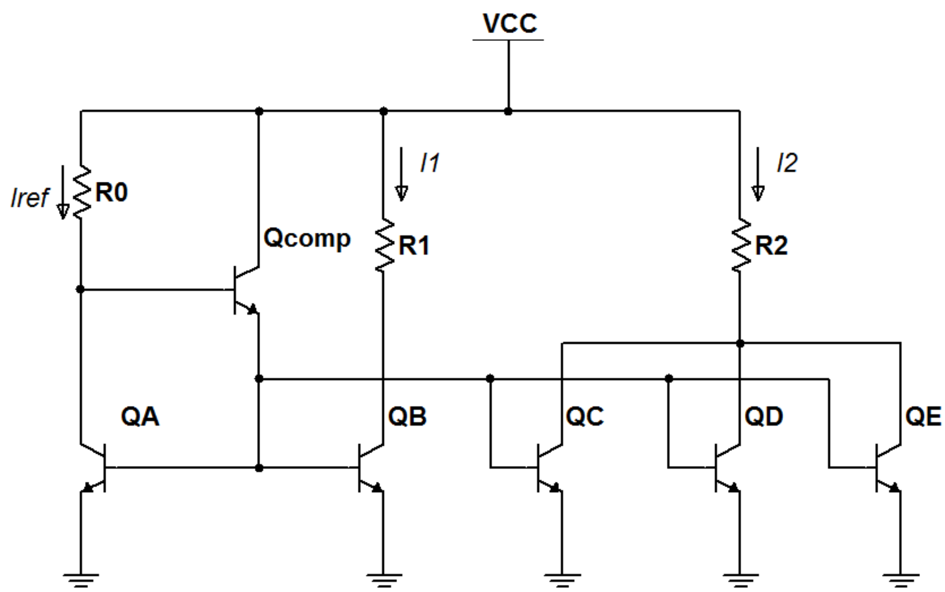


Figure 3.9: Circuit for Problem 3.8

Analysis

1. Calculate I_{REF}, I_1, and I_2.

2. Calculate the current mirror gains $\beta_1 = I_1/I_{\mathrm{REF}}$ and $\beta_2 = I_2/I_{\mathrm{REF}}$.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
BJT array, Q_{A} to Q_{E}	$\beta = 70$ and V_{BE} = 0.7 volts (active region)
BJT Q_{comp}	$\beta = 200$ and V_{BE} = 0.7 volts (active region)
Resistors	$R_0 = 4.7$ kΩ, $R_1 = 4.7$ kΩ, and $R_0 = 1.5$ kΩ

Simulation with NI Multisim

Enter the circuit of Figure 3.9 on the preceding page using the `CA3083` transistor model for Q_{A} to Q_{E}, the `2N2222A` for Q_{comp}, and the `LED_red` device. Attach measurement probes to display the required currents.

1. Determine I_{C}, I_1, and I_2.
2. Calculate the current mirror gains $\beta_1 = I_1/I_{\mathrm{REF}}$ and $\beta_2 = I_2/I_{\mathrm{REF}}$.
3. Calculate the mirrored current ratio I_2/I_1.

Additional Multisim tips for this problem:

- The CA3083 is not part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.4 on page 188 to learn how to create and add this device to your "User Database."

NI Multisim video tutorials:

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

Measurement with NI myDAQ

Build the circuit of Figure 3.9 on the preceding page using the CA3083 transistor array for Q_{A} to Q_{E} and the 2N2222A for Q_{comp}. Note that the transistor array provides five matched transistors on a monolithic (common) substrate.

IMPORTANT! *Connect the substrate pin 5 to ground.*

Use the DMM ammeter to measure the required currents.

1. Measure I_{C}, I_1, and I_2.
2. Measure I_{REF} and then calculate the current mirror gains $\beta_1 = I_1/I_{\mathrm{REF}}$ and $\beta_2 = I_2/I_{\mathrm{REF}}$.
3. Calculate the mirrored current ratio I_2/I_1.

Additional myDAQ tips for this problem:

- Use the CA3083 NPN BJT transistor array described in Appendix E. Follow the pinout diagram of Figure E.4 on page 188 to determine the base, collector, and emitter connections for each of the five transistors in the array.
- Use the 2N2222A BJT transistor described in Appendix E. Follow the pinout diagram of Figure E.1 on page 185 to determine the base, collector, and emitter connections.
- Power the circuit with myDAQ +15V for V_{CC} and AGND for the circuit ground.
- Implement the current source I_{REF} according to the circuit diagram of Figure B.4 on page 174. Use a 1 kΩ resistor for the adjustment resistor R.
- Refer to the diagram of Figure D.2 on page 182 to determine the anode and cathode connections of an LED.

NI myDAQ video tutorials:

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

3.9 Differential Pair

Figure 3.10 shows a basic *differential amplifier*; the two transistors constitute a *differential pair*. The differential amplifier creates a high gain for small differences between the inputs while applying a low gain to common-mode inputs.

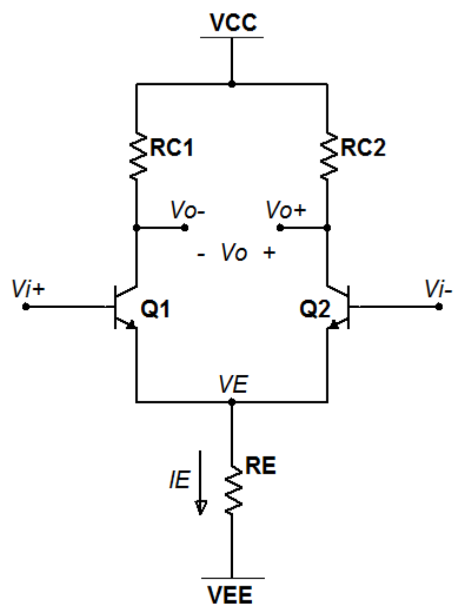


Figure 3.10: Circuit for Problem 3.9

Analysis

1. Calculate the operating point values V_{o+}, V_{o-}, V_o, V_E, and I_E with both differential inputs V_+ and V_- grounded.

2. Repeat with both differential inputs connected to 3.0 V.

3. Calculate the differential voltage gain with differential output as $A_d = g_m R_C$, $g_m = I_C/V_T$, and thermal voltage V_T = 26 mV.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	-15 volts
BJT	$\beta = 70$ and V_{BE} = 0.7 volts (active region)
R_{C1}	4.7 kΩ
R_{C2}	4.7 kΩ
R_E	4.7 kΩ

Simulation with NI Multisim

Enter the circuit of Figure 3.10 on the preceding page using the `CA3083` transistor model for Q_1 and Q_2. Attach measurement probes to display the requested current and voltages. Use a referenced measurement probe to display the differential output V_o = V_{o+} - V_{o-}.

1. Determine the operating point values V_{o+}, V_{o-}, V_o, V_E, and I_E with both differential inputs V_+ and V_- grounded.
2. Repeat with both differential inputs connected to 3.0 V, i.e., the common-mode input is 3.0 V.
3. Connect the function generator in differential mode with its center terminal grounded. Apply a 100 Hz sinusoid with 5 mVp amplitude to the differential input. Observe the differential input and differential output on the oscilloscope. Take cursor measurements to determine the differential voltage gain $A_d = v_o/(v_{i+} - v_{i-})$.

Additional Multisim tips for this problem:

- The CA3083 is not part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.4 on page 188 to learn how to create and add this device to your "User Database."
- The "5 mVp" setting on the function generator yields a 10 mV amplitude differential signal on the oscilloscope display.

NI Multisim video tutorials:

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

Measurement with NI myDAQ

Build the circuit of Figure 3.10 on page 126 using the CA3083 transistor array for Q_1 to Q_2; be sure to use the matched transistors 1 and 2 in the array.

> **IMPORTANT!** *Connect the substrate pin 5 to -15V.*

Measure the resistance of all of the resistors you have available for R_{C1} and R_{C2}, and select the two closest matching resistors.

1. Use the DMM to measure the operating point values V_{o+}, V_{o-}, V_o, V_E, and I_E with both differential inputs V_+ and V_- grounded.

2. Repeat with both differential inputs connected to 3.0 V. Use AO0 operated by the NI ELVISmx "DC Level Output" under the "Featured Instruments" tab of the instrument launcher. Alternatively, use the Function Generator with zero amplitude and DC offset of 3.0 V.

3. Use AO0 driven by the NI ELVISmx Arbitrary Waveform Generator to apply a 100 Hz sinusoid with 5 mV amplitude to input V_{i+}. Use the same waveform file for AO1, but change the gain to -1, thereby creating a differential-style input waveform. Observe the differential input and differential output on the oscilloscope. Read the peak-to-peak meters to determine the differential voltage gain $A_d = v_o/(v_{i+} - v_{i-})$.

Additional myDAQ tips for this problem:

- Use the CA3083 NPN BJT transistor array described in Appendix E. Follow the pinout diagram of Figure E.4 on page 188 to determine the base, collector, and emitter connections for each of the five transistors in the array.

- Power the circuit with myDAQ +15V for V_{CC} and AGND for the circuit ground.

NI myDAQ video tutorials:

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

3.10 Class B Push-Pull Amplifier

Figure 3.11 shows a *Class B* amplifier, also known as a *push-pull* output stage. Figure 3.11(a) shows a conceptual circuit that suffers from *cross-over distortion*, and Figure 3.11(b) shows a practical version that eliminates the distortion.

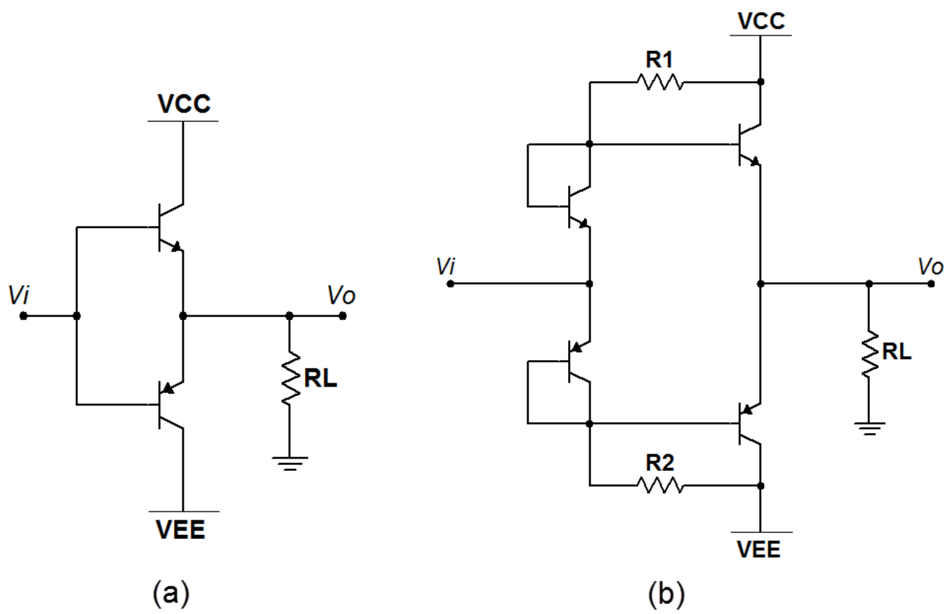


Figure 3.11: Circuit for Problem 3.10, with (a) conceptual circuit and (b) practical circuit.

Analysis

1. Plot $v_o(t)$ of the conceptual circuit of Figure 3.11(a) for an input amplitude of 5 volts; repeat for an input amplitude of 1 volt. In each case determine the peak-to-peak voltage of $v_o(t)$.

2. Plot $v_o(t)$ of the practical circuit of Figure 3.11(b) for an input amplitude of 5 volts; repeat for an input amplitude of 1 volt. In each case determine the peak-to-peak voltage of $v_o(t)$.

3. Compare the level of cross-over distortion for the two circuits.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	-15 volts
$v_{\mathrm{i}}(t)$	triangle wave, 100 Hz
BJT NPN	$\beta = 200$ and V_{BE} = 0.7 volts
BJT PNP	$\beta = 200$ and V_{EB} = 0.7 volts
R_{L}	1 kΩ
R_1	10 kΩ
R_2	10 kΩ

Simulation with NI Multisim

Enter the circuit of Figure 3.11(a) using the `2N3904G` and `2N3906G` transistor models. Drive the input with Simulate → Instruments → Function Generator and monitor the input and output with the Simulate → Instruments → Oscilloscope.

1. Plot $v_{\mathrm{o}}(t)$ of the conceptual circuit of Figure 3.11(a) for an input amplitude of 5 volts; repeat for an input amplitude of 1 volt. In each case take cursor measurements to determine the peak-to-peak voltage of $v_{\mathrm{o}}(t)$.

2. Modify the circuit to match that of Figure 3.11(b). Plot $v_{\mathrm{o}}(t)$ for an input amplitude of 5 volts; repeat for an input amplitude of 1 volt. In each case determine the peak-to-peak voltage of $v_{\mathrm{o}}(t)$.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Function generator:
  `http://youtu.be/CeOl6EzD-_c`
- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Waveform cursor measurements with the two-channel oscilloscope:
  `http://youtu.be/snBRFq1Y1q4`

Measurement with NI myDAQ

Build the circuit of Figure 3.11(a) using the 2N3904 and 2N3906 transistors. Drive the input with AO0 and the NI ELVISmx Function Generator. Monitor the input and output signals with the NI ELVISmx Oscilloscope.

1. Plot $v_o(t)$ of the conceptual circuit of Figure 3.11(a) for an input amplitude of 5 volts; repeat for an input amplitude of 1 volt. In each read the oscilloscope peak-to-peak meter under the main display to measure the peak-to-peak voltage of $v_o(t)$.

2. Modify the circuit to match that of Figure 3.11(b). Plot $v_o(t)$ for an input amplitude of 5 volts; repeat for an input amplitude of 1 volt. In each case determine the peak-to-peak voltage of $v_o(t)$.

Additional myDAQ tips for this problem:

- Use the 2N3904 and 2N3906 matched BJT transistors described in Appendix E. Follow the pinout diagrams of Figure E.2 on page 186 and Figure E.3 on page 187 to determine the base, collector, and emitter connections for each transistor type.

- Power the circuit with myDAQ +15V for V_{CC}, -15V for V_{EE}, and AGND for circuit ground.

NI myDAQ video tutorials:

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

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

3.11 Voltage-to-Current Converter

Figure 3.12 shows a *voltage-to-current* converter that uses a negative feedback op amp configuration to establish a load current I_L proportional to the input voltage V_in.

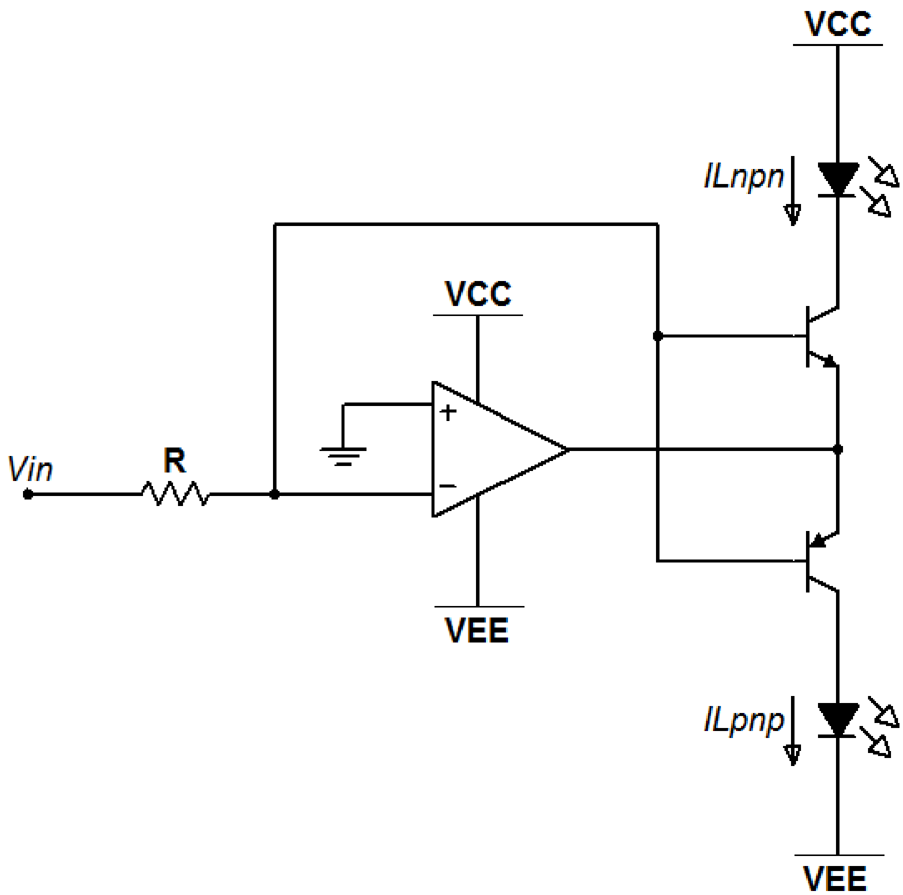


Figure 3.12: Circuit for Problem 3.11

Analysis

1. Derive an equation for I_Lnpn as a function of V_in and the transistor DC current gain β. HINTS: Assume ideal op amp constraints; negative

feedback implies the op amp inverting input is a virtual ground.

2. Repeat for I_{Lpnp}.

3. Plot I_{Lnpn} and I_{Lpnp} as a function of V_{in} over the range -10 to 10 V.

4. Determine the transconductance gain $I_{\text{Lnpn}}/V_{\text{in}}$ at $V_{\text{in}} = 4.0$ V.

5. Determine the transconductance gain $I_{\text{Lpnp}}/V_{\text{in}}$ at $V_{\text{in}} = -4.0$ V.

Use the following circuit components:

Component	Value
V_{CC}	+15 volts
V_{EE}	-15 volts
BJT NPN	$\beta = 200$ and $V_{\text{BE}} = 0.7$ volts
BJT PNP	$\beta = 200$ and $V_{\text{EB}} = 0.7$ volts
LEDs	1.8 V drop when active, ideal otherwise
R	100 kΩ

Simulation with NI Multisim

Enter the circuit of Figure 3.12(a) using the `2N2222A` and `2N3906` transistor models and the `TL072CP` op amp model. Drive the input with a `DC_POWER` source.

1. Set up a Simulate → Analyses → DC Sweep to plot I_{Lnpn} and I_{Lpnp} on the same graph as a function of V_{in} over the range -10 to 10 V.

2. Take a cursor measurement to determine the transconductance gain $I_{\text{Lnpn}}/V_{\text{in}}$ at $V_{\text{in}} = 4.0$ V; place a data label at the cursor location.

3. Find the transconductance gain $I_{\text{Lpnp}}/V_{\text{in}}$ at $V_{\text{in}} = -4.0$ V; place a data label at the cursor location.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Simulate → Analyses → DC Sweep
- Add data label to Grapher View plot:
  `http://youtu.be/uibNqFlod_Y`

Measurement with NI myDAQ

Build the circuit of Figure 3.12(a) using the 2N2222A and 2N3906 transistors and the TL072 op amp. Drive the input with AO0 and the NI ELVISmx DC Level Output found under the "Featured Instruments" tab of the ELVISmx Instrument Launcher. Monitor the currents using the DMM ammeter.

1. Collect data points at 1-volt intervals to plot I_{Lnpn} and I_{Lpnp} on the same graph as a function of V_{in} over the range -10 to 10 V. You may do this manually or use the LabVIEW "DC Sweeper" virtual instrument (VI) to collect the measurements automatically; get the VI at `https://decibel.ni.com/content/docs/DOC-20393`.

2. From your measurements calculate the transconductance gain $I_{\text{Lnpn}}/V_{\text{in}}$ at $V_{\text{in}} = 4.0$ V.

3. Find the transconductance gain $I_{\text{Lpnp}}/V_{\text{in}}$ at $V_{\text{in}} = -4.0$ V.

Additional myDAQ tips for this problem:

- Use the 2N2222A and 2N3906 BJT transistors described in Appendix E. Follow the pinout diagrams of Figure E.1 on page 185 and Figure E.3 on page 187 to determine the base, collector, and emitter connections for each transistor type.

- Use the Texas Instruments TL072 op amp described in Appendix C. Follow the pinout diagram of Figure C.1 on page 178 for either of the two available op amps in the package. You may also use an equivalent dual-supply op amp.

- Power the circuit with myDAQ +15V for V_{CC}, -15V for V_{EE}, and AGND for circuit ground.

Further Exploration with NI myDAQ

Now that you have successfully built the voltage-to-current converter circuit, you may enjoy operating the LEDs with a function generator waveform. Try a variety of waveform shapes (sine, triangle, and square), amplitudes, and frequencies. Try the triangle wave at the lowest possible frequency (0.2 Hz) and 1 Vpp amplitude to study the crossover behavior of the LEDs.

Chapter 4

MOSFET Circuits

4.1 Voltage-to-Current Converter

Figure 4.1 shows an n-channel enhancement-mode MOSFET connected for direct control of the gate-to-source voltage V_{GS} which in turn controls the current in the LED, hence this circuit is a form of *voltage-to-current* converter.

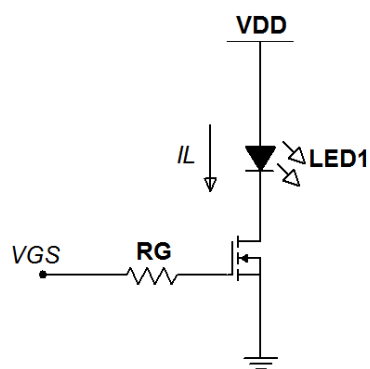


Figure 4.1: Circuit for Problem 4.1

Analysis

1. Explain why the designated input voltage V_{GS} is the same as the MOSFET gate-to-source voltage.

2. Plot the LED current I_L as a function of gate-to-source voltage V_{GS} for $1.8 \le V_{GS} \le 2.3$ V.

3. Tabulate I_L at V_{GS} equal to 1.8 to 2.3 V using 0.1 V steps.

Use the following circuit components:

Component	Value
V_{DD}	15 V
MOSFET	$V_t = 1.8$ V and $K = 50$ mA/V^2
R_G	1 MΩ
LED	Forward voltage drop = 2.0 V, ideal otherwise

Simulation with NI Multisim

Enter the circuit of Figure 4.1 on the previous page using the `ZVN3306A` transistor model and the `LED_red` LED model.

1. Set up a Simulate → Analyses → DC Sweep to plot the LED current I_L as a function of gate-to-source voltage V_{GS} for $1.8 \le V_{GS} \le 2.3$ V in 0.01 V steps.

2. Take cursor measurements to determine I_L at V_{GS} equal to 1.8 to 2.3 V using 0.1 V steps. Label these measurements directly on the Grapher View plot.

Additional Multisim tips for this problem:

- The ZVN3306A is not part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.5 on page 190 to learn how to create and add this device to your "User Database."

- Right-click on the Grapher View cursor and choose "Set X Value" to conveniently place the cursor at the exact value of V_{GS} that you require, and then choose "Cursor" followed by "Add Data Label at Cursor."

NI Multisim video tutorials:

- Find components by name:
  http://youtu.be/5wlFweh4n-c
- Plot DC circuit response with DC Sweep:
  http://youtu.be/vcYuCt9QjdI
- Add data label to Grapher View plot:
  http://youtu.be/uibNqFlod_Y

Measurement with NI myDAQ

Build the circuit of Figure 4.1 on page 135 using the ZVN3306A transistor.

> **CAUTION:** *MOSFET devices are easily damaged by "ESD" (electrostatic discharge). Carry the MOSFET in conductive foam, and touch a grounded point nearby before handling the MOSFET.*

Use the DMM ammeter to monitor the LED current I_L. Connect the transistor gate to AO0, and use the NI ELVISmx DC Level Output instrument under the "Featured Instruments" tab of the ELVISmx Instrument Launcher to set the gate-to-source voltage V_{GS}.

> **IMPORTANT!** *Do not exceed 2.3 V for V_{GS}.*

1. Measure the LED current I_L for gate-to-source voltages V_{GS} of 1.8 V to 2.3 V in 0.1 V steps. Plot your measurement results directly on your Multisim plot for easy comparison.
2. Compare your measurements for I_L at V_{GS} equal to 1.8, 1.9, 2.0, 2.1, 2.2, and 2.3 volts to your analytical and simulation results.

Additional myDAQ tips for this problem:

- Use the ZVN3306A n-channel enhancement-mode MOS transistor described in Appendix E.5 on page 190. Follow the pinout diagram of Figure E.5 on page 190 to determine the gate, drain, and source connections.
- Implement the voltage source V_{DD} with the NI myDAQ +15V supply; use AGND as the ground connection.

- Refer to the diagram of Figure D.2 on page 182 to determine the anode and cathode connections of an LED.

NI myDAQ video tutorials:

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

Further Exploration with NI myDAQ

You may have noticed that the LED current required a bit of time to stabilize from one measurement to the next, especially for the larger values of V_{GS}. The circuit's operating point shifts considerably depending on the temperature of the transistor. To experience this effect, set V_{GS} to 2.3 V and monitor the LED current as you heat the transistor; you can either pinch it between your fingers or blow warm air on it with a straw. What is the maximum voltage change you observed? Express this as an absolute value as well as a percentage change.

4.2 Triode Operation

Figure 4.2 connects an n-channel enhancement-mode MOSFET for operation in the *triode* region.

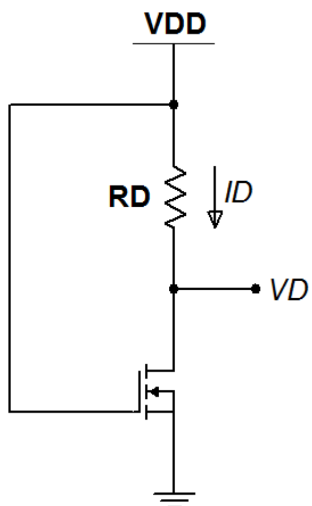


Figure 4.2: Circuit for Problem 4.2

Analysis

1. Determine the value of resistor R_D that sets the operating point voltage V_D to 125 mV. Use the closest standard resistor value from the parts list of Appendix A on page 167.
2. Determine the operating point voltage V_D and current I_D using your selected value of R_D.
3. Calculate the effective drain-to-source resistance of the MOSFET at this operating point.

Use the following circuit components:

Component	Value
V_{DD}	5 V
MOSFET	$V_t = 2.0$ V and $K = 30$ mA/V^2

Simulation with NI Multisim

Enter the circuit of Figure 4.2 on the preceding page using the `ZVN3306A` transistor model and your standard resistor value calculated above.

1. Determine the operating point V_D and I_D.

2. Calculate the effective drain-to-source resistance of the MOSFET at this operating point.

Additional Multisim tips for this problem:

- The ZVN3306A is not part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.5 on page 190 to learn how to create and add this device to your "User Database."

- Use either of the following techniques to simulate the DC currents and voltages:

  – Interactive analysis with one Simulate → Instruments → Measurement Probe indicator placed at the transistor drain configured to display voltage and current

  – SPICE-style analysis with Simulate → Analyses → DC Operating Point

NI Multisim video tutorials:

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

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

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

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

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

Measurement with NI myDAQ

Build the circuit of Figure 4.2 on page 139 using the ZVN3306A transistor.

> **CAUTION:** *MOSFET devices are easily damaged by "ESD" (electrostatic discharge). Carry the MOSFET in conductive foam, and touch a grounded point nearby before handling the MOSFET.*

1. Measure the operating point V_D and I_D.
2. Calculate the effective drain-to-source resistance of the MOSFET at this operating point.

Additional myDAQ tips for this problem:

- Use the ZVN3306A n-channel enhancement-mode MOS transistor described in Appendix E.5 on page 190. Follow the pinout diagram of Figure E.5 on page 190 to determine the gate, drain, and source connections.
- Implement the voltage source V_{DD} with the NI myDAQ 5V supply; use DGND as the ground connection to myDAQ.
- Carefully measure the resistance of R_D, and then calculate its current from a DMM voltmeter measurement of the voltage across the resistor. This method is less intrusive than inserting the ammeter and yields more accurate results.

NI myDAQ video tutorials:

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

4.3 Switch

A MOSFET operated as a switch as shown in Figure 4.3 permits a digital signal operating at very low current levels to control comparatively high-level current devices such as a light-emitting diode (LED).

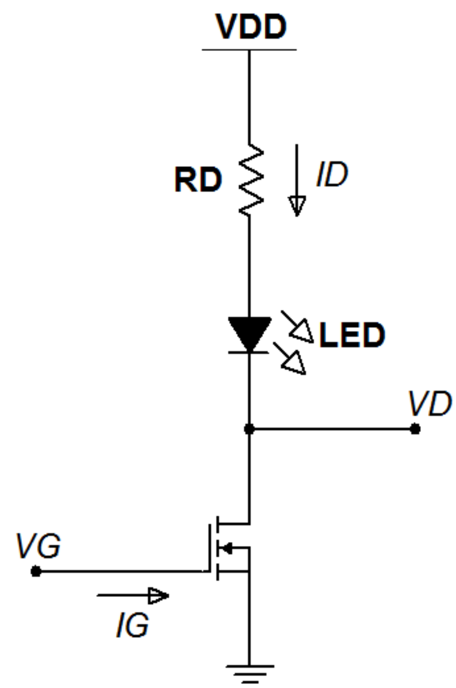


Figure 4.3: Circuit for Problem 4.3

Analysis

1. Determine I_G, V_D, and I_D when $V_G = 0$ V. The MOSFET is in what operating region?

2. Repeat for $V_G = 3.3$ V.

Use the following circuit components:

Component	Value
V_{DD}	5 V
MOSFET	$V_t = 1.8$ V and $K = 50$ mA/V^2
LED	Turn-on voltage is 1.8 V, ideal diode otherwise
R_D	330 Ω

Simulation with NI Multisim

Enter the circuit of Figure 4.3 on the preceding page using the `ZVN3306A` transistor model and the `LED_red` model.

1. Determine I_G, V_D, and I_D when V_G = 0 V.
2. Repeat for V_G = 3.3 V.

Additional Multisim tips for this problem:

- The ZVN3306A is not part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.5 on page 190 to learn how to create and add this device to your "User Database."
- Use a DC source as the input.
- Use measurement probes to determine currents and voltages.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Measure DC node voltage with a measurement probe:
  `http://youtu.be/svNGHA2-uK4`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`

Measurement with NI myDAQ

Build the circuit of Figure 4.3 on the preceding page using the ZVN3306A transistor.

CAUTION: *MOSFET devices are easily damaged by "ESD" (electrostatic discharge). Carry the MOSFET in conductive foam, and touch a grounded point nearby before handling the MOSFET.*

1. Determine I_G, V_D, and I_D when $V_G = 0$ V.
2. Repeat for $V_G = 3.3$ V.

Additional myDAQ tips for this problem:

- Use the ZVN3306A n-channel enhancement-mode MOS transistor described in Appendix E.5 on page 190. Follow the pinout diagram of Figure E.5 on page 190 to determine the gate, drain, and source connections.
- Power the circuit with myDAQ 5V for V_{DD} and AGND for the circuit ground.
- Create the 3.3 V constant voltage using digital output DIO0 operated by the NI ELVISmx Digital Writer; use either AGND or DGND as the circuit ground because they are connected internally.
- Use the NI ELVISmx DMM as a voltmeter and ammeter as required.

NI myDAQ video tutorials:

- DMM voltmeter:
  `http://decibel.ni.com/content/docs/DOC-12937`
- DMM ammeter:
  `http://decibel.ni.com/content/docs/DOC-12939`
- Digital Writer (DigOut):
  `http://decibel.ni.com/content/docs/DOC-12945`

Further Exploration with NI myDAQ

Explore what happens when you attempt to drive the comparatively high-current LED directly from the NI myDAQ digital output DIO0: remove the MOSFET and connect DIO0 directly to the cathode of the LED.

1. What DIO0 voltage causes the LED to light? If you observe that the LED is always lit, which DIO0 voltage causes the maximum brightness? Also, explain why the LED is always on to some extent.

2. What current is required from DIO0 when the LED is at its brightest level? Compare this result to the required current when the MOSFET switch is in place.

4.4 Feedback Bias

Figure 4.4 shows a simple *feedback bias* circuit for an n-channel enhancement-mode MOSFET.

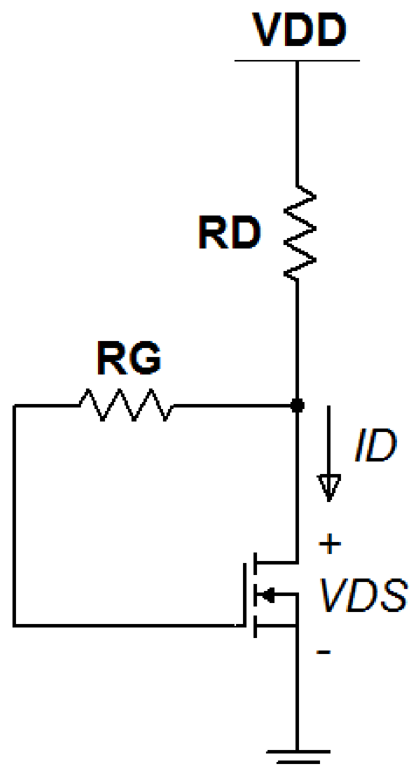


Figure 4.4: Circuit for Problem 4.4

Analysis

1. Find the Q-point (operating point) values for drain current I_D and drain-source voltage V_{DS} for $R_D = 680\ \Omega$.
2. Repeat for $R_D = 1.5\ \text{k}\Omega$.
3. Repeat for $R_D = 10\ \text{k}\Omega$.

Use the following circuit components:

Component	Value
V_{DD}	9 V
MOSFET	$V_t = 1.8$ V and $K = 50$ mA/V^2
R_G	10 MΩ

Simulation with NI Multisim

Enter the circuit of Figure 4.4 on the facing page using the `ZVN3306A` transistor model

1. Simulate the Q-point (operating point) values for drain current I_D and drain-source voltage V_{DS} for $R_D = 680\ \Omega$.

2. Repeat for $R_D = 1.5\ \text{k}\Omega$.

3. Repeat for $R_D = 10\ \text{k}\Omega$.

Additional Multisim tips for this problem:

- The ZVN3306A is not part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.5 on page 190 to learn how to create and add this device to your "User Database."

- Use either of the following techniques to simulate the DC currents and voltages:

  – Interactive analysis with one Simulate → Instruments → Measurement Probe indicator placed at the transistor drain configured to display voltage and current

  – SPICE-style analysis with Simulate → Analyses → DC Operating Point

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- VDD and VSS power supply voltages:
  `http://youtu.be/XrPVLgYsDdY`
- Measure DC node voltage with a measurement probe:
  `http://youtu.be/svNGHA2-uK4`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`
- Find node voltages with DC Operating Point analysis:
  `http://youtu.be/gXBCqP17AZs`

Measurement with NI myDAQ

Build the circuit of Figure 4.4 on page 146 using the ZVN3306A transistor.

> **CAUTION:** *MOSFET devices are easily damaged by "ESD" (electrostatic discharge). Carry the MOSFET in conductive foam, and touch a grounded point nearby before handling the MOSFET.*

Use a 1 MΩ resistor for R_{G} if a 10 MΩ resistor is not readily available.

1. Measure the Q-point (operating point) values for drain current I_{D} and drain-source voltage V_{DS} for $R_{\mathrm{D}} = 680\ \Omega$.
2. Repeat for $R_{\mathrm{D}} = 1.5\ \mathrm{k}\Omega$.
3. Repeat for $R_{\mathrm{D}} = 10\ \mathrm{k}\Omega$.

Additional myDAQ tips for this problem:

- Use the ZVN3306A n-channel enhancement-mode MOS transistor described in Appendix E.5 on page 190. Follow the pinout diagram of Figure E.5 on page 190 to determine the gate, drain, and source connections.
- Implement the voltage source V_{DD} with the adjustable DC voltage source circuit of Figure B.2 on page 172.

- Carefully measure the resistance of R_D, and then calculate its current from a DMM voltmeter measurement of the voltage across the resistor. This method is less intrusive than inserting the ammeter and yields more accurate results. Also recognize that this resistor current is the same as I_D because the transistor's gate current is essentially zero.

NI myDAQ video tutorials:

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

4.5 Voltage-Divider Bias

Figure 4.5 depicts the *voltage-divider bias* circuit for an n-channel enhancement-mode MOSFET, a popular circuit to create a stable operating point in spite of changing environmental temperature and device-to-device variations. In this problem the *load line* graphical analysis technique establishes the drain current operating point.

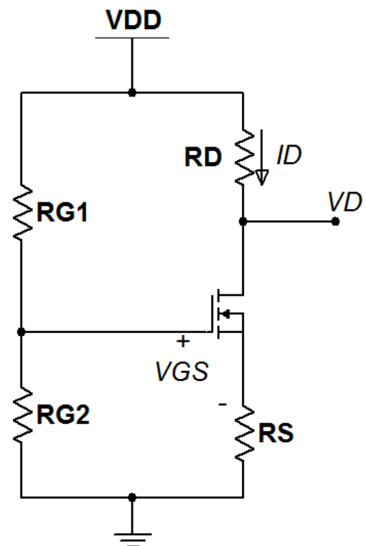


Figure 4.5: Circuit for Problem 4.5

Analysis

1. Use a suitable plotting tool such as Mathscript or MATLAB to plot the transistor drain current I_{D} as a function of gate-source voltage V_{GS} when the device operates in saturation mode; use the transistor parameters listed in the table below. Set up the plot limits as 0 to 3 volts for V_{GS} and 0 to 50 mA for I_{D}.

2. Draw the load line established by the voltage divider resistors R_{G1} and R_{G2}, the supply voltage V_{DD}, and the source resistor R_{S} directly on the transistor curve from the previous step. Identify the circuit's

operating point I_{DQ} and V_{GSQ} at the intersection of the load line and the transistor curve.

3. Determine the value of resistor R_D that sets the drain voltage V_{DQ} to 5.0 V, and then choose the closest standard resistor value from the parts list of Appendix A on page 167.

4. Determine the drain voltage V_{DQ} using your selected value of R_D.

Use the following circuit components:

Component	Value
V_{DD}	9 V
MOSFET	$V_t = 1.8$ V and $K = 50$ mA/V^2
resistors	$R_{G1} = 2$ MΩ, $R_{G2} = 1$ MΩ, and $R_S = 100$ Ω

Simulation with NI Multisim

Enter the circuit of Figure 4.5 on the preceding page using the `ZVN3306A` transistor model, the three specified resistors, and your standard resistor value calculated above. Determine the operating point values I_{DQ}, V_{GSQ}, and V_{DQ}.

Additional Multisim tips for this problem:

- The ZVN3306A is not part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.5 on page 190 to learn how to create and add this device to your "User Database."

- Use either of the following techniques to simulate the DC currents and voltages:

  – Interactive analysis with one Simulate → Instruments → Measurement Probe indicator placed at the transistor drain configured to display voltage and current

  – SPICE-style analysis with Simulate → Analyses → DC Operating Point

NI Multisim video tutorials:

- Find components by name:
  http://youtu.be/5wlFweh4n-c
- Measure DC node voltage with a measurement probe:
  http://youtu.be/svNGHA2-uK4
- Measure DC current with a measurement probe:
  http://youtu.be/uZ56byigymI
- Find node voltages with DC Operating Point analysis:
  http://youtu.be/gXBCqP17AZs

Measurement with NI myDAQ

Build the circuit of Figure 4.5 on page 150 using the ZVN3306A transistor.

> **CAUTION:** *MOSFET devices are easily damaged by "ESD" (electrostatic discharge). Carry the MOSFET in conductive foam, and touch a grounded point nearby before handling the MOSFET.*

Use two 1 MΩ resistors in series for R_{G1} if a 2 MΩ resistor is not readily available. Measure the operating point values I_{DQ}, V_{GSQ}, and V_{DQ}. Additional myDAQ tips for this problem:

- Use the ZVN3306A n-channel enhancement-mode MOS transistor described in Appendix E.5 on page 190. Follow the pinout diagram of Figure E.5 on page 190 to determine the gate, drain, and source connections.
- Implement the voltage source V_{DD} with the adjustable DC voltage source circuit of Figure B.2 on page 172.
- IMPORTANT: Use the NI ELVISmx Oscilloscope's RMS meter to measure voltages instead of the DMM voltmeter; enable only one channel of the oscilloscope. The analog input AI0 has 10 GΩ input resistance, a thousand times higher than the DMM, and therefore does not interfere with the high-resistance voltage divider created by R_{G1} and R_{G2}.
- Carefully measure the resistance of R_{D}, and then calculate its current using the measured voltage across the resistor. This method is less intrusive than inserting the ammeter and yields more accurate results.

Further Exploration with NI myDAQ

1. Demonstrate for yourself the advantage of using the analog input AO0 and the NI ELVISmx oscilloscope as a DC voltmeter compared to the DMM voltmeter. Set up the oscilloscope to monitor V_G, the voltage between the transistor's gate and ground. Be sure to enable only Channel 0 on the oscilloscope. As you observe the RMS indicator, connect the DMM voltmeter between the same points. Report the change in voltage observed by the oscilloscope and express this value as percent change.

2. Next, experience why you must enable only one channel when you use the oscilloscope as a voltmeter. Leave AI1+ and AI1- disconnected, disable Channel 1, and set the "Timebase" to its default setting of 5 ms. Observe the RMS indicator for Channel 0 and then enable the second channel. Report the change in voltage observed by the oscilloscope and express this value as percent change.

   Why the large change? Take a moment to look at the block diagram printed on your NI myDAQ. Note how the analog-to-digital converter is shared (or *multiplexed*) between the two analog inputs. With two enabled channels the multiplexer rapidly switches back and forth between the inputs. With each switch the multiplexer "steals" a small amount of charge from the circuit, a phenomenon called *charge injection*. Rapid switching therefore demands a small current from the circuit, and high impedance circuits such as the MOSFET gate voltage divider magnify this small current into a significant voltage – recall that Ohm's Law tells us voltage is the product of current and resistance.

3. With the charge injection phenomenon in mind, try other timebase settings that select different sampling rates (look in the upper right corner of the main display to see the sample rate). What is the maximum voltage error that you observed? How does the voltage error relate to sampling rate?

4.6 Common-Source Amplifier

The *common-source amplifier* circuit pictured in Figure 4.6 is studied in this problem.

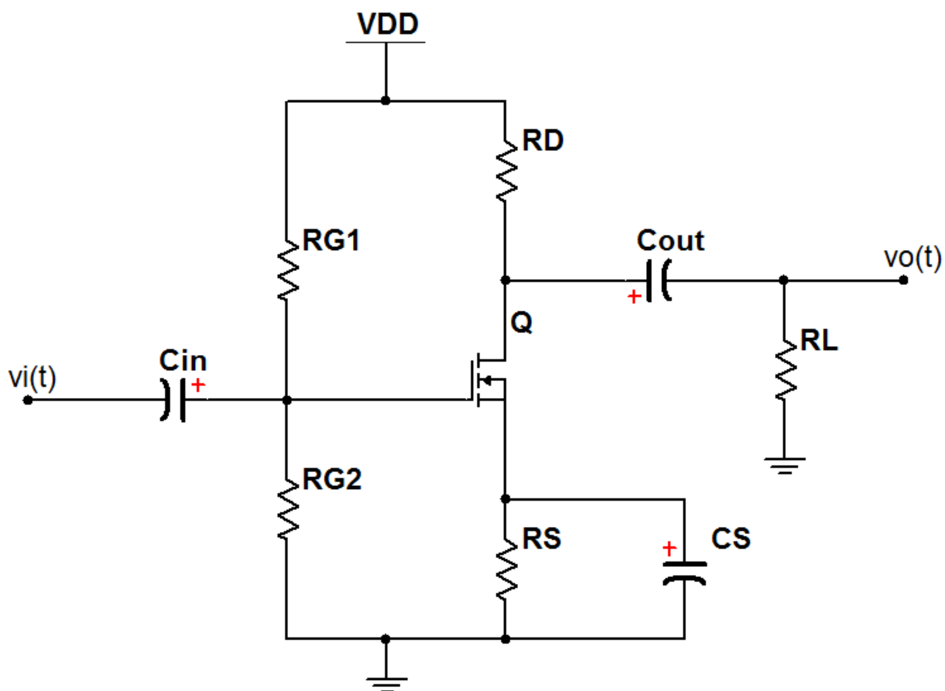


Figure 4.6: Circuit for Problem 4.6

Analysis

1. Determine the midband AC small-signal model of the circuit. Note that the portion of the bias network that controls I_{DQ} is the same as the circuit for Problem 4.5 on page 150; use the results of that problem to determine the transconductance gain g_m.

2. Calculate the open-circuit voltage gain A_{vo} with R_L unconnected.

3. Calculate the voltage gain A_v with R_L connected.

Use the following circuit components:

Component	Value
V_{DD}	9 V
MOSFET	$V_{\mathrm{t}} = 1.8$ V and $K = 50$ mA/V^2 ; $\lambda = 0$
Resistors	$R_{\mathrm{G1}} = 2$ MΩ, $R_{\mathrm{G2}} = 1$ MΩ, $R_{\mathrm{S}} = 100$ Ω, $R_{\mathrm{D}} = 330$ Ω, and $R_{\mathrm{L}} = 1$ kΩ
Capacitors	$C_{\mathrm{in}} = 1$ μF, $C_{\mathrm{out}} = 10$ μF, and $C_{\mathrm{S}} = 470$ μF

Simulation with NI Multisim

Enter the circuit of Figure 4.6 on the facing page using the `ZVN3306A` transistor model. Use the Simulate → Instruments → Function Generator to apply a 200 mV peak-to-peak sinusoid at 1 kHz as the input signal, and use the Simulate → Instruments → Oscilloscope to display the input and output signals.

1. Determine the open-circuit voltage gain A_{vo} with R_{L} unconnected.

2. Determine the voltage gain A_{v} with R_{L} connected.

Additional Multisim tips for this problem:

- The ZVN3306A is not part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.5 on page 190 to learn how to create and add this device to your "User Database."

- Place the cursors at the maximum and minimum values of the output waveform, and then read the "T2-T1" display as the peak-to-peak value.

NI Multisim video tutorials:

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

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

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

- Find the maximum value of a trace in Grapher View:
  `http://youtu.be/MzYK60mfh2Y`

Measurement with NI myDAQ

Build the circuit of Figure 4.6 on page 154 using the ZVN3306A transistor.

> **CAUTION:** *MOSFET devices are easily damaged by "ESD" (electrostatic discharge). Carry the MOSFET in conductive foam, and touch a grounded point nearby before handling the MOSFET.*

Use two 1 MΩ resistors in series for R_{G1} if a 2 MΩ resistor is not readily available.

> **IMPORTANT!** *Observe correct polarity on the electrolytic capacitors to ensure safe operation.*

Use the NI ELVISmx Function Generator to apply a 200 mV peak-to-peak sinusoid as the input signal, and use the ELVISmx Oscilloscope to display the input and output signals.

1. Determine the open-circuit voltage gain A_{vo} with R_{L} unconnected; you may also use a high-value resistor such as 100 kΩ or 1 MΩ as R_{L} to allow the coupling capacitor to charge in a reasonable amount of time.

2. Determine the voltage gain A_{v} with R_{L} connected.

Additional myDAQ tips for this problem:

- Use the ZVN3306A n-channel enhancement-mode MOS transistor described in Appendix E.5 on page 190. Follow the pinout diagram of Figure E.5 on page 190 to determine the gate, drain, and source connections.

- Implement the voltage source V_{DD} with the adjustable DC voltage source circuit of Figure B.2 on page 172.

- Use the NI ELVISmx Oscilloscope's peak-to-peak meters to measure the input and output levels for the gain calculation.

- All voltages within the amplifier can be displayed on the oscilloscope for troubleshooting purposes, however, the transistor gate voltage requires special treatment: ensure that only one channel is enabled when measuring voltages at high-impedance points in the circuit.

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

Study the behavior of your circuit under various operating conditions. Take representative screen shots of the oscilloscope and discuss your findings. Areas to consider include:

- Probe each of the nodes in the circuit, tracing how the input signal "moves" through the amplifier (see the point made earlier about how to probe the high-impedance points in the circuit),
- Investigate the triangle and square waveforms,
- Study the effect of increased input amplitude, and
- Observe the impact of reducing the value of the bypass capacitor C_S or removing it altogether.

4.7 Common-Drain Amplifier

The *common-drain amplifier* shown in Figure 4.7, also known as a *source follower*, presents a high input impedance to the source and creates a low-impedance replica of the input voltage signal, i.e., the output "follows" the input.

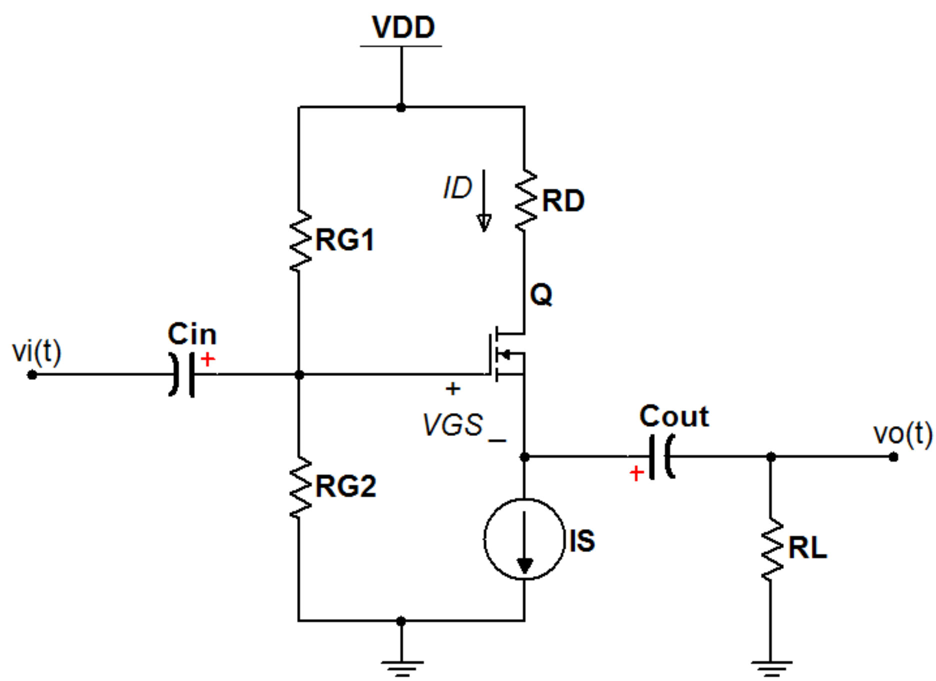


Figure 4.7: Circuit for Problem 4.7

Analysis

1. Determine the MOSFET operating point I_{DQ} and V_{GSQ}.

2. Determine the midband AC small-signal model of the circuit.

3. Calculate the open-circuit voltage gain A_{vo} with R_L unconnected.

4. Calculate the voltage gain A_v with R_L connected.

Use the following circuit components:

Component	Value
V_{DD}	9 V
MOSFET	$V_{\mathrm{t}} = 1.8$ V and $K = 50$ mA/V^2 ; $\lambda = 0$
Resistors	$R_{\mathrm{G1}} = 2$ MΩ, $R_{\mathrm{G2}} = 1$ MΩ, $R_{\mathrm{D}} = 330\ \Omega$, and $R_{\mathrm{L}} = 100\ \Omega$
Capacitors	$C_{\mathrm{in}} = 1\ \mu$F and $C_{\mathrm{out}} = 470\ \mu$F
I_{S}	12.5 mA

Simulation with NI Multisim

Enter the circuit of Figure 4.7 on the facing page using the `ZVN3306A` transistor model. Use the Simulate → Instruments → Function Generator to apply a 2 V peak-to-peak sinusoid at 1 kHz as the input signal, and use the Simulate → Instruments → Oscilloscope to display the input and output signals.

1. Determine the open-circuit voltage gain A_{vo} with R_{L} unconnected.
2. Determine the voltage gain A_{v} with R_{L} connected.

Additional Multisim tips for this problem:

- The ZVN3306A is not part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.5 on page 190 to learn how to create and add this device to your "User Database."
- Place the cursors at the maximum and minimum values of the output waveform, and then read the "T2-T1" display as the peak-to-peak value.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Basic operation of the two-channel oscilloscope:
  `http://youtu.be/qnRK6QyqjvQ`
- Function generator:
  `http://youtu.be/CeOl6EzD-_c`
- Find the maximum value of a trace in Grapher View:
  `http://youtu.be/MzYK60mfh2Y`

Measurement with NI myDAQ

Build the circuit of Figure 4.7 on page 158 using the ZVN3306A transistor.

> **CAUTION:** *MOSFET devices are easily damaged by "ESD" (electrostatic discharge). Carry the MOSFET in conductive foam, and touch a grounded point nearby before handling the MOSFET.*

Use two 1 MΩ resistors in series for R_{G1} if a 2 MΩ resistor is not readily available.

> **IMPORTANT!** *Observe correct polarity on the electrolytic capacitors to ensure safe operation.*

Use the NI ELVISmx Function Generator to apply a 2 V peak-to-peak sinusoid as the input signal, and use the ELVISmx Oscilloscope to display the input and output signals.

1. Determine the open-circuit voltage gain A_{vo} with R_L unconnected; you may also use a high-value resistor such as 100 kΩ as R_L to allow the coupling capacitor to charge in a reasonable amount of time.
2. Determine the voltage gain A_v with R_L connected.

Additional myDAQ tips for this problem:

- Use the ZVN3306A n-channel enhancement-mode MOS transistor described in Appendix E.5 on page 190. Follow the pinout diagram of Figure E.5 on page 190 to determine the gate, drain, and source connections.
- Implement the voltage source V_{DD} with the adjustable DC voltage source circuit of Figure B.2 on page 172.
- Implement the current source I_S according to the circuit diagram of Figure B.5 on page 175. Use a 100 Ω resistor for the adjustment resistor R.
- Measure $I_S = R_D$ with the myDAQ DMM ammeter and confirm that the current is close to 12.5 mA.
- Use the NI ELVISmx Oscilloscope's peak-to-peak meters to measure the input and output levels for the gain calculation.

- All voltages within the amplifier can be displayed on the oscilloscope for troubleshooting purposes, however, the transistor gate voltage requires special treatment: ensure that only one channel is enabled when measuring voltages at high-impedance points in the circuit.

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

Study the behavior of your circuit under various operating conditions. Take representative screen shots of the oscilloscope and discuss your findings. Areas to consider include:

- Probe each of the nodes in the circuit, tracing how the input signal "moves" through the amplifier (see the point made earlier about how to probe the high-impedance points in the circuit),
- Investigate the triangle and square waveforms, and
- Study the effect of increased input amplitude.

4.8 Logic Inverters: RTL and CMOS

Figure 4.8 shows two implementations of a logic inverter. The *resistor-transistor-logic* (RTL) version of Figure 4.8(a) uses a passive resistor to pull the output high when the transistor is off, while the *complementary MOSFET* (CMOS) version of Figure 4.8(b) uses another transistor to pull the output high. The two implementations differ considerably in the amount of quiescent power they require, i.e., the power required simply to maintain a constant output voltage for a constant input voltage. Note the absence of loads on the outputs of each inverter.

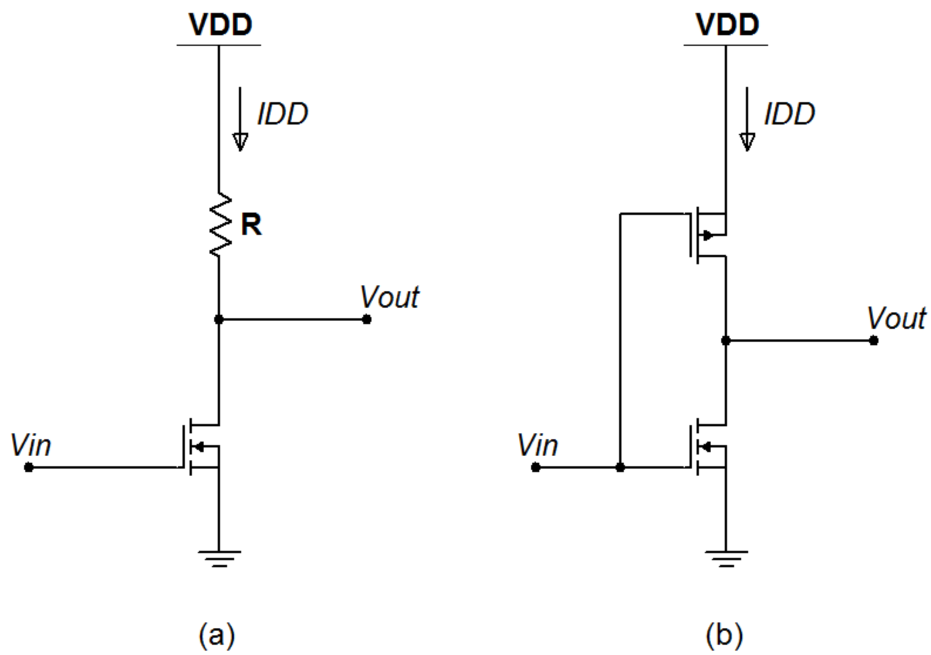


Figure 4.8: Circuit for Problem 4.8

Analysis

1. Determine V_{out} for each inverter when V_{in} is 0 V (a logical 0) and when V_{in} is 5 V (a logical 1); use these input voltages for subsequent steps.

2. Determine I_{DD} in mA for each inverter when V_{in} takes on the two possible logic levels.

3. Determine P_{DD} in mW (the power delivered by the power supply) for each inverter when V_{in} takes on the two possible logic levels.

4. Compare and contrast the two inverters in terms of output voltage and power requirements. Explain how your results point toward the fact that CMOS is the dominant circuit implementation method for VLSI (very large scale integration) integrated circuits.

Use the following circuit components:

Component	Value
V_{DD}	5 V
n-MOSFET	$V_{\mathrm{t}} = 1.8$ V and $K = 50$ mA/V^2 ; $\lambda = 0$
p-MOSFET	$V_{\mathrm{t}} = 2.9$ V and $K = 50$ mA/V^2 ; $\lambda = 0$
R	1 kΩ

Simulation with NI Multisim

Enter both circuits of Figure 4.8 on the preceding page using the `ZVN3306A` and `ZVP3306A` transistor models. Set up a DC voltage source and an `SPDT` switch to conveniently set the input to either 0 V or 5 V. Place measurement probes to display the power supply current I_{DD} and the inverter output V_{out}.

1. Determine V_{out} for each inverter when V_{in} takes on the two possible logic levels.

2. Determine I_{DD} in mA for each inverter when V_{in} takes on the two possible logic levels.

3. Determine P_{DD} in mW (the power delivered by the power supply) for each inverter when V_{in} takes on the two possible logic levels.

Additional Multisim tips for this problem:

- Neither the ZVN3306A nor the ZVP3306A transistors are part of the standard NI Multisim components database as of version 11.0. Refer to Appendix E.5 on page 190 and Appendix E.6 on page 192 to learn how to create and add these devices to your "User Database."

- Flip the p-channel MOSFET symbol vertically to ensure that its source connects to V_{DD}.

- Calculate P_{DD} as the product of V_{DD} and I_{DD}.

NI Multisim video tutorials:

- Find components by name:
  `http://youtu.be/5wlFweh4n-c`
- Measure DC current with a measurement probe:
  `http://youtu.be/uZ56byigymI`

Measurement with NI myDAQ

Build both circuits of Figure 4.8 on page 162 using the ZVN3306A and ZVP3306A transistors.

> **CAUTION:** *MOSFET devices are easily damaged by "ESD" (electrostatic discharge). Carry the MOSFET in conductive foam, and touch a grounded point nearby before handling the MOSFET.*

Power the circuit with 5V and AGND. Drive V_{in} with a wire that you connect either to ground or to 5V. Use the DMM ammeter on its most sensitive range (20 mA) to monitor the power supply current I_{DD}, and use the RMS meter of the NI ELVISmx Oscilloscope to monitor the output voltage V_{out}. Make certain that you enable only one channel on the oscilloscope.

1. Determine V_{out} for each inverter when V_{in} takes on the two possible logic levels.
2. Determine I_{DD} in mA for each inverter when V_{in} takes on the two possible logic levels.
3. Determine P_{DD} in mW (the power delivered by the power supply) for each inverter when V_{in} takes on the two possible logic levels.

Additional myDAQ tips for this problem:

- Use the ZVN3306A n-channel enhancement-mode MOS transistor described in Appendix E.5 on page 190. Follow the pinout diagram of Figure E.5 on page 190 to determine the gate, drain, and source connections.

- Use the ZVP3306A p-channel enhancement-mode MOS transistor described in Appendix E.6 on page 192. Follow the pinout diagram of Figure E.6 on page 192 to determine the gate, drain, and source connections.

- Ensure that you connect the p-channel MOSFET's *source* to V_{DD}, not its drain.

- Measure and record V_{DD} when your circuit is connected; it will likely not be exactly 5.00 V. Take this measurement for each circuit, too. Calculate P_{DD} as the product of V_{DD} and I_{DD}.

NI myDAQ video tutorials:

- 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

1. Study the input-output behavior of each inverter when you apply the the square wave and triangle waveforms available from the NI ELVISmx Function Generator. Drive the inverter input with AO0, set up the peak-to-peak voltage and DC offset so that V_{in} swings between 0 V and 5 V, and use 200 Hz for the frequency. Remove the DMM ammeter, too. Use the NI ELVISmx Oscilloscope to view the inverter input and output, and set the timebase to 2 ms.

   Discuss the behavior of the output in response to the square wave input (an obviously "digital" signal) as well as the triangle waveform that linearly sweeps the input from one input level to the other. Include representative screen shots of the oscilloscope display.

2. Investigate the dynamic current required by each inverter when it drives a small capacitive load to emulate the input capacitance of other logic gate inputs connected to the inverter output. Modify your circuit as follows: (1) add a 10 Ω current shunt resistor in line with the power supply connection (between V_{DD} and the source of the p-channel MOSFET); this is a standard technique to display a current

waveform on the oscilloscope, and (2) connect a 0.1 μF capacitor between V_{out} and ground. Set the oscilloscope timebase to 2 ms (important, use this exact setting!) and keep the function generator at 200 Hz. Display the input voltage on Channel 0 and the supply current on Channel 1. Be sure to connect AI1+ to the V_{DD} side of the shunt resistor and AI- to the other side of the shunt. Study the dynamic current for the triangle waveform and for the square wave.

Discuss your findings, and include representative screen shots of the oscilloscope display. In particular, comment on the importance of maintaining a rapid transition on the input for the CMOS inverter.

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 with the following exceptions: (1) the Digi-Key kit does not contain a 10 MΩ resistor, and (2) none of the kits contain a 2 MΩ resistor (this can be created with two series-connected 1 MΩ resistors):

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
2	100	Brown - Black - Brown
1	220	Red - Red - Brown
1	330	Orange - Orange - Brown
1	470	Yellow - Violet - Brown
1	680	Blue - Gray - Brown

Qty	Value (kΩ)	Color Code
5	1.0	Brown - Black - Red
1	1.5	Brown - Green - Red
2	2.2	Red - Red - Red
1	3.3	Orange - Orange - Red
3	4.7	Yellow - Violet - Red
2	10	Brown - Black - Orange
1	22	Red - Red - Orange
1	33	Orange - Orange - Orange
1	47	Yellow - Violet - Orange
2	100	Brown - Black - Yellow
1	470	Yellow - Violet - Yellow
Qty	Value (MΩ)	Color Code
3	1.0	Brown - Black - Green
1	2.0	Red - Black - Green
1	10	Brown - Black - Blue

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	0.01	Ceramic
1	0.1	Ceramic
1	1	Electrolytic
1	10	Electrolytic
1	100	Electrolytic
1	470	Electrolytic

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 #
2	LM317L voltage regulator, 100mA	296-17221-1-ND
2	TL072CP dual op amp	296-1775-5-ND
4	1N4148 switching diode, DO-35 case	1N4148TACT-ND
1	1N5232B 5.6 V zener diode, DO-35 case	1N5232BFSCT-ND
2	2N2222A NPN BJT, TO-92 case	P2N2222AGOS-ND
2	2N3904 NPN BJT, TO-92 case	2N3904-APCT-ND
2	2N3906 PNP BJT, TO-92 case	2N3906-APCT-ND
1	CA3083 NPN BJT array, DIP16 package	CA3083Z-ND
1	ZVN3306A n-channel enhancement-mode MOSFET	ZVN3306A-ND
1	ZVP3306A n-channel enhancement-mode MOSFET	ZVP3306A-ND

Light Emitting Diodes (LEDs)

Four LEDs required, round with domed top, through-hole mounting. Available from all listed distributors.

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

Protoboard for NI myDAQ

This breadboard connects directly to the NI myDAQ and eliminates the need for "flying test leads": Studica part number NIPCB1, `http://www.studica.com`

Carrying Case for NI myDAQ

Keep everything organized with room for the NI myDAQ, DMM and USB cables, jumper wire kit, and protoboard: Studica part number 9NICASE1, `http://www.studica.com`

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.

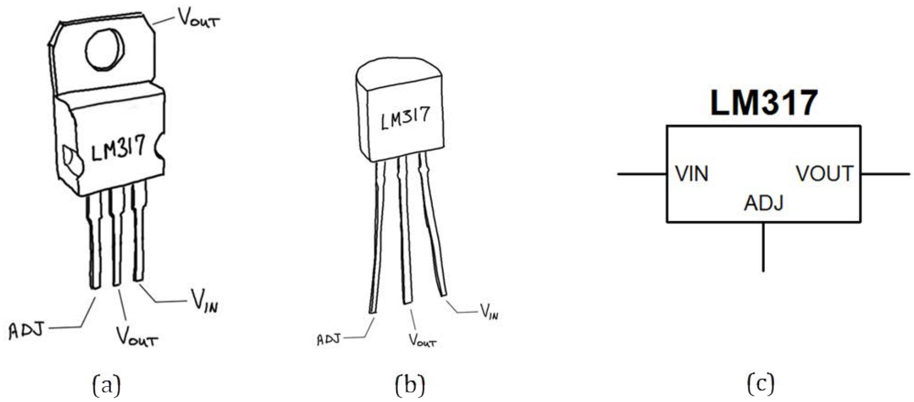


Figure B.1: LM317 adjustable voltage regulator: (a) case and terminals for 1.5-amp device (TO-220 case), (b) case and terminals for 100-mA device (TO-92 case), and (c) schematic symbol.

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.

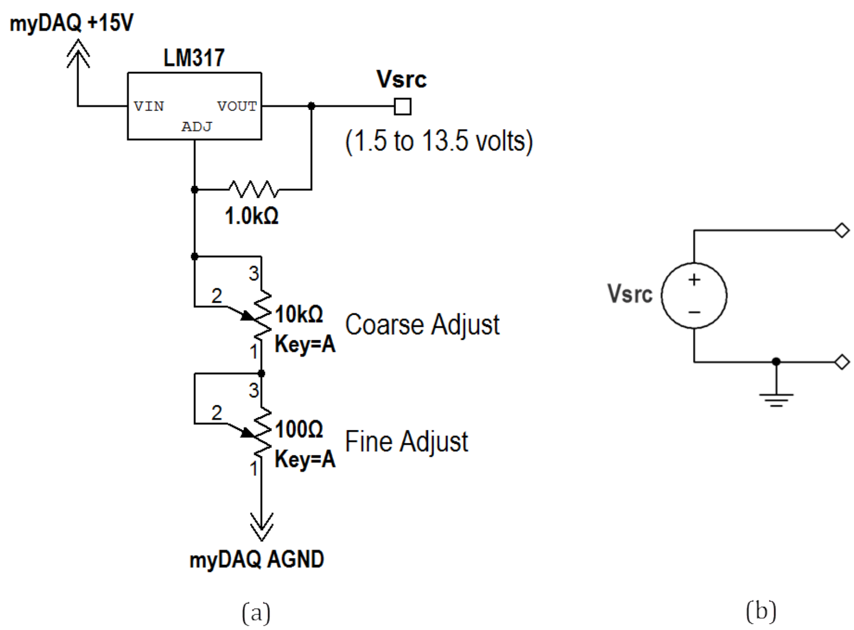


Figure B.2: LM317 as a variable voltage source: (a) schematic diagram and (b) equivalent circuit.

B.2 Current Source

The circuit shown in Figure B.4 on page 174 produces a current whose value is approximately $1250/R$ mA. This circuit configuration "sources" current

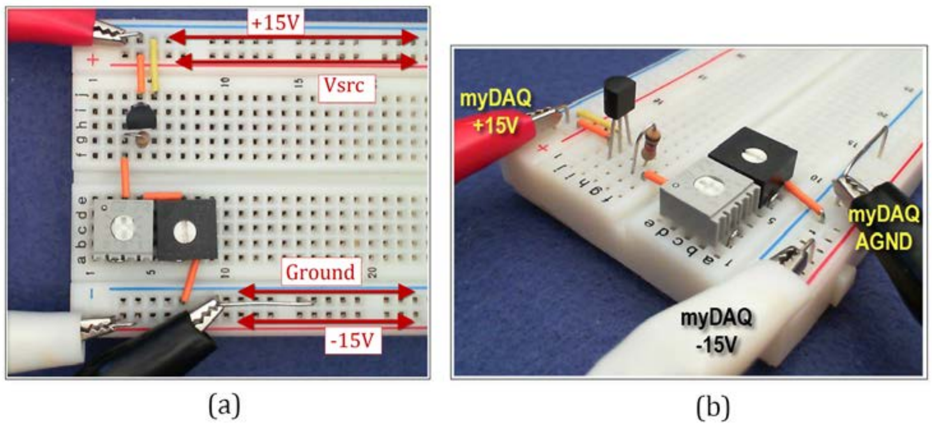


Figure B.3: Recommended breadboard layout for the LM317-based variable voltage source: (a) top view of breadboard showing the component layout and voltage rail order, and (b) side view showing test lead connections between NI myDAQ and wire loops on the breadboard.

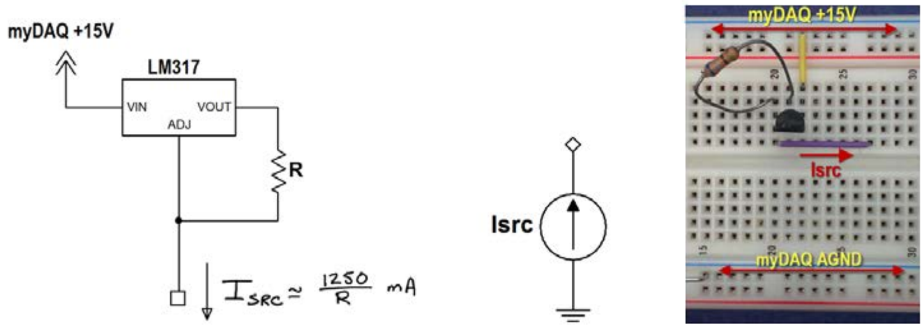
from the NI myDAQ +15V power supply and effectively operates as a current source with one terminal permanently attached to the NI myDAQ analog ground AGND.

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 175 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).

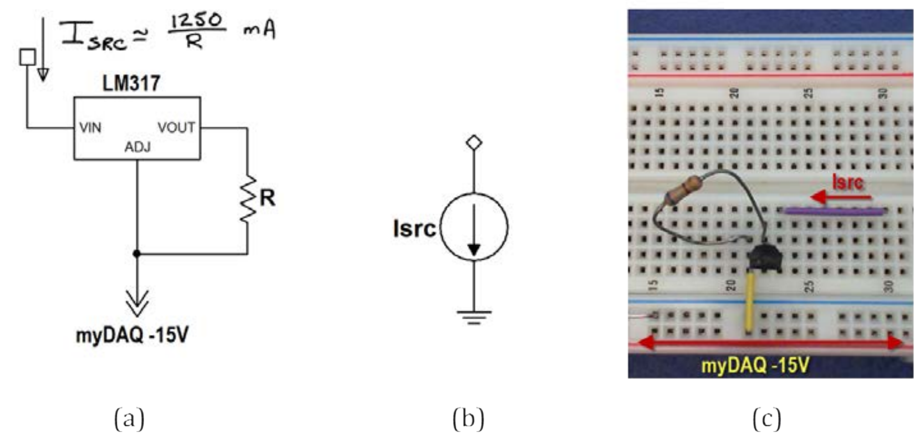


Figure B.5: LM317 adjustable voltage regulator as a grounded current source sinking current to the NI myDAQ -15V power supply: (a) schematic diagram, (b) equivalent circuit model, and (c) recommended layout with the standard breadboard layout of Figure B.3 on page 173.

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 179 shows the TL072 placed on the standard breadboard layout described in Figure B.3 on page 173, connected to power, and ready for additional circuitry.

NI Multisim provides a circuit model for the TL072: place the `TL072CP` device.

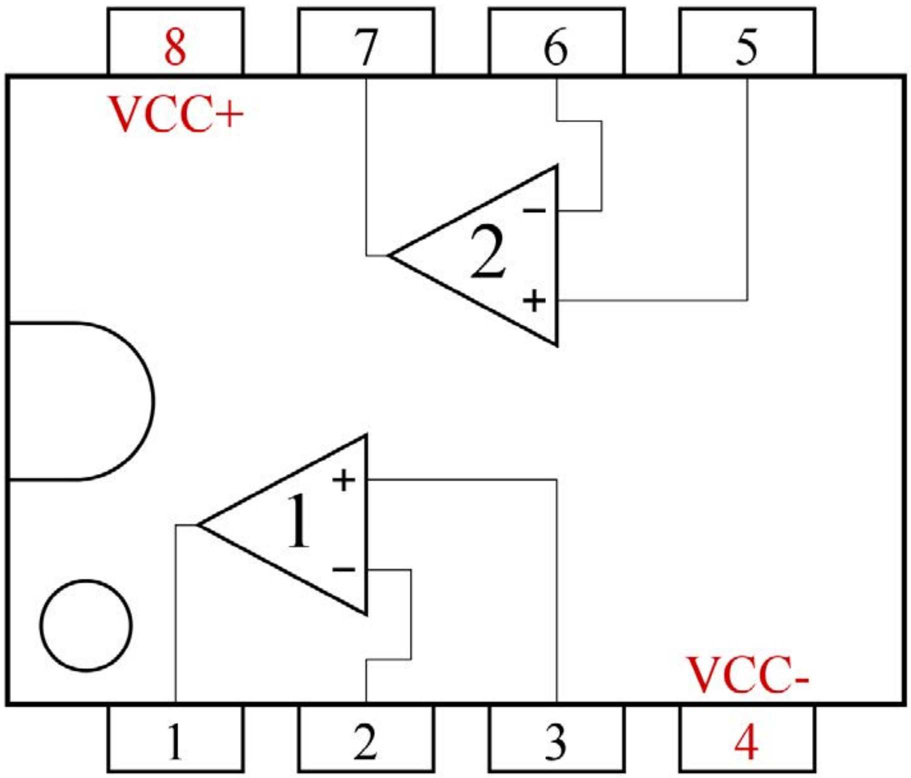


Figure C.1: Texas Instruments TL072 dual op amp pinout diagram (top view). The plastic package uses either a U-shaped cutout to indicate the left side or an indented circle to indicate pin 1. NOTE: In this book the positive supply "VCC+" is called V_{CC} and the negative supply "VCC-" is called V_{EE}.

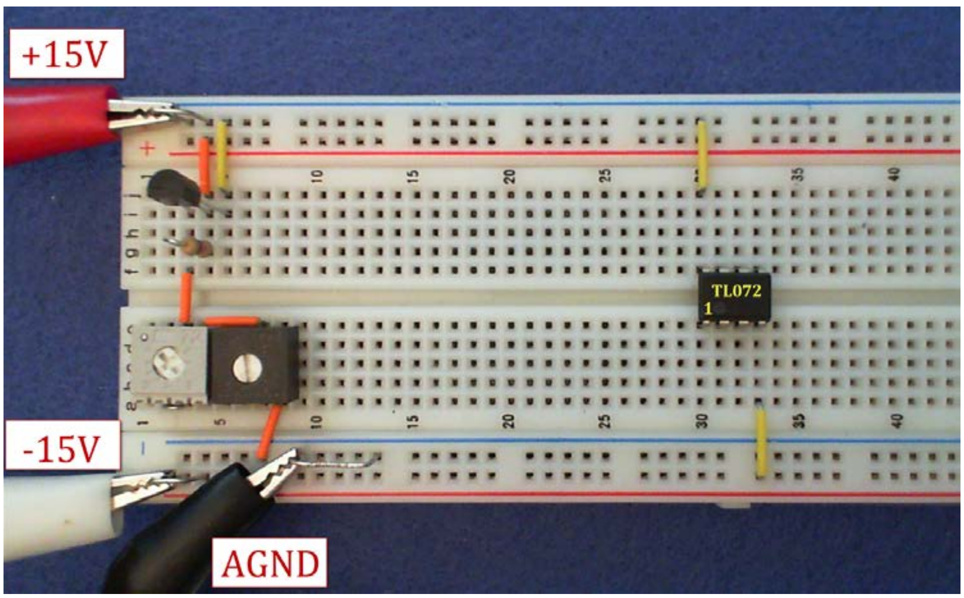


Figure C.2: Texas Instruments TL072 dual op amp placed on the standard breadboard layout, connected to power, and ready for additional circuitry.

Appendix D

Diodes

D.1 1N4148 Switching Diode

- Datasheet: NXP Semiconductors, `http://www.nxp.com`, search `1N4148`
- Multisim model: `1N4148`

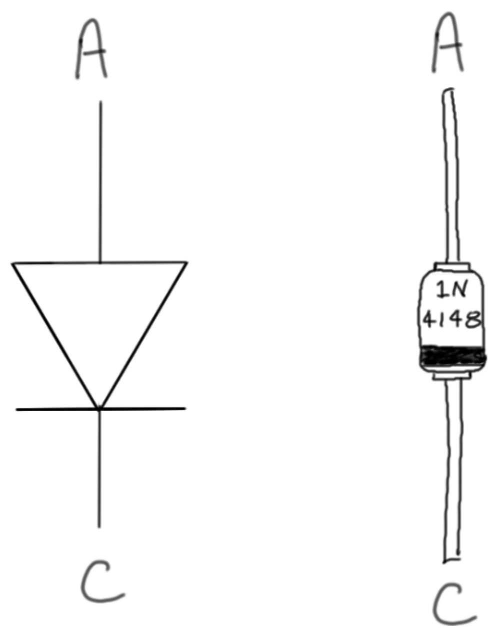


Figure D.1: 1N4148 switching diode, DO-35 case.

D.2 Light-Emitting Diode (LED)

- Datasheet: varies
- Multisim model: `LED_red` (other colors available, too)

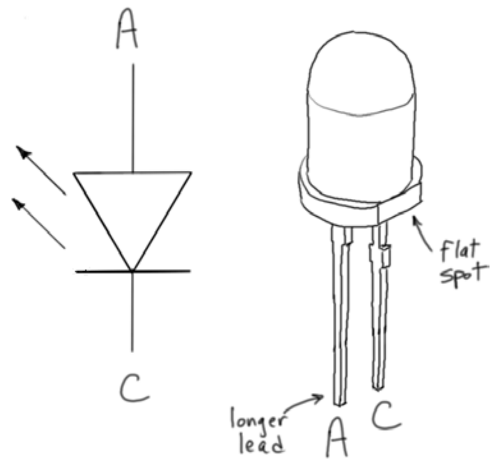


Figure D.2: Light-emitting diode.

D.3 1N5232B 5.6 V Zener Diode

- Datasheet: Fairchild Semiconductor, `http://www.fairchildsemi.com`, search `1N5232B`
- Multisim model: `ZENER_VIRTUAL`, double-click after placing and change "Breakdown Voltage" to 5.6 volts.

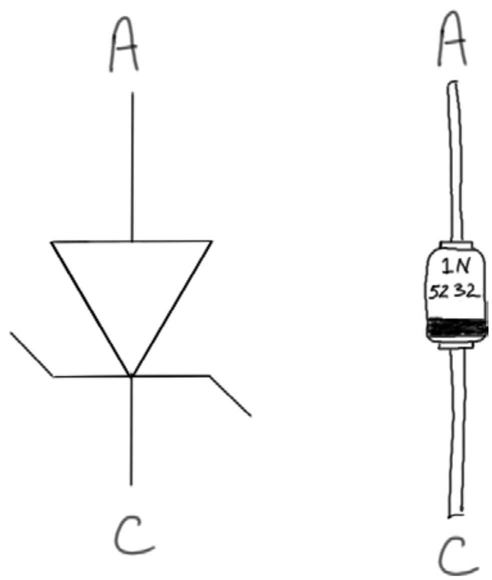


Figure D.3: 1N5232B 5.6 V zener diode, DO-35 case.

Appendix E

Transistors

E.1 2N2222A NPN BJT

- Datasheet: ON Semiconductor, `http://www.onsemi.com`, search `P2N2222A`
- Multisim model: `2N2222A`

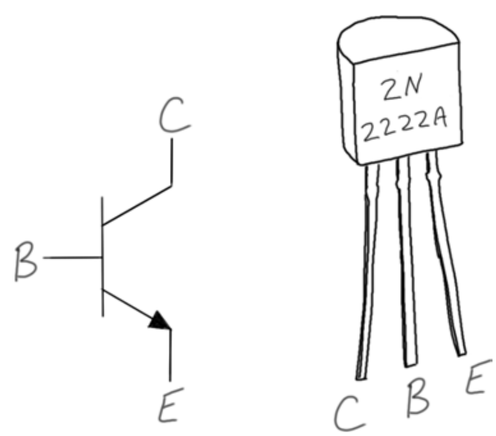


Figure E.1: 2N2222A NPN bipolar junction transistor, TO-92 case.

E.2 2N3904 NPN BJT

- Datasheet: ON Semiconductor, `http://www.onsemi.com`, search `2N3904`
- Multisim model: `2N3904`
- Complementary PNP device: 2N3906

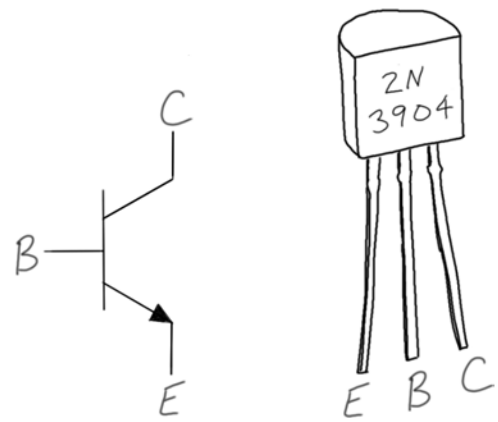


Figure E.2: 2N3904 NPN bipolar junction transistor, TO-92 case.

E.3 2N3906 PNP BJT

- Datasheet: ON Semiconductor, `http://www.onsemi.com`, search `2N3906`
- Multisim model: `2N3906`
- Complementary NPN device: 2N3904

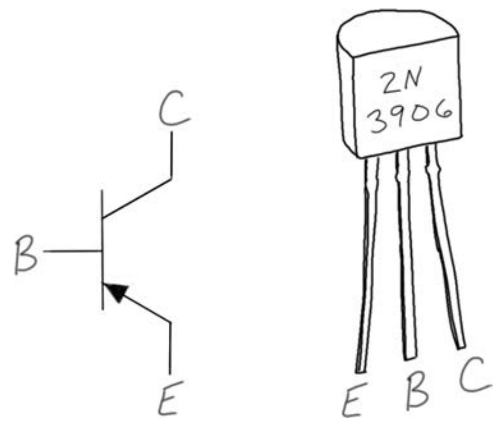


Figure E.3: 2N3906 PNP bipolar junction transistor, TO-92 case.

E.4 CA3083 NPN BJT Array

- Datasheet: Intersil, `http://www.intersil.com`, search `ca3083`
- Multisim model: see below

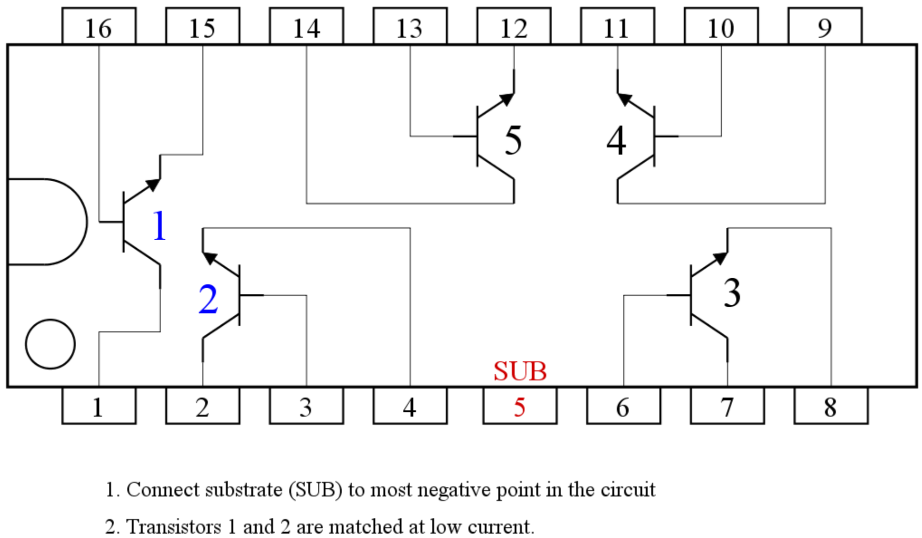


Figure E.4: CA3083 NPN BJT array, DIP-16 package.

Multisim version 11.0 does not include the CA3083 as a built-in component. Follow along with the video tutorial at `http://youtu.be/e0lObOAqgbE` to learn how to add this component to your "User Database." Copy and paste the SPICE model text on the next page when prompted for the device model; check the CA3083 product page at `http://www.intersil.com` for the most recent model.

```
*COPYRIGHT (c) 1997 INTERSIL CORPORATION
*ALL RIGHTS RESERVED
*
*CA3083 NPN PSPICE MODEL
*REV: 3-13-97
** ----- BJT MODEL -----
*
.model ca3083   NPN
+ (IS = 10.0E-15 XTI = 3.000E+00 EG = 1.110E+00 VAF = 1.00E+02
+ VAR = 1.000E+02 BF = 112.8E+00 ISE = 99.086E-15 NE = 1.410E+00
+ IKF = 120.900E-03 XTB = 0.000E+00 BR = 16.0E+00 ISC = 116.12E-15
+ NC = 1.700E+00 IKR = 29.800E-03 RC = 10.000E+00 CJC = 991.71E-15
+ MJC = 0.333E-00 VJC = 0.7500E-00 FC = 5.000E-01 CJE = 1.02E-12
+ MJE = .333E-00 VJE = 0.750E-00 TR = 10.000E-09 TF = 275.61E-12
+ ITF = .3750E-00 XTF = 91.950E+00 VTF = 8.90E+00 PTF = 0.000E+00
+ RE = 0.0E+00 RB = 0.00E+00
```

E.5 ZVN3306A N-Channel Enhancement MOSFET

- Datasheet: Diodes/Zetex, `http://www.diodes.com`, search `zvn3306a`
- Multisim model: see below
- Complementary p-channel device: ZVP3306A

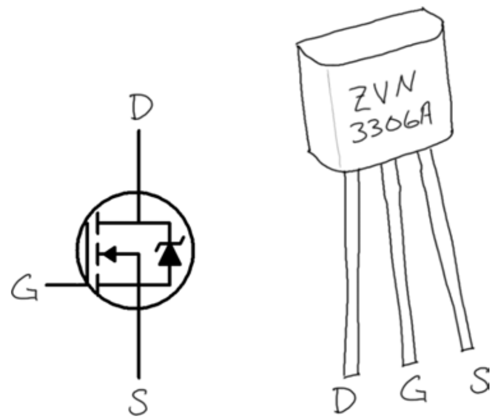


Figure E.5: ZVN3306A n-channel enhancement-mode MOSFET, E-line case.

Multisim version 11.0 does not include the ZVN3306A as a built-in component. Follow along with the video tutorial at `http://youtu.be/ffsMhTLRPaA` to learn how to add this component to your "User Database." Copy and paste the SPICE model text on the next page when prompted for the device model; check the ZVN3306A product page at `http://www.diodes.com` for the most recent model.

```
*
*Zetex ZVN3306A Spice Model v1.1 Last Revised 3/5/00
*
.SUBCKT ZVN3306A 3 4 5
*                D G S
M1 3 2 5 5 N3306M
RG 4 2 270
RL 3 5 1.2E8
C1 2 5 28E-12
C2 3 2 3E-12
D1 5 3 N3306D
*
.MODEL N3306M NMOS VTO=1.824 RS=1.572 RD=1.436 IS=1E-15 KP=.1233
+CBD=35E-12 PB=1
.MODEL N3306D D IS=5E-12 RS=.768
.ENDS ZVN3306A
*
*$
*
*                (c)  2005 Zetex Semiconductors plc
*
*   The copyright in these models  and  the designs embodied belong
*   to Zetex Semiconductors plc (" Zetex ").  They  are  supplied
*   free of charge by Zetex for the purpose of research and design
*   and may be used or copied intact  (including this notice)  for
*   that purpose only.  All other rights are reserved. The models
*   are believed accurate but  no condition  or warranty  as to their
*   merchantability or fitness for purpose is given and no liability
*   in respect of any use is accepted by Zetex PLC, its distributors
*   or agents.
*
*   Zetex Semiconductors plc, Zetex Technology Park, Chadderton,
*   Oldham, United Kingdom, OL9 9LL
```

E.6 ZVP3306A P-Channel Enhancement MOSFET

- Datasheet: Diodes/Zetex, `http://www.diodes.com`, search `zvp3306a`
- Multisim model: Not available in Multisim version 11.0; see section below to learn how to add this device to your User Database
- Complementary n-channel device: ZVN3306A

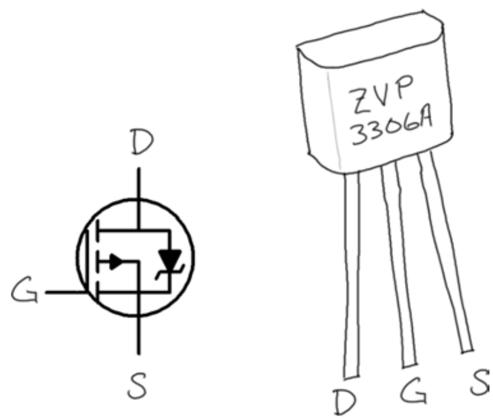


Figure E.6: ZVP3306A n-channel enhancement-mode MOSFET, E-line case.

Multisim version 11.0 does not include the ZVP3306A as a built-in component. Follow along with the video tutorial at `http://youtu.be/HJq6QHOyTsI` to learn how to add this component to your "User Database." Copy and paste the SPICE model text on the next page when prompted for the device model; check the ZVP3306A product page at `http://www.diodes.com` for the most recent model.

```
*
*Zetex ZVP3306A Spice Model v1.1 Last Revised 3/5/00
*
.SUBCKT ZVP3306A 3 4 5
*                D G S
M1 3 2 5 5 P3306M
RG 4 2 252
RL 3 5 1.2E8
C1 2 5 28E-12
C2 3 2 3E-12
D1 3 5 P3306D
*
.MODEL P3306M PMOS VTO=-2.875 RS=5.227 RD=7.524 IS=1E-15 KP=.145
+CBD=35E-12 PB=1 LAMBDA=6.67E-3
.MODEL P3306D D IS=5E-12 RS=.768
.ENDS ZVP3306A
*
*$
*
*                (c)  2005 Zetex Semiconductors plc
*
*   The copyright in these models  and  the designs embodied belong
*   to Zetex Semiconductors plc (" Zetex ").  They  are  supplied
*   free of charge by Zetex for the purpose of research and design
*   and may be used or copied intact  (including this notice)  for
*   that purpose only.  All other rights are reserved. The models
*   are believed accurate but  no condition  or warranty  as to their
*   merchantability or fitness for purpose is given and no liability
*   in respect of any use is accepted by Zetex PLC, its distributors
*   or agents.
*
*   Zetex Semiconductors plc, Zetex Technology Park, Chadderton,
*   Oldham, United Kingdom, OL9 9LL
```

Appendix F

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
- Place and operate linear potentiometer:
  http://youtu.be/oazwGLzWvhs
- Place dual op amp second device:
  http://youtu.be/-QDFEf-KdEw

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
- VCC and VEE power supply voltages:
  http://youtu.be/XkZTwKD-WjE

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

Grapher View environment:

- Add data label to Grapher View plot:
  `http://youtu.be/uibNqFlod_Y`
- Plot mathematical expression:
  `http://youtu.be/qYVf_rkTF-o`
- Adjust axis limits and tick marks on a Grapher View plot:
  `http://youtu.be/nxHHMpJd5Ow`
- Plot second variable on its own axis in Grapher View:
  `http://youtu.be/rULFKRTphcI`
- Export simulation data to an Excel spreadsheet:
  `http://youtu.be/s6ezjb8Xrhc`

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

Word Generator:

- Create digital sequences with Word Generator:
  http://youtu.be/vp1l-TeRB3s

Transient response:

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

Temperature Sweep:

- Evaluate circuit behavior with Temperature Sweep:
  http://youtu.be/74zZaWqTcyU
- Vary the temperature of one component with a Parameter Sweep:
  http://youtu.be/OfXXPmpdlEU

Voltage-controlled switch:

- Voltage-controlled switch:
  http://youtu.be/BaEBjhD4TOw

DC Sweep:

- Plot DC circuit response with DC Sweep:
  http://youtu.be/vcYuCt9QjdI

Transistor and diode characteristic curves:

- Plot NPN BJT base characteristic:
  http://youtu.be/PJZ8i5O7daw

- Plot NPN BJT collector characteristic:
  `http://youtu.be/LgjaxmC7NB8`
- Plot diode current-voltage characteristic:
  `http://youtu.be/fGYyYm7J3JA`
- Plot n-channel enhancement MOSFET drain characteristic:
  `http://youtu.be/R-05ETpv-fc`
- Plot n-channel enhancement MOSFET transfer characteristic:
  `http://youtu.be/nkI1vxw1qP4`

Add components to User Database:

- Add ZVN3306A component to User Database:
  `http://youtu.be/ffsMhTLRPaA`
- Add ZVP3306A component to User Database:
  `http://youtu.be/HJq6QHOyTsI`
- Add CA3083 component to User Database:
  `http://youtu.be/e0lObOAqgbE`

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