EE302 Lab Manual

Fall

2016

Lab Manual for the Introduction to Electrical Engineering

UT EID:

Name:

The University of Texas at Austin
Department of Electrical & Computer Engineering

Table of Contents

1.0 Introduction

This manual is designed to accompany the EE302, Introduction to Electrical and Computer Engineering, lab section. You will find complete descriptions of the labs for EE302 and a great deal of information about your projects. Read this manual carefully.

2.0 EE302 Lab Schedule

The following sections describe each of the circuit laboratory sessions for the semester.

LAB ACTIVITY	DAY YOUR LAB MEETS				
	Mon.	Tues.	Wed.	Thurs.	Fri.
Lab 1: Intro to LabVIEW 1 and myDAQ	8/29	8/30	9/1	9/2	9/3
Lab 2 : Intro to LabVIEW 2	9/12	9/6	9/7	9/8	9/9
Lab 3: Digital Multimeter and Voltage Generator	9/19	9/13	9/14	9/15	9/16
Lab 4: Multisim Simulation of Circuits	9/26	9/20	9/21	9/22	9/23
Lab 5: Audio Equalizer	10/3	9/27	9/28	9/29	9/30
Lab 6: Breadboard and Measurements	10/10	10/4	10/5	10/6	10/7
Lab 7: Soldering and Kit Assembly	10/17	10/11	10/12	10/13	10/14
Lab 8: Solar Power and Data Logging	10/24	10/18	10/19	10/20	10/21
Lab 9: Final Project – Robot Car I	10/31	10/25	10/26	10/27	10/28
Lab 10: Proficiency Exam	11/7	11/1	11/2	11/3	11/4
Lab 11: Thevenin Equivalent Circuits	11/14	11/8	11/9	11/10	11/11
Lab 12: Final Project – Robot Car II	11/21	11/15	11/16	11/17	11/18
Lab 13: Final Project – Robot Car III	11/28	11/29	11/30	12/1	12/2

3.0 Grading

The laboratory for EE302 counts as **20%** of your final class grade. Your grade for the laboratory will be determined as follows:

Category	% of Lab Grade
Attendance	5%
Participation	5%
Pre-labs	5%
Lab Reports	25%
Lab Notes	5%
Proficiency Exam	15%
Final Project	40%
Total	100%

Lab 1: Introduction to LabView 1 & myDAQ

Lab Goals:

- Learn the basics of LabView programming and to design a simple VI (Virtual Instrument).
  - Compute the area of a triangle, How to use MathScript
- Learn the LabView Environment and Graphical Programming

Required Lab Material:

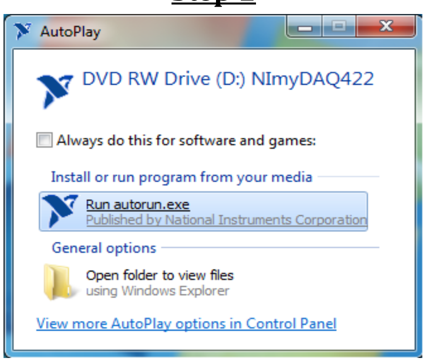
- NI myDAQ kit

Due at the end of Lab:

- A copy of the simple VI you designed

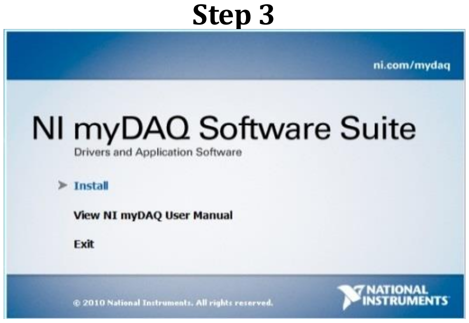
Background Knowledge: Installing NI LabView Software and Drivers

Step 1

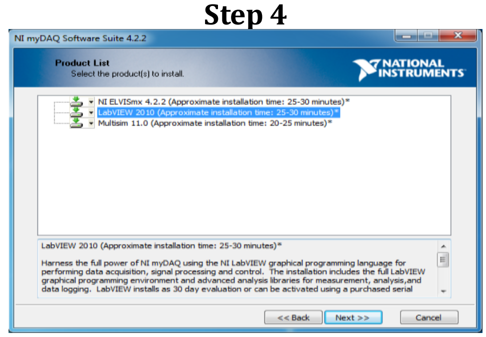


Step 2

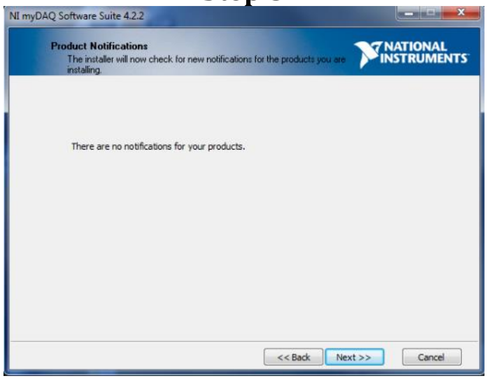
Step 3



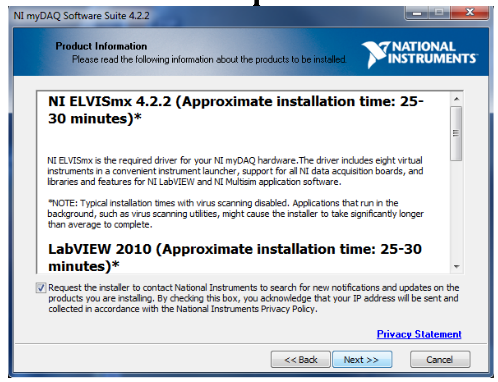
Step 4



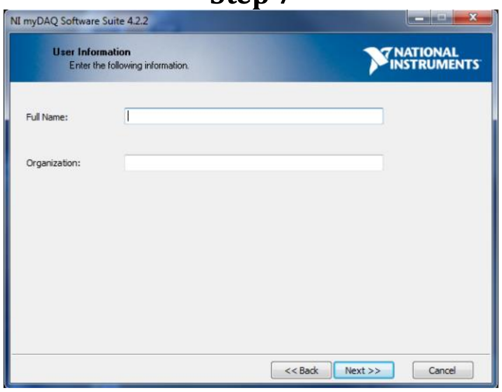
Step 5



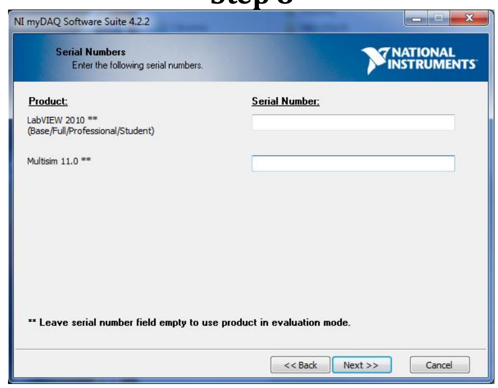
Step 6



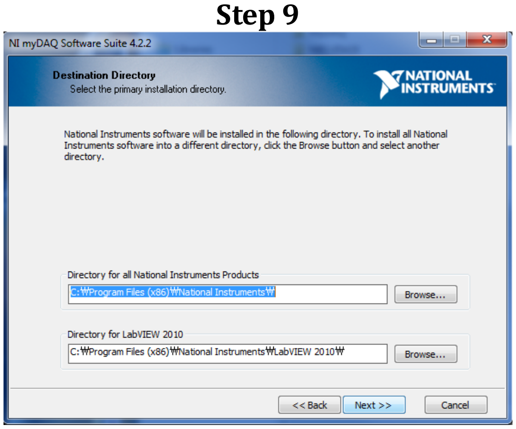
Step 7



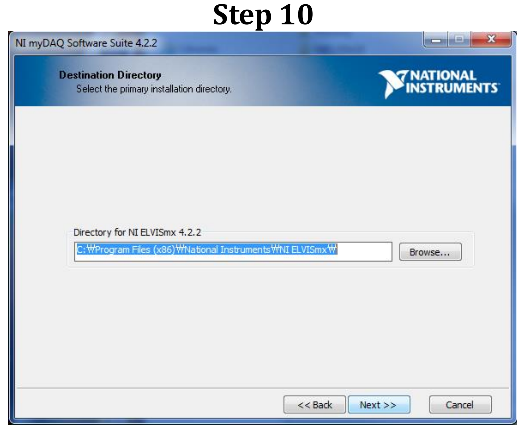
Step 8



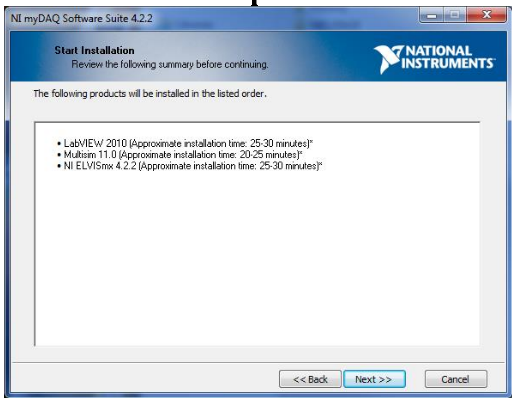
Step 9



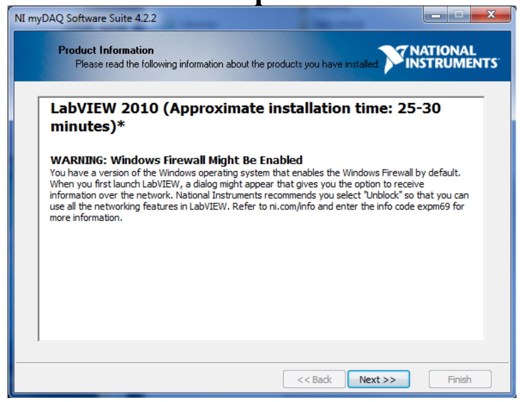
Step 10



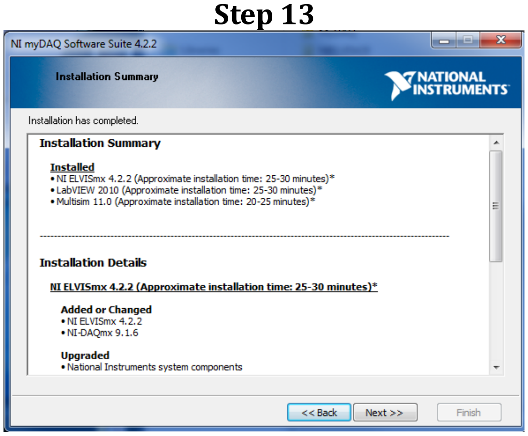
Step 11



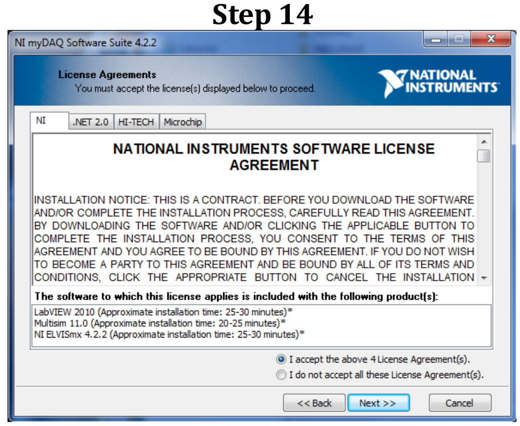
Step 12



Step 13



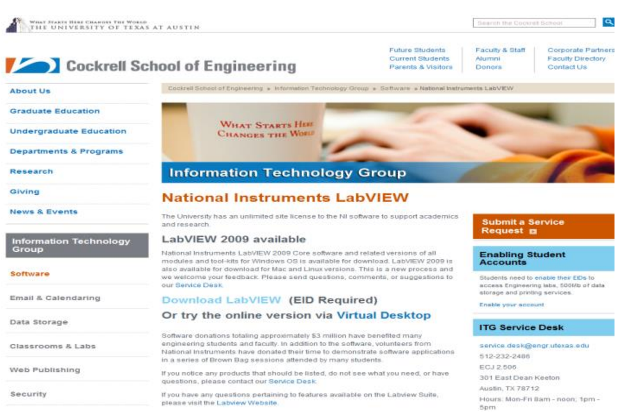
Step 14



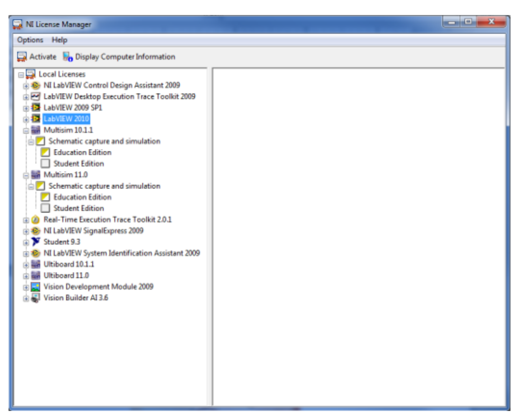
After installing the SW and drivers, go to the UT Engineering webpage and from the Information Technology Group (ITG) there are directions on how to obtain a student license to Labview and Multisim. http://www.engr.utexas.edu/itg/products/435-labview

You do not need to download the software from the website again. Just obtain the license and go to *NI License Manager* under **National Instruments** in *All programs.* Click active on the top left hand corner and apply the license information you obtained from ITG

UT School of Engineering Webpage



NI License Manager

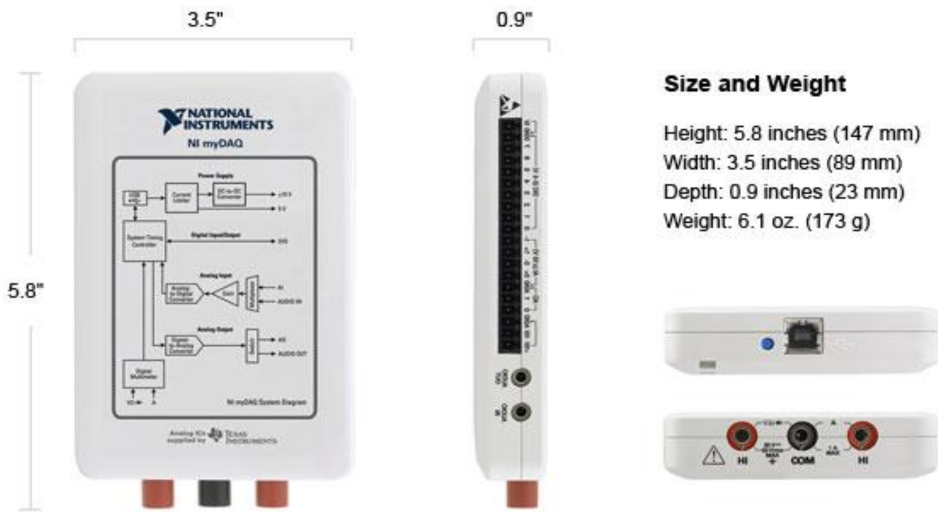


Background Knowledge: NI myDAQ

Refer to https://decibel.ni.com/content/community/academic/products_and_projects/mydaq to gain more ideas on how to utilize the NI myDAQ outside of the lab.

Reference:

- LabVIEW Introduction 3-Hour Courseware
- **https://lumen.ni.com/nicif/us/academiclv3hr/content.xhtml**
- **http://www.ni.com/mydaq/**
- **http://decibel.ni.com/content/groups/mydaq**



Assignment: Build VI to Calculate Area of a Triangle

If you are using a school/lab computer, skip to step 4.

1. Google "**utexas LabVIEW**" and install LabVIEW in your system or use this link to download LabVIEW http://www.engr.utexas.edu/itg/products/435-labview.

   **Note: Check here to see if your OS is compatible with the most recent version of LabVIEW: http://www.ni.com/labview/os-support/. Although LabVIEW is compatible with both Mac and Windows, it's recommended to use Windows with LabVIEW for full functionality

2. On the same page you'll find "**Student License Instructions for Windows**", use that to get LabVIEW license on your system. If you don't do this your LabVIEW license will expire in 30 days.
3. Download "LabVIEW MathScript" from ni.com or http://www.engr.utexas.edu/itg/products/435-labview
4. Open this link: www.ni.com/lv101 .
5. Go to Learn LabVIEW.
6. Read through the "Detailed Explanation" for the following modules: LabVIEW Environment, Graphical Programming, MathScript and Text-Based Programming, and Programming Tools.

7. Now, it's time to develop your own VI that calculates the area of a triangle

Part A (Using Blocks):

1. Open LabView and click File →New VI
2. On the front panel (the gray window), right click and drag and drop the "Numeric Control" from the "Control Palette" into the front panel. Notice the arrow is on the right in the block diagram; this means it's an input. Label this control as "Base"
3. Repeat step 2 and label the second control as "Height"
4. Insert the "Numeric Indicator" from the "Control Palette" into the front panel. Notice the arrow is on the left in the block diagram, this means it's an output. Label this indicator "Area."
5. Your front panel should look like Figure 1 (You can add decorations, colors, and other things to your VI). Be sure to put your name and UTEID.
6. Go to the block diagram (white window), right click and insert the appropriate math functions and numeric constants you need in the "Numeric" section of the "Function Palette". You can hover on a math function and press "ctrl+h" to get a help on explaining the inputs and outputs of the math function
7. Connect wires by clicking on a node on any of your structures in the block diagram and drag and click the wire to another node.
8. Your block diagram should look like Figure 2 (left).
9. After connecting your wires, go to the front panel and test your program by entering values into the base and height controls and press the "Run Arrow"

Part B (using MathScript):

1. Go back to the front panel and add another set of Controls and Indicators with different variable names for the Base, Height, and Area.
2. In the "Function Palette", find MathScript (drag and drop) in the "Structures" section. Note: If you do not see a MathScript, you need to re-install MathScript
3. Right click on the blue outline of the MathScript box, go to "Add Input" and type the variable name for one of your inputs in the box
4. Repeat step 3 for your other input
5. Connect the two controls to the inputs you just created on the MathScript box
6. Inside the MathScript box, type the formula in this form: "Output = equation" with the input variables on the blue outline of the MathScript box
7. Right click on the blue outline of the MathScript box and add an output variable
8. Connect the output of the MathScript box to your indicator
9. Your block diagram should look like Figure 2 (right).
10. After connecting your wires, go to the front panel and test your program by entering values into the base and height boxes and press the "Run Arrow"

8. Save the VI. Ex: your_name_lab1.vi.
9. Call your TA over to verify the functionality of your VI.

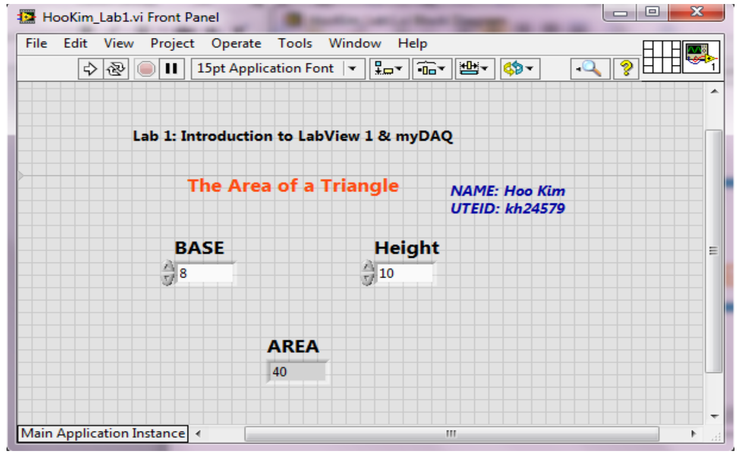


Figure 1: Area of triangle, Front panel

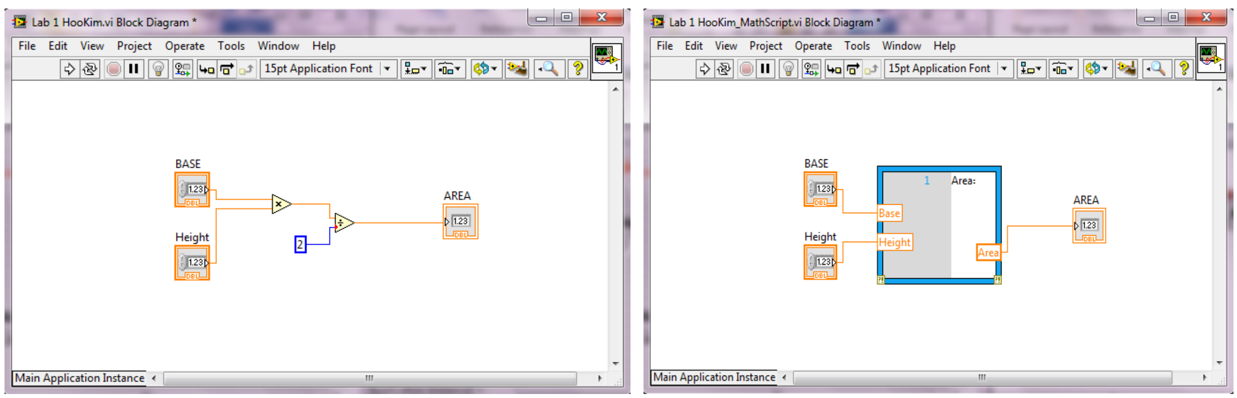


Figure 2: Area of triangle, Block diagram
(Left: using arithmetic functions, Right: using MathScript)

TA Signature:____

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Lab 2: Introduction to LabView 2

Lab Goals:

- Learn about LabView Programming Tools, Debugging and Handling Errors, Data Types and Structures, and Execution Structures.
- Design a Temperature Convertor using the **Case Structure**, **Enum,** and **While Loop** data structures.

Due at the end of Lab:

- A copy of the simple VI you designed

Assignment: Build a VI to Convert Temperature

1. Go to http://www.ni.com/lv101 and read the sections on Debugging and Handling Errors, Data Types and Structures, and Loops and Execution Structures.
2. Design a VI that utilizes Enum, Case Structure, MathScript, and While Loop to do the following conversions:

    - Celsius to Fahrenheit
    - Fahrenheit to Celsius
    - Celsius to Kelvin
    - Fahrenheit to Kelvin

** Note: Look up the appropriate temperature conversion equations (via Google, Bing, etc.) **

3. Open a new VI in LabVIEW
4. In the block diagram, insert the Case Structure (drag and drop) from the “Structures” section of the “Function Palette”
5. In the front panel, drag and drop an “Enum” (Enumerated list) from “Ring & Enum” section of the “Control Palette”
6. Right click on the “Enum” that you just placed, go to “Properties”, and click “ Edit Items”
7. Type in one of the conversion options (e.g. Celsius to Fahrenheit)
8. Click Insert and type the rest of the conversion options
9. Make sure to delete all blank cases you have in your list or else there will be an error
10. Insert a “Control” in the front panel and label it as “Input”
11. Insert an “Indicator” and label it as “Output”
12. Position your items in the block diagram like in Figure 2 (next page).
13. Drag and drop a “While Loop” from the “Structures” section around everything that you have placed so far. The while loop is used so that you don’t have to continually press the “Run Arrow” every time you change your Enum state.
14. Right click on the “Stop Sign” in the bottom right bottom corner of the “While loop” and click “Create Control”. A stop button will appear on the front panel. It is used to stop the loop.
15. Connect your Enum box to the green question mark on the case structure box. This is the selector terminal that determines which case executes
16. Right click on the gray region of the case structure box and click “Add Case for Every Value”
17. Notice that there are three arrows near the top center of your case structure box. The right and left arrows allow you to move from case to case. The down arrow allows you to see all of your cases.
18. Insert MathScript into one of your case structure boxes
19. Type the correct conversion formula with the appropriate Input and output variables
20. Wire your Input box to your Input MathScript node and your output MathScript node to your Output box like in Figure 2.
21. Repeat steps 18-20 for each of your four cases, making sure that you change the formulas each time to make sure the temperature conversions are correct.
22. Test your program to make sure it works by going to the front panel, entering a number in your Input and pressing the Run arrow.
23. Optional: To add any text to the front panel, make sure your program isn’t running and double click anywhere on the grid. A cursor will appear and you can type. To change the font size and color use the font tab underneath the Operate and Tools tab at the top.

24. Save the VI. Ex: *your_name_lab2.vi*
25. Call your TA over to verify the functionality of your VI

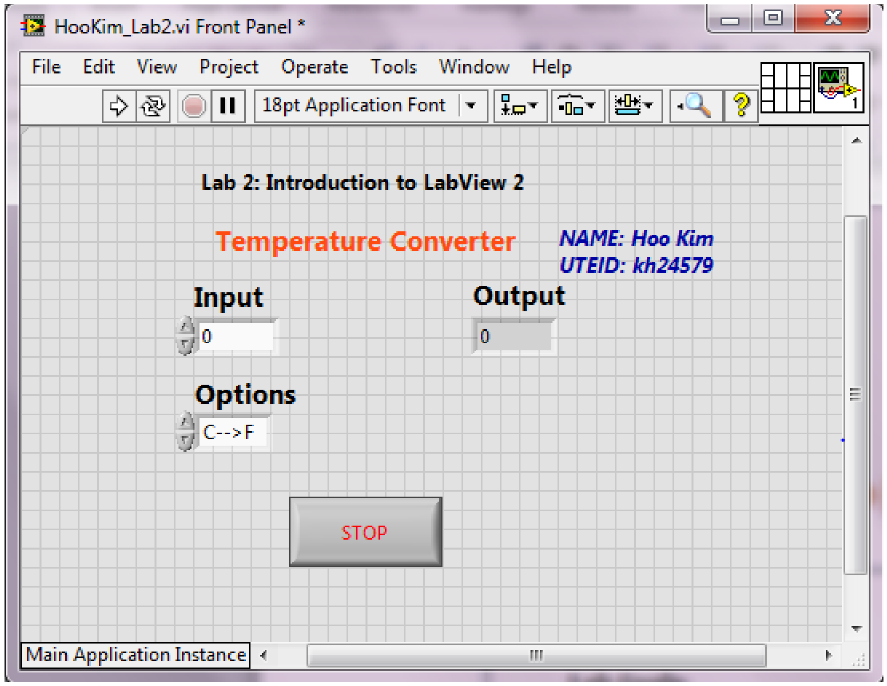


Figure 1: Temperature Converter, Front panel

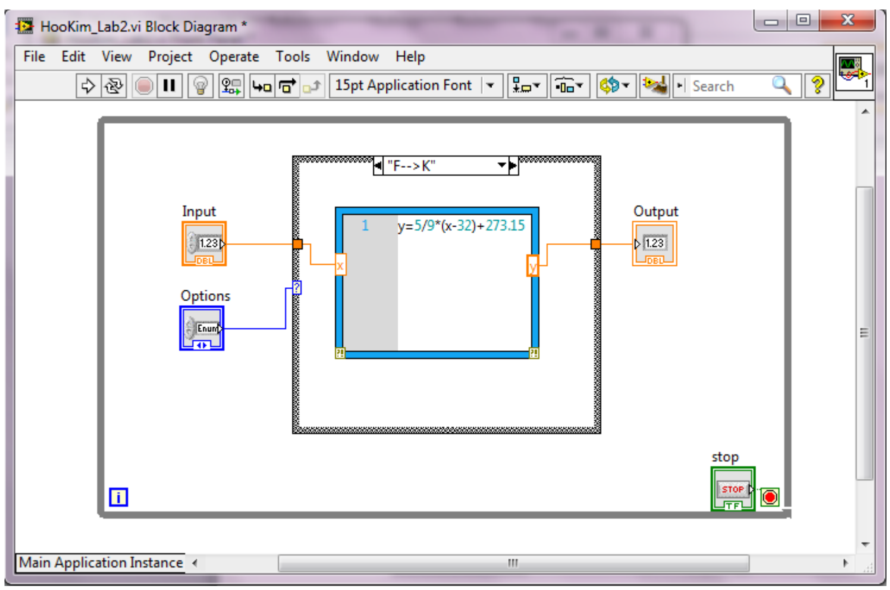


Figure 2: Temperature Converter, Block diagram

TA Signature:____

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Lab 3: Digital Multimeter and Voltage Generator

Lab Goals:

- Learn how to use your myDAQ as a Digital Multimeter (DMM)
- Learn how to output a signal to a specified output port on the myDAQ and verify its functionality.
- Design a Digital Multimeter and Voltage Generator to be used on later labs.

Required Lab Material:

- NI myDAQ kit: NI myDAQ, red and black banana DMM probes, USB-to-myDAQ power cable, 20-position black screw terminal connector, and a screwdriver

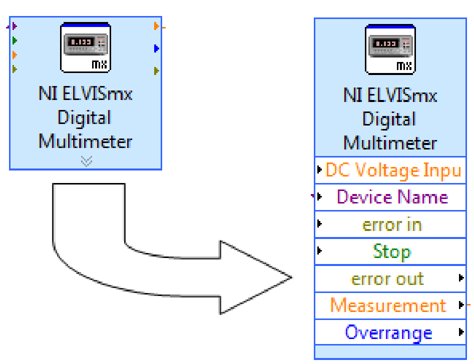
Due at the end of Lab:

- A copy of the simple VI you designed

Assignment: Build a Digital Multimeter

Create a voltmeter

1. If you have not done so already, download NI ELVISmx from ni.com or http://www.engr.utexas.edu/itg/products/435-labview
2. On the block diagram, use “Quick Drop” (Ctrl-Space) to find Multimeter → Select NI ELVISmx Digital Multimeter. It’s called an Express VI because it has its own configuration screen which comes up automatically.
3. Configure for DC Voltage measurement: right click on the NI ELVIS Digital Multimeter → properties. Select “Auto scale” under the mode of “Measurement Settings”.
4. Expand the Express VI down to see all the input/output terminals



5. Right click on "Device Name" input and Create→Control (this places control on front panel)
6. On the front panel, use "Quick Drop" to find Waveform Chart and Numeric Indicator
7. On the block diagram, wire together as shown.
8. On the block diagram, use Quick Drop to find While Loop. Drag loop around your code
9. Right click on Conditional terminal and Create→Control (this places stop button on front panel)

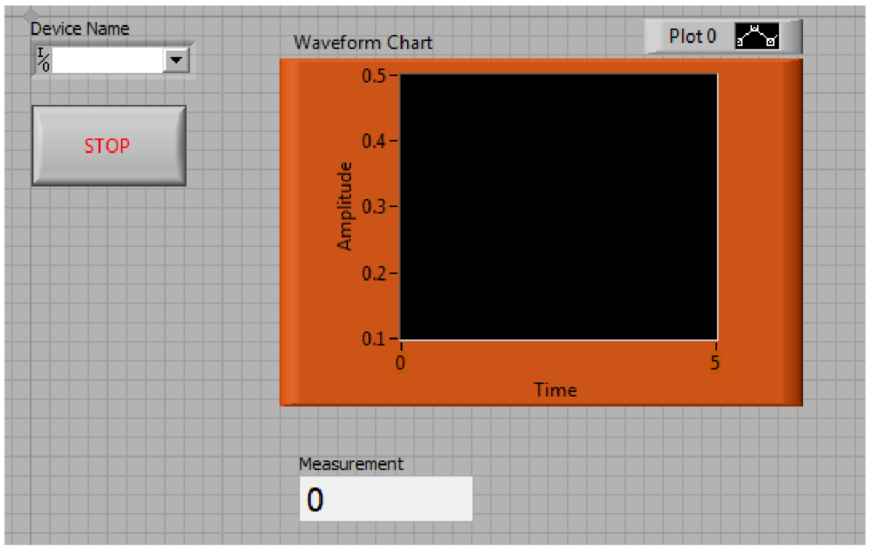


Figure 3.1: Multimeter, Front Panel

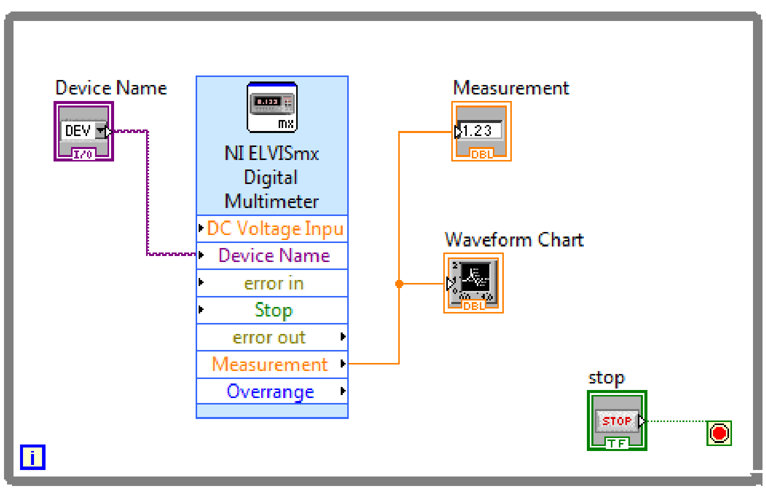


Figure 3.2: Multimeter, Block Diagram

10. Test by measuring a battery, or 5V / DGND pins on myDAQ using DMM probes.
11. Save your VI as your_name_myDMM.vi

<u>Add Current and Resistance measurements to your DMM</u>

12. On block diagram, drag a Case Structure around the DAQ Assistant.

13. On front panel, use Quick Drop to create an Enum (an enumerated list). Label as “Measurement Type” (Figure 3.3, 3.4)
14. On front panel, right click on Enum and select Edit Items (Figure 3.5)
15. Add Voltage, Current ,and Resistance as the three items on the list (add them the same way you added your four temperature conversions in Lab 2)
16. On block diagram, wire the Enum to the Case Selector (denoted by a “?” on border of loop). Also make sure that no blocks are on the outside of the For Loop.
17. Right click on border of Case and select “Add Case for Every Value” (this will create all three cases you need, neatly labeled to match the “Measurement Type” list).

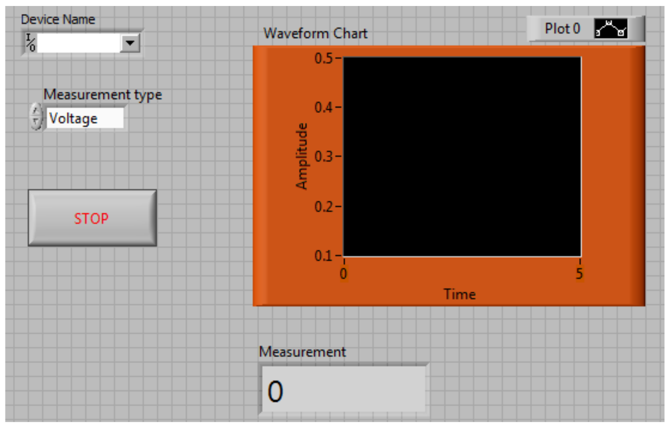


Figure 3.3: Front Panel

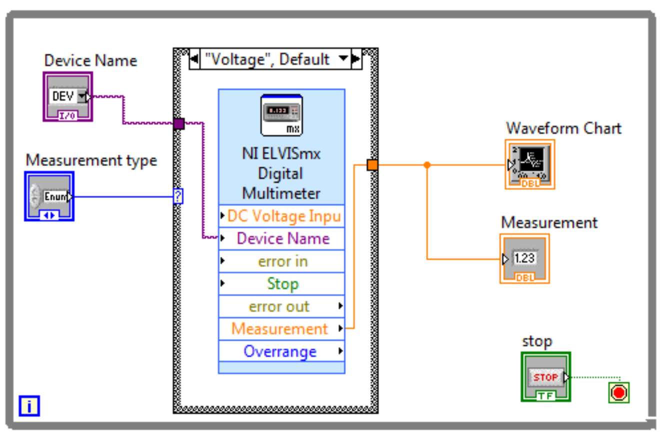


Figure 3.4: Block Diagram

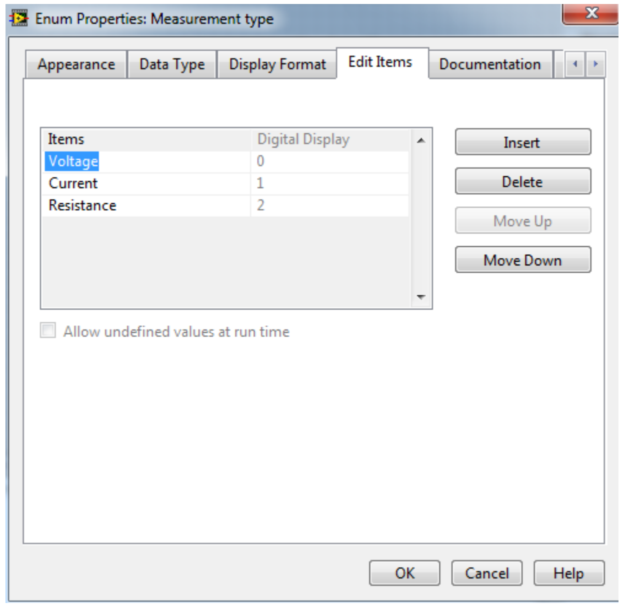


Figure 3.5: Enum Properties Menu

18. Add NI ELVISmx Digital Multimeter.vi to the "Current" case (Figure 3.6), and setup the measurement as shown (need to add a NI ELVIS block to each case, i.e. current, resistance, and voltage). The NI myDAQ has an internal shunt resistor to measure current through the analog-to-digital converter.

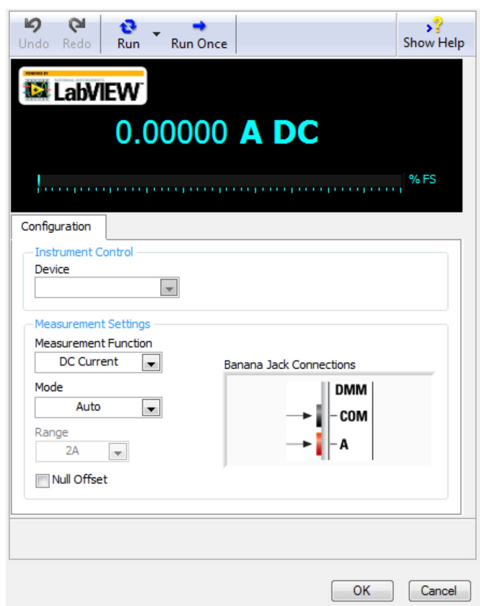


Figure 3.6: Multimeter VI Properties Menu

19. Add NI ELVISmx Digital Multimeter.vi to the “Resistance” case (Figure 3.7), and setup the measurement as shown. Note that in order to measure Resistance using an Analog-to-Digital converter, you supply an excitation current (Iex) through a Shunt Resistor. myDAQ has an Internal current source, allowing the measurement of Resistance.

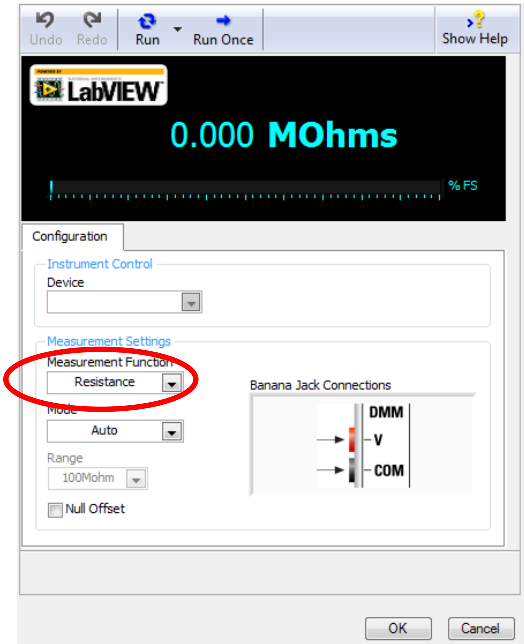


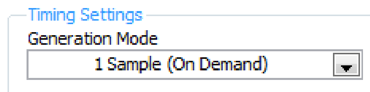
Figure 3.7: Multimeter VI Properties Menu

20. Be sure to wire the “Data” output of each DAQ Assistant to the white tunnel on the Case Structure. When all Cases are properly wired, the tunnel will turn solid and your VI should run.
21. Now you are ready to test your DMM for all three measurements: Voltage, Current, and Resistance.
22. Congratulations! You have just created your own Digital Multimeter using LabVIEW and myDAQ!
23. Remember to switch the DMM probes to the correct when you want to measure current.
24. Test Resistance by touchi8ng probes together. Resistance should go to zero. Or get a resistor and touch the probes to either side of the resistor.
25. Save your VI.

Assignment: Building a voltage generator

We can either create a new VI to generate voltage from myDAQ, or add this function to your DMM.vi. The steps below show how to add the Voltage Generator to your DMM.vi

1. Use Quick Drop (Ctrl-Space) to find the new functions: DAQ Assistant, and Select.
2. Set the DAQ Assistant to Generate Signal →Analog Voltage on AO0 (analog output channel 0). Also set Timing Settings to 1 Sample (On Demand). This tells myDAQ to generate a new output voltage each time the loop executes.



3. Select OK to accept all DAQ Assistant settings

4. On the Block Diagram, right click → Comparison → click on the "Select" block and place it in an appropriate location on the Back Panel. Then go to the Front Panel, right click → Numeric → Dial → place the dial on the Front Panel. Then go back to the Back Panel and wire the Dial block to the Select block as shown in Fig. 3.8.
5. Right click on the bottom node of the Select block (the "false" node) → Create → Constant → type 0.
6. Go to the Front Panel, right click → Numeric → Meter → place on Front Panel. Then on the Back (Block diagram) Panel, wire the Meter block to the Select block as shown in Fig. 3.8

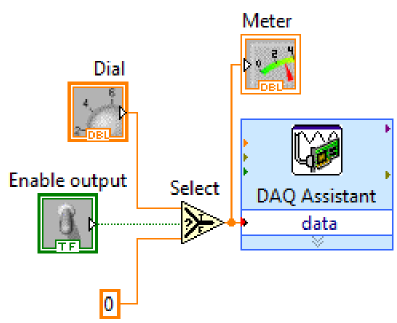


Figure 3.8: Block Diagram for Voltage Generator

** The Select function allows you to shut off the Voltage generator while the VI is still running.

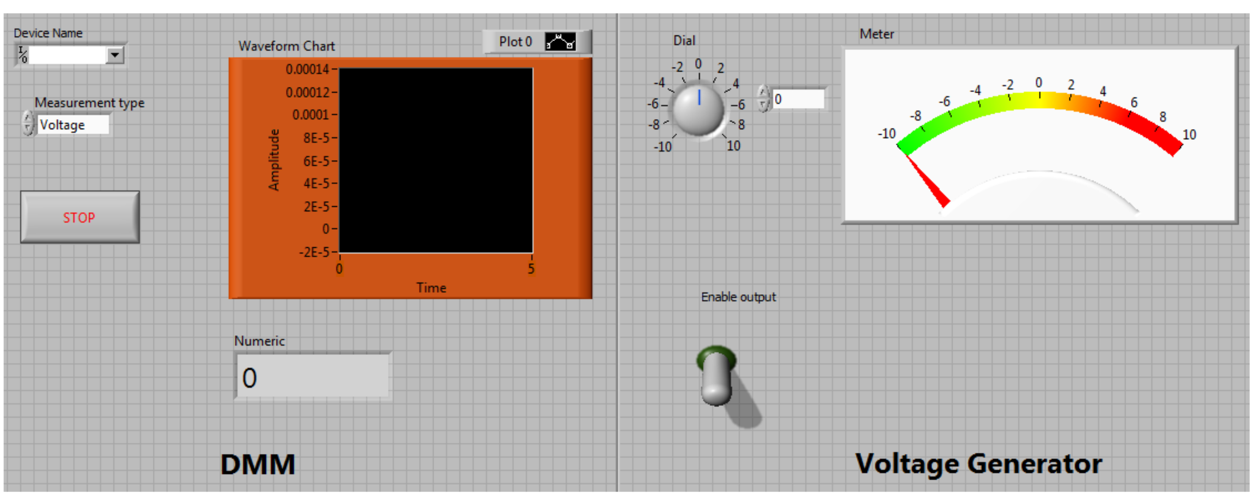


Figure 3.9: Final Front Panel

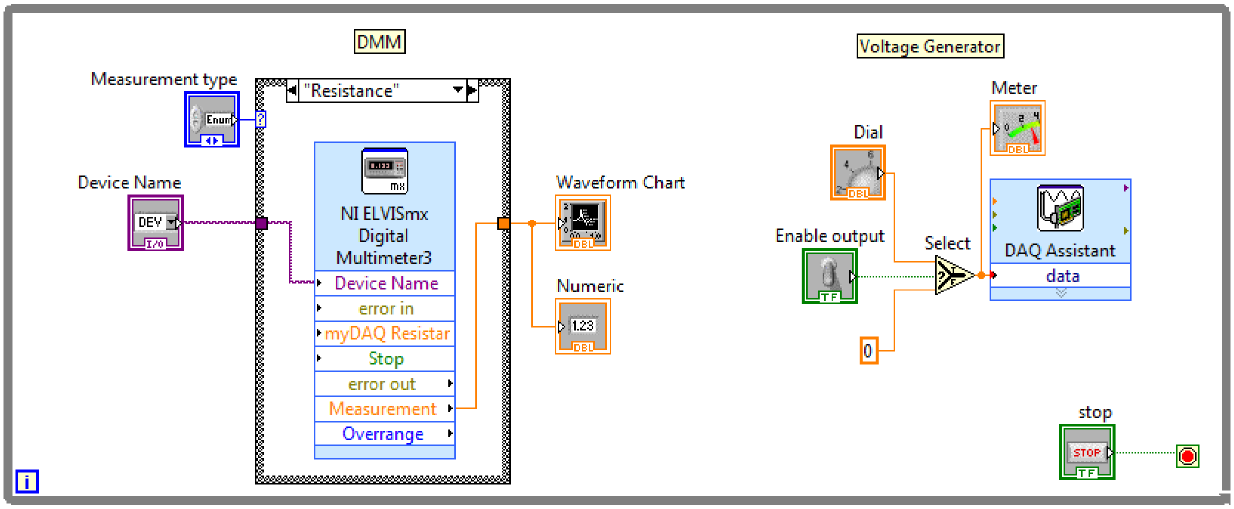


Figure 3.10: Final Block Diagram

7. Test your VI by generating a voltage on AO0 (Analog Output channel 0), and testing it with the DMM. When testing, make sure that your Toggle Switch (Boolean switch for the Select block) is up (in the "true" position)
8. Call your TA over to check you off for completion of the lab.

TA Signature: ____

Acknowledgement: A Special thanks to Eric Dean (Academic Field Engineer of NI) for the example DMM and voltage generator.

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Lab 4: Multisim Simulation of Circuits

Due beginning of lab:

- **Completed pre-lab questions from the end of this section.** One copy is to be turned in and keep a copy in this lab manual
- **Blank copy of the Lab Report.** One copy of the lab report will be turned in. You can either place another copy of the lab report into your notebook, or copy down the data collected

Lab Goals:

- Learn how to build resistive circuits in Multisim, a circuit simulation program.
- Learn how to use Multisim to obtain currents and voltages from elements in circuits
- Observe how the values obtained with simulation differ from those measured from real components.
- Gain an intuitive understanding of how the tolerance of a resistor affects the characteristics of a DC circuit.
- Understand how to obtain and graph the IV plot for a circuit element.

Due at the end of Lab:

- A copy of the simple VI you designed.
- Turn in your completed lab report to your TA.

Background Knowledge: Brief introduction to MultiSim

Multisim, a product of National Instruments, (for more information, visit the MultiSim WWW page at http://www.ni.com/academic/multisim.htm) is a drag-and-drop schematic capture and simulation program that allows you to quickly create complete circuits and analyze them. With Multisim, circuits containing independent and dependent sources, resistors, and other circuit elements can be instantly simulated at the click of a single button.

The simulation engine of Multisim is BSPICE/XSPICE compliant which is the defacto standard in circuit simulation. Other important features of Multisim include:

1. The ability to interact with circuits using Virtual Instruments that look and function like their real-world counterparts. Some of the instruments included are:

- Ammeter — For fast, simple read-outs of current.
- Function Generator — Produces square, triangular, or sinusoidal voltages.
- Multimeter — Measure AC and DC current and voltage, resistance and decibel loss (auto-ranging).
- Voltmeter — For fast, simple indications of voltage.

2. Multiple ways to analyze circuits so that you can explore and understand them in-depth. Some of the analysis methods include:

   - DC Sensitivity — Displays sensitivity to a particular parameter and predicts how variances in manufacturing can affect circuit performance.
   - DC Sweep — Computes the DC operating point of nodes in the circuit for various values of voltage or current sources.
   - Worst Case — Determines the most extreme values to be expected in your circuit, given the specified tolerance for each component.

3. Multisim contains a parts database that includes 16,000 different components including all the standard components plus several categories of unique "parts" that enhance each design.

It is not the goal of this lab to introduce you to all aspects of Multisim. Instead we will focus on some basic analysis techniques. You are free to continue to explore the features of Multisim on your own since it is available on all Learning Resource Center computers. We will also use MultiSim in the pre-labs for the remaining circuits labs.

Background Knowledge: Brief discussion of diodes and Zener Diodes

During this lab, you will learn how a semiconductor device called a zener diode can be used as a reasonable alternative to a voltage source. As a brief introduction to these devices, let's start by discussing a silicon diode. A diode is a semiconductor device that conducts current in only one direction. The symbol for a diode is shown at the top of Figure 1. If we place a voltage of slightly more than 0.7V across the diode with the reference shown in the figure, significant current will flow. This is called the forward bias region and current flows in the direction shown. The two terminals of the diode are called the anode and the cathode. If we apply a negative voltage across the diode, no current flows and we enter the reverse bias region. The diode characteristic of conducting current in only one direction makes it useful in many devices including converting AC voltages to DC.

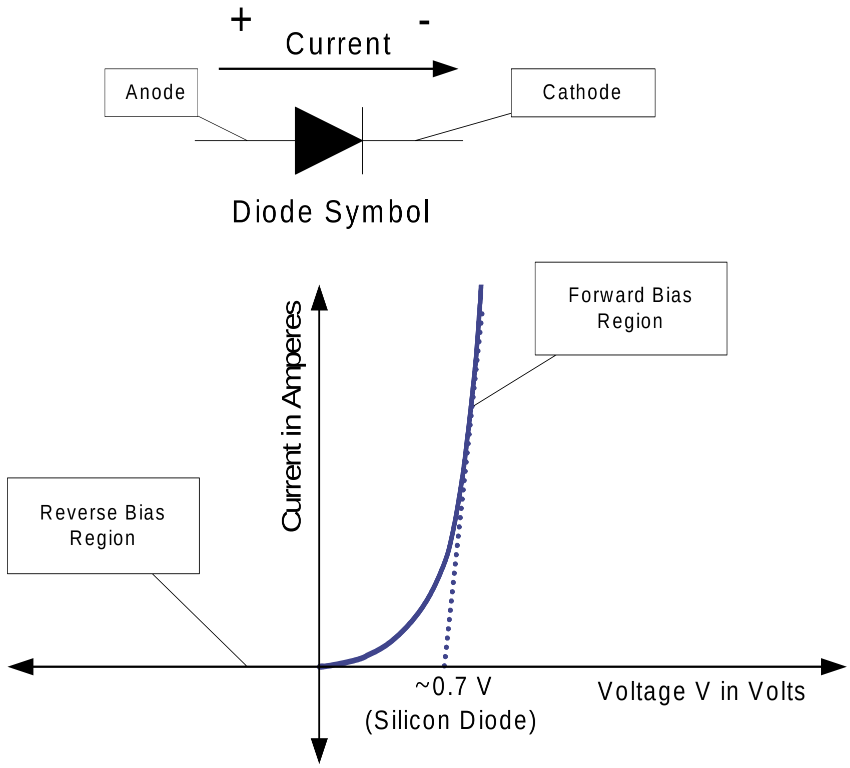


Figure 1: Diode Symbol and Characteristic

Let's now examine what happens if we continue to increase the negative voltage across the diode. As shown in

Figure 2, the diode eventually enters the avalanche breakdown or "Zener Region". Note that this drawing is not to scale. This region is characterized by a near constant voltage regardless of the current flow. While all diodes exhibit this characteristic, some diodes are precisely designed to exploit this behavior. These devices are called Zener diodes and are designed to deliver a wide array of precise voltage references, much like a voltage source.

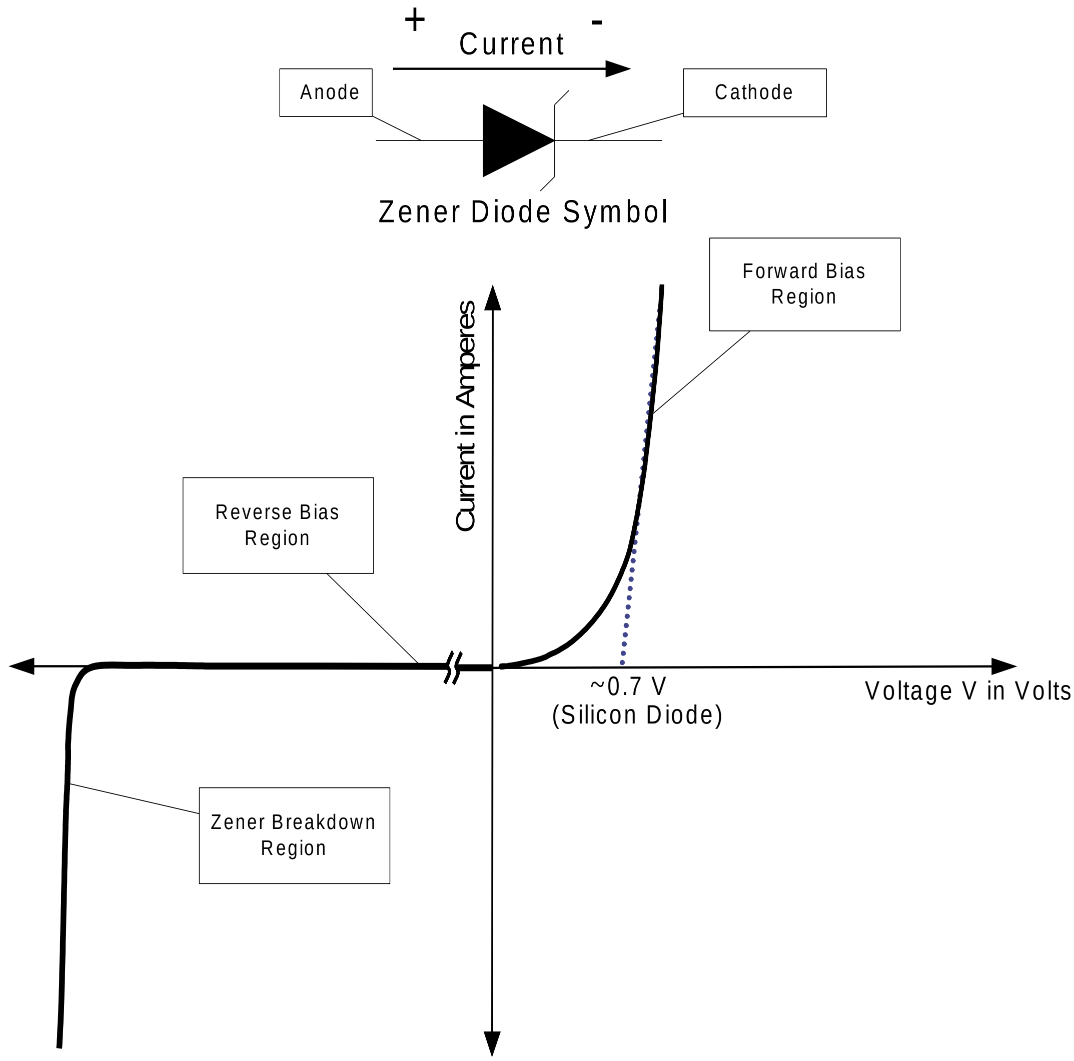


Figure 2: Zener Diode Symbol and Characteristic

Assignment 1: Getting oriented to MultiSim

1. Please read the FAQ's on page 33 (at the end of this lab's instructions) for helpful information.
2. Start Multisim by going to *Programs > Multisim*.
3. Press Control-W.

This will open up the Parts menu. We will need the following components for this lab: resistors, voltage sources, and a means of measuring current (multimeter or ammeter) and voltage (voltmeter). We can get to each of these as follows:

- The voltage source should be on the current screen. It is called DC_POWER.
- Under Groups, select Basic. Under the basic group we are going to use resistors. These resistors can be set to any value we want with any tolerance we want. They do not directly relate to resistors that you would use in a hands-on lab.
- Under Groups, select indicators to get to the voltmeter and ammeter.
- Instruments are under the menu *Simulate > Instruments*.

We can also get to these by selecting the appropriate images at the bottom of the menu bar or the rightmost side of the workspace. You can move the cursor over these images and the text explaining which group they belong to will be shown. From this point on, you are left to yourself to make the circuits and take the measurements. Some images are shown for the first circuit. Keep the following things in mind when building circuits and taking measurements:

- When an item is selected, it can be rotated with Control-R
- When the cursor is above the wire of a component, it will turn into a circle. Left-clicking on the mouse will start the wiring process. Move the cursor to where you want the wire to go and left-click the mouse again to end the wiring. To complete a specific path for the wire, you can left-click the wire at each point where you want a straight path to go.
- For nodes that contain more than two components, you need to create a junction by going to *Place>Junction*.
- All circuits need a ground wired to them before simulation can begin.
- *Function key F5* begins and ends a simulation

The schematic for the first circuit is shown below. In this simulation, we will be measuring current either using ammeters or a multimeter set to the current setting. You may choose either method you wish.

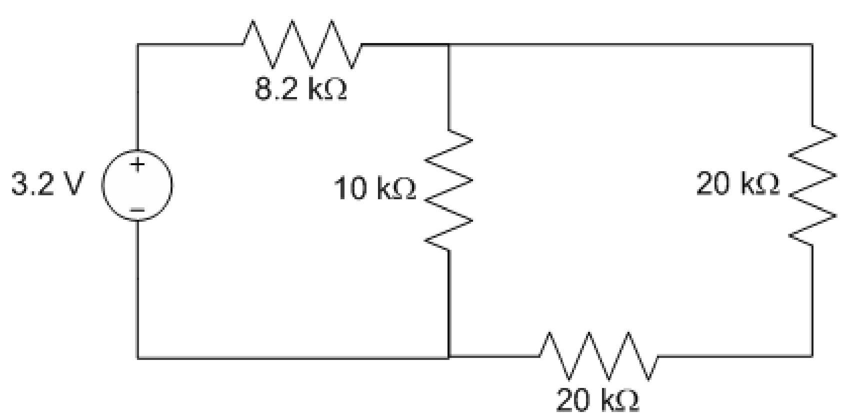


Figure 3: Example Circuit for Simulation

4. Place the appropriate components from Figure 3, including a ground, onto the workspace in Multisim. You can use Control-W to open the Parts Window or go to appropriate icons around the workspace. When you use virtual resistors, all of them will default to a value of 1 kΩ.
5. Double-click on a resistor.

This opens a window which allows you to change some properties of the resistor. We are interested in changing the value of each resistor to the ones shown in Figure 3. This is done under the Value tab of the open window.

6. Double-click on each resistor and change their values to the ones needed for the circuit shown.
7. Rotate (using Control-R) the 10 kΩ resistor and one of the 20 kΩ resistors so that they are vertical.
8. Double-click on the voltage source and change its value to 3.2 volts
9. Figure 4 shows what your screen might look like at this point.

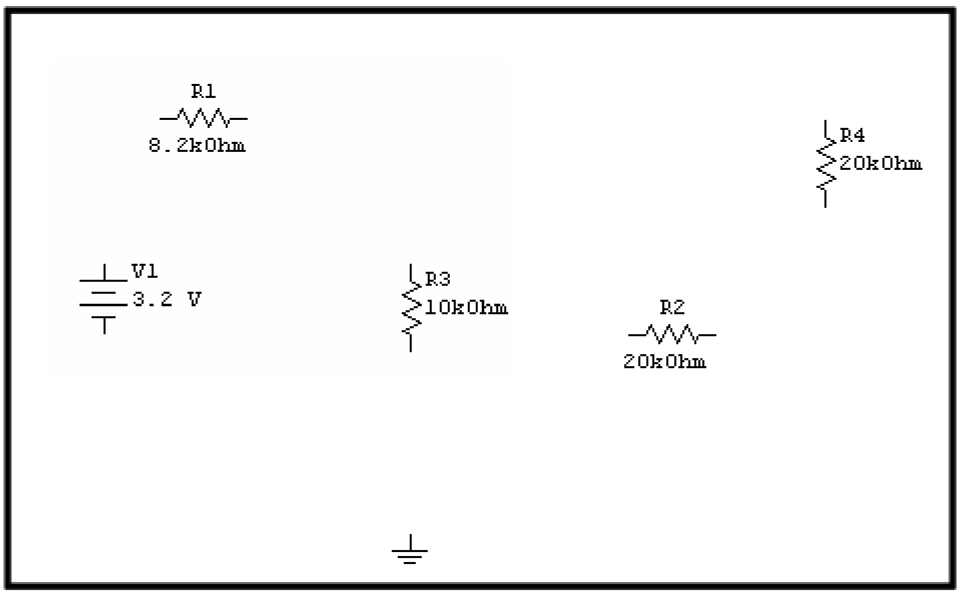


Figure 4: Typical display in MultiSim for Example Circuit

We now need to decide whether we want to use ammeters or multimeters to measure the current. Unlike the measurements you did previously, you can place multiple ammeters or multimeters so that all currents can be measured at once. **Remember that ammeters are connected in series with circuit components that you want to measure. This makes sense because the resistance of the ammeter itself is 1E-9 Ohms and will not affect your circuit if placed in series but will greatly affect your circuit if placed in parallel with a resistor or other component. The opposite is true for a voltmeter, which has a resistance of 10 Mega ohms. It should be placed in parallel with the circuit component(s) that you are measuring.**

10. Place the appropriate number of ammeters or multimeters on the workspace. Again, ammeters can be found by hitting Control-W and then selecting indicators under groups. Multimeters can be found under the Simulate menu (select the Instruments sub-menu).

11. Wire the circuit up so that the ammeters/multimeters are in series with the path for which you want to measure the current. When each component is connected to a wire, the wire will turn to red and a number may appear.

12. Figure 5 and Figure 6 show what the circuit may look like using either ammeters or multimeters.

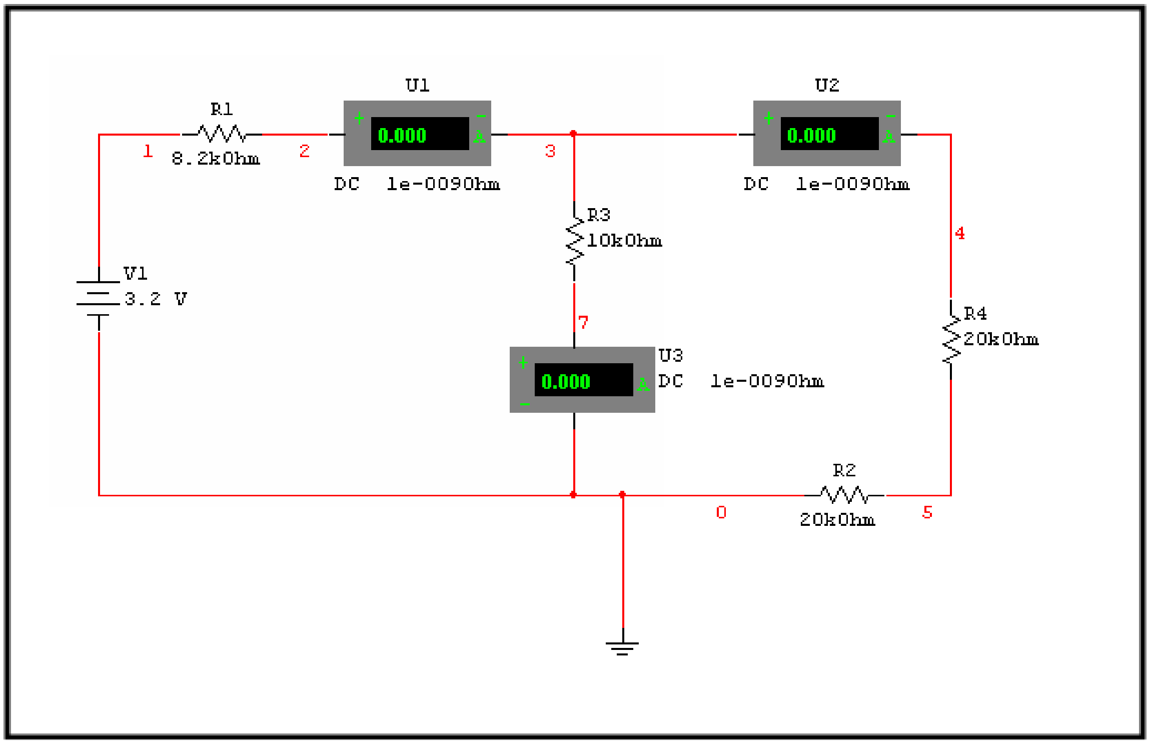


Figure 5: Simulation of Example Circuit using ammeters

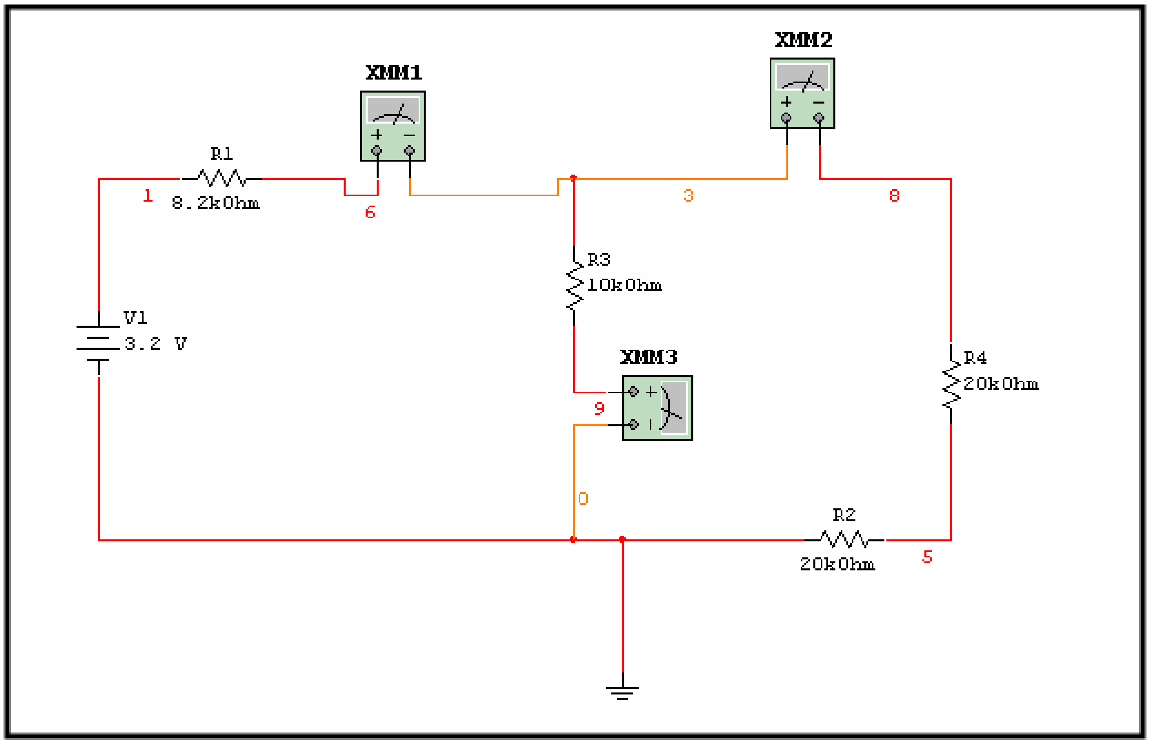


Figure 6: Simulation of Example Circuit using multimeters

If you chose to use multimeters, you need to do the following steps to set it up to measure current. If you chose to use ammeters, go to Step 15.

13. Double-click on each multimeter. This will open up a new window.
14. Select "A" to set this to measure current. Leave these windows open so that you can observe the current value. Once we are at this point, we are ready to simulate the circuit to obtain the appropriate current measurements.
15. Select *Use Tolerances* from the Simulate Menu.
16. This opens up a window where we can control the variation in certain parameters. Since we have a battery and resistors in our circuit, we can change the tolerances of these elements
17. Set the tolerance for batteries to 0 % and the one for resistors to 0 %.

To get the results of the simulation, do the following steps.

18. *Press Function Key F5* to start the simulation
19. *Press Function Key F5* to stop the simulation after the values for the ammeters or multimeters change. Note the values that each ammeter/multimeter has is retained after the simulation is completed.
20. Write down the current values obtained for each resistor in the appropriate column of **Table 1** and in your lab note section.
21. Change one of the 20 kΩ resistors to a 40 kΩ resistor. You can do this by double-clicking on the resistor and change the value under the value tab.
22. Run the simulation again (*Function Key F5*) and write down the simulation values in the appropriate columns of **Table 1** and in your lab note section.
23. Show your TA the results of your simulation and get his or her signature before continuing.

Assignment 2: Simulating the second circuit

We will now create a different circuit which is shown below.

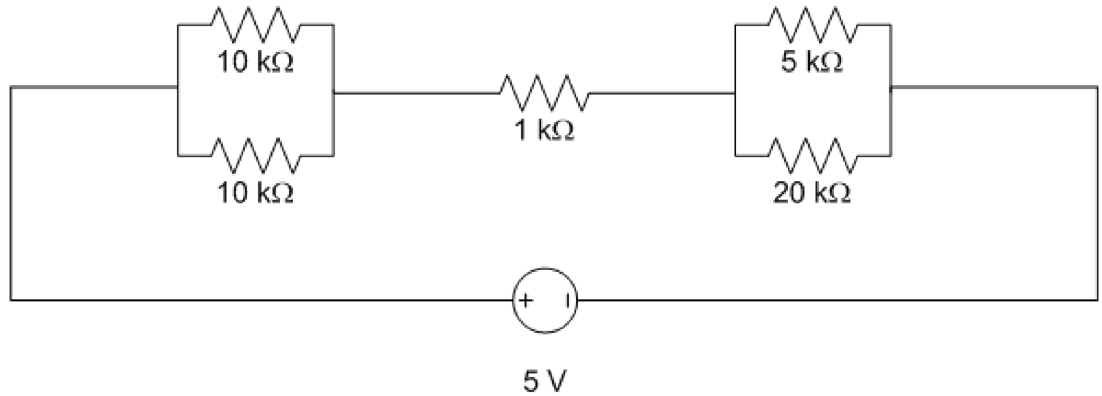


Figure 7: Second Circuit to Simulate

24. Build and wire the circuit shown in Figure 7 using virtual resistors.
25. Use either ammeters or voltmeters to measure the following quantities in the circuit: current flowing through the top 10 kΩ resistor, the voltage across the 1 kΩ resistor, and the current flowing through the 5 kΩ resistor. For clarity, assume that the positive references for voltages are on the left and positive currents flow left to right through the resistors
26. Change the resistor tolerance to 0 % by double clicking on the component and finding the tolerance option under the value tab. When using tolerances, be sure to go to Simulate → click on "Use Tolerances" and make sure that there is a check mark by "Use Tolerances".
27. Run the simulation five times and write down values for each quantity in the appropriate place of **Tables 2a, 2b, or 2c**.
28. Repeat Step 25-26 with resistor tolerances of 1 % and 5 %. Set ALL resistors to have a tolerance of 1% and then ALL to have a tolerance of 5%, but leave the power source (battery) at 0% tolerance.
29. Show your TA the results of your simulation and get his or her signature before continuing.

Assignment 3: Simulating the Third circuit

The last circuit you will simulate is shown below in Figure 8. As discussed in the introduction, we will briefly introduce a Zener diode which can act similar to a voltage source under the right conditions. The Zener diode is a non-linear device which means its current-voltage characteristic curve cannot be modeled as a straight line as demonstrated in. The Zener diode which we will use in this simulation produces approximately 4.3 V under the right conditions.

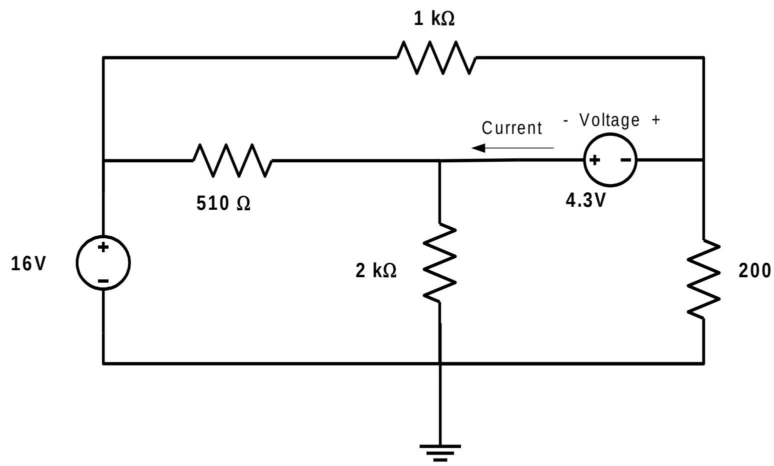


Figure 8: Third Simulation Circuit

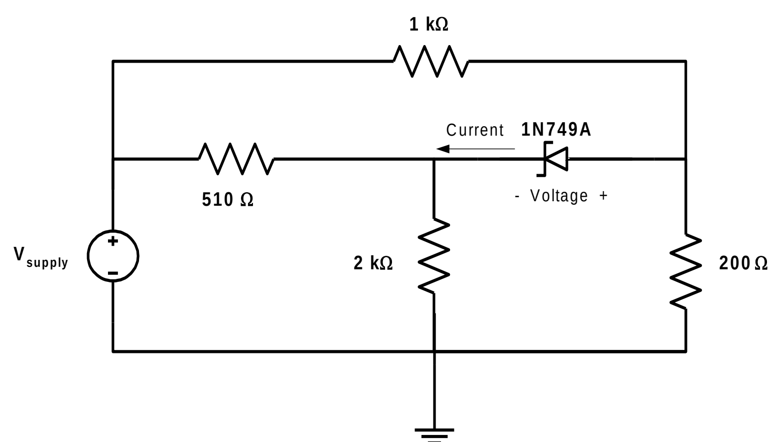


Figure 9: Third Circuit Using a Zener Diode

30. Build and wire the circuit shown in Figure 9. Include a voltmeter to measure the voltage across the 4.3 V source with the reference shown and an ammeter to measure the current flowing right to left through the 4.3 V source. (Note: for easy wiring, you can right-click on the meters in your circuit, and flip them horizontally.) Please pay special attention to the direction of the positive and negative terminals of the voltmeter and ammeter as you can see from the labeling (current arrow and + and – sign) in Fig. 8 and Fig. 9. You can see that the voltmeter terminals are in the opposite direction of the battery terminals. And for the ammeter, place the positive terminal so that the current arrow is going into the positive terminal (+ on the right side, - on the left).
31. Change the resistor tolerance to 0 %.
32. Run the simulation. Enter the voltage and current into the appropriate place of **Table 3** of the Lab Report.
33. Replace the 4.3 V source with the Zener diode (model number 1N749A) as shown in Figure 8. Repeat Step 31 with supply voltages (V_{supply}) of between ±20 V in steps of 4 V (i.e., -20V, -16 V, -12 V, etc.) and place your results in Table 4 in the lab report and in your lab note section.
34. Plot the I-V characteristics of the diode using the data from **Table 4** of the Lab Report. You should either do a **CLEAR** plot by hand or use a program like EXCEL and paste the results in the space provided.
35. Turn in your completed lab report to your TA.

FAQ's

Q: When I simulate my values, why are all the values the same all the time even when I have tolerances?
A: You must have **Use Tolerances** option checked under the Simulate menu.

Q: Why are some of my values reading 0?
A: Check your connections on your circuit. Chances are something's not connected. Move around some of your components to make sure the correct parts are connected.

Q: Why am I not getting my calculated values?
A: Chances are you probably wired a component or multimeter wrong. In addition, check you're measuring the appropriate value. If you're measuring current through something, make sure your meter is connected in series with it. If you're measuring voltage, be sure your meter is connected parallel with the voltage you're measuring.

Q: Why can't I find tolerance in my resistors?
A: Do not use **RESISTOR_RATED**. Go to **BASIC=>RESISTOR**, and pick out the appropriate value from there.

Q: Which voltage source should I use? Where is ground?
A: Use **DC_POWER** under **SOURCES** (it has two parallel lines). For ground, it's under **SOURCES** as well.

Q: Where do I find a voltage source?
A: Control-W → Group: Sources → Family: Power_Sources → DC_POWER

Q: Where can I find a resistor?
A: Control-W → Group: Basic → Family: Resistor → just click on any number (you can change the resistance value once you place it). Also after your place the resistor, double click on it and make sure that you can change its resistance value and its tolerance. If you can, then you are using the right type of resistor.

Q: Where can I find an ammeter / voltmeter and which one do I use?
A: Control-W → Group: Indicators → Family: ammeter / voltmeter → any voltmeter or ammeter is good. Some have their nodes vertical (V) or horizontal (H) and some are high-resolution (HR). Ammeters are used to measure current and voltmeters are used to measure voltage.

Q: Where can I find a ground node?
A: Control-W → Group: Sources → Family: Power_Sources → Ground

Q: Where can I find the correct Zener diode to add?
A: Control-W → Group: Diodes → Family: Zener → 1N749A

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Name: ____ EID: ________________

By placing by name and EID above, I am certifying that I determined the answer to the questions posed below and did not copy my answers from a fellow student.

*** Due at the beginning of your lab session ***

You will need 2 completed copies of this pre-lab. One is to be turned in to your TA at the beginning of the lab session. The other one is to be done in your lab manual.

1. Find a description on the **<u>operation of a diode</u>** on the internet (Places to check: How Stuff Works, Encyclopedia.com, etc.). You need to refer to **<u>at least 3</u>** different sources. Provide a summary of the information that you find. At the end of your description, provide the URLs for the sources that you read to write your description

Description of Zener Diode Operation

URLs Referenced

http://____

http://____

http://____

http://____

http://____

Multisim Simulation of Circuit
Lab Report

Submitted by (Print name)

Table 1: Simulated current values in the circuit in Figure 3 of the Lab. Assume that positive currents flow left or right or up to down as appropriate. Make sure you use an appropriate number of significant figures and include your units.

	Simulated Value (Step 19)	Simulated Value (Step 21)
Current through 8.2 kΩ resistor		
Current through 10 kΩ resistor		
Current through 20 kΩ resistor		

TA Signature (Step 22): ____

Table 2a: Simulated values for the current flowing through the top 10 kΩ resistor in Figure 7 of the Lab. Make sure you use an appropriate number of significant figures and include your units.

	0% tolerance	1% tolerance	5% tolerance
Run #1			
Run #2			
Run #3			
Run #4			
Run #5			

Table 2b: Simulated values for the voltage across the 1 kΩ resistor circuit in Figure 7 of the Lab. Make sure you use an appropriate number of significant figures and include your units.

	0% tolerance	1% tolerance	5% tolerance
Run #1			
Run #2			
Run #3			
Run #4			
Run #5			

Table 2c: Simulated values for the current flowing through the 5 kΩ resistor in Figure 7 of the Lab. Make sure you use an appropriate number of significant figures and include your units.

	0% tolerance	1% tolerance	5% tolerance
Run #1			
Run #2			
Run #3			
Run #4			
Run #5			

TA Signature (Step 28): ____

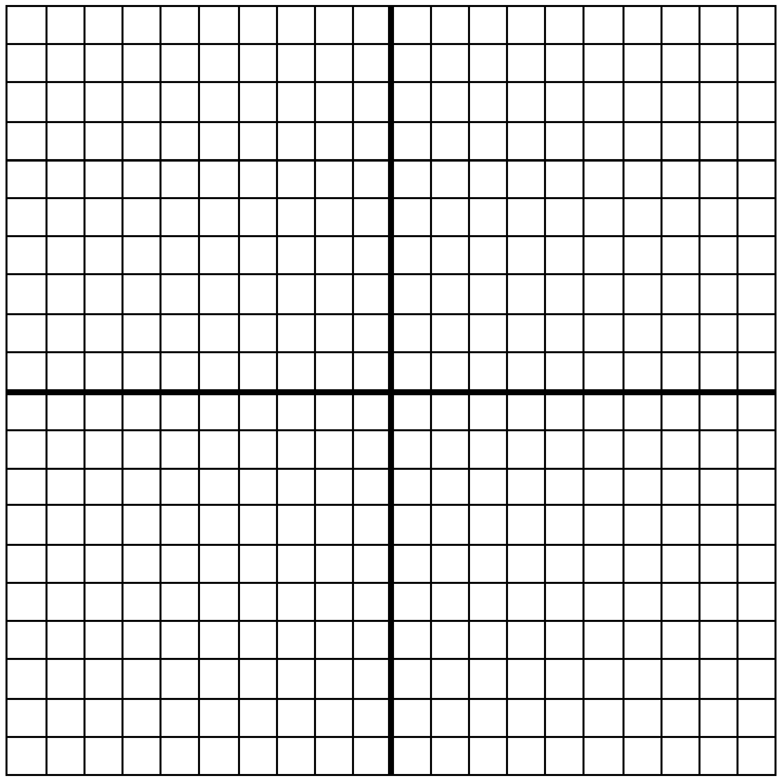
Table 3: Simulated values for Figure 8 of the lab

4.3 V Source Voltage	4.3 V Source Current

Table 4: Voltage and Current Values for the Diode in Figure 9 of the lab.

Supply Voltage Value (V)	Diode Voltage (V)	Diode Current (mA)

Plot the I-V characteristics of the diode using the data from Table 4. You should either do a **CLEAR** plot by hand or use a program like EXCEL and paste the results in the space provided below.



Can you use a Zener Diode to replace a voltage source in a circuit? If so, how well does it match the operation of an ideal voltage source?

Lab 5: Audio Equalizer

Lab Goals:

- Learn how to use Labview to program an audio equalizer
- Understand how filters are used in signal processing.

Required Lab Material:

- NI myDAQ kit with the audio cable
- A sound source (Please, bring your any sound source such as iPod, phone, etc.) and earphones

Due at the end of Lab:

- Demonstration of a working audio equalizer

Background Knowledge: Basic concepts of filter and equalizer

Low-pass Filter: A low-pass filter is a filter that passes low-frequency signals but attenuates (reduces the amplitude of) signals with frequencies higher than the cutoff frequency. The actual amount of attenuation for each frequency varies from filter to filter.

High-pass Filter: A High-pass filter is a filter that passes high frequencies well but attenuates (reduces the amplitude of) signal with frequencies lower than the filter's cutoff frequency. The actual amount of attenuation for each frequency is a design parameter of the filter.

Band-pass Filter: A band-pass filter is a filter that passes frequencies within a certain range and rejects (attenuates) frequencies outside that range.

Audio Frequency: An audio frequency (abbreviation: AF), or audible frequency is characterized as a periodic vibration whose frequency is audible to the average human. While the range of frequencies that any individual can hear is largely related to environmental factors, the generally accepted standard range of audible frequencies is 20 to 20,000 hertz. Frequencies below 20 Hz can usually be felt rather than heard, assuming the amplitude of the vibration is high enough. Frequencies above 20,000 Hz can sometimes be sensed by young people, but high frequencies are the first to be affected by hearing loss due to age and/or prolonged exposure to very loud noises

Assignment 1: Design the audio equalizer

Design an audio equalizer as shown below to adjust the output audio signal to the speaker.

Use the filter express vi under signal analysis from the functions palette. Also use the Spectral Measurement express vi to display your signal.

Figure 1 shows the template for the Front Panel of your Audio Equalizer. This VI will be provided by TA. Although the given template provides general concepts to implement an Audio Equalizer, you need to follow the instructions, modify, and finish building the corresponding block diagram in Figure 2.

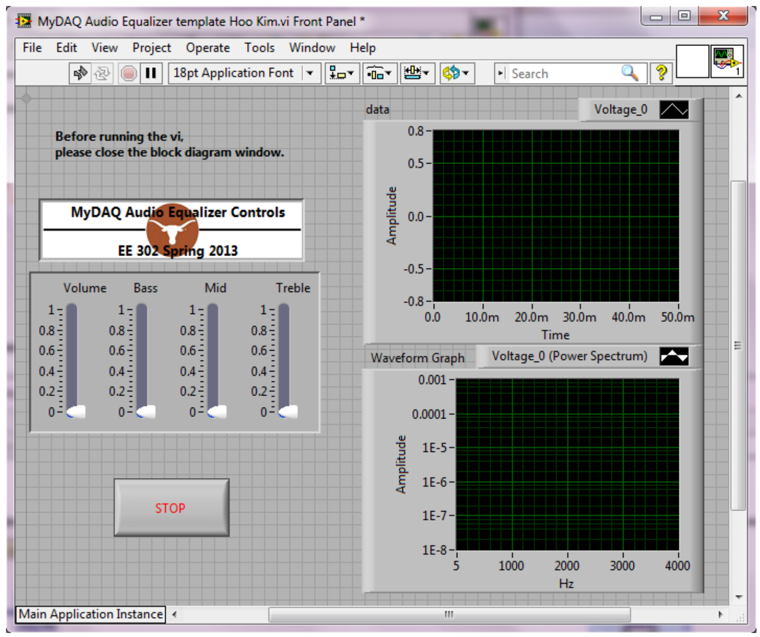


Figure 1: Audio Equalizer VI template, Front Panel

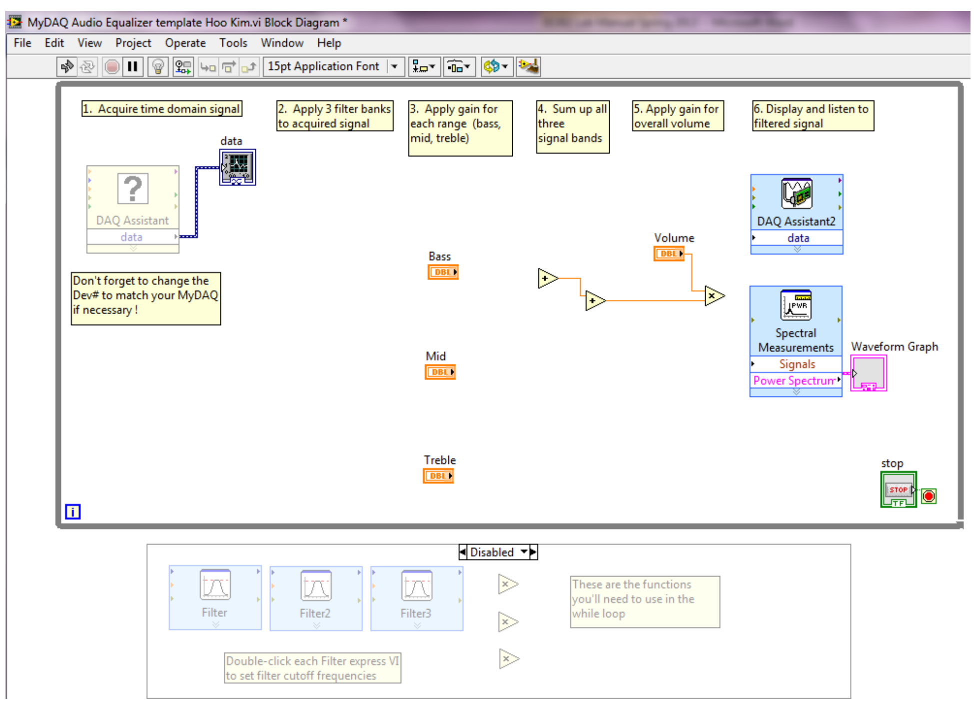


Figure 2: Audio Equalizer VI template, Block Diagram

Refer to the frequency range given below.

Bass, midrange, and treble frequencies

Bass

```
     Sub bass:   31 Hz  to   62 Hz (and below)
     Mid bass:   62 Hz  to  125 Hz
   Upper bass:  125 Hz  to  250 Hz
```

Midrange

```
Lower midrange:  250 Hz  to  500 Hz
  Mid midrange:  500 Hz  to   1K Hz
Upper midrange:   1K Hz  to   2K Hz
```

Treble

```
  Lower treble:   2K Hz  to   4K Hz
    Mid treble:   4K Hz  to   8K Hz
  Upper treble:   8K Hz  to  16K Hz (and above)
```

TA Signature: ____

Acknowledgement:
Thanks to Eric Dean (Academic Field Engineer of NI) for the example audio equalizer vi.

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Lab 6: Breadboard and Measurements

Due beginning of lab:

- **<u>Completed pre-lab questions from the end of this section.</u>** One copy is to be turned in and one copy should be in this lab manual.
- **<u>Blank copy of the Lab Report.</u>** One copy of the lab report will be turned in.
- You can either place another copy of the lab report into your notebook, or copy down the data collected.

Lab Goals:

- Learn how to build a circuit on a breadboard from a circuit schematic.
- Learn how to measure voltage and current accurately from a circuit on a protoboard.
- Observe how the values obtained from real components differ from those calculated from ideal components.
- Gain an intuitive understanding on how changes in resistance affect characteristics of a DC circuit.

Equipment needed:

- Two resistors, with 5 % precision, with each of the following values: 2 kΩ, 3 kΩ, 5.1 kΩ, 10 kΩ, and 20 kΩ.
- One (1) breadboard
- Your lab notebook
- MyDAQ with connection probes (Black and Red)
- A copy of the Resistor Organization Sheet (Search the internet)

Due at the end of lab:

- Turn in your completed lab report to your TA.

<u>Background Knowledge: Prototyping with a breadboard</u>

A breadboard (or prototyping board) is used to quickly construct circuits for testing and evaluation. You will be using different types of protoboards during your various labs at UT. This section provides some general comments and suggestions for these breadboards.

The bread board has many strips of metal (copper usually) which run underneath the board. The metal strips are connected as outlined in **orange** below. These strips connect the holes on the board. This makes it easy to connect components together and build circuits. To use the bread board, the legs of components are placed in the holes (the sockets). Each hole is connected to one of the metal strip s running underneath the board. A **node** is a point in a circuit where two components are connected. Connections between different components are formed by putting their legs in a common node. On the breadboard, a node is the row of holes that are connected by the strip of metal underneath. The long read & blue row of holes are usually used for power supply connections.

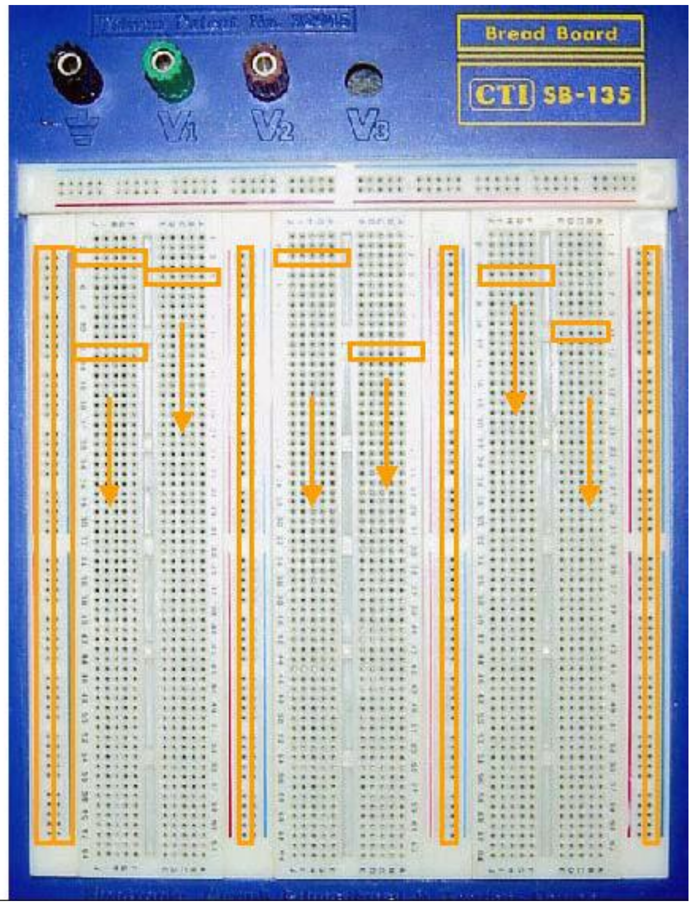


Figure 1: Typical Breadboard

Here are two example circuits connecting resistors in series and parallel.

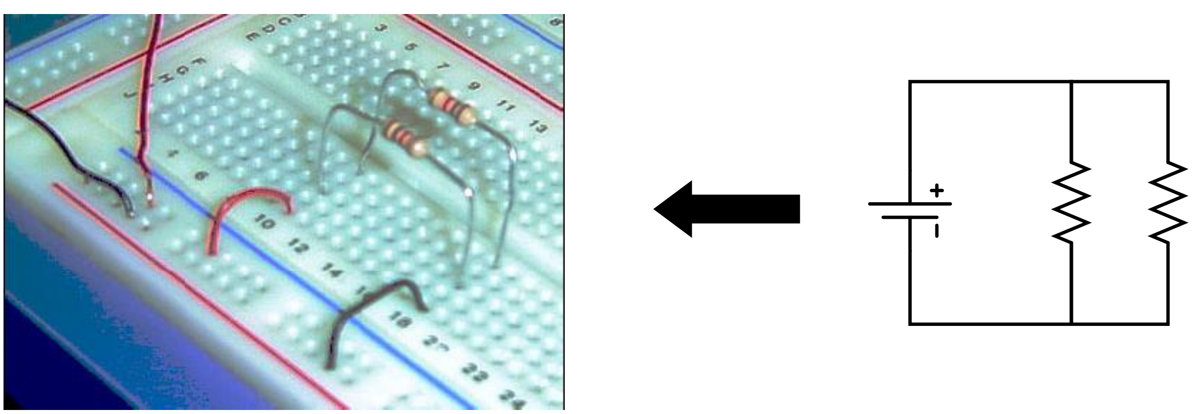


Figure2: Resistors in Parallel

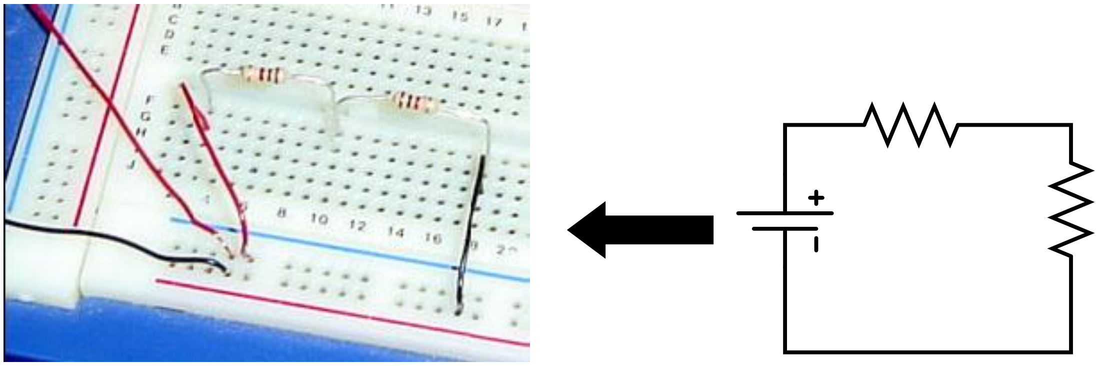


Figure 3: Resistors in Series

The Concept

When building a "permanent circuit" the components are together (as in an integrated circuit), soldered together (as on a printed circuit board), or held together by screws and clamps (as in house wiring). In lab, we want something that is easy to assemble and easy to change. We also want something that can be used with the same components that "real" circuits use. Most of these components have pieces of wire or metal tabs sticking out of them to form their terminals.

How it Works

The heart of the solderless breadboard is a small metal clip that looks like this:

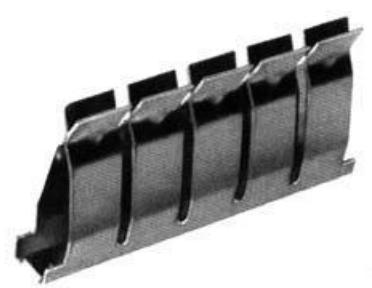


Figure 4: Metal Clip

The clip is made of nickel silver a material which is reasonably conductive, reasonably springy, and reasonably corrosion resistant. Because each of the pairs of fingers is independent, we can insert the end of a wire between any pair without reducing the tension in any of the other fingers. Hence each pair can hold a wire with maximum tension. To make a breadboard, an array of these clips is embedded in a plastic block which holds them in place and insulates them from each other, like this:

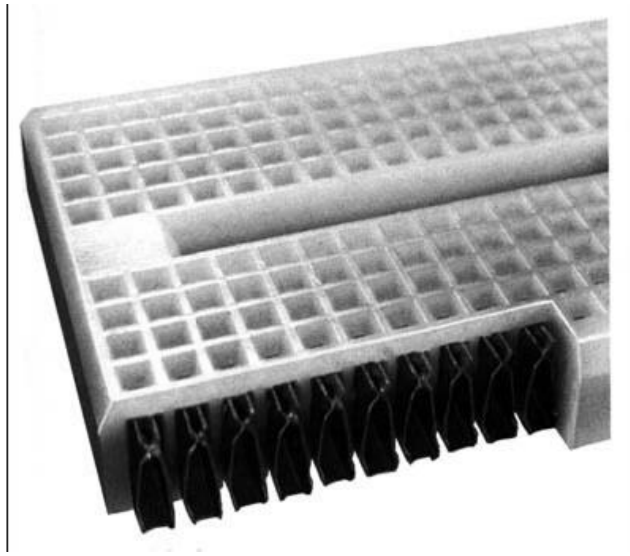


Figure 5: Breadboard Holes with clips

The hole in the block above each pair of fingers holds the wire accurately centered in the clip. Depending on the size and arrangement of the clips, we get either a socket strip or a bus strip. The socket strip is used for connecting components together. It has two rows of short (5 contact) clips arranged one above another (Figure 6).

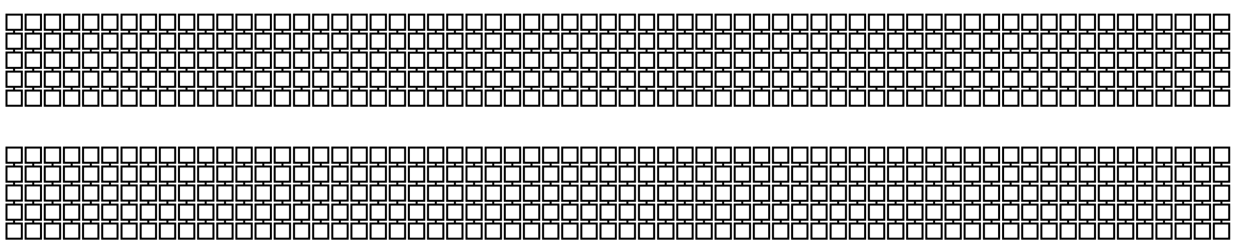


Figure 6: Bus Strip

The bus strip is used to distribute power and ground voltages through the circuit. It has four long (25 contact) clips arranged lengthwise (Figure 7).

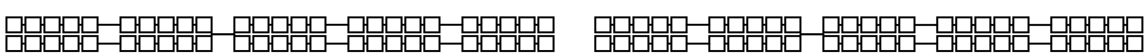


Figure 7: Bus Strip

Note that the manufacturer elected *not* to join the adjacent 25 contact strips into a single, full-length, 50 contact strips. If this is what you want, you will have to bridge the central gap yourself.

When we combine two socket strips, three bus strips, and three binding posts on a plastic base, we get the breadboard:

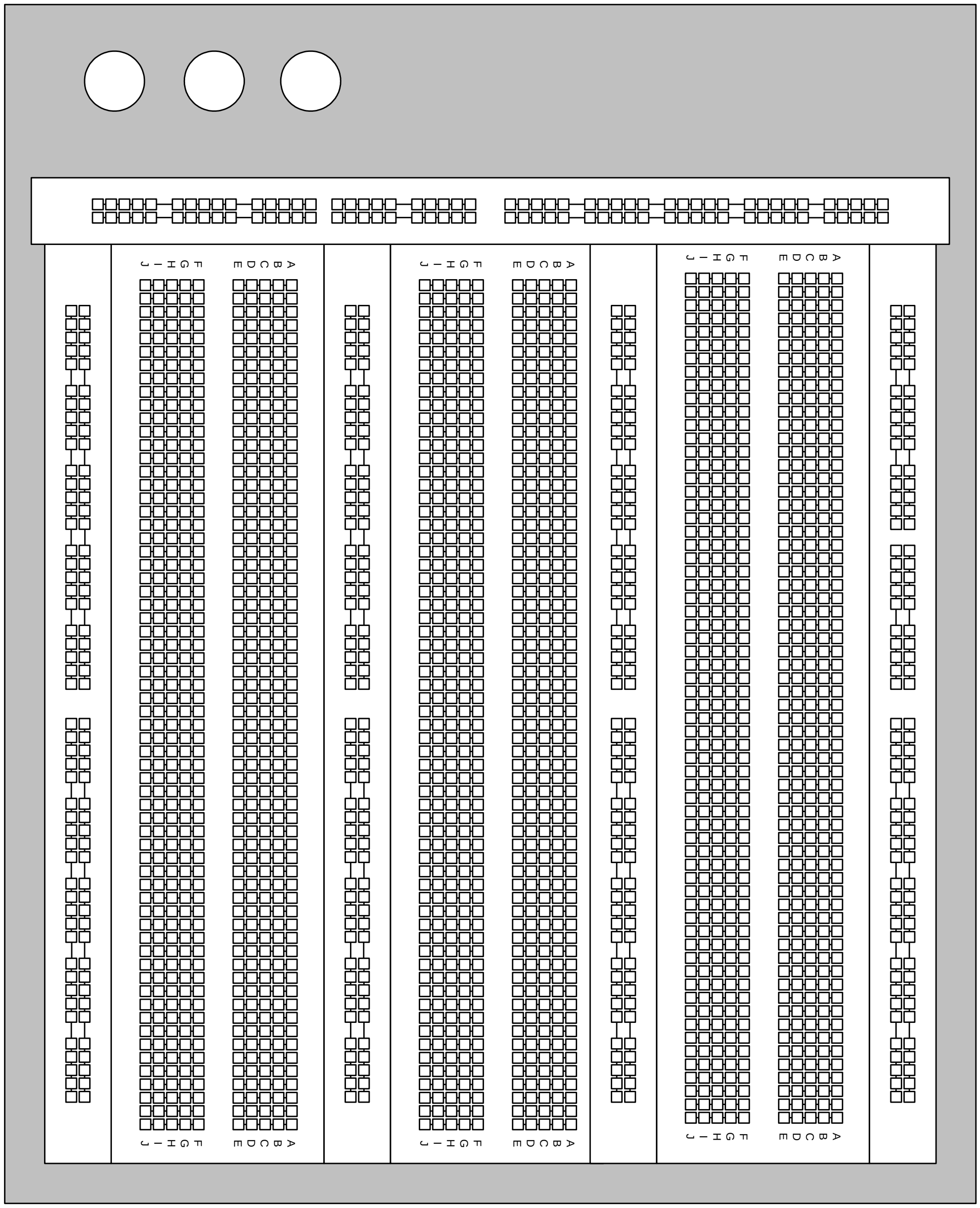


Figure 8: Breadboard

The breadboard lets us connect components together and by wiring the bus strips to the binding posts and the binding posts to the power supply, to connect the power supply to the circuit. Now what we need is a way to bring connections from the rest of the instruments into the breadboard.

Assignment 1: Build the circuit

One of the goals of this lab is to give you experience building a circuit on a proto-board. You studied some aspects of this process as part of the pre-lab assignment. If you still have questions about this, review the information in your lab manual or discuss the process with your partners. The schematic for the first circuit is shown below in Figure 4.

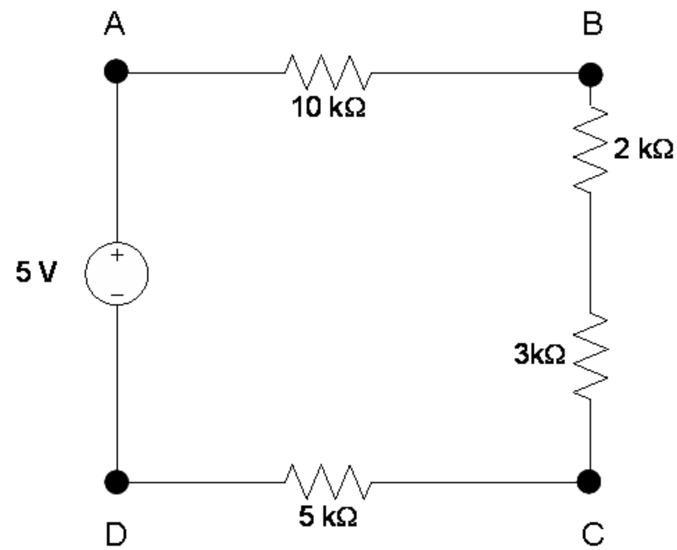


Figure 9: Breadboard and Measurements First Circuit

4. Build this circuit on the breadboard. Get TA signature.
5. A port for the power supply will be the Analog Output port of your NI myDAQ. The connections will be *AGND* and AO0.

6. Connect a wire from the *AGND* to the point D in Figure 9. You may need to use multiple wires to make this connection. For clarity, you should use wires of the same color.
7. Connect another wire (of a different color) from the *AO0* to point A of Figure 9. You may need to use multiple wires to make this connection. For clarity, you should use wires of the same color.
8. Get your TA to verify that your circuit is correct. If it is, your TA will sign the appropriate place in the lab report.

9. We will now measure voltages between points A, B, C, and D. These voltages can be used to calculate the current through the resistors. You calculated the expected values for these voltages as part of your pre-lab assignment. We will compare those results to those you will measure.

10. Draw the circuit shown in Figure 9 above in the box provided in the Lab Report. Include where you will place voltmeters (denoted by a “V” inside a circle; include the positive and negative terminals) to measure V_{AB}, V_{BC}, and V_{CD}. You will have more than one voltmeter in your circuit.
11. Enter the values you calculated from the pre-lab into the appropriate location in **Table 1** of the Lab Report.
12. Run your vi from lab 3 to generate a DC voltage to your circuit.

13. Set the DC voltage value to 5 volts. You can either use the knob on the front panel of your vi or type the value 5 V.

14. Measure V_{AB}, V_{BC}, and V_{CD} using your myDAQ and the Digital Multimeter (DMM). (Make sure your probes are connected to the Voltage slot and not the current slot). Enter the voltage measured into your notebook and in the appropriate place of Table 1 of the Lab Report. Make sure you use an appropriate number of significant figures and the units. Confirm that KVL is satisfied by summing your measured values in the table to find V_{AD}.
15. Disassemble the circuit you have built.

Assignment 2: Measurements on the second circuit

We will now create a different circuit which is shown in Figure 5.

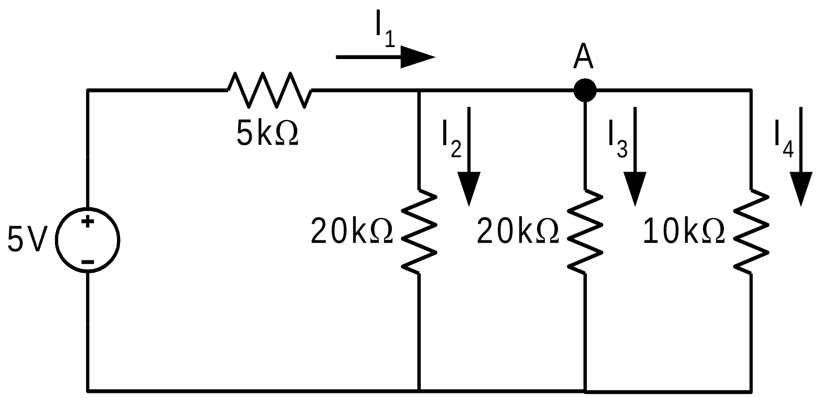


Figure 10: Second Circuit for Circuits I Lab

1. Build this circuit on the breadboard.
2. Connect the *AGND* and AO0 wires to the appropriate places in the circuit.
3. Get your TA to verify that your circuit is correct. If it is, your TA will sign the appropriate place in the lab report.
4. This time, we will measure the currents I_1, I_2, I_3, and I_4 flowing through its resistor by measuring the voltage across it and then calculating the current from this measurement.
5. Draw the circuit shown in Figure 10 above in the box provided in the Lab Report. Include where you will place ammeters (denoted by an “A” inside a circle; include the positive and negative terminals) to measure I_1, I_2, I_3, and I_4. You will have more than one ammeter in your circuit.
6. Enter the values you calculated from the pre-lab into the appropriate location in Table 2 of the Lab Report.
7. Set the DC voltage value to 5 volts. You can either use the knob on your vi or type the value 5 V into the space provided.
8. Measure the current value flowing through each resistor. Use the current setting on myDAQ to do this. Enter the current measured into both your notebook and in the appropriate place of Table 2 of the Lab Report. Assume that positive currents flow from left to right or up to down as appropriate. Make sure you use an appropriate number of significant figures and the units. Remember that KCL must be satisfied.

** **Note:** If you are having trouble getting a non-zero current measurement, go to the properties menu for your multimeter and select "Specify Range" to 20 mA instead of "Auto" **

9. Disassemble the circuit you have built.
10. Turn in your completed lab report to your TA.

FAQ's

Q: Why won't my circuit work?
A: Check your connections. Only the 5 holes in a row are connected; the columns are not connected, and rows across the middle crevasse are not connected!

Q: Why am I not measuring current correctly?
A: In the block diagram VI, edit the range option for current to 20mA.

Q: How do I measure current anyway?
A: Break the connection, and connect your myDAQ probes in series to complete the connection, as you would with a resistor. Also, be sure to change your myDAQ red probe into the correct plug.

Q: Why do my values fluctuate so much and why am I not getting any good reading?
A: You need to make a good contact with your probes to the metal leads of the resistors in order to minimize contact resistance and give a good reading. You may need to press the meter leads to the resistor terminals or do some other intervention to make it work.

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Name: ____ EID: ________________

By placing by name and EID above, I am certifying that I determined the answer to the questions posed below and did not copy my answers from a fellow student.

*** Due at the beginning of your lab session ***

You will need 2 completed copies of this pre-lab. One is to be turned in to your TA at the beginning of the lab session. The other one is to be done in your lab manual.

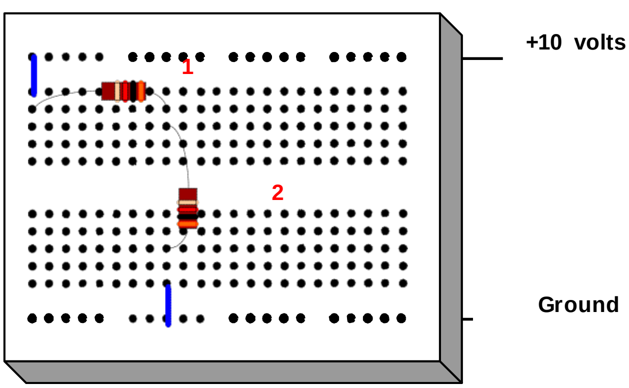
Required Readings:

- APPENDIX A: MULTIMETER BASICS of the lab manual
- APPENDIX B: KEEPING AN ENGINEERING NOTEBOOK of the lab manual

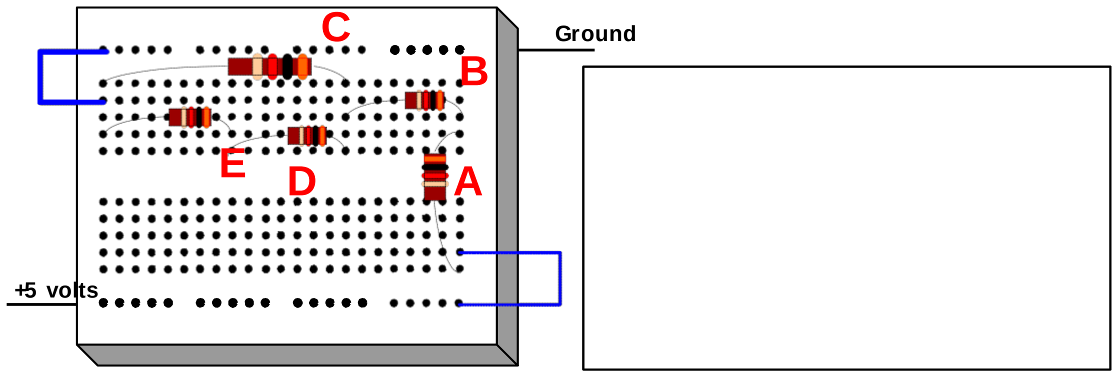
1. Determine what the appropriate color bands are for the resistors to be used in this lab. Place your answers in the table below.

Resistor	First Color	Second Color	Multiplier Color
1 kΩ			
2 kΩ			
3 kΩ			
5 kΩ			
10 kΩ			
20 kΩ			

2. Draw the appropriate schematic diagram in the box provided for each of the following protoboard circuits.

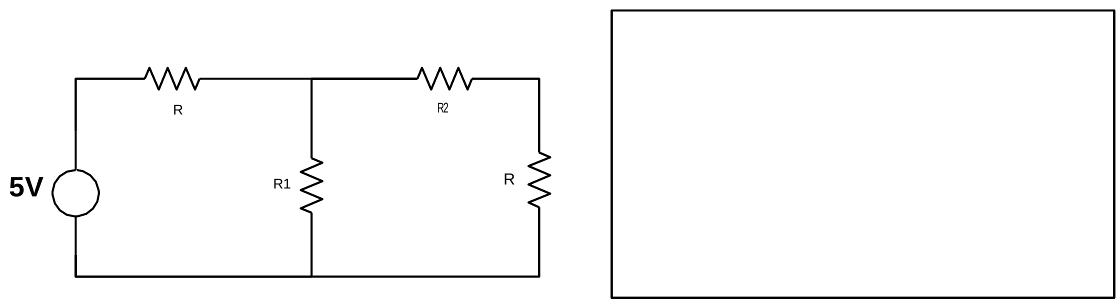


(a)

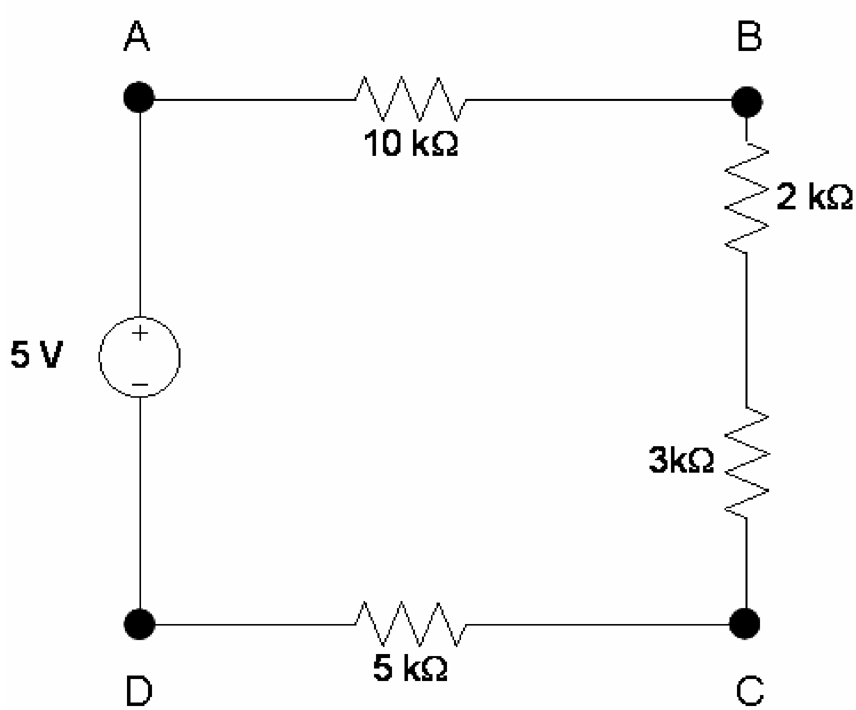


(b)

3. Indicate where you'd place a ammeter to measure the current through the voltage source and where you'd place a voltmeter to measure the voltage of resistor R2 in the diagram below. Place you re-drawn circuit in the box provided.



4. Use the three circuits below to calculate the quantities requested. Assume that positive current flows either left to right or up to down as appropriate. **Make sure you include units and an appropriate number of significant figures.**



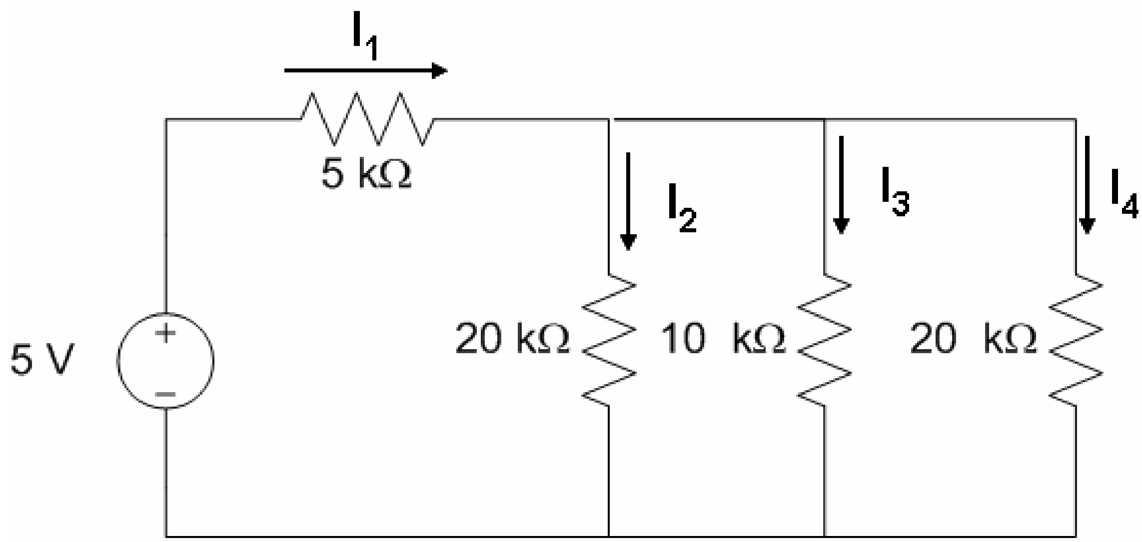
Paramet	Val
Voltage between points A(+) & B(-) $\equiv V_{AB}$	
Voltage between points B(+) & C(-) $\equiv V_{BC}$	
Voltage between points C(+) & D(-) $\equiv V_{CD}$	

(a)

Re-do part (a) assuming that the 5 kΩ resistor is 5.1 kΩ.

Paramet	Val
Voltage between points A(+) & B(-) $\equiv V_{AB}$	
Voltage between points B(+) & C(-) $\equiv V_{BC}$	
Voltage between points C(+) & D(-) $\equiv V_{CD}$	

(b)



Parameter	Value
I_1	
I_2	
I_3	
I_4	

(c)

Re-do part (c) assuming that the 5 kΩ resistor is 5.1 kΩ.

Parameter	Value
I_1	
I_2	
I_3	
I_4	

(d)

Circuits II - Breadboarding and Measurements Lab Report

Submitted by (Print names)

____

____

____

Assignment 1: Build the circuit

TA Signature (Step 4): ____

Place the drawing asked for in Step 5 in the box below. Also, put your Ammeters with appropriate polarity for the measurement.

Table 1: Measured and Calculated voltage values in the circuit in Figure 9 of the Lab. Make sure you use an appropriate number of significant figures and include your units.

	Calculated Voltage	Measured Voltage
V_{AB}		
V_{BC}		
V_{CD}		
$V_{AD}=V_{AB}+V_{BC}+V_{CD}$		

TA Signature (Step 13): ____

Assignment 2: Build the circuit

TA Signature (Step 3): ______________________________

Place the drawing asked for in Step 2 in the box below. Also, put your Ammeters with appropriate polarity for the measurement.

Table 2: Measured and Calculated current values in the circuit in Figure 10 of the Lab. Make sure you use an appropriate number of significant figures and include your units.

	Calculated Current	Measured Current
I_1		
I_2		
I_3		
I_4		

Lab 7: Soldering and Kit Assembly

Due beginning of lab:

- Before coming to lab, review these on-line resources for learning to solder or find your own using a web-search:
  - http://en.wikipedia.org/wiki/Soldering
  - http://www.instructables.com/id/E30LR180T4EWP872BS/?ALLSTEPS (This site contains videos which you may find helpful.)

Lab Goals:

By the end of this lab, students will:

- Learn how to solder electronic components to a “printed circuit” board.
- Learn how to use tools to build the siren and flashing LED circuit.

Required Lab Material:

- A kit to assemble along with the following tools and instrument:
- Soldering iron, stand, solder, and solder wick
- Screwdrivers, wire cutters, and pliers

At the completion of this lab please return the equipment

Due at the end of lab:

- Demonstration of your assembled and working kit.

Assignment 1: Practice soldering

Using the instructions starting on page 4 of the **kit instructions**, practice soldering using the practice pads on the edge of the circuit board as shown in Figure 2 in the instructions. Before proceeding, have your TA inspect your soldering technique.

Assignment 2: Solder components to the board

Using the instructions starting on page 6 of your kit documentation, solder the components to the board. **Pay special attention to the orientation of the following components, as they must be installed in the correct orientation:**

- C6 - 100 μF electrolytic capacitor
- IC1 - 555 or 1455 timer integrated circuit
- C1, C2, C3 - 10 μF electrolytic capacitor
- Q1, Q2 2N3904 transistors
- LEDs

Demonstrate your working board to your TA. If you have problems, ask him or her for assistance in diagnosing the cause.

FAQ's

Q: Which parts have polarity?

A: The electrolytic capacitors have a negative end marked by a white stripe. The LED's have a flat end that is the negative end; you may have to hold up to the light to see. Also, the negative terminals of both these devices have shorter leads. Ceramic capacitors (orange), resistors, and speakers have no polarity.

Q: How do I remove solder? I soldered a part on incorrectly!

A: Use the copper soldering wick by sandwiching it between the soldering iron and the solder lump. Use pliers to remove the affected part once you remove enough solder (you will know).

Q: My soldering iron won't heat the solder to melt it! What should I do?

A: Chances are your tip is badly oxidized, so either clean the tip or get a new iron.

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Lab 8: Solar Power – Data Logging

Due at the beginning of lab:

- **Read the background information to this lab.**
- **Blank copy of the Lab Report.** One copy of the lab report will be turned in and another copy will be in your notebook

Lab Goals:

- Determine the I-V and P-V characteristics of a solar panel
- Learn about a renewable energy source application
- Determine the maximum power delivered by a solar panel
- Learn to use Microsoft EXCEL to display data graphically.

Required Lab Material:

- Solar Panel (provided)
- myDAQ kit
- Computer with Microsoft Excel

Background Knowledge: Solar Panels

Incident sunlight can be converted into electricity by photovoltaic conversion using a solar panel, which consists of individual cells that are large-area semiconductor diodes. Light is absorbed in the silicon, generating both excess holes and electrons. These excess charges can flow through an external circuit to produce power and the equivalent circuit of a single cell amounts to a current source as shown in Figure 5.

In electrical engineering, it is often of interest when characterizing a device to look at its current and voltage relationship, known as the "I-V characteristics" of the device because of the current versus voltage graphs that are commonly used to represent this relationship. Similarly, another relationship that is of interest for a device is the power-voltage or "P-V characteristics".

The I-V characteristics of a solar panel are nonlinear and follow the general shape and equation shown in Figure 6. The current is maximum for the short circuit condition (i.e., V = 0), and the voltage is maximum for the open circuit condition (i.e., I = 0). Maximum power, P_{max}, corresponds to V_m and I_m.

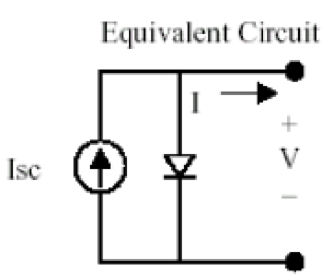


Figure 5. Cell Equivalent Circuit

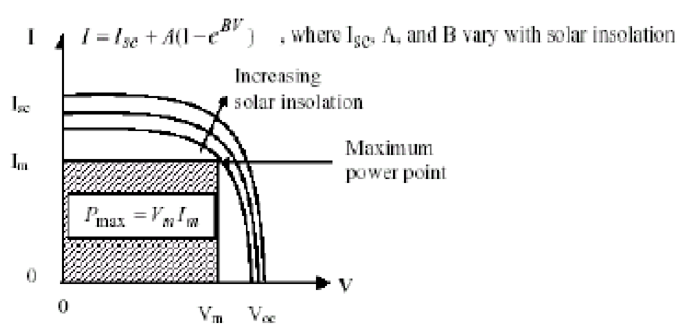


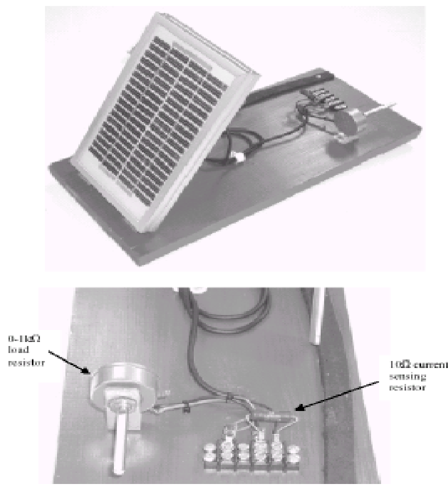
Figure 6. I-V Characteristics of a Solar Panel

Solar Power and Typical Panel Facts:

- Clear day incident solar energy (i.e. solar insolation): 1kW/m^2
- Solar panel power output facing a bright sun: 140W/ m^2
- For 24/7 power availability, deep-discharge batteries store energy for nighttime or overcast daytime use. - Solar panel efficiency: ≈14%
- Efficiency decreases with high temperatures.
- An everyday application of solar power is some of the LED flashing signs from the TxDOT in school zones, for instance.
- V_{oc} per cell (refer to figure 6): 0.5-0.6V
- A 12V-battery-charging panel has (refer to figure 6): ≈ 36 cells, a total V_{oc}≈19V, a V_m≈14V
- Each series cell in a solar panel may contain multiple individual cells in parallel to increase the total surface area and power generating capability.
- Solar power cost: $4-5/W per panel + $4-5/W for batteries and electronics
- V_m, I_m, and the Thévenin equivalent resistance vary with light level, making operation at maximum power difficult.
- DC-DC converters often used to match the load resistance to the Thévenin equivalent resistance for maximum power output and charge storage batteries in a way that maximizes battery life.
- Ideally, a solar panel should be perpendicular the incident rays of the sun and track it like a sunflower to maximize energy capture. Sun positions for Austin are shown in Figure 7.
- Typically, panels are fixed in position facing true south due to high wind survivability and adjusted only seasonally for altitude (winter angle = latitude+20º, summer angle = latitude-10º). For Austin's suggested seasonal panel angles, see Figure 8.

Your Solar Panel System:

- Looks like Figure 1.
- Translates to an equivalent circuit as in Figure 2
- Should have characteristics on overcast and sunny days as in Figure 3
- Should have a V_{oc} = 19V (open circuit voltage), I_{sc} = 0.1A (short circuit current), and P_{max} = 1.2W (maximum power)
- Uses a current sampling/sensing resistor of 10Ω
- Has a potentiometer (or variable resistor) as a load with a range from 0-1kΩ
- Can be used for trickle-charging automobile and boat batteries to replenish leakage losses and power drawn by dashboard electronics, theft detectors, etc.
- Is identical to commercial-grade (40-100W) panels used for powering remote communication sites, school zone flashers, and other loads where conventional power is not readily available or is expensive, except for the current scale factor.



1W Solar Panel with Adjustable Load and Current Sensing Resistors

Figure 1. Solar Panel System

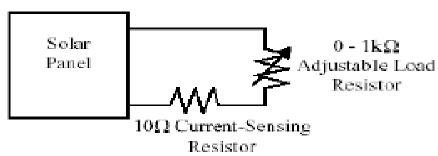
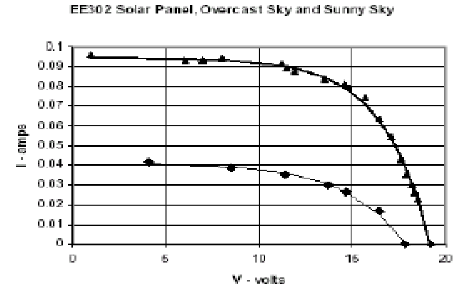
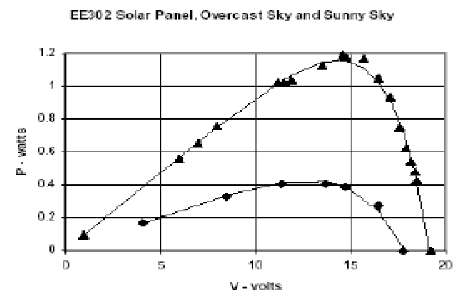


Figure 2. Solar Panel Equivalent Circuit



Measured Data Points and Excel Estimate $I = I_{sc} + A(1 - e^{BV})$
(Estimate for Overcast: Isc = 0.0408, A = 0.0001082, B = 0.332;
Estimate for Sunny: Isc = 0.0945, A = 0.0000476, B = 0.396)



Power Plot (P = V * I)

Figure 3. Sunny and Overcast I-V and P-V Plots

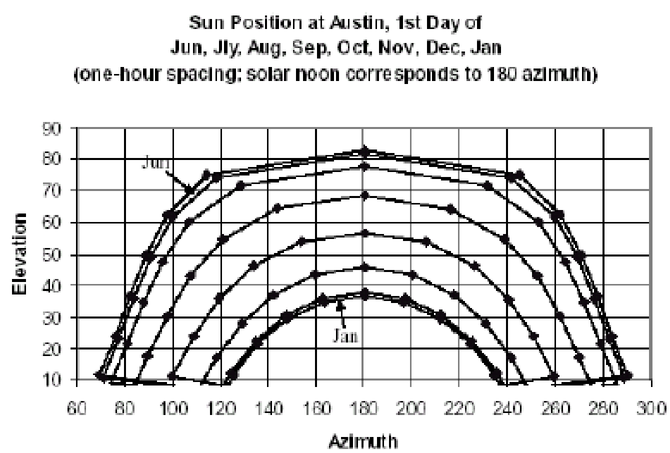


Figure 7. Sun Position of Austin (altitude is in degrees above the horizon and azimuth is in degrees from true north)

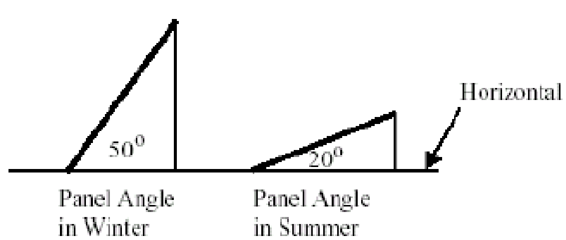


Figure 8. Seasonal Solar Panel Angles for Austin (latitude = 30°) with Respect to the Horizontal

Assignment 1: Experimental Procedures

1. If possible, perform the experiments outside in full sunlight. An alternative to the sun is to use the lamps provided in ENS 212 (the power lab).
2. If you are performing the experiment in sunlight, orient the panel so that it is perpendicular to the sun. Shadows projected by the sides of the panel will help you align it. Holding a pencil alongside and perpendicular to the panel is also helpful. The shadow of the pencil will be a minimum when the pencil points directly towards the sun.
3. If you are using a lamp, visually orient the panel so that it is perpendicular to the lamp rays. Keep the panel at least 12 inches from the light bulb to avoid overheating the panel. If you are using a focused beam, do not concentrate the light on a small part of the panel, but instead, spread the beam over the entire panel face.
4. Once the experiments begin, do not move the panel.
5. Add a voltage divider circuit across the panel voltage, which consists of two high value resisters connected in series. The TA will provide this.
6. Run wires from AI0 to one of the voltage divider resistors and also run wires from AI1 to the sensing resistor (10 Ω)
7. You will use the VI you created for pre-lab to acquire data.
8. Turn the potentiometer counterclockwise until it cannot be turned.
9. Start the vi and slowly turn the potentiometer in a clockwise direction until you reach the end.
10. Stop the vi.
11. You will use the logged data to create an I-V characteristic plot and P-V characteristic plot of the Solar panel on MS Excel.
12. Based on the data acquired you will also use the equation in Figure 3 to generate a theoretical I-V and P-V plot.
13. Have both the measured plot and the theoretical plot on the same graph for comparison.
14. Complete the Solar Power Lab Report using the hard copy you brought to lab and turn a copy in to your TA at the end of lab.

Acknowledgments:

- The solar panels and associated hardware used in this lab were donated by TXU Electric.
- The introductory material was taken from Dr. Ewald Fuchs' lecture notes for ECEN3170, "Energy Conversion I," University of Colorado at Boulder.

For More Information:

Three excellent web sites for information on solar power are Jade Mountain, www.jademountain.com, the Texas Solar Energy Society, www.txses.org, and Southwest PV Systems, www.southwestpv.com.

This lab was created by Dr. Grady on 11/16/01 and modified by Seunghyun Chun on 12/18/2010.

FAQ's

Q: What am I supposed to measure?

A: Remove the current sensing resistor and measure the voltage across the terminals where the resistor used to be. This is your open circuit voltage. Replace the resistor. While sweeping the variable load resistor, measure the voltage across the load resistor, V_L, then measure the voltage across the current sensing resistor V_R. Divide V_R by 10 (the value of the current sensing resistor, and this value will be your measured current I_R. Take several data points of (V_R, I_R).

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Solar Power – Data Logging
Lab Report

Submitted by (Print names)

Please, state the Weather Condition (Cloudy, Sunny, or Using a light source in the lab)

Solar Power: I-V plot (Both Theoretical and Measured values in the same plot)

Solar Power: P-V plot (Both Theoretical and Measured values in the same plot)

Explain why there is a difference between the theoretical values and measured values.

Lab 9: Instructions for Final Project (Robot Car) and Training module 1 (Rock, Paper, Scissors)

Due at the beginning of lab:

- **Read through the Robot Car instructions (all of lab 9 section)**
- **Complete pre-lab questions at the end of this section for the training-module**
- **Blank copy of the Lab Report.** One copy of the lab report will be turned in and another copy will be in your notebook

Lab Goals:

- Learn about flex sensors
- Understand voltage dividers (especially with varying resistances)
- Begin building Robot Car

Required Lab Material:

- Robot Car Kit
- myDAQ (number of myDAQs can be varied by the Robot Car functions)
- Laptop with USB for myDAQ

Introduction

Training Module

For this final project, you will need to brainstorm with your team to determine the best design for your robot to complete the obstacle course. In order to help you accomplish this goal, we have designed special training modules that each student must complete before starting to work on your final project. Each subsystem will have its own associated training module that is meant to expose the students to some of the concepts and skills necessary to complete their subsystem.

Background Knowledge: Understanding a Flex Sensor

A flex sensor has the unique property that it changes its internal resistance based on how much it is bent. A typical flex sensor will have a resistance of 20kOhms when it is flat. This resistance gradually increases and the sensor is bent further. Typical ranges of bent resistances are between 40k and 60kOhms. Figure 1 shows the SpectraSymbol Flex Sensor. Notice the how there is a grid on one side of the sensor. The grid should be facing outward when the sensor is bent, as the resistance will not change if bent the other way.

Flex sensors can be used in a variety of applications, from robotic bumpers to playing the air guitar, and once you understand the basics, with a little creativity the possibilities are endless!

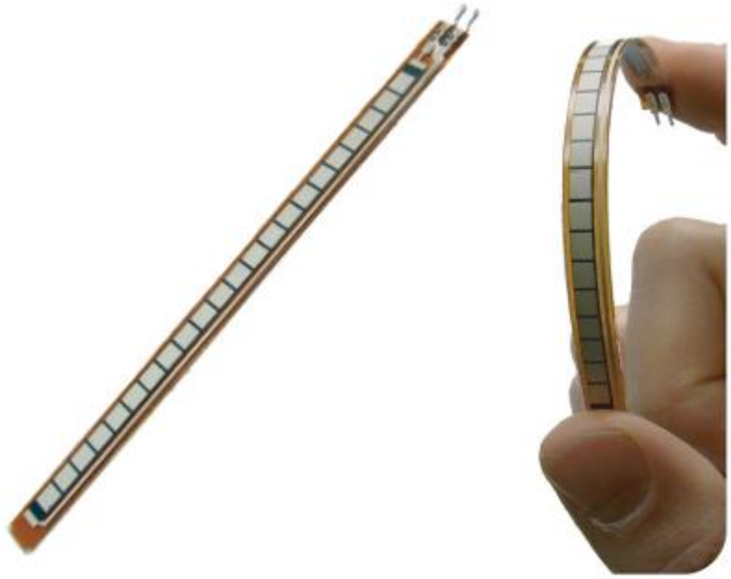


Figure 1: SpectraSymbol Flex Sensor

The Flex sensor has two leads, and since it acts much like a resistor, the orientation of the leads does not matter. An additional resistor is required in order to turn the sensor into a simple voltage divider. The output voltage is sent to an Analog Input pin.

Assignment 1: Building the Circuit

1. Fill in Chart 1 in your lab report. Make sure to measure resistance when the flex sensor is not powered by a source.
2. An example circuit is shown in Figure 2. Build two copies of this circuit on your breadboard. Make sure to attach the output of one of them to AI_0 as indicated in Figure 2 and the other to AI_1.

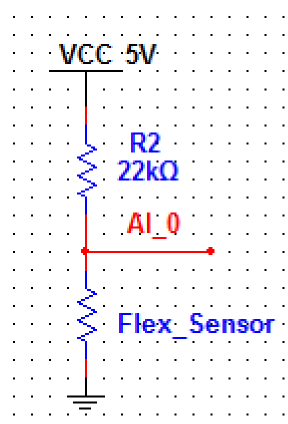


Figure 2: Flex Sensor schematic made in Multisim

3. Fill in Chart 2 of your lab report.
4. Call a TA over to sign your lab report to verify the functionality of your circuit.

5. Download the VI for this lab from blackboard. It should resemble the following.

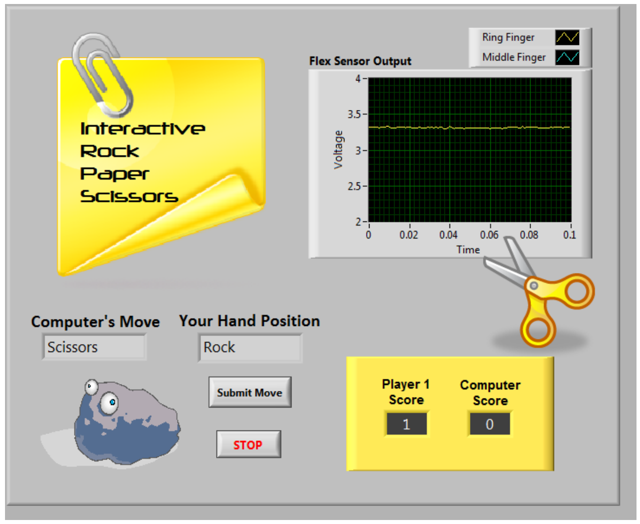


Figure 3: Front Panel

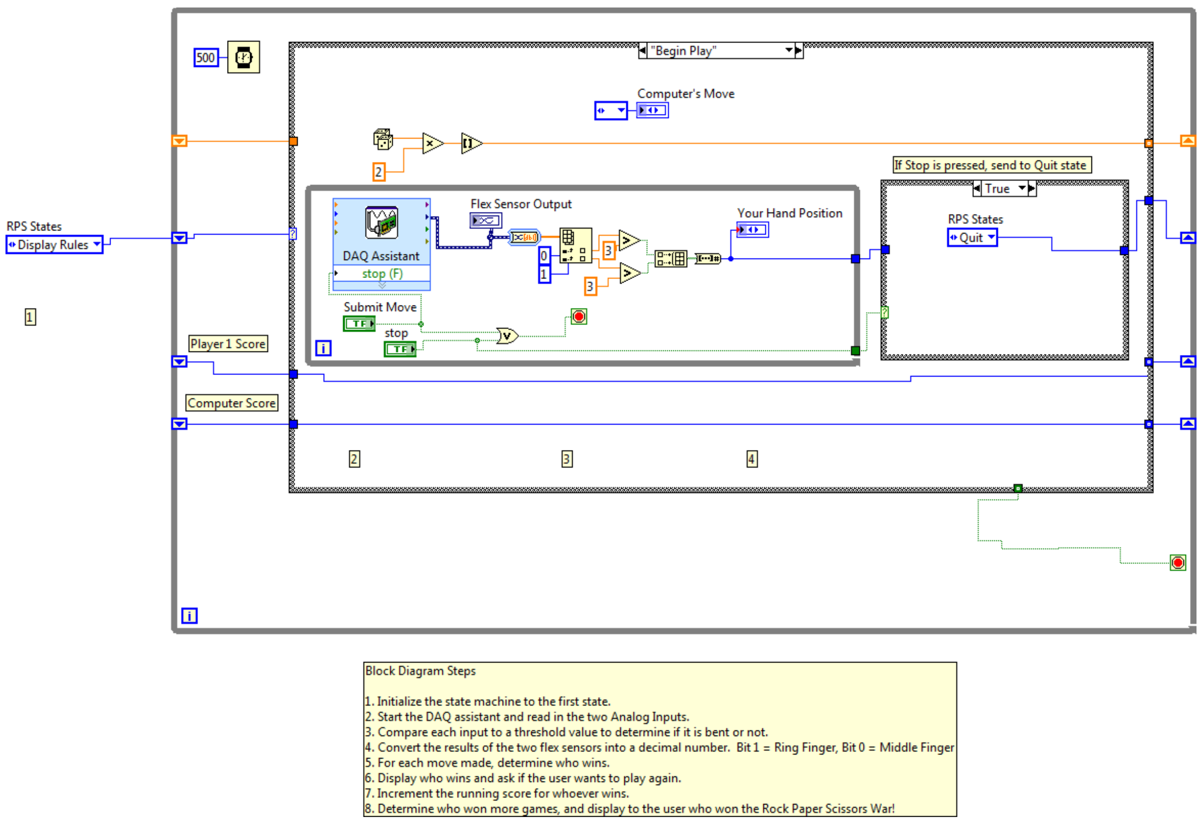


Figure 4: Block Diagram

6. Attach one flex sensor to your middle finger and the other to your ring finger. Refer to the chart below and the VI to determine which sensor should go on which finger. The values of these two sensors will determine the position according to Table 1. A 1 represents a bent finger. For example, if the move is Paper, then neither sensor will be bent, and if the move is Scissors, then only the ring finger should be bent.
7. Connect the 0- and 1- pins to ground.
8. **Ensure that the grid side of the sensor is facing upwards. You can attach the sensors to your fingers with tape or rubber bands. Experiment with what positioning of the sensor on the finger works best.

Table 1

Ring Finger Bit 1	Middle Finger Bit 0	Decimal Number	Game Move
0	0	0	Paper
0	1	1	Invalid
1	0	2	Scissors
1	1	3	Rock

9. Write in your lab note section to indicate which finger goes to which input port (AI_0, AI_1).
10. The set up screen for the DAQ Assistant is shown in figure 5.

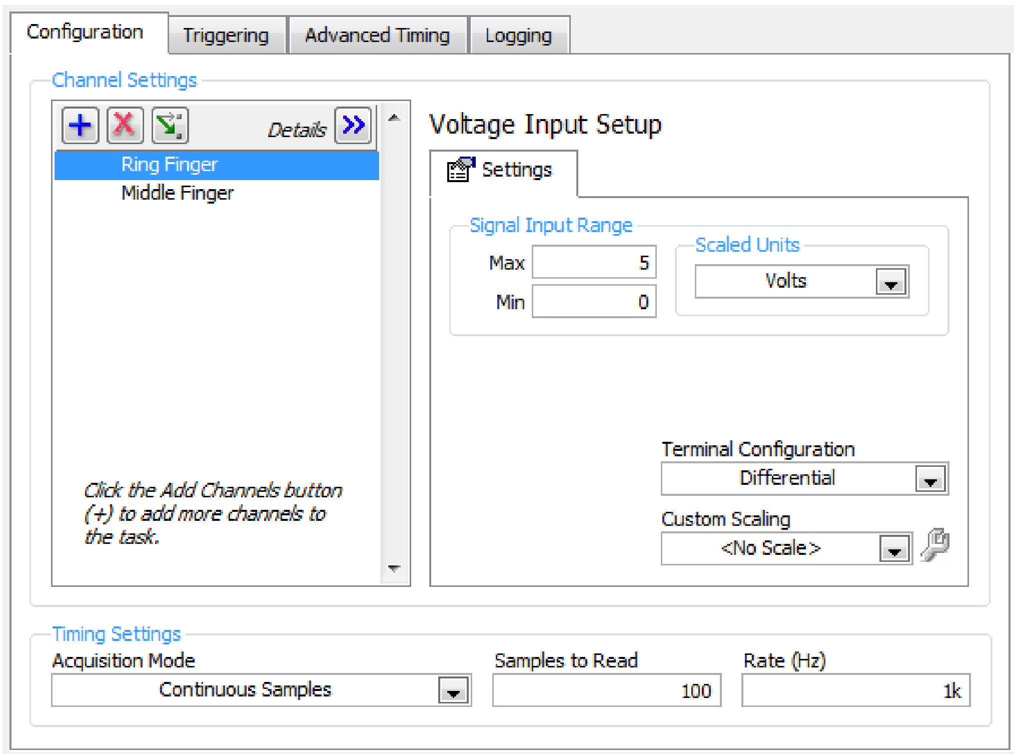


Figure 5: DAQ Assistant set up

11. A state machine was used to program the Rock Paper Scissors game. At the start of each game, the value of the flex sensors is read in on the Analog In lines using the DAQ Assistant. The voltages from the flex sensors are then compared to a threshold voltage that determines whether the sensor is bent or not. This may vary depending on the resting and bent resistance of the particular flex sensor. In our case, a good threshold was 3V. Once the move of the user is determined, the software determines who wins the round and displays a pop up to the user. If the player presses stop, then the state machine will go to a Quit state, and the program will determine who won the most Rock Paper Scissors battles. The front panel and block diagram are shown in Figure 4 and 5. Take time to look through the various states in this vi. When you understand how it works, play the game!
12. Play with the program and then when you are satisfied with its performance, call a TA over to verify the functionality of your game.
13. Measure resistance or voltage for other sensors.
    (Photoresistor: resistance measurement, IR sensor: direct Voltage measurement with white line)

Acknowledgement: A special thanks to Jackie Leverett (Summer ELP Intern of NI) for the example Rock Paper Scissors game.

Robot Car Instruction Guide (used in labs 9, 12, and 13)

For your final project, your team of 4-6 people will be building a robot car that can navigate a track on its own using the program you write in LabView for your myDAQ. There are six phases in the track.

Grade Breakdown:

Phase	Your grade can be up to
1	50
2	80
3	90
4	100
5 (Bonus)	105
6 (Bonus)	110

Track:

The track your car will navigate is shown below:

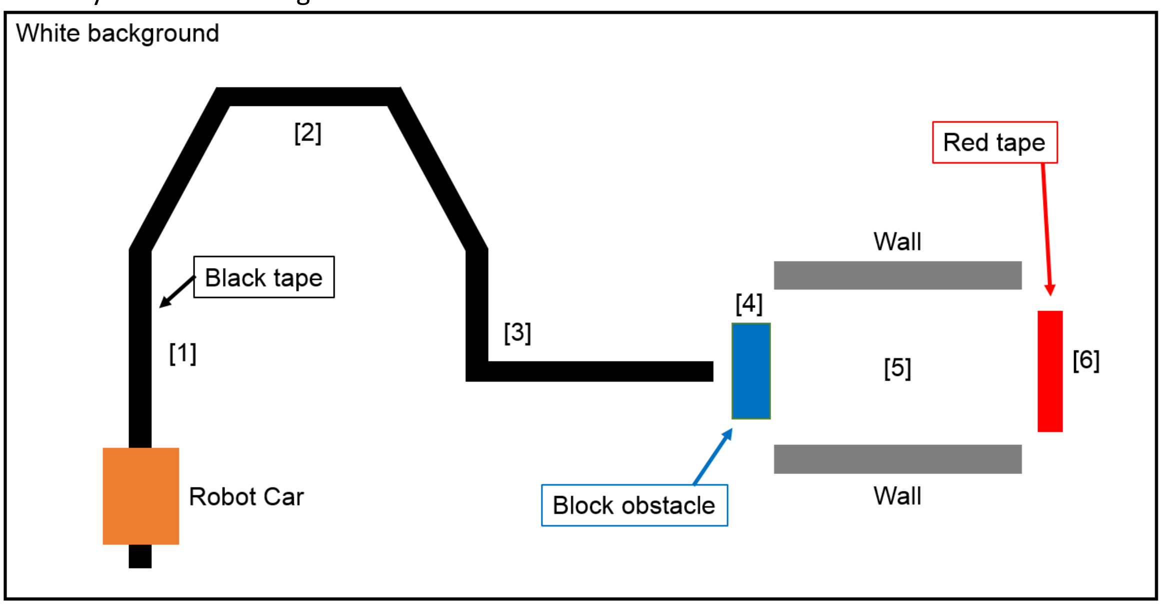


Fig. 1.

Phase [1]: The car must be able to follow a straight line for a distance.
Phase [2]: The car must be able to take several large angle, slow turns
Phase [3]: The car must be able to make a 90 degree turn
Phase [4]: At a certain section of the course, the TA will introduce an obstacle (Robot Car kit box) in the path your car is travelling. The car must be able to sense this and stop moving while the obstacle is there. The TA will then remove the obstacle and the car should resume its path.
Phase [5]: The car will navigate through a section of the maze with no lines on the ground, only walls. The IR (distance) sensor should prevent the car from hitting a wall.
Phase [6]: At the end of the course, the car should stop when it sees a red line.

Block diagram:

A very simple block diagram of how this will work is shown below:

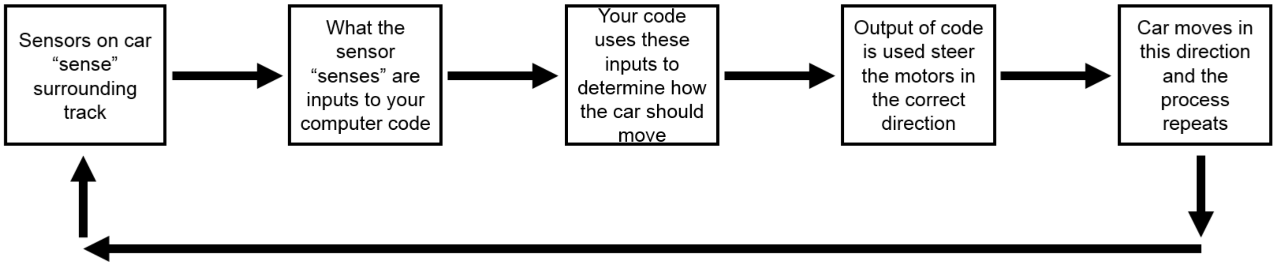


Fig. 2.

Sensors:

Below are the 2 types (you have 3 of each) of sensors in your box:

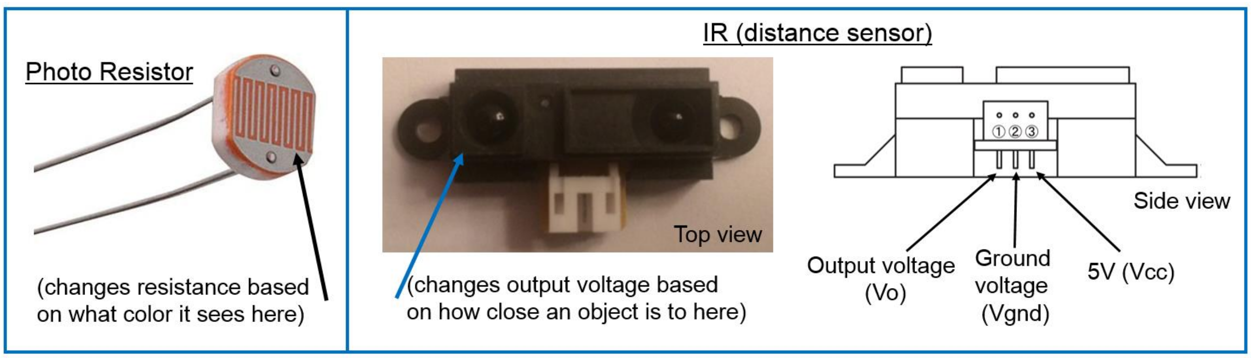


Fig. 3.

Color (in front of photo-resistor)	Resistance (ohms)
Nothing in front of PR	~ 4000
White	~ 5000
Red	~ 9000
Black	~20000

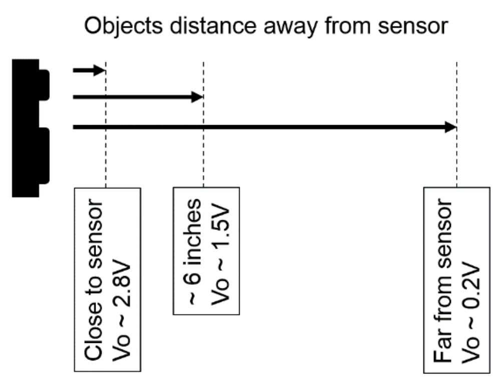


Fig. 4.

Inputs and outputs (and voltage divider for the photoresistors):

The second block in the block diagram says that what the sensor "senses" are inputs to your computer code. The myDAQ and Arduino read input voltages and input currents. This means that the input that your code will analyze must be a voltage or a current. Luckily, the IR (distance) sensor's output (Vo) is already a voltage. The output voltage, Vo, from each of the three IR (distance) sensors will be three of the six total inputs to your myDAQs or Arduino. The other three inputs will be from the photoresistors (PRs). You will need to convert the changing resistance from the PR into a changing voltage. In order to do this, you will make a voltage divider using the PR. A diagram of the inputs is shown below:

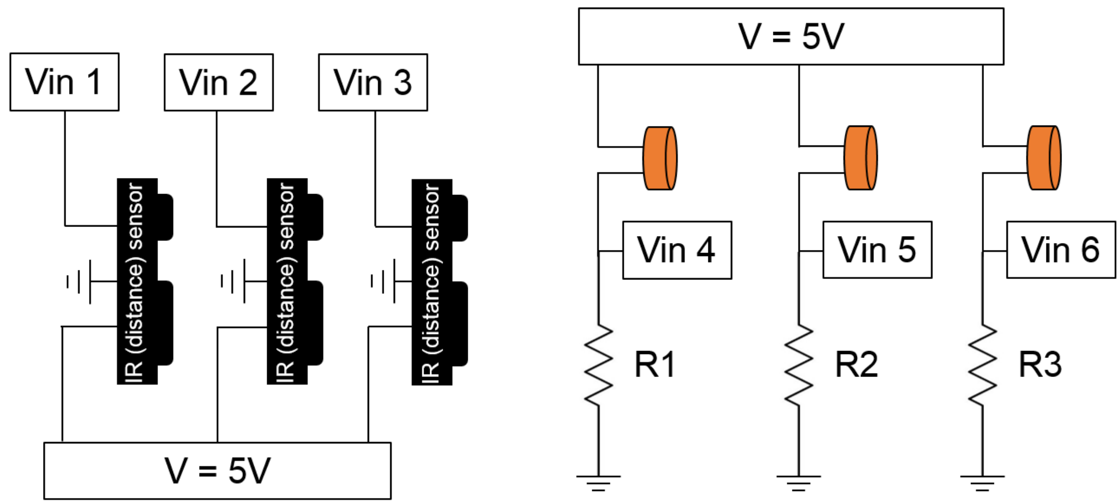


Fig. 5.

How will these inputs be connected to your myDAQ? The myDAQ has two analog ports that can be used as analog inputs or outputs. It also has eight digital ports that can be used as digital inputs or outputs. We will use then in their voltage mode (meaning the read and supply voltages). Four of the digital ports will be used for the digital outputs that will connect from the myDAQ to the L293D (H-Bridge). Four of the digital ports will be used as inputs. And the two analog ports will be used as inputs as well. Further in this section, we will see why this is the case. However for now let us just look at the s etup shown below. Connect the inputs to the myDAQ as seen below. If you are using an Arduino, connect the inputs into the correct ports.

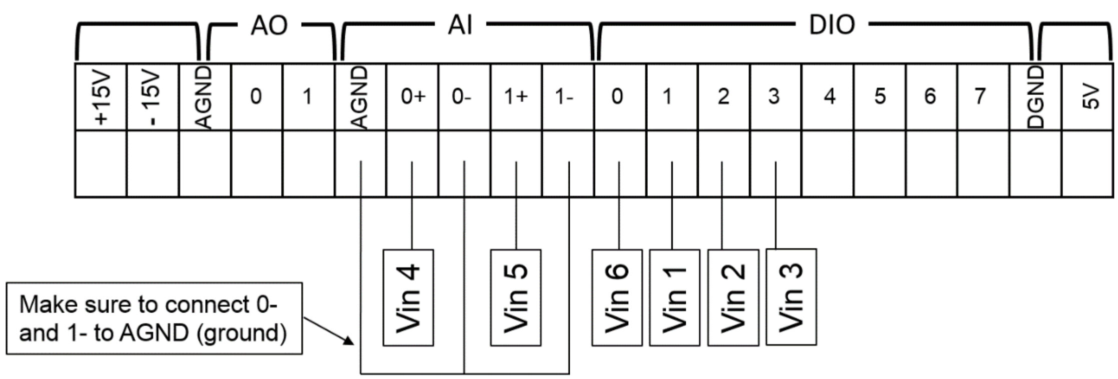


Fig. 6.

Now we will look at the outputs of the myDAQ. Each of the two motors has two leads for a total of four leads. This is why we need four outputs (output voltages) from the myDAQ that will control the motor. But why can they be *digital* outputs? This is because for the simplest case, we will only care that we can turn on the motors at a fixed speed, and that we can also turn them off. Below are the four cases (forward, left, right, and stop) that we are interested in. We see that using the four leads of the mot ors, all four cases can be obtained by setting some leads to a fixed "high" voltage and the remaining leads to a negative (zero) voltage. Please take notice of the plus and minus signs at each particular lead.

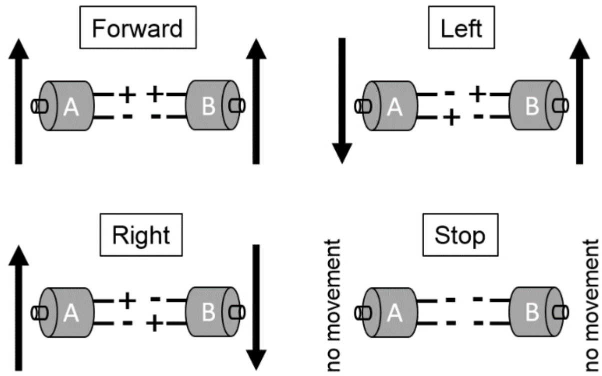


Fig. 7.

However, the myDAQ cannot supply enough power to power the two motors. The L293D (also known as the H-bridge) will be used to take your binary outputs from your myDAQ and supply enough current and voltage to the leads of the motors to turn them on or off. The L293D supplies power to the motors from the 6V battery pack or 9V battery that is attached to it. A diagram of the L293D and its pins is shown below:

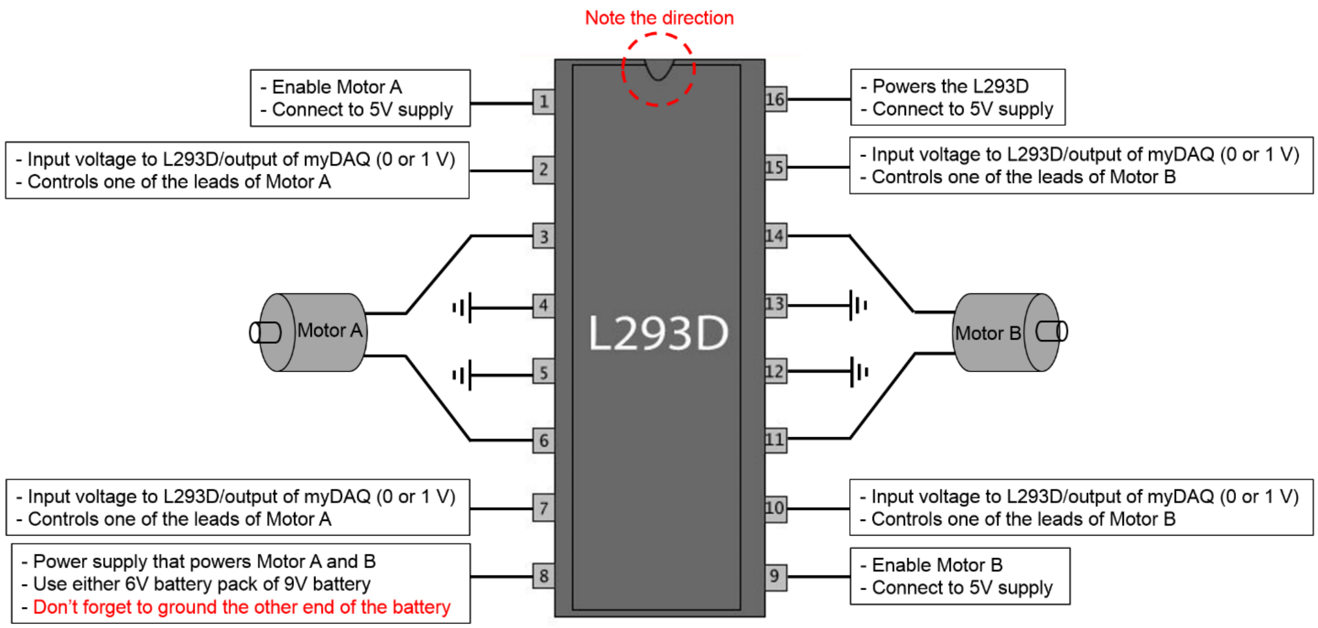


Fig. 8.

Below is a diagram showing the inputs and outputs connected to the myDAQ as well as the L293D (H-bridge). Please connect the correct ports from the myDAQ to the L293D using wires. Using different colored wires is suggested so that you can tell each connection apart.

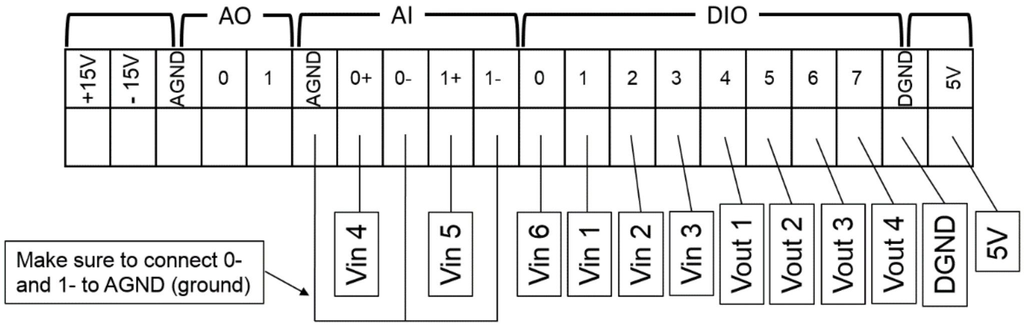


Fig. 9.

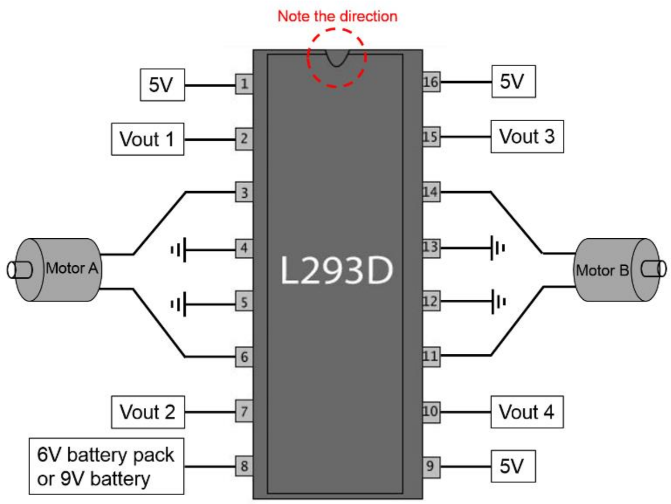


Fig. 10.

The L293D (H-bridge) and the voltage dividers for the photoresistors will need to be mounte d on a breadboard. The myDAQ and the breadboard will need to be mounted to your car.

Coding using LabView:

Because we are using binary outputs (either we turn on a motor or turn it off), we can use binary logic to analyze our inputs and determine which motors we should turn on and in which direction they should rotate. The digital inputs (for the three IR (distance) sensors and the one "middle" photoresistor (PR)) have a "threshold" voltage of about 1.5 volts. Look at the picture below. It is a screen shot of a portion of the Robot Car LabView template.

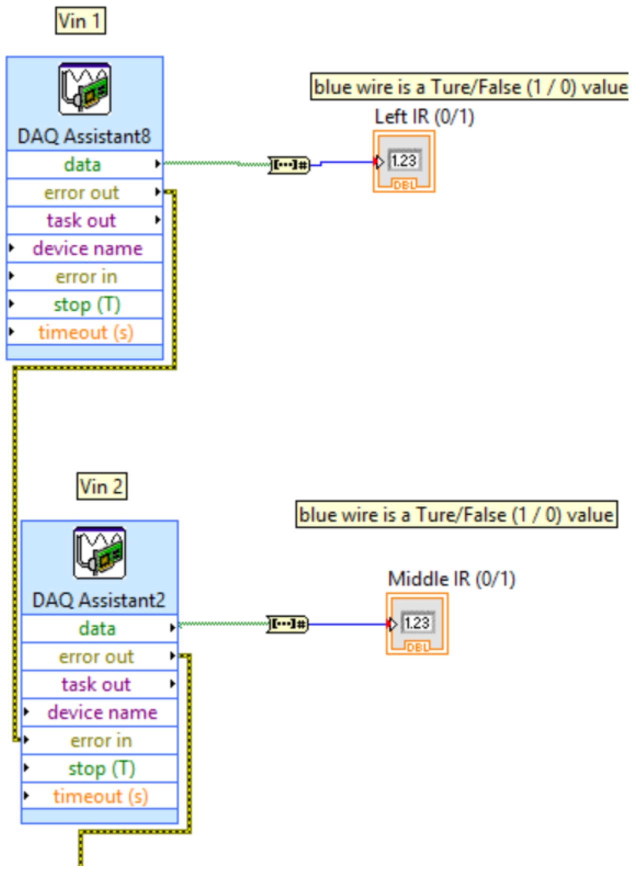


Fig. 11.

What does it mean that the digital inputs have a threshold voltage of about 1.5V? Let's take an IR (distance) sensor as an example. From Fig. 3, we see that when an object is close to the sensor, Vo ~ 2.8V. When an object is far away from the sensor, Vo ~ 0.2V. Let's say that Vo of the sensor we are talking about at the moment is input Vin 1. If "Vin 1" is < 1.5V, then the indicator "Left IR (0/1)" will be a 0. If "Vin 1" is > 1.5V, then then indicator "Left IR (0/1)" will be a 1. Luckily, when an object is ~ 6 inches (0.15 meters) from the sensor, Vo is ~1.5V. The walls will be placed accordingly (more than 6 inches apart from either side of your car) on the track such that the IR (distance) sensors are not always reading just 0 or just 1 the entire time. You will need to build an appropriate voltage divider for the "middle" IR (distance) sensor such that the output voltage of the divider will be around 1.5V (because Vo of the divider needs to be able to produce both a 0 and 1 when read as an input to the myDAQ).

We want to turn the two analog inputs (i.e. Middle and Right PR sensor) to digital inputs as well. To do this we just read in the voltages from the voltage dividers from the Left and Right PR sensors, then we do a comparison with a voltage that you will choose. See the screen shot below:

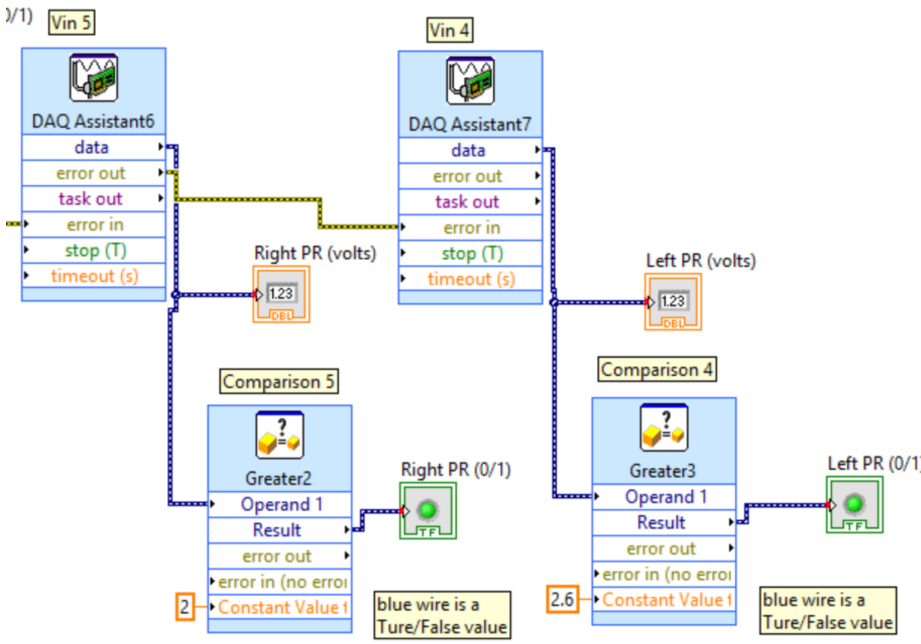


Fig. 12.

In this case the input voltage "Vin 5" is being compared to V = 2 and will be a 0 if "Vin 5" < 2 and a 1 if "Vin 5" > 2. This is the same scenario for "Vin 4" except this time the comparison voltage is V = 2.6. You can change these comparison voltages (also known as threshold voltages) by double clicking on them and changing them to a different constant.

Shown below are the four Boolean arrays that represent the outputs. Each array is four 0/1 bits long. They represent the sequences of 0/1 combinations that will control the leads of the motor. The green boxes representing the arrays are located on the back panel (block diagram panel) and the right most picture with the arrays with gray circles is located on the front panel. If you press on the circles, you can turn them from gray to black. Gray represents a 0 and black represents a 1. An example (**not the correct combinations**) is show below on the right.

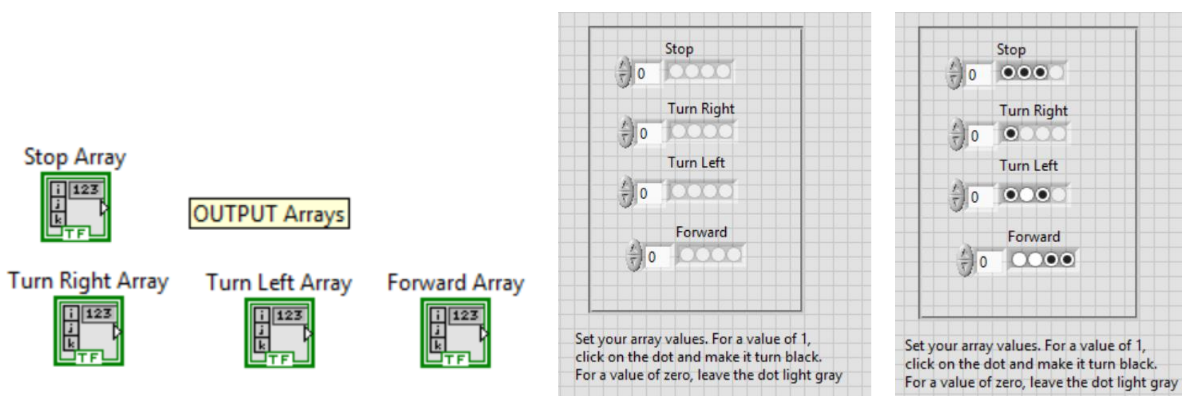


Fig. 13.

The main work of you coding involves connecting the wires that represent the inputs to the digital logic blocks that then connect to the "if true, else false" blocks (that are connecting the output arrays to the DAQ Assistance that represents the 4-bit output array). See the below screen shot. The output arrays consists of the correct "Vout 1", "Vout 2", "Vout 3", and "Vout 4" for each of the forward, right, left, and stop operations. Below, inputs are circled in red, and what you need to connect your logic blocks to are circled in blue.

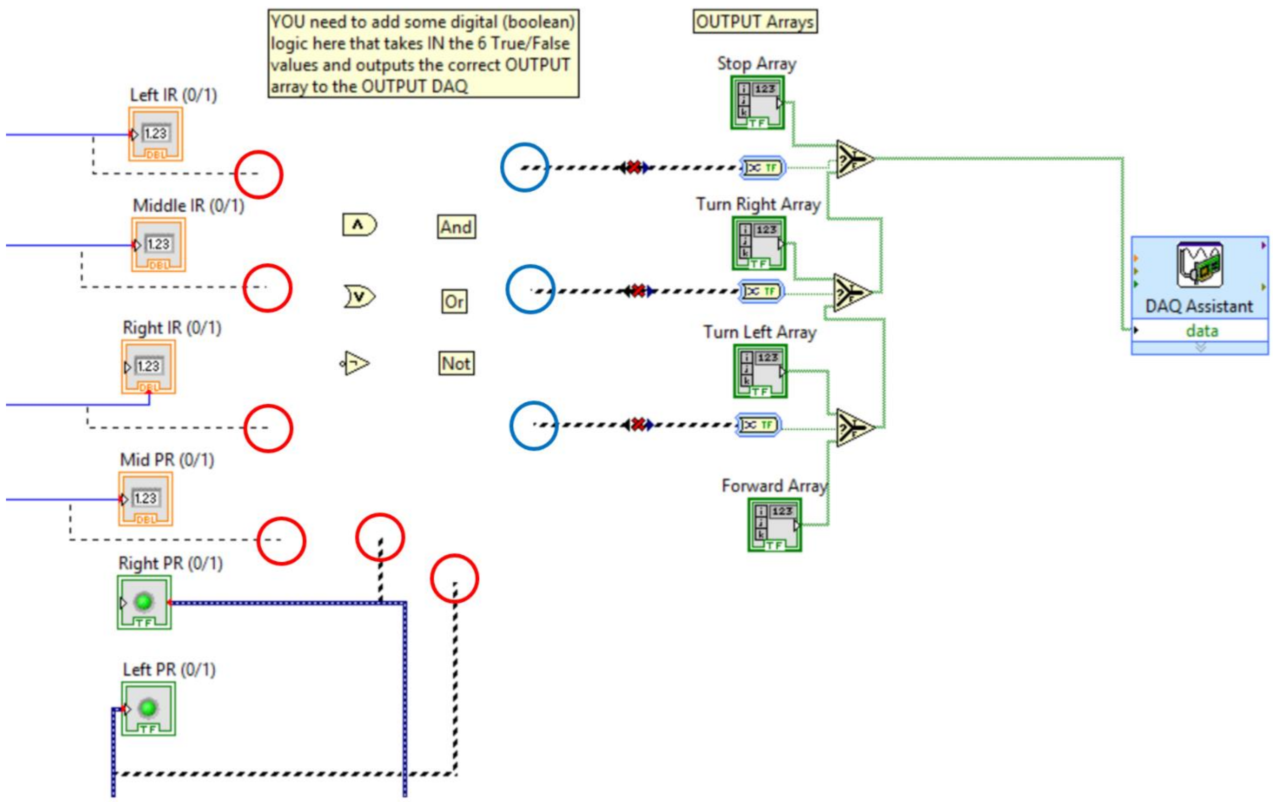
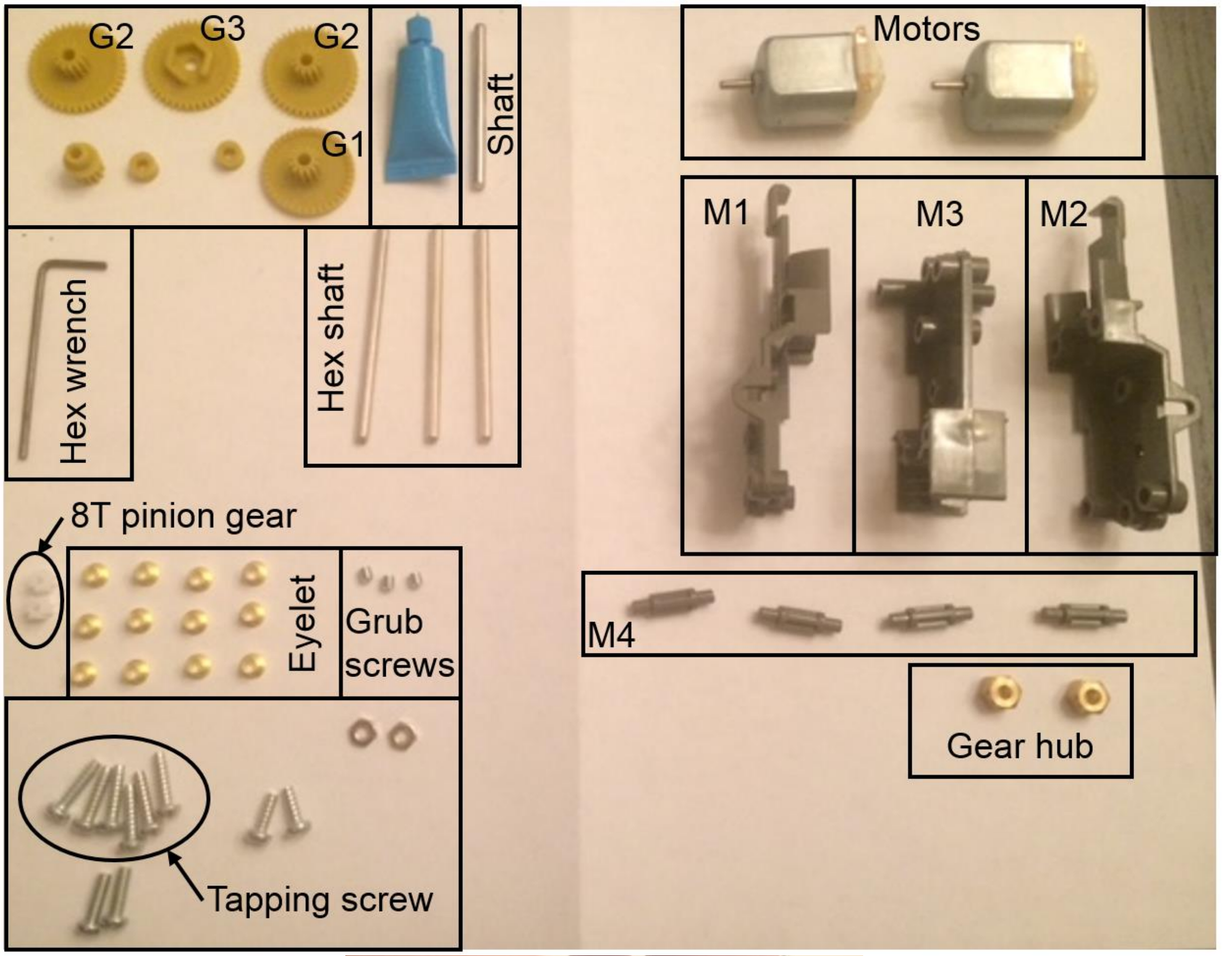


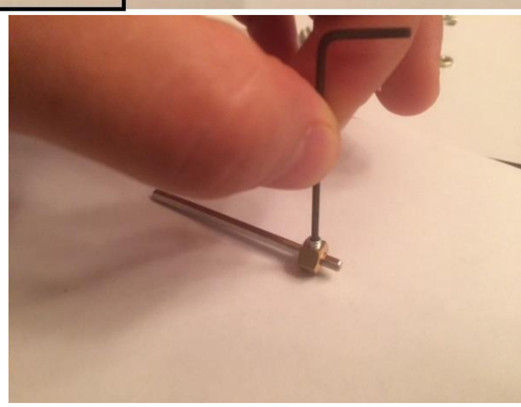
Fig. 14.

Building the Gear-Box:

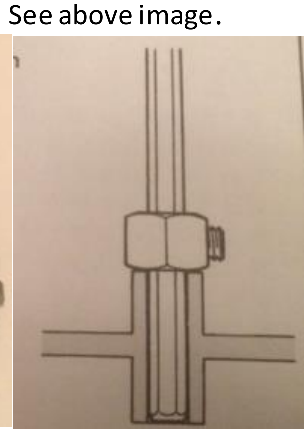
Constructing the cars chassis and treads/wheels is fairly simple and I have not had questions regarding this during my semesters as a TA. However, I have received many questions regarding the construction of the gear box, so detailed instructions are listed below to supplement the instructions already given in the Robot Car Kit.

Because the Robot Car is really a Robot Tank (two treads), you will need to build the gear box that holds the two motors. This gearbox is located in the smaller box with the label “Twin-Motor Gearbox”. **Do not build the gear box that holds only one motor**, you will not use it. It is recommended to build the twin-motor gear box using the lowest speed and the steps for doing so are displayed below. (note: the terminology is that of what is in the instruction sheet). Please note also that there will be some parts in the Twin-motor Gearbox that will not be used.



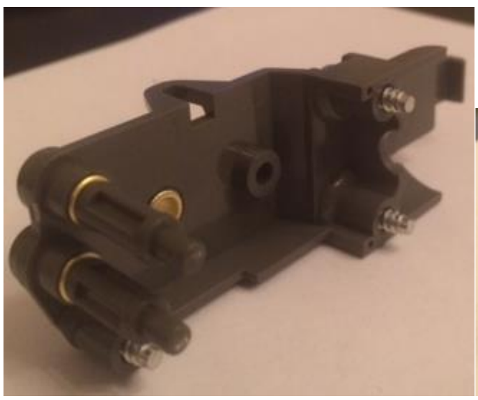


1. Place a gear hub though a hex shaft and screw the grub screw lightly into the gear hub (don't screw in all the way). **Do this same process for one more hex shaft**. See above image.



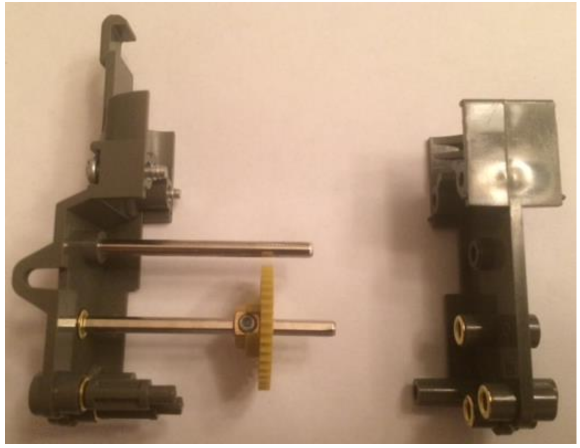
2. To know what location to tighten the gear hub on the hex shaft, place through the hole labeled "C" on component M3.

3. Place three eyelets into M1.

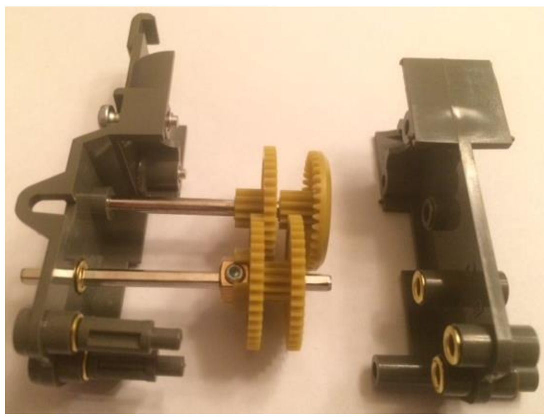



4. Place two M4 rods and three tapping screws as seen above.

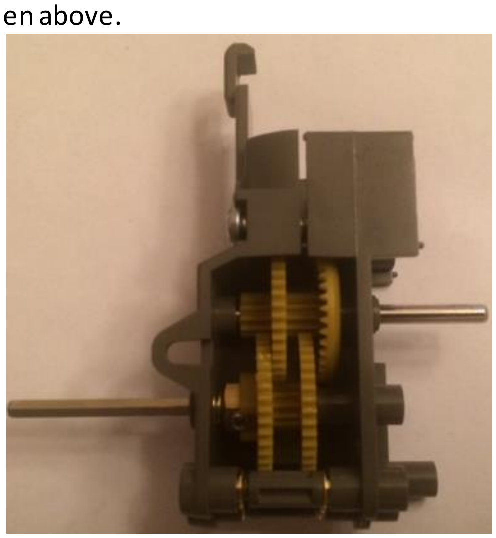
5. Place eyelets into M3 as seen above.



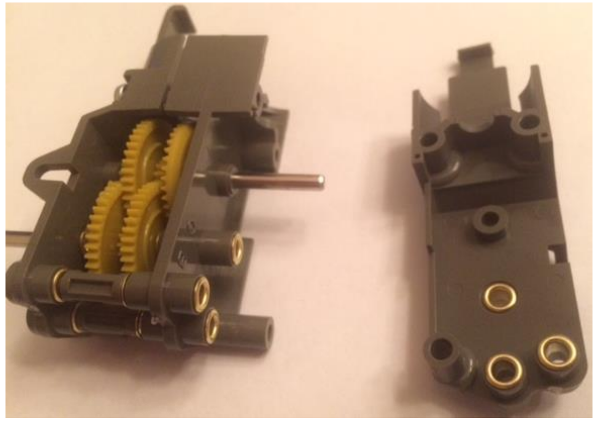
6. Place the shaft (upper rod) and the hex shaft (created in step 1) as seen above.



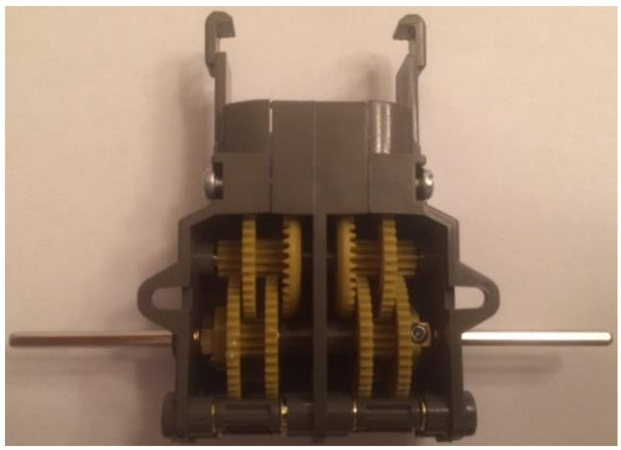
7. Place gears G2 and G1 as seen above.



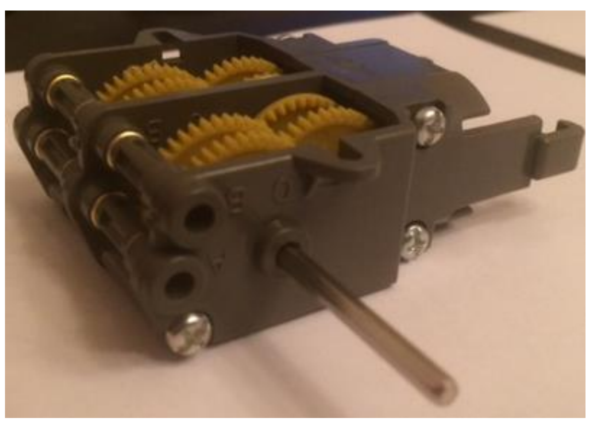
8. Bring M1 and M3 together and tighten the tapping screws slightly so that M1 and M3 will not come apart.



9. Place eyelets into the other side of M3 as well as M2 as seen above.



10. Put gears G1, G2, and the hex shaft and G3 that you assembled in step 1 in the correct position as seen above.



11. Screw three tapping screws in slightly to make sure that M3 and M2 do not come apart.

12. Push the 8T pinion gears onto the motors as seen above.

13. Insert the motors with their leads facing out as seen above and tighten all tapping screws.

Building rest of the Robot Car:

As was mention in the previous section, you will need to assemble the wheels and treads to the wooden chassis block that is provided in the Robot Car Kit. You will also need to attach your sensors to this chassis in a way that you and your group decide on. You will also need to mount the b readboard as well as your myDAQ to the chassis (wooden block as well). Groups in the past have taken advantage of UT's Maker-Space and have used 3D printers to print mounting brackets. You TA will provide more details on this construction but for most groups in the past, building the gear-box, debugging, and programming the algorithm have been the most time consuming.

Final comments:

Be neat about your wiring and use different colored wires to help the debugging process. Make sure that the sensors are mounted well and will not move around. The distance that the PRs and IRs (distance sensors) are from the track and walls is important so that is why you want to mount them properly.

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Name: ________________________________ EID: ________________

By placing by name and EID above, I am certifying that I determined the answer to the questions posed below and did not copy my answers from a fellow student.

*** Due at the beginning of your lab session ***

You will need 2 completed copies of this pre-lab. One is to be turned in to your TA at the beginning of the lab session. The other one is to be done in your lab manual.

1. Read the background knowledge section of Lab X.
2. Write down the equation for the voltage at AI_0 for the circuit in Figure 1. Let R_{flex} be the resistance of the Flex Sensor.

V_{ai_0} = ____

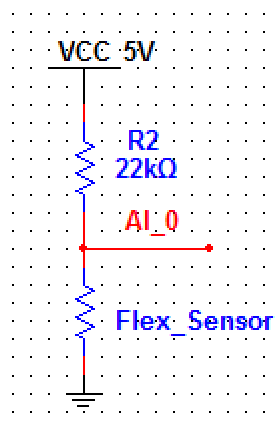


Figure 1

3. Draw a circuit diagram for the following. Omit the AI_0 wire. Instead, draw where you would place your voltage meter to obtain the voltage at AI_0

4. Make this circuit in Multisim.
5. Fill out chart 1 by simulating this circuit using multisim. Replace the Flex_Sensor with a normal resistor. Assume tolerance 0% for all resistors.

R_{flex} (kOhms)	V_{ai_0} (calculated from Q2)	V_{ai_0} (Multisim)
20		
40		
60		

6. Do the simulated values in Multisim match the calculated values from your equation in Q2? Why or why not?

Final Project: Training Module 1

Rock, Paper, Scissors and Sensor level measurements

Submitted by (Print names)

____

____

____

Chart 1: Resistance Measurements of Flex Sensor

How much to bend the Flex Sensor	Resistance of the Sensor (ohms)
Unbent	
Halfway bent	
Fully bent	

Chart 2: Voltage Measurements at the Output

How much to bend the Flex Sensor	Voltage at Output (v)
Unbent	
Halfway bent	
Fully bent	

TA Signature (Step 4):____

TA Signature (Step 12):____

Question: Do the values calculated in the pre-lab from an analytical approach, simulated in Multisim and measured on the breadboard match? Why or why not?

Chart 3: Resistance Measurements of Photoresistor

Color Line below the photoresistor	Resistance of the Sensor (ohms)
Black	
White	
Red	

Chart 4: Voltage Measurements of IR sensors (Direct measurement of white line)

Distance from the sensor	Voltage at Output (v)
10 cm	
20 cm	
30 cm	

TA Signature:____

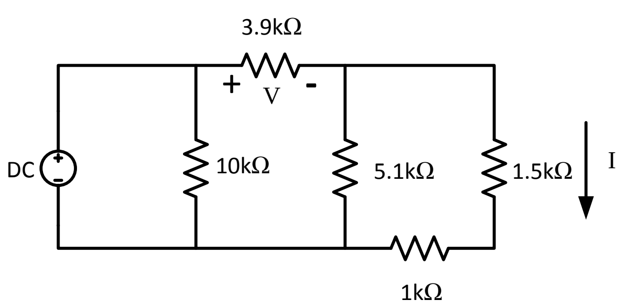
Question: How do you apply above measured values of your sensors for the sensor system for your robot car? Please, describe your thoughts concisely.

Lab 10: Proficiency Exam

EE 302 Lab Proficiency Exam

Name: ______________________________ EID: ______________
Professor: ____________________________

The goal of this exam is to measure your abilities to build a circuit on a breadboard based on a circuit schematic and accurately measure currents and voltages from this circuit. The circuit you are to build is shown below:



Circuit Correct 1st Time?	
Circuit Correct 2nd Time?	
Circuit Correct 3rd Time?	
Circuit Correct 4th Time?	
Circuit Correct 5th Time?	

Build this circuit using the breadboard and resistors provided. Use the multimeter provided to fill in the table below. **The value of DC will be provided to you by your TA**.

Quantity	Value (include units)	Verification By TA
R_{eq} seen by the voltage source		
V		
I		

Notes

1. All resistors have a tolerance of ±5%
2. You may write any information that you want on this sheet. You CANNOT use MultiSim, the circuit simulation package
3. All values placed in the table above MUST BE CONFIRMED by TAs. NO EXCEPTIONS!
4. If you blow a fuse in the multimeter when taking a measurement, you will not receive any credit for that measurement or any measurements that would still need to be taken.
5. All resistors must be returned to the packet at the end of the exam.

Lab 11: Thevenin Equivalent Circuits

Due at the beginning of lab:
- **<u>Your completed Pre-lab.</u>** One copy is to be turned in to your TA. A second copy should be placed into your engineering notebook.
- **<u>Blank copy of the Lab Report.</u>** One copy of the lab report will be turned in and another copy will be in your notebook

Lab Goals:
- You will construct and verify Thévenin equivalent circuits using experimental measurements.
- You will practice measuring current and voltage using a myDAQ.
- You will calculate load resistance and derive the maximum power from a circuit.

Required Lab Material:
- Your lab notebook
- Wires for building circuits (available in lab)
- MyDAQ setting for Digital Multimeter (DMM) with connection probes
- The following resistors:

Quantity	Value
1	100 Ω
1	330 Ω
1	470 Ω
1	1 kΩ
5	1.5 kΩ
1	3.9 kΩ
1	5.1 kΩ
1	10 kΩ

Due at the end of the lab:
- A completed copy of your lab report. One copy is to be turned in and one copy is to be placed into your notebook.

Assignment 1: Construct the first circuit

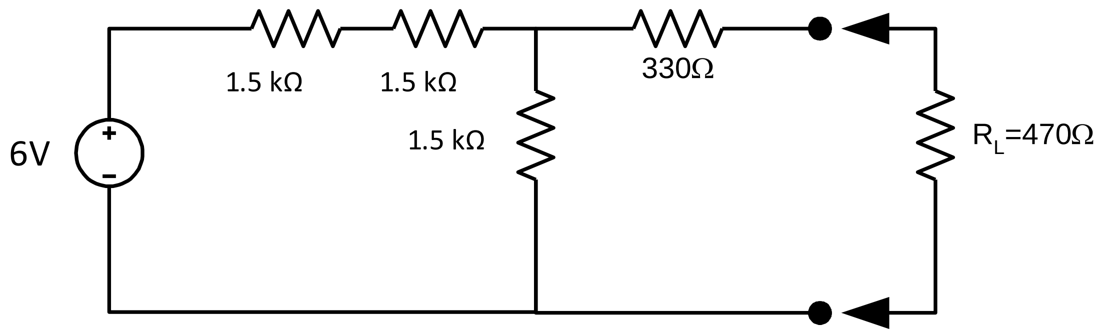


Figure 1: First Circuit for Circuits V Lab

1. Construct the circuit shown in Figure 1 on a breadboard. Provide the wiring needed to use the positive variable power supply.
2. Confirm with your TA that your circuit is correct and have your TA sign your lab report.
3. Set the power supply voltage to 6 volts. Use the vi you created from Lab 2.
4. Measure the output voltage across the load resistor R_L = 470 Ω and record your reading in your notebook and in Table 1 in your lab report.
5. Change the load resistor R_L to 100 Ω and measure the new output voltage. Record your reading in your notebook and in Table 1. **Remember that you need to turn off the power supply applied using the vi from Lab 2 each time you remove a resistor.**
6. Repeat Step 5 for load resistance values of 1 kΩ, 3.9 kΩ and 10 kΩ.
7. Turn off the power and remove the 10 kΩ load resistor from the circuit.
8. Turn on the power and measure the open circuit voltage (V_{OC}) and short circuit current (I_{SC}). Using these values calculate Thévenin voltage (V_{TH}) and Thévenin resistance (R_{TH}). Record the results in Table 1 of your lab report. *(Hint: to measure short-circuit current remember that the internal resistance of an ideal ammeter is zero ohms. To measure the short-circuit current, you can simply place the ideal ammeter across the load terminals.)*
9. Draw the equivalent circuit in the space provided in your lab report. Disassemble your circuit.
10. Using the V_{TH} and R_{TH} values construct a Thévenin equivalent circuit using the breadboard. Provide the wiring needed to use the positive variable power supply.
11. Add the load resistance R_L =470 Ω to your circuit.
12. Set the positive power supply to the value of V_{TH} calculated in Step 9. Measure the voltage across the load resistor. Record your reading in your notebook and in Table 2.
13. Using the Thévenin equivalent circuit what you found in step 8, change the load resistor R_L to 100 Ω and measure the new output voltage. Record your reading in your notebook and in Table 2. **Remember that you need to turn off the power each time you remove a resistor.**
14. Repeat Step 12 for load resistance values of 1 kΩ, 3.9 kΩ and 10 kΩ.
15. Disassemble your circuit.

Assignment 2: Construct the second circuit

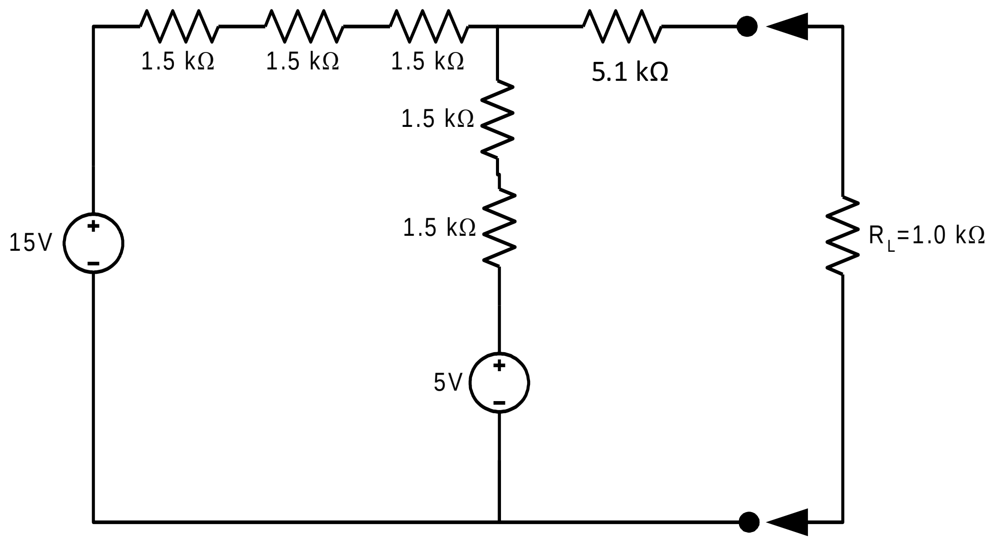


Figure 2: Second Circuit of Circuit V Lab

1. Construct the circuit shown in Figure 2 using the breadboard. For this step, use the standard **+15V and AO** channel on your myDAQ**. (Connect the ground only to the load resister!) Also NOTE: Do not use both the +15V and 5V preset sources. Use AO to generate the 5V.**
2. Confirm with your TA that your circuit is correct.
3. Turn on the power.
4. Measure the output voltage across the load resistor R_L = 1.0 kΩ. Record your reading in your notebook and in Table 3 in your lab report.
5. Repeat Step 4 for load resistance values of 100, 470 Ω, 3.9 kΩ, and 10 kΩ. Record your reading in your lab note section and in Table 3 in your lab report
6. Turn off the protoboard power and remove the 10 kΩ load resistor from the circuit.
7. Turn on the protoboard power and measure the open circuit voltage (V_{OC}) and short circuit current (I_{SC}). Using these values calculate Thévenin voltage (V_{TH}) and Thévenin resistance (R_{TH}). Record your readings and calculations in your notebook and in Table 3.
8. Draw the Thévenin equivalent circuit in your lab report in the space provided.
9. Using the measured output voltages recorded in this task, compute the power in the load resistor and record these values in Table 4. Plot the power versus load resistance on the provided graph.
10. At what value of load resistance is the maximum output power realized? Mark this value on your plot.
11. **Find the Thévenin equivalent for an unknown circuit**
12. You have been provided with a black box that contains an unknown circuit. Determine its Thévenin equivalent circuit. Record the values you measure in Table 5 of the lab report and in your notebook.
13. Draw this circuit in the space provided in your lab report.
14. Complete the questions in your lab report and turn in a copy to your TA.

FAQ's

Q: Why am I getting a notice saying I'm drawing too much current?
A: For circuit 2, you actually have 3 power supply terminals: +15, +5, and GND. Since the rails on the side can only accommodate only 2 supply voltages, you need to designate another node somewhere on your breadboard as the +5V source.

Q: Why aren't my values close to what I calculated?
A: Try adjusting the range settings on your multimeter to 20 V and 20 mA or so.

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Name: ____ EID: ________________

By placing my name and EID above, I am certifying that I determined the answer to the questions posed below and did not copy my answers from a fellow student.

*** Due at the beginning of your lab session ***

You will need 2 completed copies of this pre-lab. One is to be turned in to your TA at the beginning of the lab session. The other one is to be done in your lab manual.

Questions

1. Select the correct choice to complete the following two sentences:

   a. To find the Thévenin equivalent voltage of a circuit, one must measure the open circuit (i.e. no load) output (voltage / current).

   b. To calculate the Thévenin equivalent resistance of a circuit, all voltage source are replaced by a/an sources are replaced by a/an (open / short) circuit and all current (open / short) circuit.

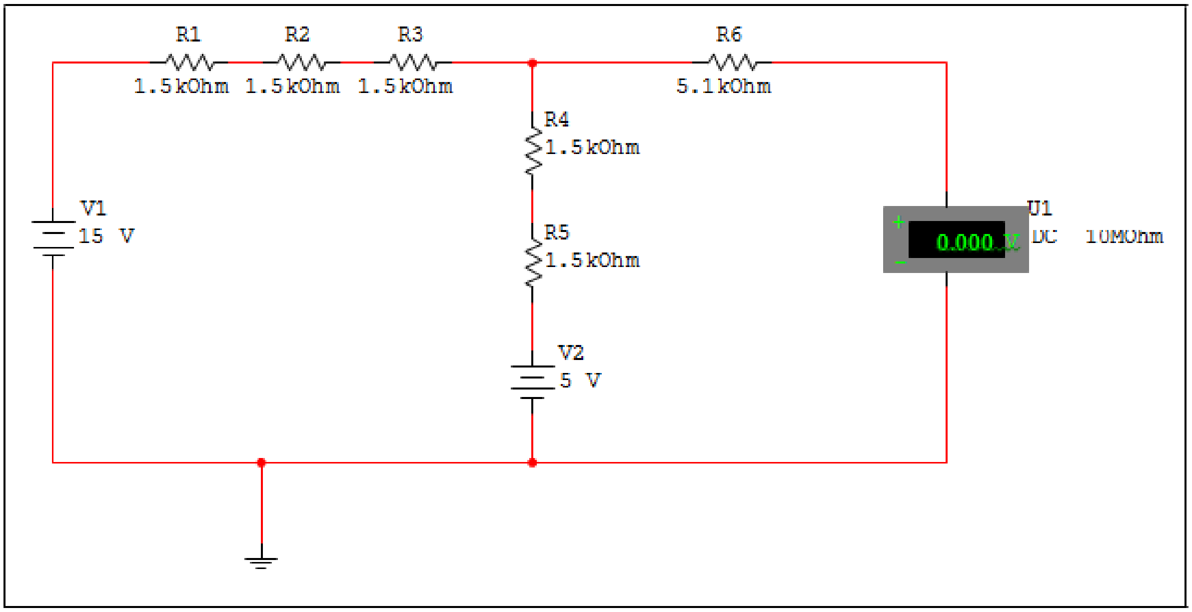
2. Derive and draw the Thévenin equivalent circuit (i.e. find Thévenin voltage and Thévenin resistance) for the circuit in Figure 4 of your lab manual, that has a load resistance of 470 Ω. (The 470 Ω resistor is the load resistor and should NOT be included in your Thévenin equivalent circuit calculations.)
   Show your work in order to get full credit.

3. Using Multisim, derive and draw the Thévenin equivalent circuit (i.e. find Thévenin voltage and Thévenin resistance) for the circuit of Figure 5. To use Multisim, you will need to enter the circuit of Figure 5 and use a virtual voltmeter to measure the open-circuit voltage. The diagram below shows what your simulation should look like. Double click on the virtual voltmeter and change the resistance to 10 MΩ. Print your simulated results showing your voltage measurement and attach this print to the Prelab. To complete the derivation of the Thévenin equivalent circuit, you will need to measure the short-circuit current using a virtual ammeter, measured at the output terminals. Print your simulated circuit showing the current measurement and attach it to the Pre-lab.

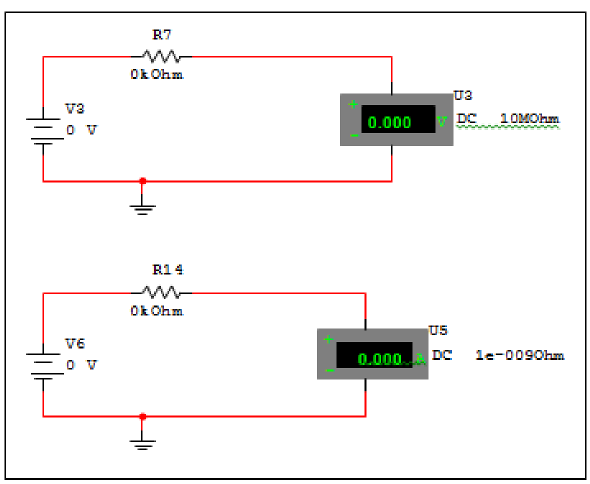
   Hints:

- *To measure short-circuit current remember that the internal resistance of an ideal ammeter is zero ohms. To measure the short-circuit current, you can simply place the ideal ammeter across the load terminals.)*
- *In Multisim, you can select the circuit you have entered, copy it, and paste to create an exact copy. You can then use the original circuit and the copy to simulate V_{OC} and I_{SC} simultaneously.*

Remember that $V_{TH} = V_{oc}$, the open-circuit voltage and $R_{TH} = V_{oc}/I_{SC}$, the short-circuit current.



4. Enter your Thévenin equivalent circuit into Multisim. Your circuit will look like the one below with your derived values for V_{TH} and R_{TH} substituted in the circuit. Using a virtual voltmeter and virtual ammeter, measure the open-circuit voltage and short-circuit current. Confirm that these measurements are identical to those measured in step 3. Print out your simulated results and attach them to the pre-lab.



Circuits VI - Thévenin Equivalent Circuits Lab Report

Submitted by (Print names)

Assignment 1: Construct the first circuit

1. TA Signature (Step 2): ____

Table 1: Quantities asked for in the assignment 1

Quantity	Value (include units)
Open Circuit Voltage V_{OC}	
Short Circuit Current I_{SC}	
Thévenin Equivalent Voltage	
Thévenin Equivalent Resistance	
Output voltage across R_L = 100 Ω	
Output voltage across R_L = 470 Ω	
Output voltage across R_L = 1 kΩ	
Output voltage across R_L = 3.9 kΩ	
Output voltage across R_L = 10 kΩ	

2. TA Signature (Step 8): ____

3. Draw the Thévenin Equivalent circuit you derived in the assignment 1 (Step 9).

Table 2: Quantities asked for in the assignment 1

R_L	Output Voltage (V)
100 Ω	
470 Ω	
1 kΩ	
3.9 kΩ	
10 kΩ	

4. TA Signature (Step 14): ____

Assignment 2: Construct the second circuit

1. TA Signature (Step 2): ____

Table 3: Quantities asked for in the assignment 2

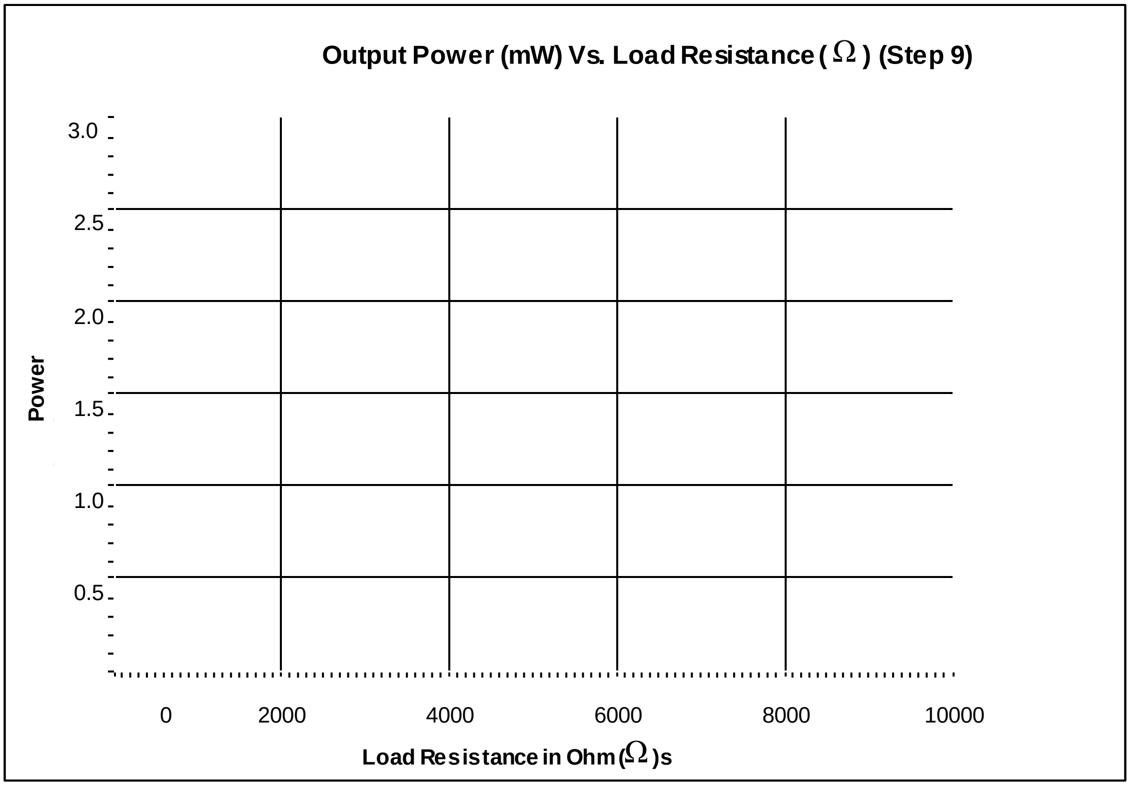
Quantity	Value (include units)
Open Circuit Voltage V_{OC}	
Short Circuit Current I_{SC}	
Thévenin Equivalent Voltage	
Thévenin Equivalent Resistance	
Output voltage across R_L = 100 Ω	
Output voltage across R_L = 470 Ω	
Output voltage across R_L = 1 kΩ	
Output voltage across R_L = 3.9 kΩ	
Output voltage across R_L = 10 kΩ	

2. TA Signature (Step 5): ____

3. Draw the Thévenin Equivalent circuit you derived in Task 2 (Step 8).

Table 4 Measured Voltages and Computed Power (Step 9)

	R_L	Output Voltage (V)	Power (mW)
Assignment 2	100 Ω		
	470 Ω		
	1 kΩ		
	3.9 kΩ		
	10 kΩ		



4. Record your measured values for Step 11 in the following table. Be certain to record the quantities measured and their units.

Table 5. Measurements for Unknown Circuit

5. Draw the Thevenin Equivalent Circuit in the table 5. (Step 13)

Lab 12: Final Car Project continued I

Due at the beginning of lab (1 for each group):

- **<u>Completed Gear box</u>**
- **<u>Completed pre-lab questions.</u>** One copy is to be turned in and one copy should be placed in your engineering notebook

Lab Goals:

- Continuing working on Robot Car Project

Required Lab Material:

- MyDAQ, L293D H-Bridge, 6V battery

Description

During this lab section, you will continue building and testing your Robot Car. Refer back to the Robot Car Guide in Lab 9 for instructions on the Robot Car.

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

Name: ______________________________________ EID: _______________

By placing by name and EID above, I am certifying that I determined the answer to the questions posed below and did not copy my answers from a fellow student.

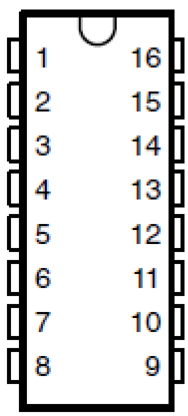
*** Due at the beginning of your lab session ***

You will need 2 completed copies of this pre-lab. One is to be turned in to your TA at the beginning of the lab session. The other one is to be done in your lab manual.

1. Review the H-Bridge: http://en.wikipedia.org/wiki/H_bridge

2. Review the L293D H-Bridge datasheet: http://www.parallax.com/Portals/0/Downloads/docs/prod/compshop/603-00019-L293D-Datasheet.pdf

3. Label pins' functions of the H-bridge below in a way that makes sense with the H-bridge from the previous problem:



4. Draw a breadboard connection of the H-bridge including myDAQ ports:

Lab 13: Final Project continued II: Training Module 2 (Optical Theremin)

Due beginning of lab (1 for each group):

- **<u>Completed pre-lab questions.</u>** One copy is to be turned in and one copy should be placed in your engineering notebook.
- **<u>Blank copy of the Lab Report.</u>** One copy of the lab report will be turned in and another copy will be in your notebook
- Download Labview Vis for Path Logic

Lab Goals:

- Learn about the functionality of an Op Amp.
- Learn about the behavior of a photodiode.
- Implement Path Logic

Equipment needed:

- 2 x Photodiode
- 2 x 741 Op-Amps
- 4 x 5.1MOhm resistors
- NI myDAQ kit

Due at the end of lab:

- Turn in your completed lab report to your TA.

** **Each group needs to turn in Final Report by the last day of lab. Please, also refer BB and appendix C for the report.**

Background Knowledge: Photodiodes

A photodiode converts light into either current or voltage, depending on the mode of operation. When a photon of sufficient energy strikes the diode, it excites an electron (photoelectric effect).

Below is a circuit diagram for a photodiode:

Review the photodiode data sheet for more information on how to wire the photodiode and its characteristics.

http://www.optekinc.com/datasheets/OP950.PDF

Background Knowledge: Using the Optical Theremin VI

On the myDAQ Optical Theremin VI use the Max and Min Pitch knobs to set the highest and lowest frequencies to be generated. Use the Max and Min Level sliders for Pitch and Gain to configure the amplitude range that will be seen by each sensor. The program scales the output frequency range and gain to be equally distributed across the amplitude ranges for those respective inputs.

To configure visual pitch markers select the notes your want to display in the Pitch Markers array control. To configure auto-tune select the notes you want to tune to in the AutoTune Pitches array control. Enable auto-tune with the AutoTune toggle switch.

Assignment 1: Building the Circuit

Build two instances of the photodiode amplifier circuit as show in Figure 1:

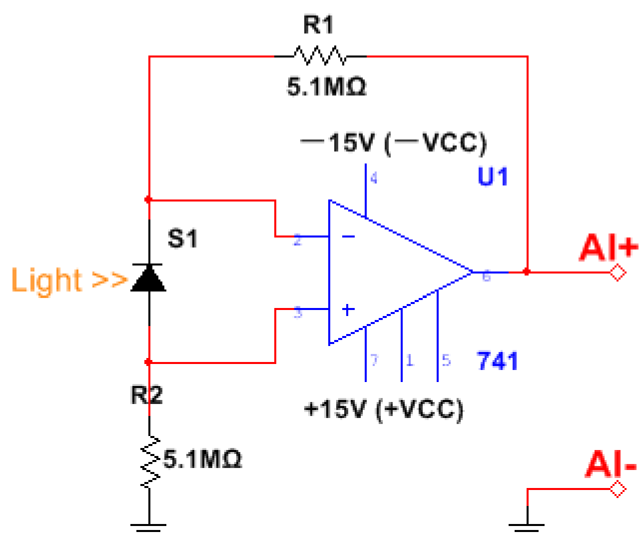
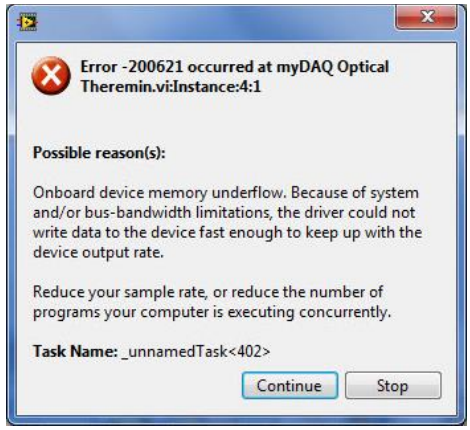


Figure 1: Photodiode Amplifier Circuit Diagram.

1. Use the +15V, -15V, and GND sources on the myDAQ to power the circuit.
2. The input for AI0- and AI1- should both be tied to ground.
3. The output of the amplifier circuit to control pitch should be connected to AI0+ and the amplifier circuit to control gain should be connected to AI1+.
4. Call your TA over to verify your circuit.
5. Download the Optical Theremin VI from blackboard.
6. Remember to read the Background Knowledge section on using the program.

**Note: The DAQ Assistant is configured for the myDAQ unit to be named “Dev1”. If you have configured your myDAQ differently you will need to modify both DAQ Assistant VIs.

Also, the DAQ Assistant VI performing the analog output is configured by default to generate at 100kS/s and write 6.5k samples to the myDAQ at a time. Depending on the performance of your computer you may experience a buffer underflow error -200621 because the program cannot generate data fast enough for the myDAQ to maintain continuous analog output.



If you experience this error you will need to modify one, or both, of the following settings:

1. **Increase** the "**Samples to Write**" in the second DAQ Assistant VI and **increase** the "**Number of Samples**" in the Simulate Signal Express VI. Both of these values should be the same.
2. **Decrease** the "**Rate (Hz)**" in the second DAQ Assistant VI and **decrease** the "**Samples per Second**" in the Simulate Signal Express VI. Both of these values should be the same.

It will take a little bit of fiddling to figure out which settings work best with your computer. You want to have the largest sampling rate and smallest number of samples to write as possible because it will minimize the system delay from input to output.

1. Test your system according to Figure 2

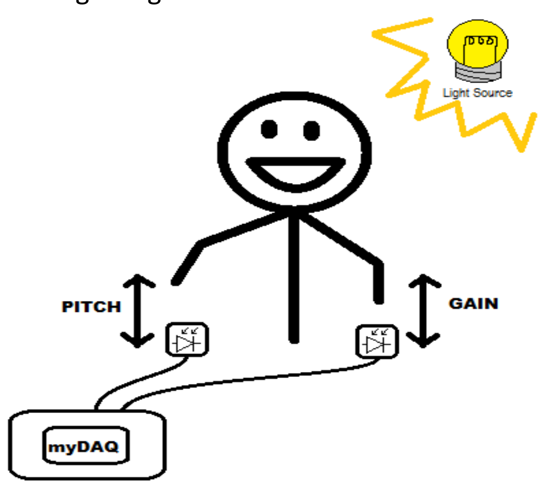


Figure 2: Usage Diagram

2. When you are done, call a TA over to sign you off.

TA Signature:____

Acknowledgement: A special thanks to Liz Savage (Employee of NI) for the example optical theremin circuit.

Lab Note

Date . . 2016

Important concepts / Key ideas
Procedure / DATA
Try and error / Thoughts

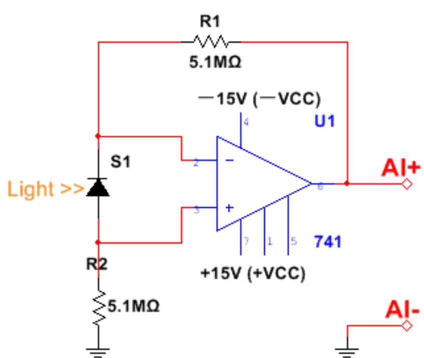
Name: ______________________________ EID: ______________

By placing by name and EID above, I am certifying that I determined the answer to the questions posed below and did not copy my answers from a fellow student.

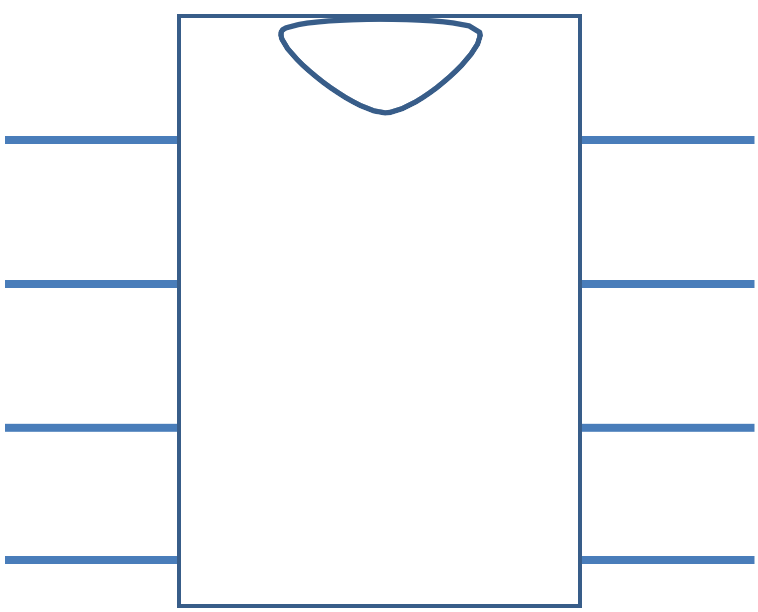
*** Due at the beginning of your lab session ***

You will need 2 completed copies of this pre-lab. One is to be turned in to your TA at the beginning of the lab session. The other one is to be done in your lab manual.

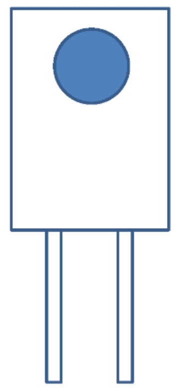
1. Review the photodiode datasheet: http://www.optekinc.com/datasheets/OP950.PDF
2. Review the LM741 OpAmp datasheet: http://www.national.com/ds/LM/LM741.pdf
3. Draw a breadboard connection of the following (including myDAQ ports):



4. Label the OpAmp below in a way that makes sense with the OpAmp from the previous problem:



5. Label each leg of the photodiode as either cathode or anode and indicate where it should be connected on the Op Amp.



6. (Path Logic) Describe your group's path logic or algorithm. Please, specify the sensor types (Photoresistor, IR) and those Input/Output ports in your myDAQ. You may mention the number of myDAQs which you need. Moreover, please, describe your labview algorithm which combines sensor signals and motor controls.

5.0 Appendices

Appendix A: Multimeter Basics

What is a Multimeter?

A meter is a measuring instrument. An ammeter measures current, a voltmeter measures the potential difference (voltage) between two points, and an ohmmeter measures resistance. A multimeter combines these functions, and possibly some additional ones as well, into a single instrument.

Typically it has three ports; one for measuring voltage and resistance, one for measuring current while the rest is used as a common port for measuring all of the above.

Most of the multimeters used nowadays are digital multimeters and they can display measurements up to one or two digits after the decimal point.

☠MAINS VOLTAGE☠

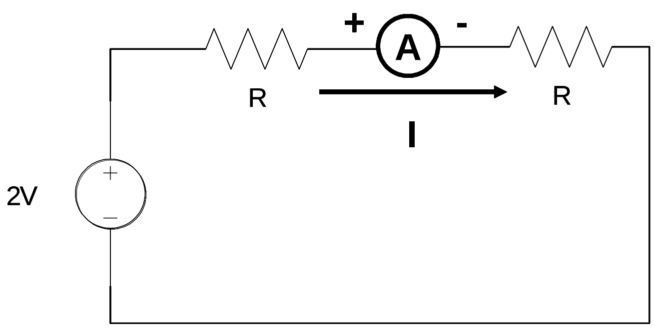
For safety reasons, you must NEVER connect a multimeter to the main supply.

Measuring Current

To start with, you **may** need to **break the circuit** so that the **multimeter can be connected in series**. All the current flowing in the circuit must pass through the ammeter. An ammeter has a LOW resistance. We can use the multimeter as an ammeter **by rotating the knob and setting it in the current measuring portion of the multimeter.**

Now we must connect one end of the 'red' wire to the port with 'mA' or 'A' sign. The other end of this wire should be connected to the 'current entering node' (red node). The 'black' or 'white' wire should be connected to the multimeter port with 'COM' sign. The other end of this wire should be connected to the 'current leaving node' (black node).

A schematic diagram showing the placement of the ammeter is shown below.

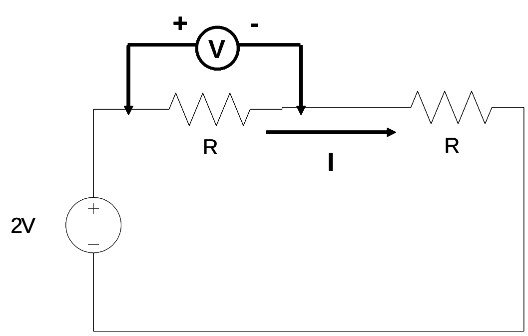


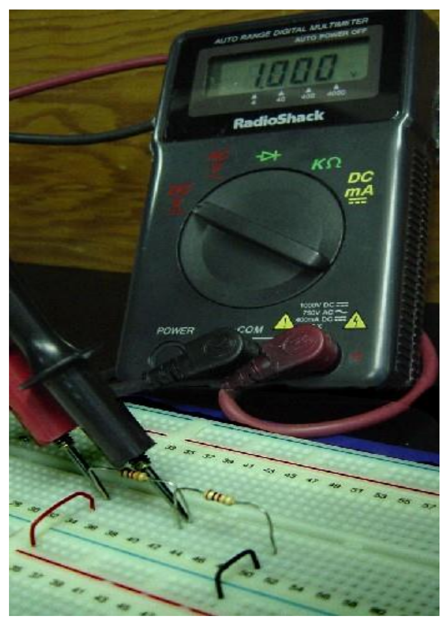
Measuring Voltage

For voltage measurements, you **do not need to break the circuit**. To measure *voltage drop* across a resistor in an electrical circuit, one **must connect the Multimeter in parallel** with that particular resistor. In this case, a multimeter behaves more like an actual voltmeter. A voltmeter has a very HIGH resistance inside it. We can use the multimeter as a voltmeter by **rotating the knob and setting it in the voltage-measuring portion of the multimeter**.

Now we must connect one end of the 'red' wire to the port with 'V' sign. The other end of this wire should be connected to the 'current entering node' (red node) of the resistor. Now the 'black' or 'white' wire that often represents 'Neutral' should be connected to the multimeter port with 'COM' sign. The other end of this wire should be connected to the 'current leaving node (black node) of the re sistor.

A schematic diagram showing the placement of the voltmeter is shown below.



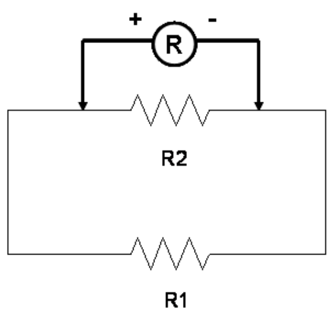


Typically, voltage measurements are used much more often than current measurements. The processing of electronic signals is usually thought of in voltage terms. It is an added advantage that a voltage measurement is easier to make. The original circuit does not need to be changed. Often, the meter probes are connected simply by touching them to the points of interest.

Measuring Resistance

An ohmmeter does not function with a circuit connected to a power supply. If you want to measure the resistance of a particular component, you **must take it out of the circuit altogether and test it separately**. A multimeter works by passing a current through the component being tested. The pictures below show the resistance of 2 – 1 k resistors in parallel.

A schematic diagram showing the placement of the ohmmeter is shown below.



Appendix B: Contribution Chart for Final Project

Check each box indicating who contributed to which sub-modules and place this chart in your final report. You may download the electronic version in BB.

	Group Member		Motor Control	Collision Detection	Path Detection	Other(Specify):
Member 1						
Member 2						
Member 3						
Member 4						
Member 5						

Now, out of a raw score of 100, assign each group member a grade based on your perception of how much he/she contributed. Please be honest and explain, especially if you feel like a certain group member is over- or under-deserving.

	Member 1	Member 2	Member 3	Member 4	Member 5
Contribution					

I, the undersigned, certify that this information stated above is accurate and completed to the best of my knowledge.

Student Name:____

Student Signature:____