ECE 220 Laboratory 2

Mesh and Node Analysis, AC Sources, and d'Arsonval Meters

Michael W. Marcellin

Please follow all rules, procedures and report requirements as described at the beginning of the document entitled ECE 220 Laboratory 1.
Always wear your safety glasses when performing your lab experiments.

Prelab Part 1

This part and the next part reinforce the circuit analysis techniques that we have learned in class.

Use mesh analysis to solve the circuit shown below in Figure 1. For the purpose of your analysis, assume that the LED is forward biased (turned on). As before, we will approximate its behavior by a constant voltage drop of 2.1 V. Hint: only two equations and two unknowns are required.

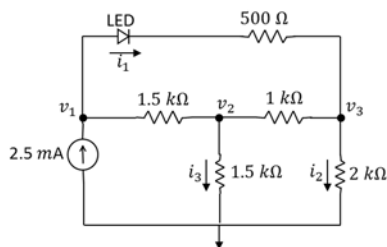a) Calculate i_1, i_2, i_3, v_1, v_2, and v_3.

Figure 1. Solve using mesh analysis.

We will now verify your calculations using PSpice. Start PSpice and create a new project as you did for the PSpice homework and Lab 1. Draw the circuit from Figure 1, as shown below in Figure 2. Again, we use three D1N4002 diodes in place of the LED. Note that the bottom node in Figure 1 has been labeled as the reference node (0 V). As discussed in the PSpice homework and in class, any essential node can be chosen as the reference node. The PSpice ground should be thought of as the reference node. This can be seen via a comparison of Figures 1 and 2.
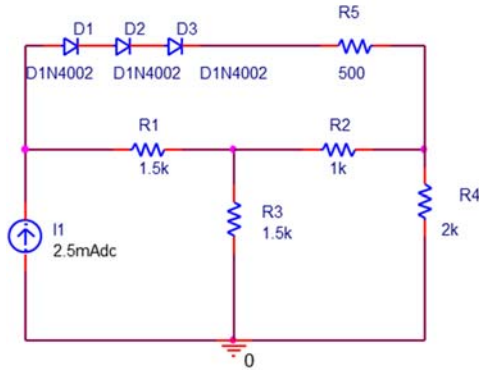
Figure 2. PSpice schematic.

We will use PSpice to find the DC voltages and currents in the circuit, as we did in the PSpice homework.
Click
PSpice -> New Simulation Profile
Enter a name for this profile
Click Create
A new window will open. You might have to look at the icons at the bottom of the screen to find it.
In the drop down menu entitled Analysis type:, choose Bias Point
This tells PSpice that we are going to compute the static DC values of voltages and currents in the circuit.
Check the box below the drop down menu that says "Save Bias Point."
Click "OK"
Click
PSpice -> Run
A new window will open. You may need to look at the icons on the bottom of the screen to find it. If there are no errors, you may close this window and return to your schematic.

Near the top of the screen there are 2 green buttons, labeled **V** and **I**, respectively. Use these buttons to view the currents and voltages in your circuits. The values should be slightly different than your calculated values.

b) Print a copy of your schematic with the voltage labels turned on (File -> Print).
c) Print a copy of your schematic with the current labels turned on (File -> Print).
d) Why do the PSpice values differ (slightly) from your calculations?

Prelab Part 2

Use node analysis to solve the circuit shown below in Figure 3.

Hints:
1) Again, only two equations and two unknowns are required.
2) Recognize that $5\sin(200\pi t) = 5\cos(200\pi t - \pi/2)$ and use complex phasor analysis.

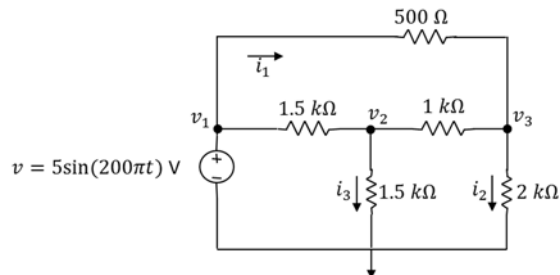a) Calculate $i_1(t)$, $i_2(t)$, $i_3(t)$, $v_1(t)$, $v_2(t)$, and $v_3(t)$.

Figure 3. Solve using node analysis.

Again, we will verify your calculations using PSpice. Start a new PSpice project as you have done before, and draw the circuit of Figure 3. The PSpice part name for the sinusoidal voltage source is VSIN. Set VAMPL=5, and FREQ=100, which is consistent with the sin in Figure 3. In that figure the angular frequency $\omega = 200\pi$ radians/second. You must input the frequency to PSpice in Hz. Since $\omega = 2\pi f$, we have $f = 100$ Hz. VOFF is an optional DC offset. Set this to 0. Add voltage markers (PSpice -> Markers -> Voltage Level) at each of the three nodes of interest, as shown in Figure 4 below.
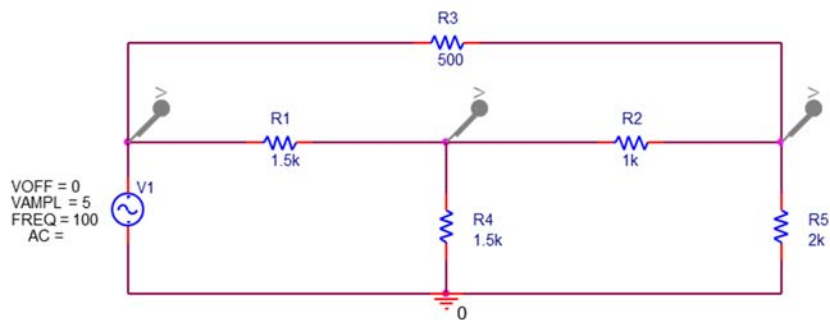

Figure 4. PSpice schematic.

We will use PSpice to plot $v_1(t)$, $v_2(t)$, and $v_3(t)$. We have not done this type of analysis in PSpice before.
Click
PSpice -> New Simulation Profile
Enter a name for this profile
Click Create
A new window will open. You might have to look at the icons at the bottom of the screen to find it.
In the drop down menu entitled Analysis type:, choose Time Domain (Transient)

This tells PSpice that we are going to create a time domain plot of the values associated with the markers in the circuit.
Enter the following parameters:
Run to time: 25m
Maximum step size: 0.1m
Click "OK"
This tells PSpice we want to create 25 miliseconds worth of data for our graph, with data points no more than 0.1 miliseconds apart.

Click
PSpice -> Run
A new window will open. You may need to look at the icons on the bottom of the screen to find it. This window should contain a time plot (of duration 25 ms) showing the three voltages associated with the markers. These plots should agree with your calculations in a) above.

b) Print the plot.

Prelab Part 3

In class, we learned how to make a simple (but not very good) voltage meter using a d'Arsonval meter movement. The circuit diagram for such a meter is shown below in Figure 5. This meter includes (of course) a d'Arsonval meter movement and a resistor. The purpose of the resistor is to limit the current through the meter movement. The meter movements that we have in the lab have a maximum current of $i_{max} = 1\ mA$. The internal resistance of the meter movements is $R_d = 155\ \Omega$. You are to design a volt meter that gives full scale reading (100% deflection of the meter needle) at 5 V.

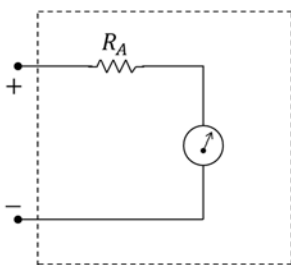a) Calculate the required value for R_A.

Figure 5. Voltage meter.

The amount of deflection of the meter needle will now indicate the voltage being measured. Since we have designed for full (100%) deflection at 5 V, 25% deflection indicates 1.25 V, 50% deflection indicates 2.5 V, etc. It is worth noting that the current flow is actually what causes the needle deflection. Since $i_{max} = 1\ mA$, 0.25 mA will cause 25% deflection, 0.50 mA will cause 50% deflection, etc.

We will now verify the operation of your meter design using PSpice. We will "connect" your

meter to a voltage source. We will then sweep the value of the voltage source from 0 to 5 V and see that the current through the meter movement varies linearly from 0 to 1 mA.

Create a new project in PSpice and draw the circuit shown in Figure 6 below. We model the meter movement as a resistor with value R_d.
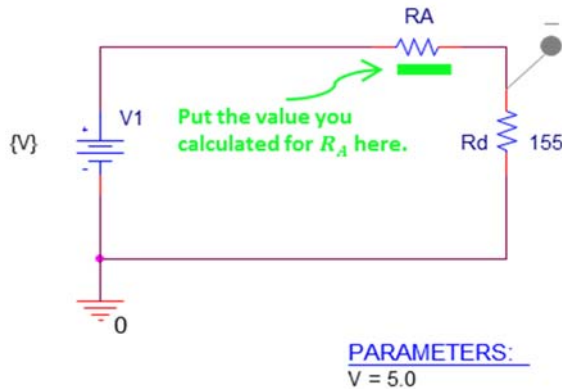

Figure 6. Meter simulation.

We will now sweep the voltage source using a variable V. This is very similar to when we did a sweep of the pot value in Lab 1. If you have not already done so, place the "Part" PARAM, which appears as PARAMETERS in Figure 6. Double click on PARAMETERS to open the Property Editor Spreadsheet. Click the "New Property" button in the upper left portion of the screen. A new screen will open. Enter V in the field labeled Name:. Below that enter 5.0 for Value:. Click "OK"

Click on the V Heading (that we just created) in the Property Editor Spreadsheet. You may need to scroll to the right or left to find this heading. Now click the button above labeled "Display…"

A new window will open. Click the button for Name and Value and click "OK"

Return to the schematic by clicking the tab marked "PAGE1"

Below the PARAMETERS part, you should now see V=5.0 as in Figure 6.

We now need to associate our parameter V with the voltage source. Double click carefully on the value of the voltage source (probably 0Vdc). A window will open. Replace the value 0Vdc with {V} Don't forget the {} braces. Click OK. On the circuit symbol for the voltage source, you should now see {V} as in Figure 6.

Here we will specify the parameters to "sweep" the value of V from 0 to 5 in steps of 0.1. Click PSpice -> New Simulation Profile

Enter a name for this profile (e.g., voltage sweep)

Click Create

A new window will open. You might have to look at the icons at the bottom of the screen to find it.

In the drop down menu entitled Analysis type:, choose DC Sweep.

Enter the following parameters:

Sweep Variable: Global parameter

Parameter name: V (no curly braces this time)
Sweep type: Linear
Start value: 0
End value: 5
Increment: 0.1
Click "OK"

Add a current marker to the top pin of R_d as shown in Figure 6.
(PSpice -> Markers -> Current Into Pin)
Now run PSpice. A new window will open. You may need to look at the icons on the bottom of the screen to find it. This window will contain a graph of the meter movement current as a function of the voltage V. If the graph does not look as expected from the discussion above, something has gone wrong. Check your work carefully and try again.

b) Print the graph.

Recall from above that we said our meter is not very good. So then why does it seem to work perfectly? The reason is that we have used it to measure an ideal voltage source, which holds a constant voltage regardless of current.

We have learned (or will learn soon) in class that circuits can be modeled by a Thevenin equivalent circuit, as shown in Figure 7 below. In our PSpice simulation above, v_{Th} was varied from 0 to 5 V, but always with $R_{Th} = 0$. What we will find out below is that our meter works well whenever $R_M = R_A + R_d$ is very large compared to R_{Th}. Indeed, an ideal volt meter would have $R_M = \infty$.
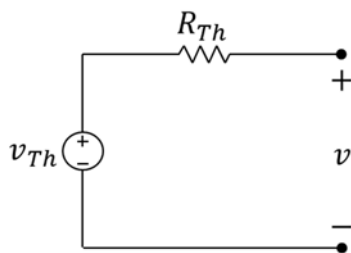

Figure 7. Thevenin circuit.

Let's first look at a simple example when this is not true. Let's connect our meter from Figure 5 to the Thevenin circuit of Figure 7 with $v_{Th} = 5$ V, and $R_{Th} = 4.7\ k\Omega$, as shown below in Figure 8. If our meter were not connected, v would be 5 V. If our meter were ideal, v would still be 5 V even with our meter connected. However, our simple meter loads the circuit and alters the value of v. A certain amount of this is unavoidable, but it should be minimal for a good meter.
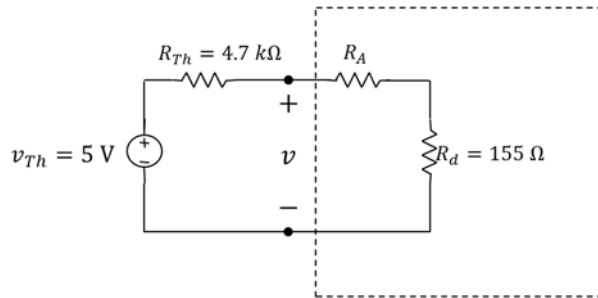
Figure 8. The loading effect (again).

c) Calculate v in Figure 8.

d) Calculate the percent error. Let v_t be the true value of the voltage we are trying to measure. Then, %error = $\frac{v-v_t}{v_t} \times 100\% = \frac{v-5}{5} \times 100\%$.

We observe that the %error is relatively large (and negative).

e) Recalculate v in Figure 8, but with $R_{Th} = 47\ \Omega$, rather than 4.7 $k\Omega$.

f) Recalculate %error for the value of v in e).

We observe that the %error is small (and again negative).

The difference between d) and f) is that R_M is large compared to R_{Th} for the case of f), but not for that of d). We could use a transistor to make R_M appear larger, similar to what we did in Lab 1. However, we will learn soon how to do even better using something called an "Op Amp."

Prelab Grading

Part 1
a)   3 pts (1/2 pt each)
b)   3 pts
c)   3 pts
d)   2 pts

Part 2
a)   3 pts (1/2 pt each)
b)   3 pts

Part 3
a)   1 pt
b)   3 pts
c) – f)   4 pts (1 pt each)
                _____
                25 total pts

Lab 2 Data Sheet

Please follow all rules, procedures and report requirements as described at the beginning of the Lab 1 Data Sheet.

Lab Grading: Data Sheet = 15 pts; Explanation = 10 pts.

Lab Part 1

$i_1 =$	$i_2 =$	$i_3 =$
$v_1 =$	$v_2 =$	$v_3 =$

0.5 pt each

Lab Part 2

v_2	$v_p =$	$T =$	$v_2(t) =$
v_3	$v_p =$	$T =$	$v_3(t) =$

1 pt each

Lab Part 3

$R_d =$	$R_A =$

1 pt each

$R_{Th} = 4.7\ k\Omega$	$x =$	$v =$
$R_{Th} = 47\ \Omega$	$x =$	$v =$

1 pt each

Lab Experiments

Lab Part 1

Build the circuit shown in Figure 1. Note that one of the resistors in this figure has a non-standard value. You will have to make the equivalent resistance using the standard resistors available in the lab. Hint: You do not need more than two resistors. The power supplies in the lab may have a current source mode. However, the value of the current cannot be controlled precisely enough for our purposes here. So, we use a "trick" to achieve the desired effect. We will use a "voltage supply" and initially set the value of the voltage to 0 V. Connect it into your circuit (in place of the current source) with the positive terminal connected to the node marked v_1. Connect the DMM so as to measure the current coming from the positive terminal of the power supply. Now, slowly increase the voltage until the measured current reaches the desired value of 2.5 mA. You can now remove the DMM (reconnecting the circuit of course). Now, use the DMM to measure i_1, i_2, i_3, v_1, v_2, and v_3. Record these values on your Lab 2 data sheet. Have your TA initial your results.

Online students: take a photo of your circuit and your DMM showing each of the values recorded on your data sheet.

Lab Part 2

For the circuit shown in Figure 3, you will verify some of your calculations, and will gain some experience using an oscilloscope. Figure 9 shows a simulated screen shot for an oscilloscope that is measuring a sinusoidal voltage. Items shown in black (in the figure) are things you will see on the oscilloscope. Colored items are for illustration of the concepts described here. The horizontal axis represents time. Each horizontal division on the grid represents a certain number of seconds, which can be adjusted by altering the "horizontal scale" on the oscilloscope (scope). Similarly, the vertical axis represents voltage. Each vertical division on the grid represents a certain number of volts, which can be adjusted by altering the "vertical scale" on the scope. Assume that the scope is set for 10 mV per vertical division. The sinusoid shown in the figure then has an amplitude of (2 divisions)*(10 mV/division) = 20 mV. Now assume that the scope is set for 2.0 ms per horizontal division. The sinusoid shown in the figure then has a period of $T = $ (4 divisions)*(2.0 ms/division) = 8 milliseconds. The amplitude is sometimes referred to as the peak voltage, v_p. The peak-to-peak voltage is double this value, or $v_{pp} = 2v_p$, as shown in the figure. The frequency in cycles/second, or Hz, is $f = \frac{1}{T} = 125$. The frequency in radians/second is $\omega = 2\pi f$. Thus, the equation for the voltage shown on the scope screen of Figure 9 is $v(t) = v_p \sin(\omega t) = 20\sin(250\pi t)$ mV.
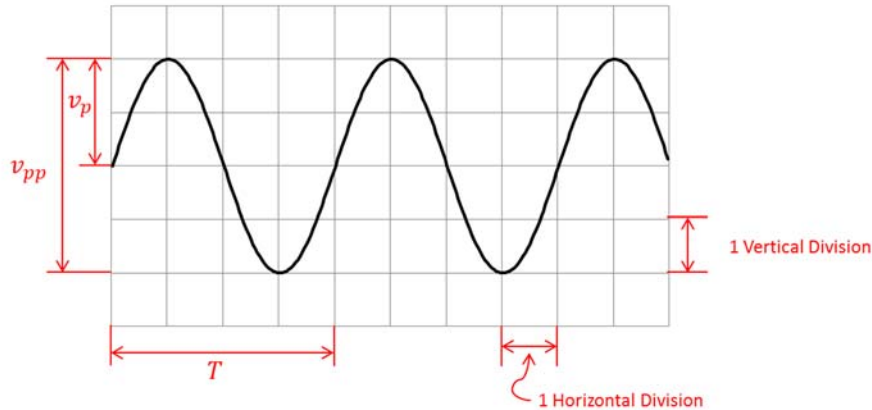
Figure 9. Oscilloscope "screen shot."

Build the circuit shown in Figure 3. Use the function generator to generate the sinusoidal voltage. Set $v_{pp} = 10$ V and set the frequency to 100 Hz. Our scope has two channels. Here we use channel 1. Connect the two leads for this channel together. You should now see a flat line on the scope. You can adjust the level of this line up or down to some convenient location using the "vertical position" setting. Set it to lie on one of the grid lines near the middle of the screen. This grid line then represents 0 V. Set the scope coupling mode for channel 1 to DC. We will always use DC mode unless otherwise noted. AC mode would remove any DC offset present in the signal. Now connect the ground lead of the scope probe to the reference node in Figure 3. Connect the other lead to the node in your circuit marked v_1 in Figure 3.

It is important to understand "triggering" of the scope. The trigger determines when the scope begins to paint the trace of the voltage being measured. Set the scope to trigger on channel 1 in normal mode. In this mode, you can set the trigger to occur at any given voltage "level." You can also choose whether it triggers when the voltage passes through that level going up (positive slope), or going down (negative slope). Set it to trigger at 0 V with positive slope. Set the trigger hold off (delay) to minimum. When you have the trigger set accordingly, the trace on the screen should look like a sine wave (not a cosine, etc.). Play with the trigger settings until you understand what they do, then set them back to the values discussed above.

Verify that the period and amplitude shown on the scope are as you expect for $v_1(t)$. Now measure the amplitude and period of the voltage at the node labeled v_2 in Figure 3. Record these values on your data sheet, and write an equation for $v_2(t)$. Repeat for the node labeled v_3.

Have the TA initial your results.

Online students: take a screenshot showing the oscilloscope screen for each of the measurements you are asked to make above.

Lab Part 3

The value of R_d varies somewhat from panel meter to panel meter. Use the DMM to measure R_d for your panel meter. Recalculate R_A based on the measured value of R_d. Record these values on your data sheet.

Choose an appropriate standard resistor, and connect it in series with a $1\ k\Omega$ pot acting as a variable resistor. Adjust the pot until the series combination equals the new value of R_A from your data sheet. Use this series combination to build a volt meter (as per Figure 5).

Build the Thevenin circuit shown in Figure 7 using $v_{Th} = 5$, and $R_{Th} = 4.7\ k\Omega$. Connect your meter to the Thevenin circuit. Estimate as best you can the fraction, say x, of full scale that the needle deflects. Since you designed your meter for a full scale of 5 V, your voltage reading is then $v = 5x$. Record your values of x and v on your data sheet.

Recall at this point in the prelab we replaced the 4.7 $k\Omega$ resistor with a 47 Ω resistor. The internal (Thevenin) resistance of the power supply is on the order of 50 Ω. Thus, just replace the 4.7 $k\Omega$ resistor in your circuit with a piece of wire and repeat the measurements. Record the new values of x and v.

Have the TA initial your results.

Online students: take a photo of your circuit showing the d'Arsenval panel for each of the two measurements recorded on your data sheet.