LABORATORY 1: LTSpice/Analog Discovery Introductory Circuits

PART A – Introduction to LTSpice

Materials need: Laptop

Choose a partner:
1) Pick a partner. Remember, you will likely have to work with this person for the rest of the semester
2) Only one laboratory report is needed for each pair of partners. Make sure you include both names on the front of the report. Also remember to add your partner to Gradescope. (Only one person needs to submit it).
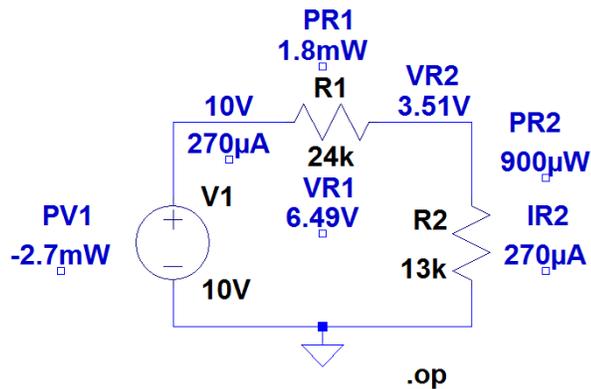
Installation:
1) LTSpice
   a. Use the link http://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html
      i. Or find (LTSpice Download) on the Circuits webpage under Resources.
   b. Run the executable and follow the default installation.

LTSpice Starting a new projects:
1) Start "LTSpice XVII" (or earlier version)
2) Start a new Project under the File -> New Schematic
    a. Make sure your files are saved in a convenient directory. The root directory (C:\) or Desktop are probably not good choices. I would suggest creating a directory "C:\Circuits" and saving your work there.
    b. Pick an appropriate name (Laboratory_01 is a good choice)
3) To place components:
    a. The first time you start, click Edit in the task bar. Here you'll find symbols and their matching labels.
    b. You can choose to go down the menu and click Component or simply use the Toolbar above the schematic frame from this point on OR
        i. There are shortcut keys for adding components as well.
        ii. http://www.analog.com/media/en/simulation-models/spice-models/LTspice_ShortcutFlyerC.pdf?modelType=spice-models

A1: Bias Point Measurements for Resistive Circuits
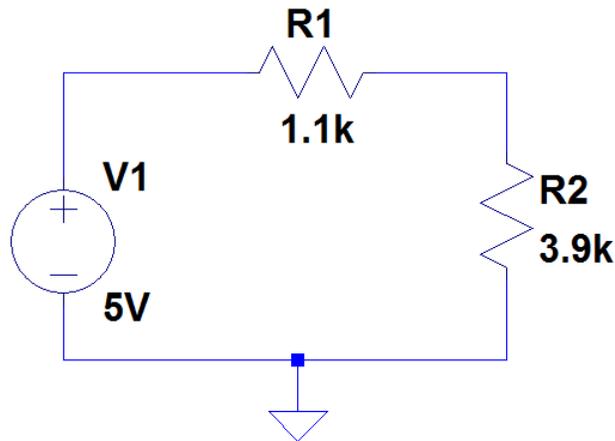
1) Create the circuit above in LTSpice and display the current, voltage and power values for the circuit similar to the diagram below. *The sample image below has different component values but gives an idea of what you need to show.*

PR1
1.8mW
R1   VR2
10V   3.51V
270µA   PR2
24k   900µW
V1   VR1
PV1   6.49V   R2   IR2
-2.7mW   13k   270µA
10V

.op

*Note: If you are confident that you can do this **without step-by-step instruction you can skip to A2.** Step-by-step instructions are below.*

2) Find and Add the Component (shortcut "F2"), VDC, by typing "voltage". Press "OK".
   a. A DC voltage source should follow your cursor.
   b. Left click to place. Press ESC to keep from duplicating. (If you make a mistake, you can press CTRL-V to cut the component).
   c. Move your cursor over the component. When a hand pointing left appears, right click, to edit the value of the DC voltage. Change the value to 5V.
3) Add resistors. (shortcut "R"). Press CTRL-R to rotate. Right click to place and ESC to keep from duplicating.
   a. They should be default labeled R1 and R2.
   c. Change values of R1 and R2 to 1.5k and 3.2k respectively by right clicking the component.
4) Connect the DC source and the two resistors in a closed loop by placing wire connections between each component. (shortcut "F3") Click on one of the square boxes connected to a circuit component and then move the mouse and click on another square box to place a wire connection.
5) Add a ground connection. (shortcut "G") You MUST use a ground in every circuit you simulate. Be sure to wire this as well.

Written by J. Braunstein   *Revised by S. Sawyer Fall 2018: 8/23/2018*
Rensselaer Polytechnic Institute   *Troy, New York, USA*

3

Setting Operating Point DC Analysis

1) To add operating point analysis (.op) you have a few options
    a. Click ".op" in the tool bar on the far right and add text ".op" in the text box as shown below
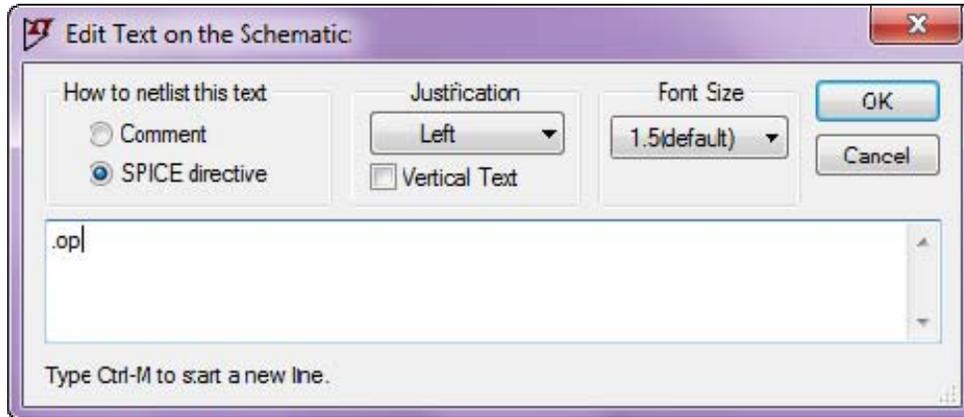
    b. Press shortcut "t" on your keyboard then change radio button to "SPICE directive" then add text ".op" in the text box below it
    c. Press "s" on your keyboard which automatically selects "SPICE directive" then add text ".op" in the text box below it
2) Place .op anywhere near the circuit
3) Run the simulation. (Click the "Running person" in the toolbox.)
4) Results will appear in a dialog box.
5) Label the circuit with voltage, current, and power by placing operating point data labels.
    a. FOR VR2:
        i. Right click on the wire to the right of R1 (this is the voltage above R2, Vn002 to ground which is the voltage across R2, VR2).
        ii. Right click to change the number of significant figures. For two decimal places use round(***data label****100)/100. For three significant figures multiply and divide by 1000. For four 10000 etc.
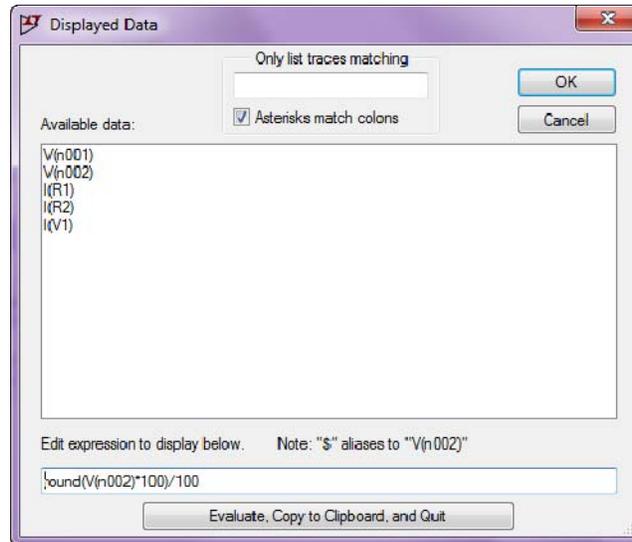
      iii. Press ok, and move the label anywhere convenient using the "Move" hand on the toolbar

  b. FOR IR2

      i. Right click on the wire to the right of R1 again. It will show voltage but you will change it by right clicking on it and choosing I(R2) which will show in the text box

      ii. Be sure to round this value to significant figures.

Note: LTSpice assumes conventional current flows from positive to negative terminal of the power supply (pointing down). Therefore all resistors current will be labeled as negative in LTSpice. To make positive simply add a negative in front of I(R1) and I(R2).

  c. For PR2

      i. An equation can be written P=VI with rounding to significant figures is included round(I(R2)*V(n002)*10000)/10000

  d. Add text above all the numerical labels by pressing "t". The radio button should automatically be on "Comment". Type VR2 in the text box for example and place near the value in the figure.

  e. Repeat this process for the voltage current and power for R1 and power for the voltage source in your figure. ***Remember VR1 is the difference between points V(n001) and V(n002) in the diagram. Also remember power supplied must equal power absorbed by the resistors!***

Results

1) Include a screen shot of your results in your report. Report template can be found https://ecse.rpi.edu/~ssawyer/CircuitsSum2018_all/Templates/

A2: Making Changes
1) Change R1 to 15k and R2 to 32k. Rerun the simulation
    a. Comment on the new voltage and current measurements. Are the results consistent with expectations? Why?
    b. Did the power levels go up, down or stay the same? Why?
    c. Include a screen shot of your results in your report.
2) Replace the voltage source with a 5mA current source. (Component "current")
    a. What can you say about the current as R1 and R2 change?
    b. What happens to the power if both resistors are reduced back to their 1.5k and 3.2k values (from the 15k and 32k values)?
    c. Include a screen shot of your results in your report.

A3:   Unlimited-All Access Homework Solutions and Practice Problem Solutions!

1) Use LTSpice to implement the circuit in HW1, problem 5 (shown below but values are in HW1).
2) https://ecse.rpi.edu/~ssawyer/CircuitsFall2018_all/Homework/Unit1/HW01. pdf
3) Using LTSpice, find the current through R5. When you complete HW1, compare your calculated answer with your simulated answered. (The LTSpice simulation does not constitute a solution when submitting HW1. When you hand in HW1, you must include your work demonstrating that you can obtain the solution by hand.)
4) Discuss how to find answers to ANY problem (team assignments, homework, previous exams, problems from other books), when you don't have an answer key.
5) Include a screen shot of your results in your report.
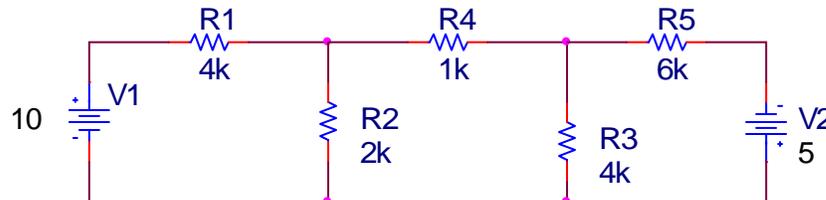
Note: This circuit does not include a ground. In order to run a LTSpice simulation, your circuit must have a ground. Add a ground at an appropriate node. It will not affect the current calculation.

6) Change the location of your ground (move it to a different node) and verify that the current through R5 has not changed.

Written by J. Braunstein   *Revised by S. Sawyer Fall 2018: 8/23/2018*
Rensselaer Polytechnic Institute   *Troy, New York, USA*

7

Part B: Analog Discovery Introductory Circuits
Materials need: Analog Discovery Board and Circuits Kit, Resistors

Installations:
1) Analog Discovery Board 1 (or Analog Discovery Board 2)
   a. In a web browser, open up the page
      https://store.digilentinc.com/waveforms-previously-waveforms-2015/
      Click on the link, "DOWNLOAD HERE"
   b. Follow the default install. If it asks about any Microsoft runtime libraries, let it install them.
   c. If you are using Windows 7, then open the Start Menu and find the Digilent tab and start the Waveform application. If you are using Windows 8 or Windows 10, find the Waveform application using the search tool and start it.
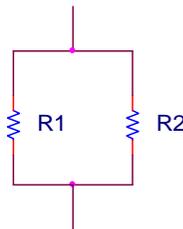
Background Circuit Concepts:
1) Series Resistors

Resistors in series 'add' to an equivalent total resistance. For the two resistors shown above, the equivalent resistance is $R_{EQ} = R1 + R2$. For N resistors in series, this expression extends to $R_{EQ} = R1 + R2 + \cdots + RN$

2) Parallel Resistors

Resistors in parallel have an inverse relationship. For the two resistors shown above, the equivalent resistance is $\frac{1}{R_{EQ}} = \frac{1}{R1} + \frac{1}{R2}$. More generally, this expression is written as $R_{EQ} = \left(\frac{1}{R1} + \frac{1}{R2} \right)^{-1}$. For N resistors in parallel, this expression extends to $R_{EQ} = \left(\frac{1}{R1} + \frac{1}{R2} + \cdots + \frac{1}{RN} \right)^{-1}$

Discovery Board basics:
1) Plug in the Discovery board using the USB connector
2) Start the Waveform applications (the Welcome tab should open).
3) In the window that opens, click on the Supplies icon (the third icon). You should see a DC power control window open.
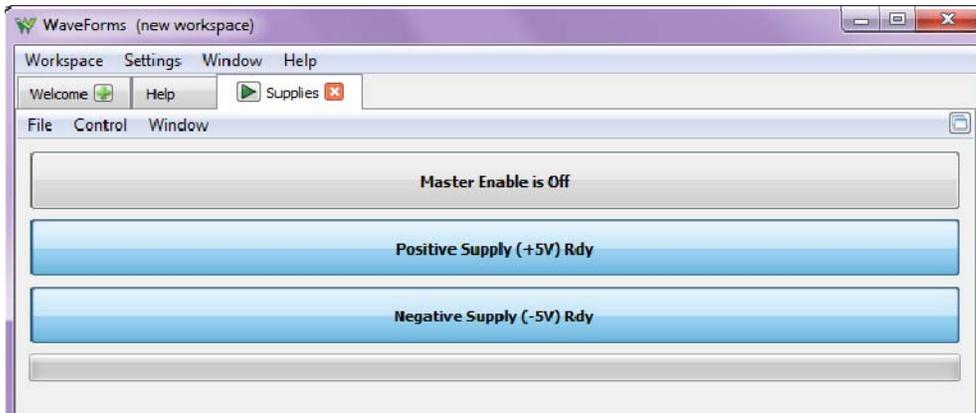
    a. Click the Master Enable is Off button to turn DC voltage on. It should now say Master Enable is On and be highlighted in blue.

    b. This provides access to a 5V/-5V supply using the V+/V- connections on the Discovery board.

    c. You find the connections and see that they are the solid red wire and the solid white wire, respectively.

4) To find Voltmeter, go back to the Welcome Tab

    a. Choose the forth icon, Voltmeter, and a new window should open.

    b. In the upper left corner of the Voltmeter window, click the Run box so that they you can use the channel inputs for digital measurements.

    c. For now, we only be using the DC measurements.
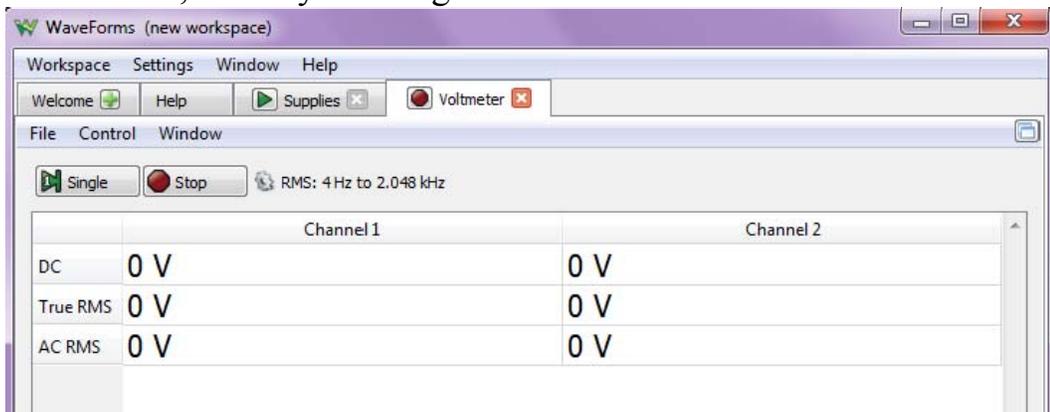
d. There are two input channels, Channel 1 and Channel. They are represented as 1+/1- and 2+/2- on the discovery board.
e. Channel 1 is the solid orange wire (1+) and the orange wire with a white stripe (1-).
f. Channel 2 is the solid blue wire (2+) and the blue wire with a white stripe (2-).
g. Each channel needs two connections, to measure across a circuit component (equivalent to using a voltmeter)

B1: Simple Check - Power Supply Measurement

As an initial experiment to make sure everything is working correctly, we will measure the DC voltage using the channel 1 inputs.

Hardware
1) Connect six pin header to your proto-board, making sure that each pin is connected to a different row.
2) Connect the V+ (red wire) to one of the header pins and a ground (any of the black wires) to another pin.
3) Connect another six pin header to a different area on the proto-board. (This is for convenience, if you really want you can use only one six pin header).
4) Connect the 1+ (orange wire) to a header pin and the 1- (orange white stripe wite) to another header pins.
5) On your proto-board, connect the V+ row to the 1+ row and connect the ground row to the 1- row.

On the Voltmeter (computer)
1) Check the DC measured voltage. If everything is connected correctly and your software is setup, you will see a measurement of approximately 5 [V]

B2: Voltage Divider/Series Resistors

First off, we will look at a simple voltage divider. Build the following circuit
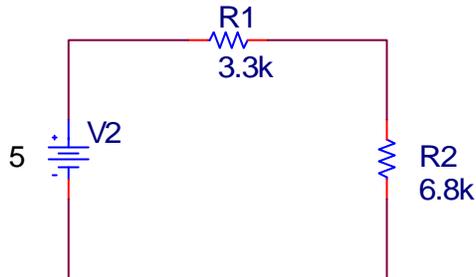
In this circuit, the positive side of the voltage source is the V+ connection (red wire) and the negative side of the voltage source is any of the grounds (black wires).

We will use the Discovery Board to make voltage measurements across each resistor.

Connect 1+ (orange wire) and 1- (orange white stripe) across R1.

Voltage across R1: _____

Connect 2+ (blue wire) and 2- (blue white stripe) across R2.

Voltage across R2. _____

   a. Are these voltages consistent with expectations from KVL? In other words, does the voltage around the closed loop sum to zero? Note, the polarity of your connections matters. 1+ and 1- correspond to your 'guessed' polarity for Channel 1. Similarly for Channel 2.

Replace R2 with a much larger resistor (100kΩ or more). Measure the voltage across R1 and R2 again. In this case we can <u>symbolically</u> say, VR2 ≈ _____

   b. With that in mind, circuit reduction in this simple circuit would result in an equivalent resistance when R2 >> R1, then REQ = R1+R2 ≈_____ (symbolically, not numerically).

B3: Effects of Measurement Equipment
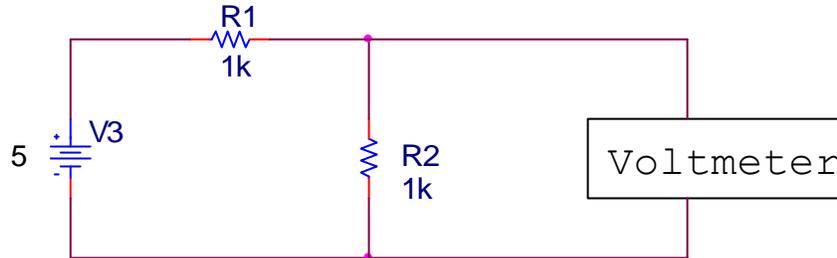
1) Build the simple voltage divider circuit shown above. In a more complete model, the Voltmeter is actually part of the circuit. Ideally it has no affect on the circuit.
2) Measure the voltage across R2.
    a. Does the measurement agree with expectations for the voltage divider circuit?
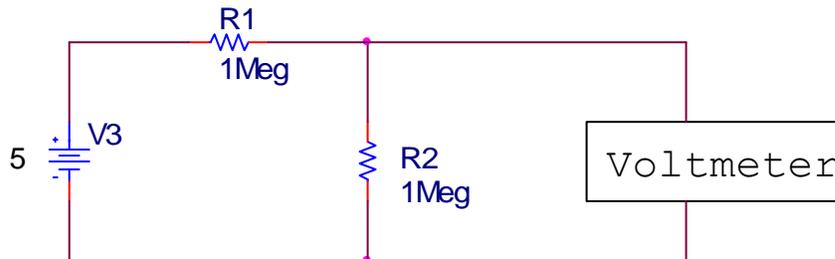
3) Replace the 1k resistors with 1Meg resistors and measure the voltage across R2 again.
    a. Does the measurement agree with expectations? What affect did the Channel 1 probes (the Voltmeter) have on the circuit? Why?

Written by J. Braunstein   *Revised by S. Sawyer Fall 2018: 8/23/2018*
Rensselaer Polytechnic Institute   *Troy, New York, USA*

13

Part C: Alpha Laboratories Applications

Can you start the design of a game controller? The full context of the circuit design below can be found in the introduction of Beta Laboratories Design Stage 1 found in the link below.
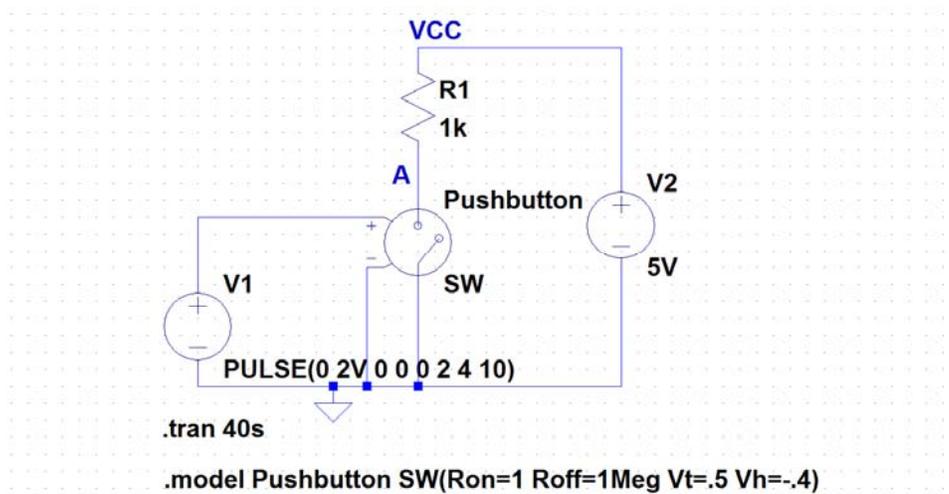https://ecse.rpi.edu/~ssawyer/CircuitsFall2018_all/Labs/Unit1/BetaDS1.pdf

C1: Application of Ohm's Law - Push and Pull

To create a game controller you need push button switches. Push button switches are a way to momentarily open and close a circuit. Often, pull-up resistors are used with push button switches to make sure power and ground are never shorted while ensuring discrete high and low output states (rather than some ambiguous state in the middle).

To model a push button on the game controller in LTSpice follow the steps using the schematic below:
1) Find and place the voltage controlled switch component, "sw". *Note: To mirror or flip a component horizontally, press "CTRL-E".*
2) The physical act of pushing the button is modelled with voltage pulses. Find the "voltage" component and place in the schematic as shown below. Right click the voltage source and click "Advanced". Select "Pulse". You are free to write any reasonable values into the text box list on the left.
3) The switch model must be defined using a SPICE directive. Follow instructions found http://www.analog.com/en/technical-articles/ltspiceiv-voltage-controlled-switches.html *Note: If you are getting an error that "LTSpice cannot find model definition ___", remember you've flipped the component so the value or name of the switch is now in a different position.*

VCC

R1
1k

A
Pushbutton

V2

V1

SW

5V

PULSE(0 2V 0 0 0 2 4 10)

.tran 40s

.model Pushbutton SW(Ron=1 Roff=1Meg Vt=.5 Vh=-.4)

4) Using Ohm's law, determine the appropriate value of R1 to produce 1mA of current through the push button when the switch is closed. Write the calculation.
5) Provide the plot of the current through R1 as the pushbutton is pushed (voltage is pulsed to open and close the switch).
6) Build the circuit with the Analog Discovery board. Remember, the voltage pulse is only simulating your finger pushing the button so you don't need it in your physical circuit. Compare your experimental results to the LTSpice results.
7) Questions:
   a. Describe what happens to the voltage at point A when the switch opens and closes.
   b. Discuss the important factors to consider when choosing the value of resistance for the pull-up resistor. (Can the resistor be too large or two small? What happens in each case? Are there other important conditions that affect your design choice?)

C2: Application of Voltage Dividers –Reference Voltage

You have a 5V source and two resistors. Design a circuit using these components that will provide 2.7V reference for the next part of the circuit using LTspice. (The next part of the circuit is arbitrary for now.)
1) Show the bias point analysis results.
2) Build the circuit and measure using the Analog Discovery Board. Provide screen shot of the results.

VCC = 5V

R1

R

Ref voltage

Next Circuit

R2

R

Written by J. Braunstein   *Revised by S. Sawyer Fall 2018: 8/23/2018*
Rensselaer Polytechnic Institute   *Troy, New York, USA*

16