Lab#4 General DC Circuits

This lab covers the general DC circuits. In addition to KCL, KVL and absorption power, this lab also covers the concepts of a reference node and using voltage division to design the circuit in Figs (2.2 and 2.4)

Multisim

Please follow the steps below to setup the simulation environment and build your circuits

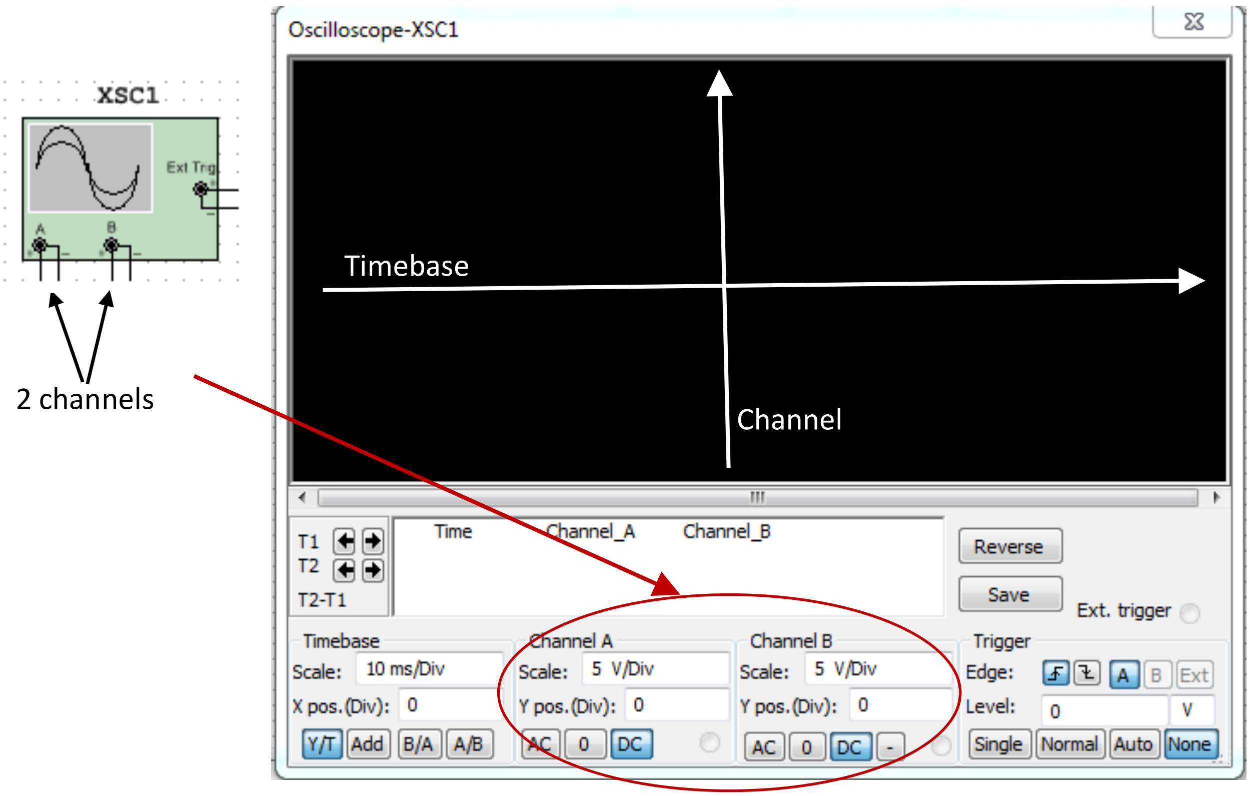
1. Setting up simulation environment
   - Simulate -> Analyses and simulation
   - Select "Interactive Simulation", then "Save"
2. Place component on the circuit board to build your circuit
   - Place -> Components
   - Change Database to Master Database
   - Change Group to <All Groups>
   - Change Family to <All families>
   - Type RESISTOR_RATED under Component, then click OK to place it on the board

   Repeat the steps above to place all components for your circuits

Circuit	Devices	Power	GROUND (Reference Node)
Fig (4.1)	2 RESISTOR_RATED	DC_POWER	1
Fig (4.1)	1 RESISTOR_RATED 1 CAPCITOR_RATED	DC_POWER	1

3. Placing
   a. Multimeters (First instrument icon on the top right)
   - Alternatively,
     - Simulate -> Instruments -> Multimeter

   b. Oscilloscope (**Fourth** instrument icon on the top right)
   - Alternatively,
     - Simulate -> Instruments -> Oscilloscope



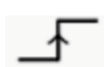
Timebase: Setting the x-axis display

- Scale: sensitivity setting for time/division [Zoom in and out of time scale]
- X pos. (Div): moving the signal left or right [NOT USED]
- Y/T: showing both channel in Y-time (Y/T) mode [Normal display]
- A/B: plotting V_B against V_A for phase measurement in XY-mode [See Lab4 App. B]

Channel A or B: Setting the y-axis display of the channel

- Scale: sensitivity setting for voltage/division [Use **Up** and **Down** arrow to enlarge or reduce signal amplitude]
- Y pos. (Div): moving the signal up or down
- AC : AC coupling [DC component will not be displayed]
- 0 : Beam off [No signal will be displayed]
- DC : DC coupling [Showing both AC and DC components]

Trigger: Display reference

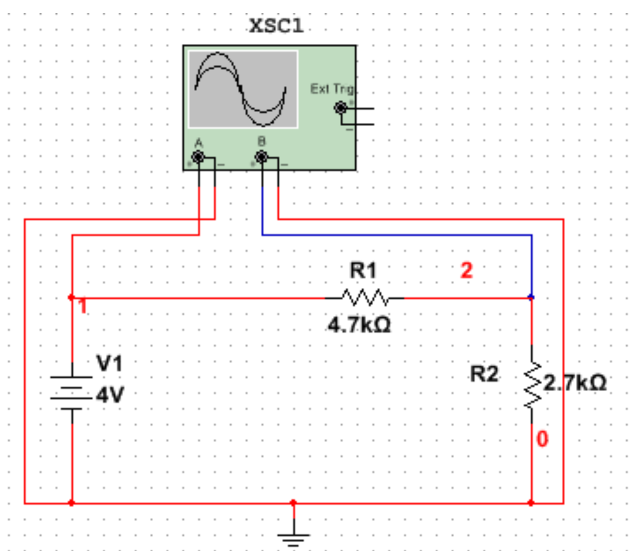
- Edge: Use rising button 
- A : trigger source is channel A

Reverse: Reverse background colour between black<-> white [Use white background so it is easier for your TA to see the display]

Lab Procedure

DC Power

1. Build the circuit as in Fig. (4.1). Connect CH A to node (1) and CH B to node (2) on the oscilloscope and the following setting:



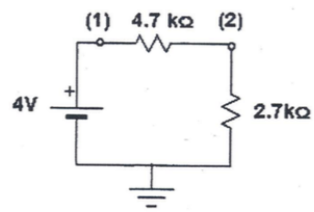
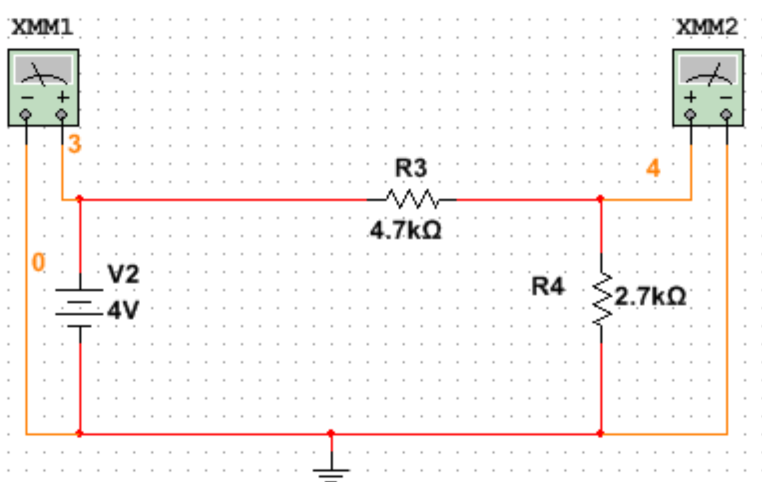


Fig (4.1)

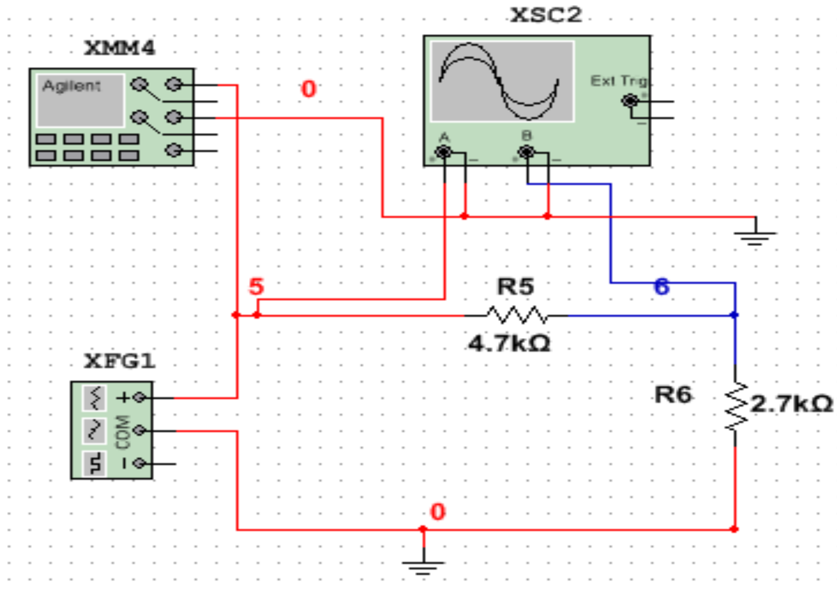
	Channel A	Channel B
Scale	2 V/Div	2 V/Div
Coupling	DC	DC

3. Use DMM to measure voltages at nodes (1) and (2)



AC Power

Steps 4-7 shows the difference between reading the voltage peak (Vp) value on the oscilloscope and the RMS values on the **Agilent Multimeter.** A table of RMS/V_{peak} ratio is available in the Conclusion section. [Beam off CH B. You will not need it.]



4. Use the function generator as your AC power source. Check the voltage source signal on the oscilloscope to see if it is correct with the following setting:

	Timebase	Channel A
Scale	200 µs/Div	2 V/Div
Coupling	n/a	DC

Set the function generator to produce the following AC voltage:

Waveform	Sinusoidal
Frequency	1 kHz
Amplitude	2 Vp

NOTE: You should see one cycle every 1 ms (1000 µs) with max. of voltage amplitude 2V.

Agilent Multimeter
- Measure AC voltage: Press <AC V> button
- Measure Frequency: Press <Freq>

NOTE: If you don't see anything, press <Single>

5. Change the function generator waveform to Square wave and read the voltage and frequency from the Agilent multimeter.

6. Change the function generator waveform to Triangular wave and read the voltage and frequency from the Agilent multimeter.

7. Add 2V DC offset to the function generator from step 6 as below:

Waveform	Sinusoidal
Frequency	1 kHz
Amplitude	2 Vp
Offset	2 V

Multisim does not provide a DC+AC reading at the same time, you need to use the formula below to calculate it:

$$RMS\ voltage = \sqrt{(V_{dc}^2 + V_{ac}^2)}$$

NOTE:
- For more information, please refer to https://zone.ni.com/reference/en-XX/help/372062L-01/multisim/multimetercontrols/
- $V_{dc+ac\ RMS}$ DOES NOT equal to $V_{dc\ RMS}$ + $V_{ac\ RMS}$

8. Change the Amplitude of the function generator to 20 mV

Waveform	Sinusoidal
Frequency	1 kHz
Amplitude	20 mVp
Offset	2 V

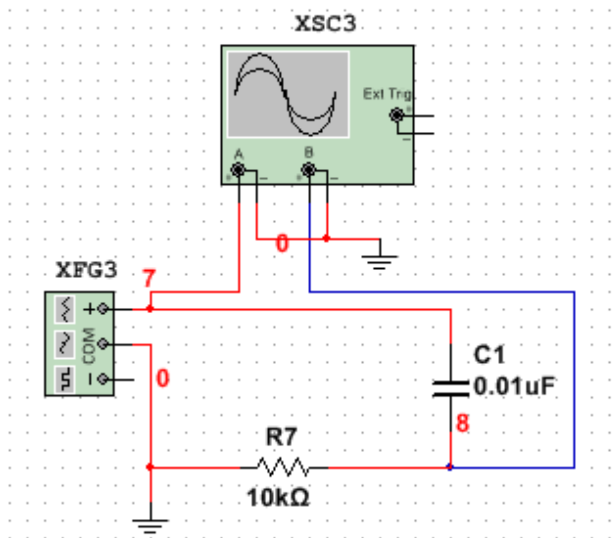
Using AC coupling on the oscilloscope to display the max using **Scale = 20 mV/Div**.
Using DC coupling to see the combined AC+DC voltage using **Scale = 2 V/Div**.

9. Change the function generator frequency according to Table 4.1 and record the results.

NOTE: Unlike Multisim, the multimeter in the physical lab is not capable of measuring at 1000 kHz. You will get erroneous results using physical instrument in the lab.

Phase Angle Measurement
Steps 10 and 11 allows you see the phase difference between voltage and current. Since oscilloscope measures voltage, your current can be measured using voltage across the 10 kΩ resistor, i.e. $i_L = \frac{V_{10k}}{10k}$ A.

Step 10: Build the circuit



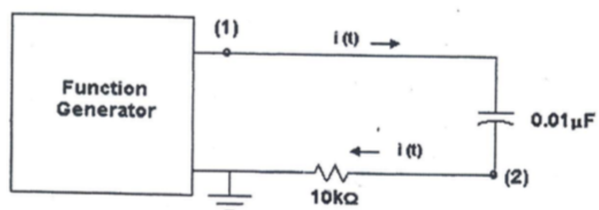


Fig (4.2)

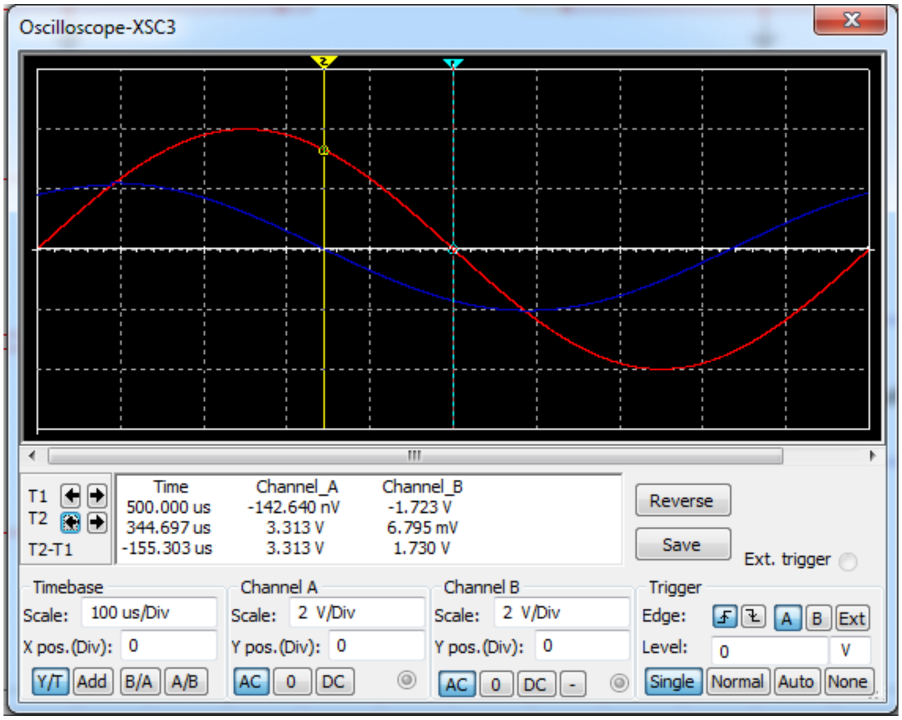
Use the following function generator setting:

Waveform	Sinusoidal
Frequency	1 kHz
Amplitude	4 Vp

Use the following oscilloscope setting:

	Timebase	Channel A	Channel B
Scale	100 µs/Div	2 V/Div	2 V/Div
Coupling	n/a	AC	AC

Step 11: Use the function generator to measure the phase angle in Y-Time mode and XY mode.



Procedure to find phase angle in Y-Time mode:

1. Find DeltaT: Measure the time difference between the 2 voltage signals
2. Each cycle time (T) has 360°
3. Phase angle = DeltaT/T x360 °
4. Since T = 1/Freq,

   Phase angle = DeltaT x freq x 360
5. If CHB is leading, the phase angle = +ve.

NOTE:

- Both have to be on the falling part of the waveform. Or, both rising.
- The leading channel is the one on the left of the other channel.

To measure phase angle using the XY mode, please follow the manual’s Appendix B.