SARDAR VALLABHBHAI NATIONAL INSTITUTE OF TECHNOLOGY, SURAT
ELECTRONICS ENGINEERING DEPARTMENT

Expt. No:	**1**	**Introduction to Multisim**
Date:	**13-08-2020**	

AIM: To study the Multisim software interface and the tools thereby get acquainted with implementing and simulating circuits using Multisim Live Simulator.

SOFTWARE TOOLS / OTHER REQUIREMENTS:

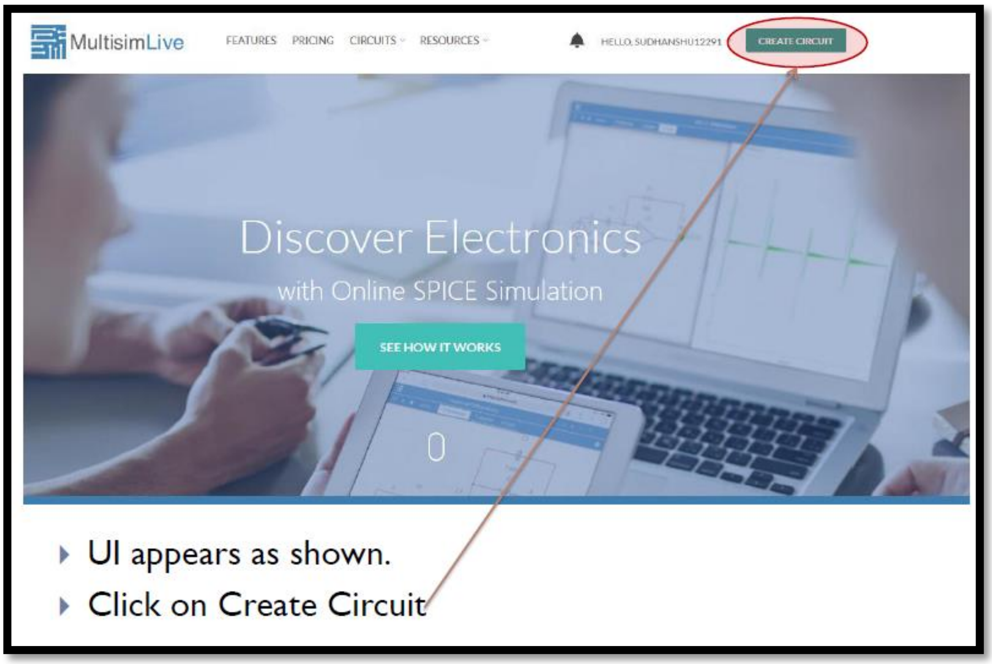
1. Multisim Live (Online Interface)

WORKING ON MULTISIM LIVE SIMULATOR:

NI Multisim (formerly MultiSIM) is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard. Multisim is one of the few circuit design programs to employ the original Berkeley SPICE based software simulation. Multisim was originally created by a company named Electronics Workbench, which is now a division of National Instruments. Multisim includes microcontroller simulation (formerly known as MultiMCU), as well as integrated import and export features to the printed circuit board layout software in the suite, NI Ultiboard. Multisim is widely used in academia and industry for circuits education, electronic schematic design and SPICE simulation.

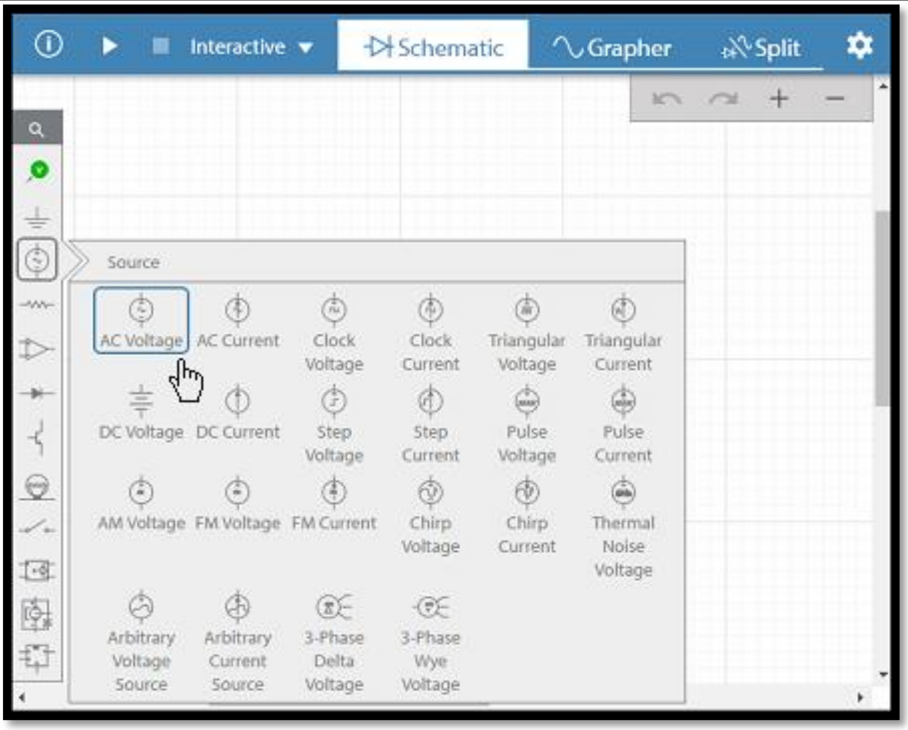
Multisim Live is a free online circuit simulator that includes SPICE software, which lets you create, learn and share electronics circuits online.

Creating Circuits on Multisim:



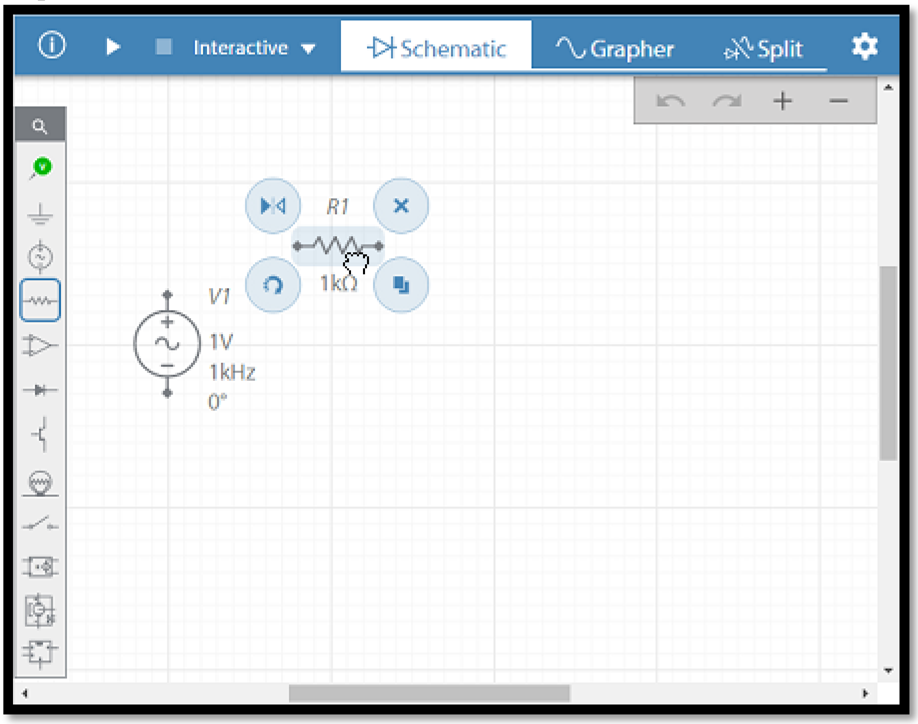
Placing Voltage Source:

Tap the Source subpalette and tap AC Voltage and tap on the workspace or Type V if you are using a device with a keyboard, and tap to place the source.



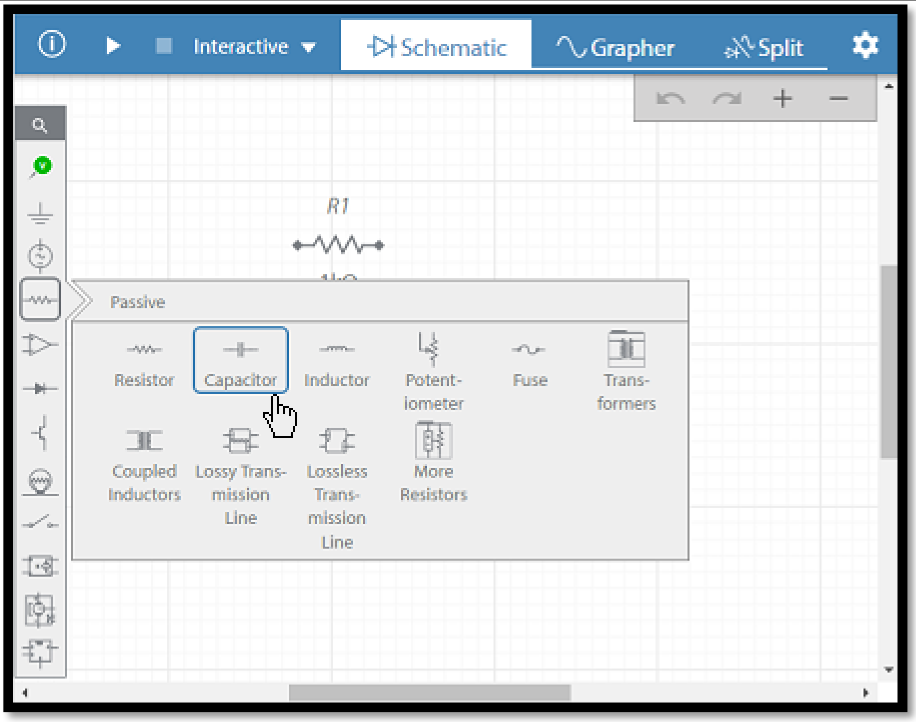
Placing Resistor:

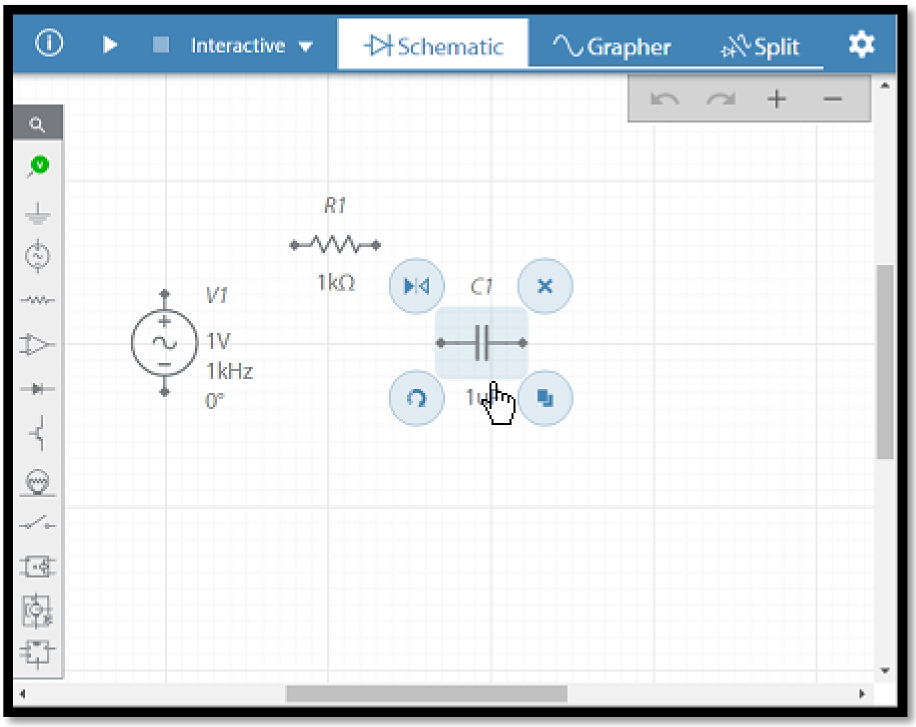
Place a resistor by dragging from the Passive subpalette or Type R if you are using a device with a keyboard, and tap to place the resistor.



Placing Capacitor:

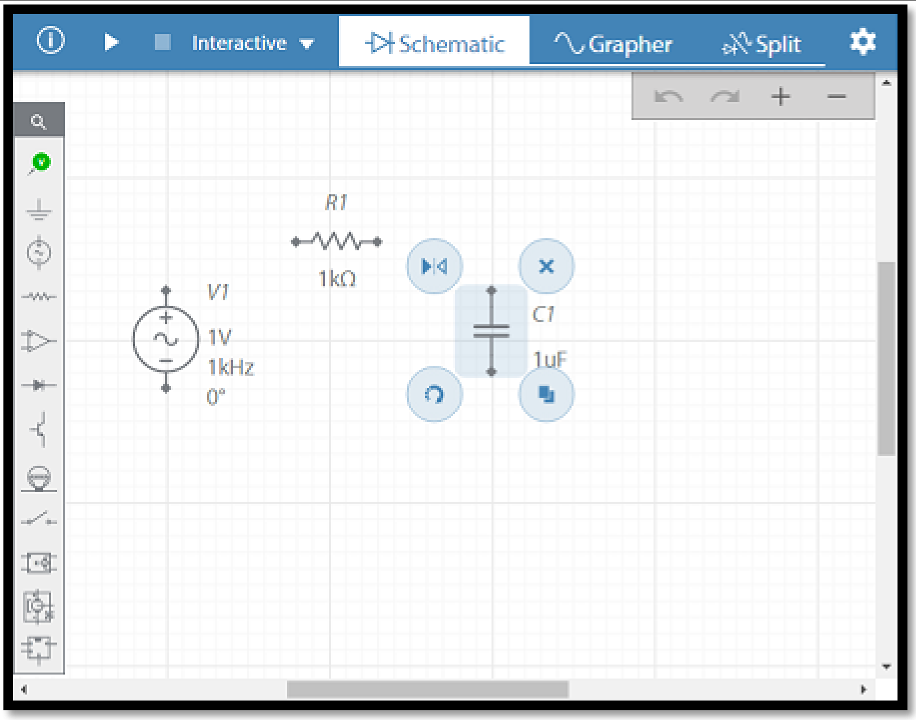
Type C if you are using a device with a keyboard, and tap to place the capacitor.





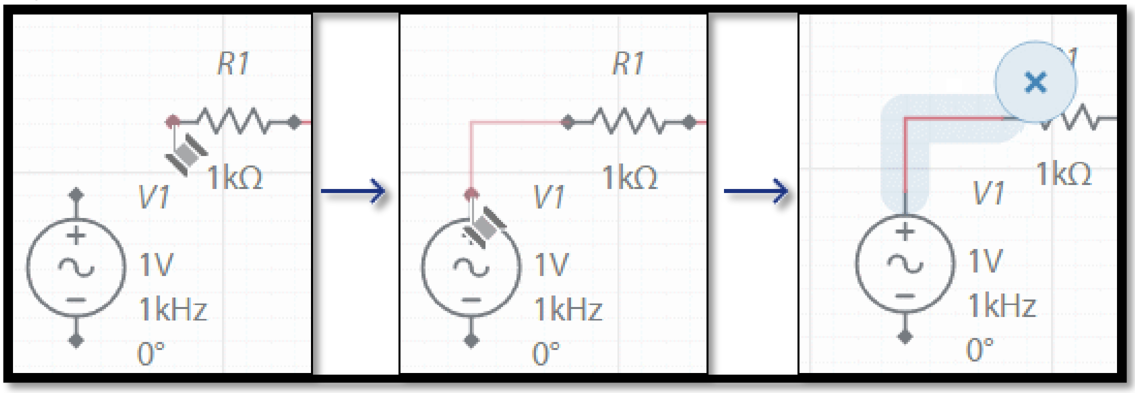
Rotating Components:

Tap  to rotate the capacitor and other components.



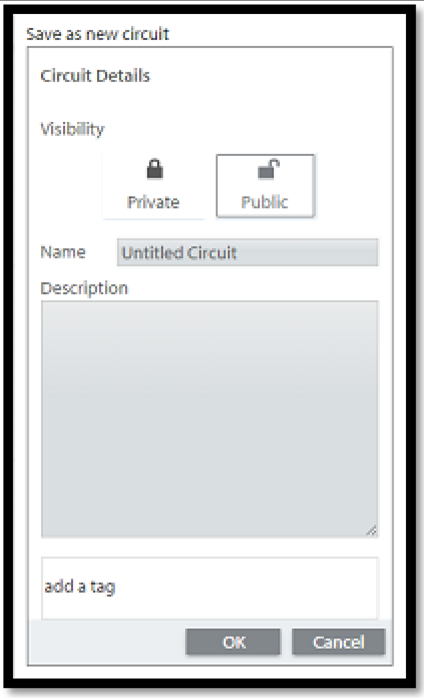
Wiring the components:

Tap a component's wiring point (black diamond) and tap another wiring point. The connection is automatically made, and the new wire is selected.



Saving the design:

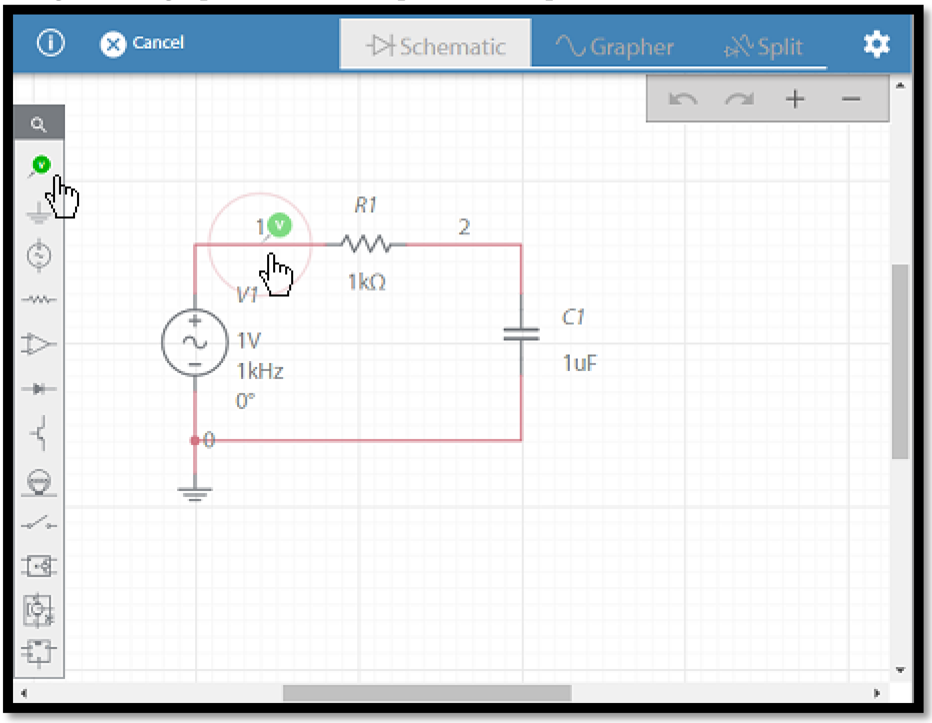
Tap  in the title bar and select Save as.



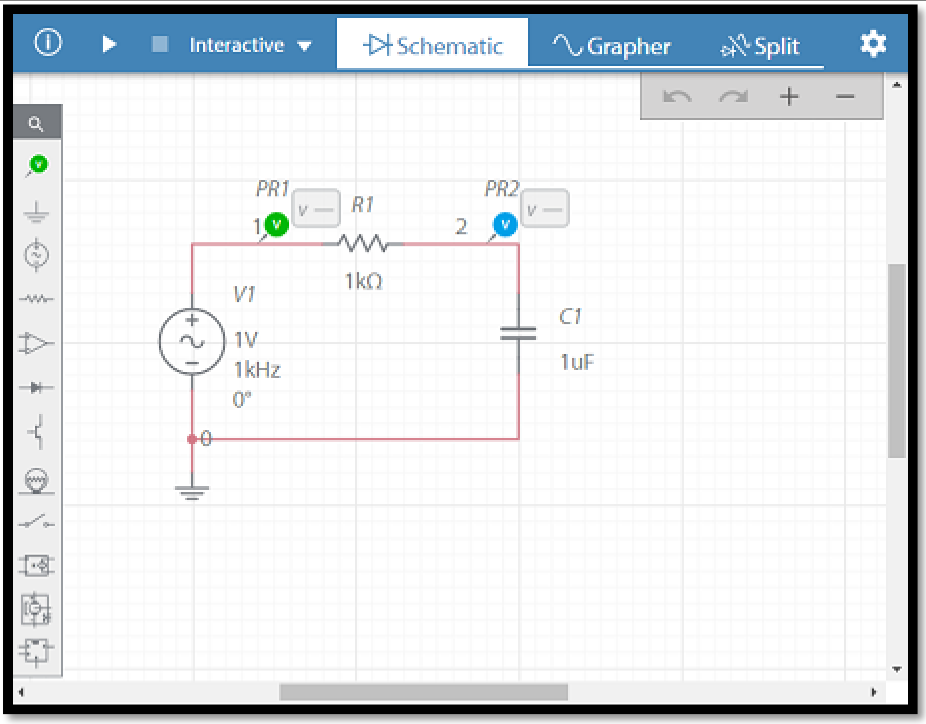
Simulating a Design:

To run a simulation, you must place at least one probe.

1. Drag a voltage probe from the palette and place as shown below.

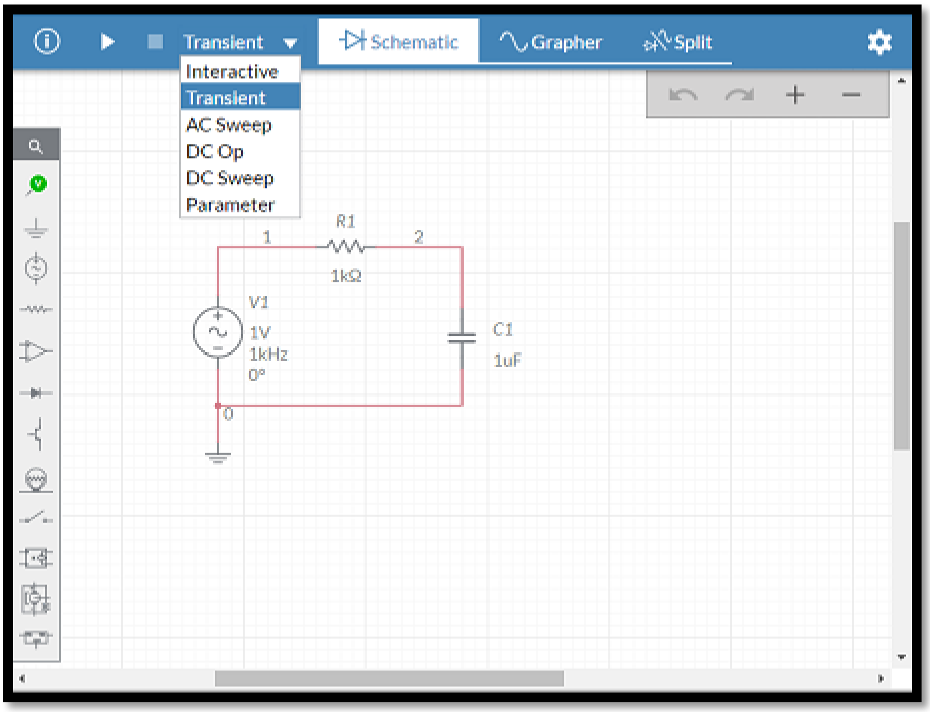


2. Place a second probe.

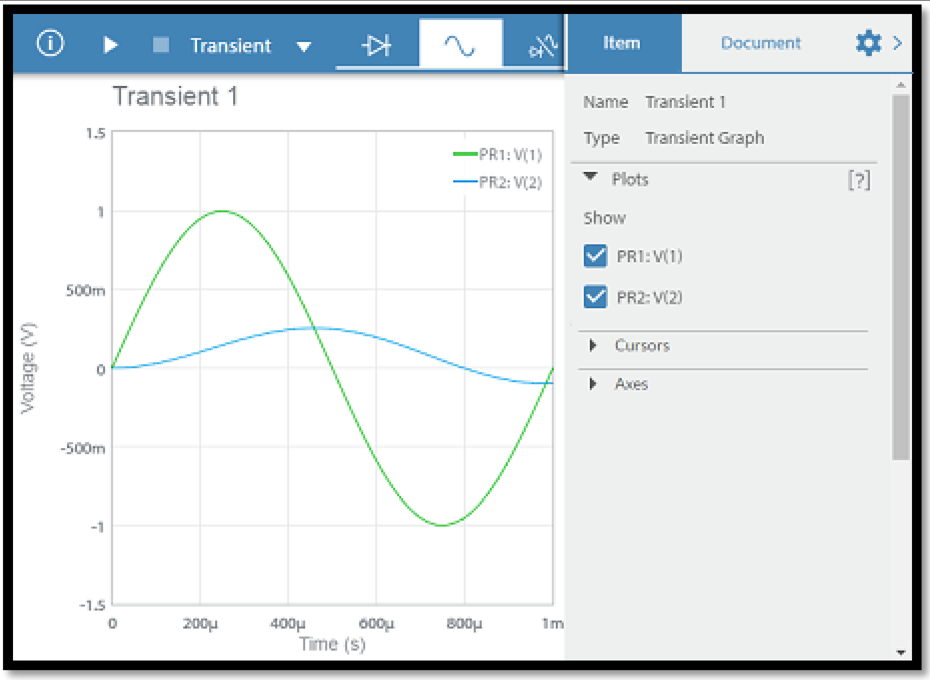


Select and run simulation

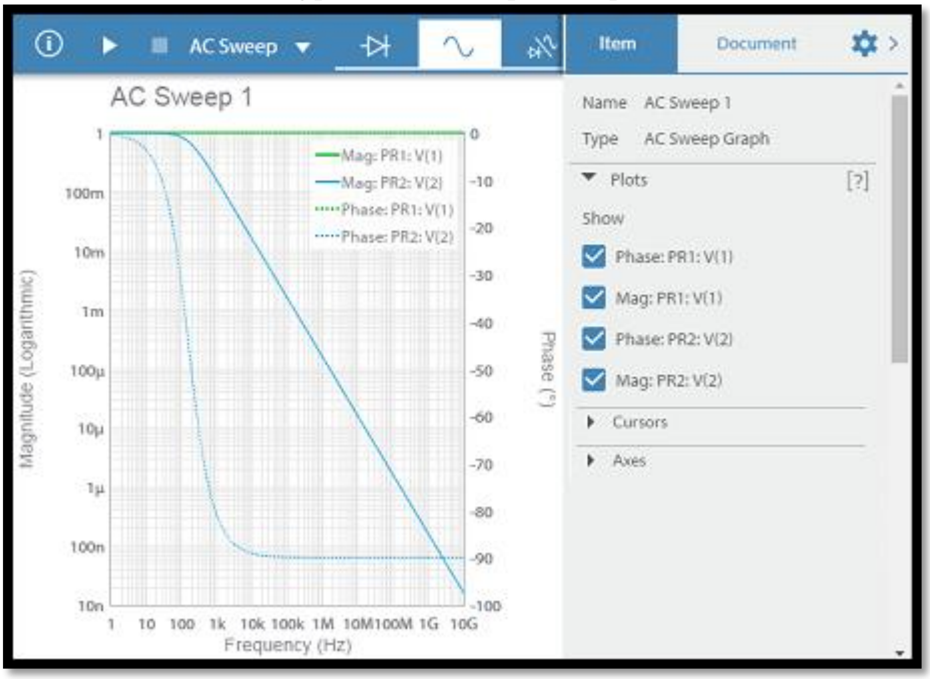
1. Select Transient from the toolbar.



2. Tap ▶ in the toolbar. For transient simulation, the view switches to the grapher.
3. Tap ⚙ in the toolbar to open the configuration pane. You can also double-tap on the grapher.
4. Use the Plots and Axes sections to manipulate the grapher as desired.

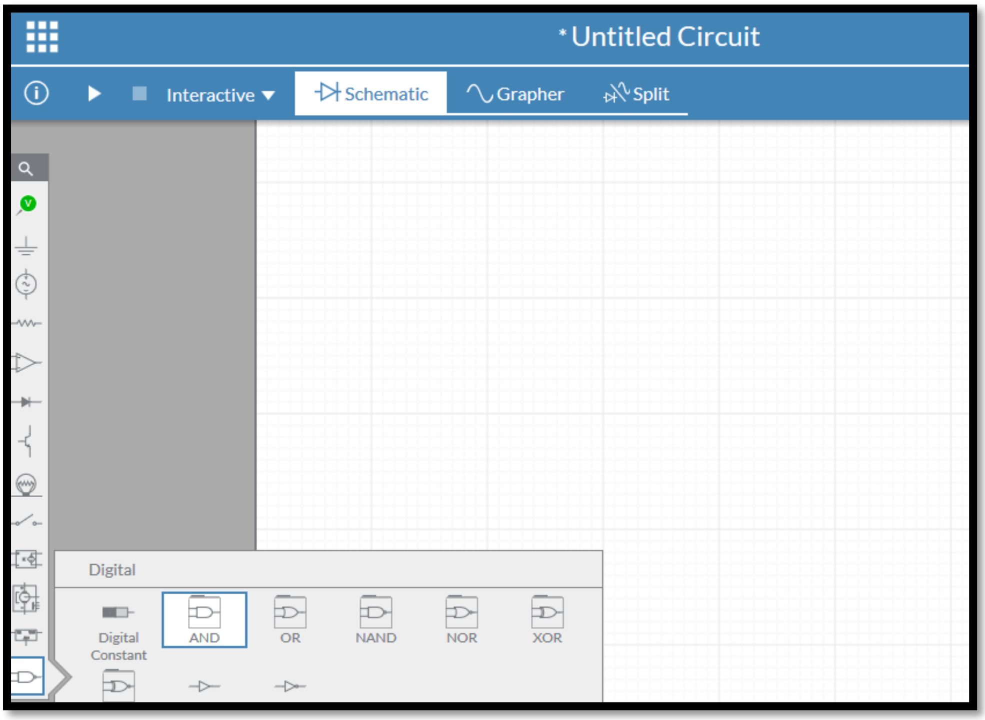


5. Switch the simulation type to AC Sweep and tap .

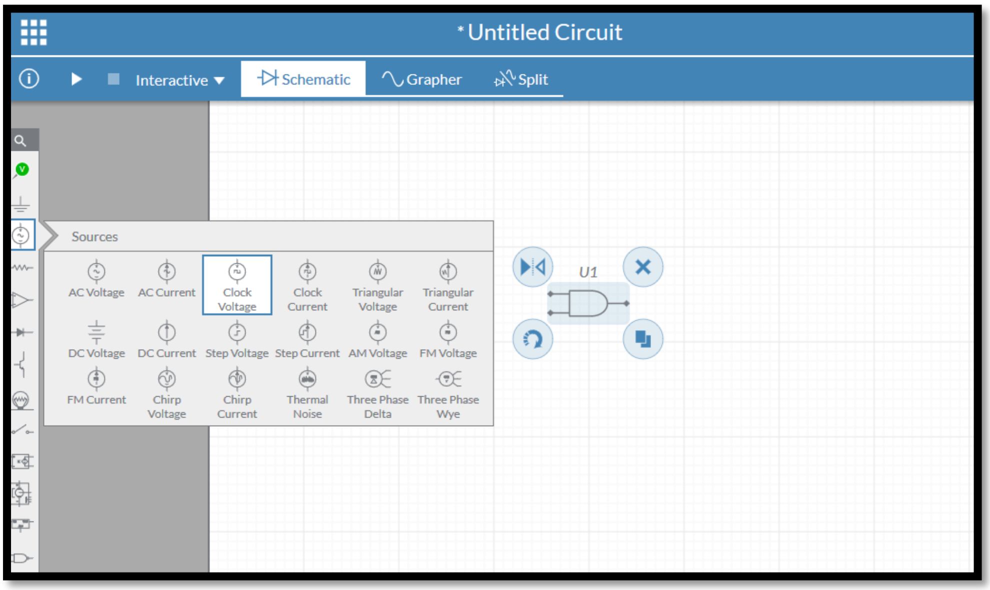


Simulating a simple Digital Circuit:

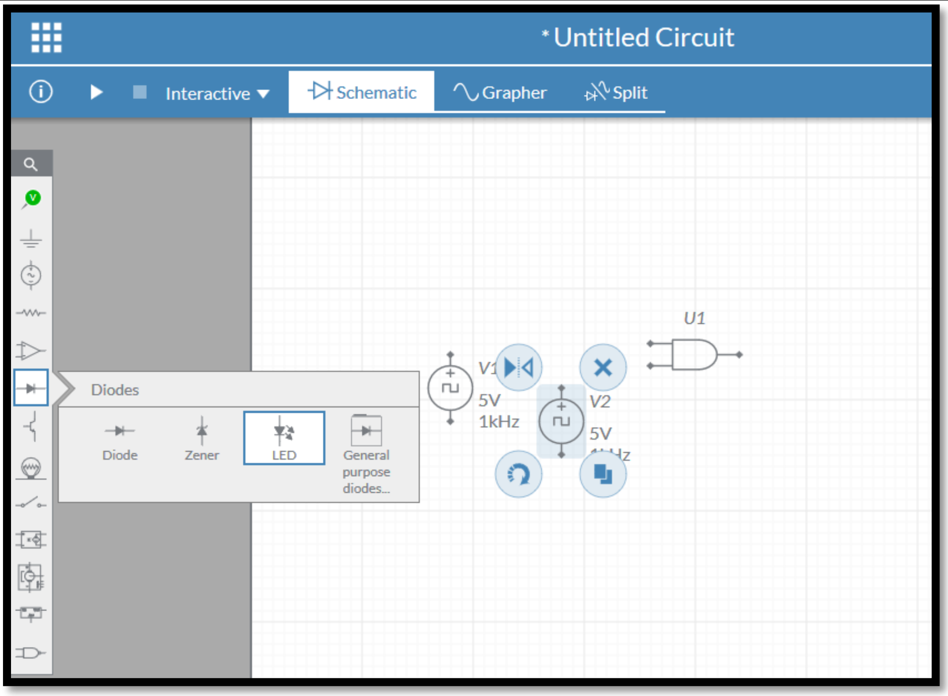
Step 1: Selecting AND Gate



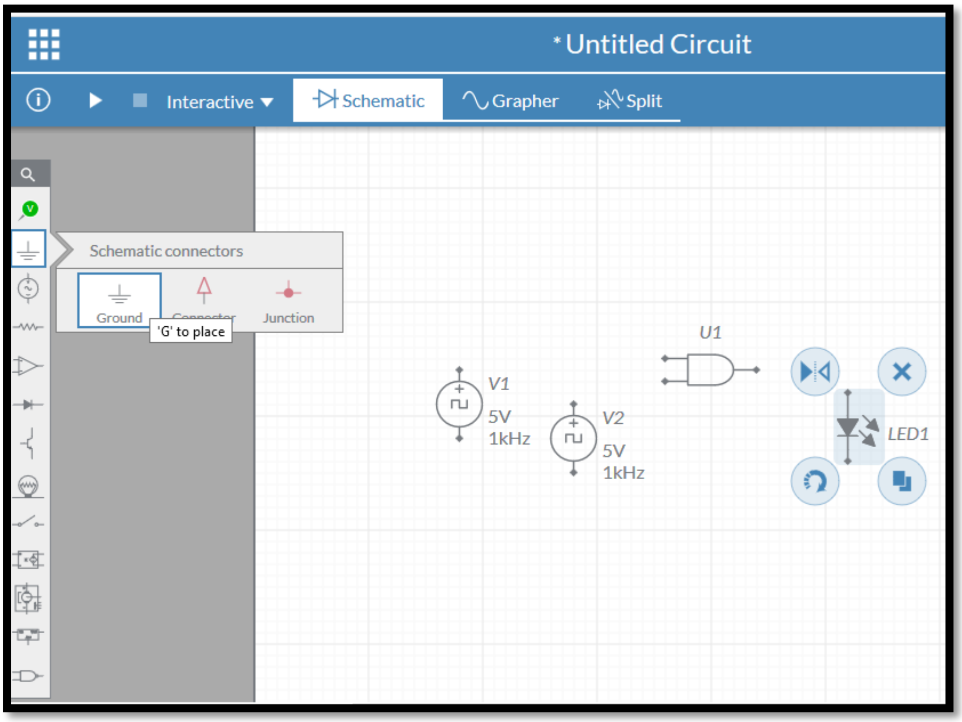
Step 2: Adding Source (Clock Voltages)



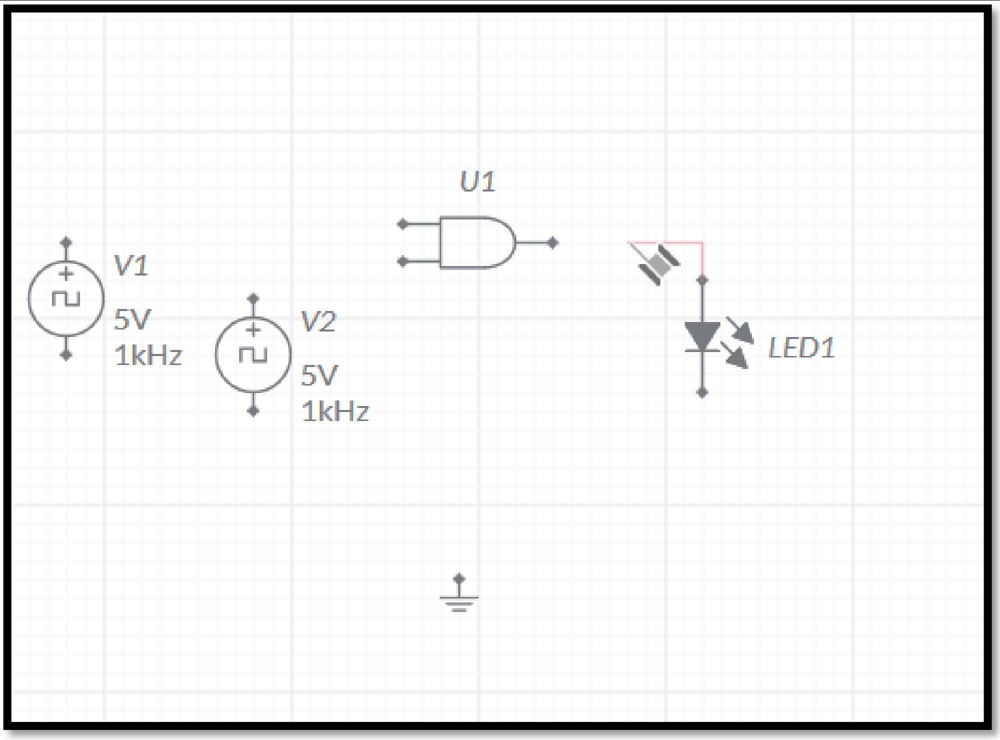
Step 3: Adding Load (LED)



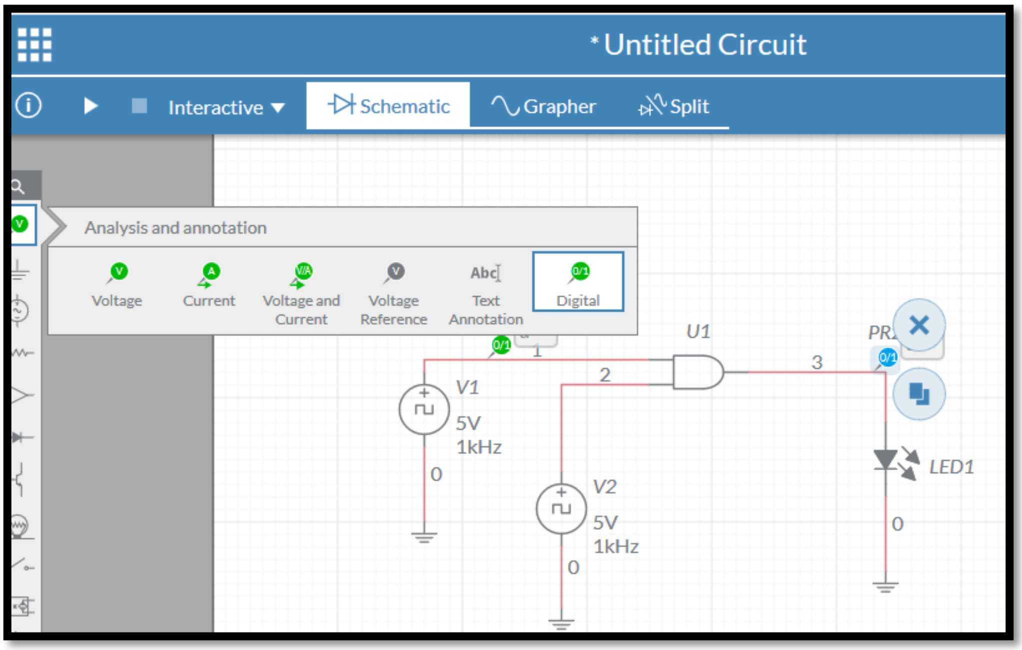
Step 4: Grounding the Components:



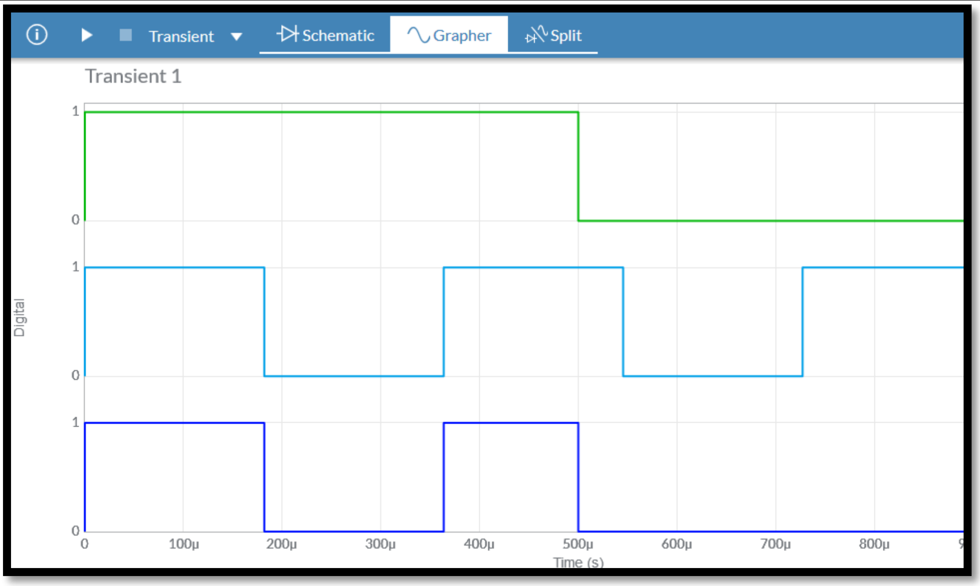
Step 5: Connecting Components



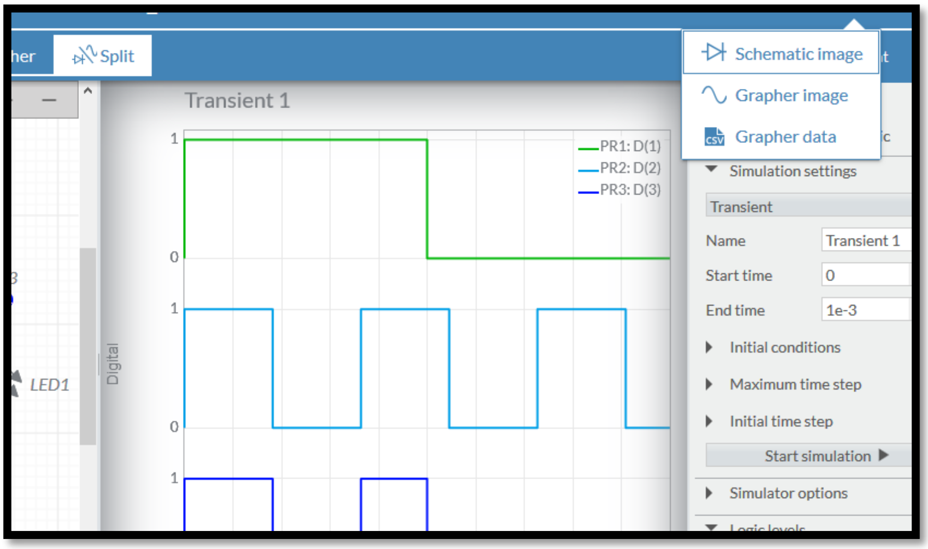
Step 6: Adding Probes



Step 7: Finally we Simulate the Design



Step 8: Exporting Schematic/Grapher Images/Screenshots

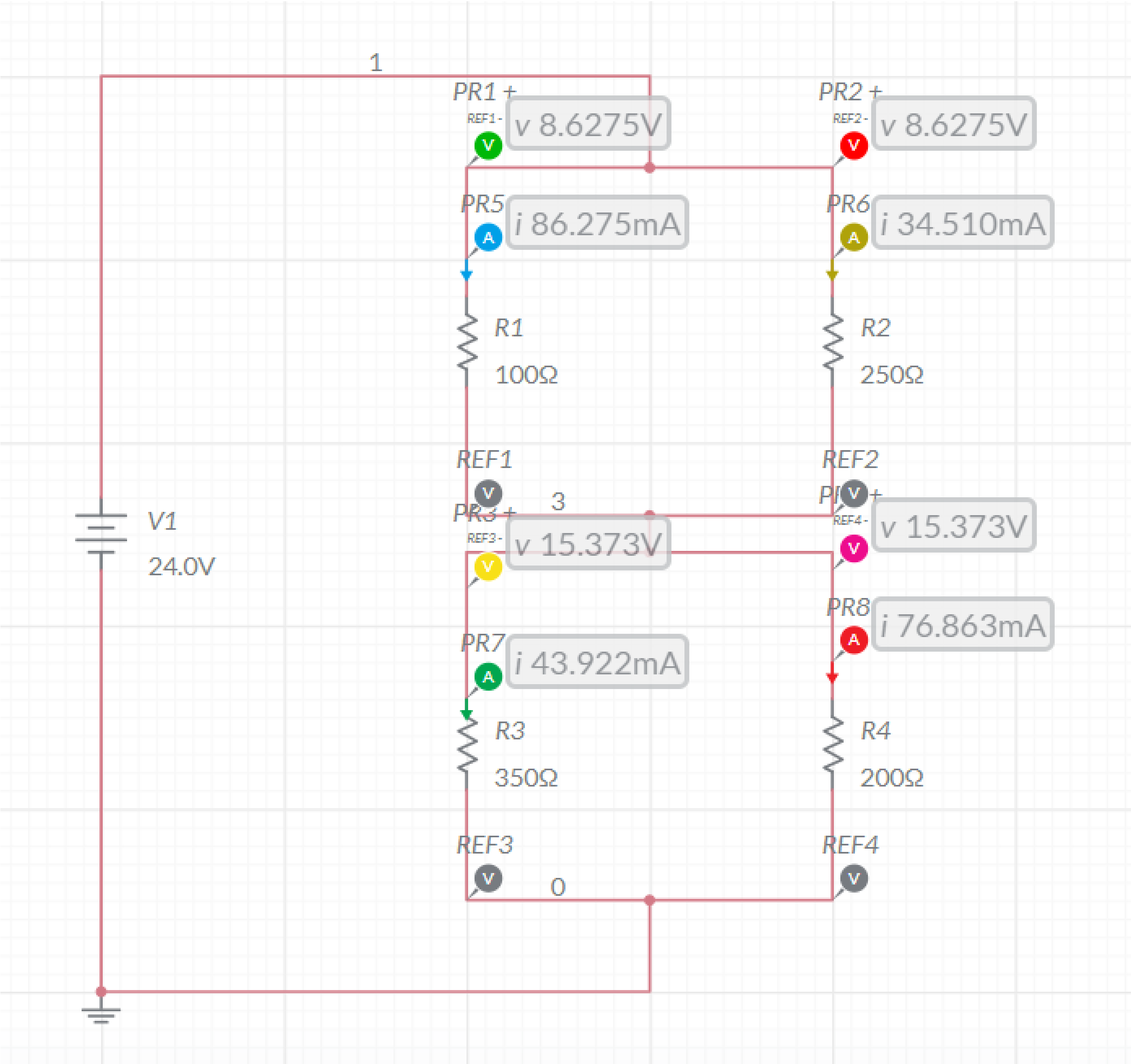


CONCLUSIONS:

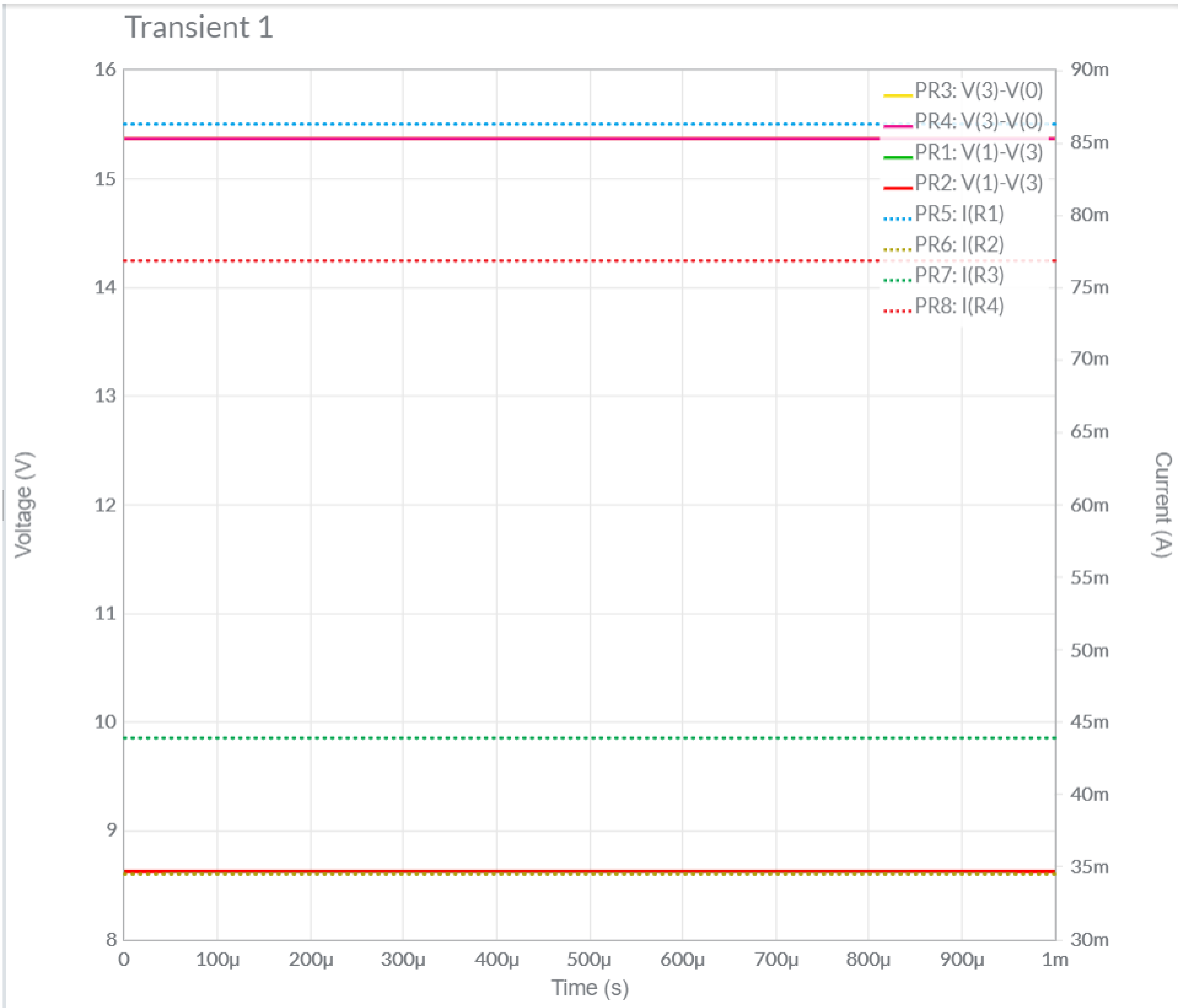
1.) We learned and Implemented *Multisim Software interface* and used different electronic tools available in the simulator to create circuits.
2.) We used *resistor, wires, D.C. voltage source, Voltmeter, Ammeter* and other electronic devices to **verify** *current and voltage* across resistor by *theoretical* and simulated data with Multisim values obtained.
3.) We also learned how to Export Schematic Image, grapher Image and its Data from Multisim.

Assignment -1 Q1

a.) Implement the circuit as shown in Figure in Multisim online.



b.) Evaluate the current and voltage across each resistor using simulator.



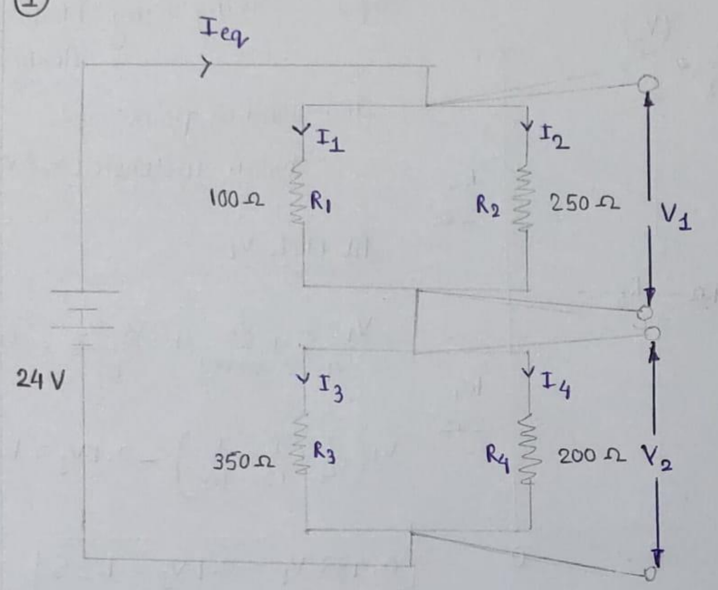
DC OP 1

Signal	Value
PR3: V(3)-V(0)	15.373V
PR4: V(3)-V(0)	15.373V
PR1: V(1)-V(3)	8.6275V
PR2: V(1)-V(3)	8.6275V
PR5: I(R1)	86.275mA
PR6: I(R2)	34.510mA
PR7: I(R3)	43.922mA
PR8: I(R4)	76.863mA

c.) Compare with theoretical values.

ASSIGNMENT - 1 (U19CS012)

1



Step 1 : $R_{eq} = \left(\frac{R_1 \times R_2}{(R_1 + R_2)}\right) + \left(\frac{R_3 \times R_4}{R_3 + R_4}\right) \Omega$

$R_{eq} = \left(\frac{250 \times 100}{350} + \frac{350 \times 200}{550}\right) \Omega$

$= 71.4286 + 127.\overline{27} \ \Omega$

$\doteq 198.7013 \ \Omega$

$\approx \underline{198.7 \ \Omega}$

Step 2 : $I_{eq} = \frac{V}{R_{eq}} = \frac{24 V}{198.7 \Omega}$

$= 0.120785 \ A$

$= \underline{120.785 \ mA}$

Step 3 : Parallel

$V_1 = I_1 R_1 \ -(i) \quad V_1 = I_2 R_2 \ -(ii)$

$\therefore I_1 R_1 = I_2 R_2 \ -(1)$

$I_1 + I_2 = I_{eq} \ -(2)$

Derivation (Just for info)

$I_1 = I_{eq} - \left(\frac{I_1 R_1}{R_2}\right) \Rightarrow I_1\left(1 + \frac{R_1}{R_2}\right) = I_{eq} \Rightarrow \left[I_1 = \frac{I_{eq}(R_2)}{R_1 + R_2}\right]$

Similarly $\left[I_2 = \frac{I_{eq}(R_1)}{(R_1 + R_2)}\right]$

$I_1 = \frac{(I_{eq}) R_2}{(R_1 + R_2)} = (120.785) mA \times \frac{(250)}{(350)} = \boxed{86.275 \ mA}$

$I_2 = \frac{(I_{eq}) R_1}{(R_1 + R_2)} = (120.785) mA \times \frac{(100)}{(350)} = \boxed{34.51 \ mA}$

$V_1 = I_1 R_1 = (86.275 \times 10^{-3} A) \times (100 \Omega) = \boxed{8.6275 \ V}$

Similarly;

$I_3 = \frac{(I_{eq}) R_4}{(R_3 + R_4)} = (120.785) mA \times \left(\frac{200}{550}\right) = 43.9218 \ mA \approx \boxed{43.922 \ mA}$

$I_4 = \frac{(I_{eq}) R_3}{(R_3 + R_4)} = (120.785) mA \times \left(\frac{350}{550}\right) = \boxed{76.8632 \ mA}$

$V_2 = I_3 \times R_3 = (43.922 \times 10^{-3}) A \times 350 \ \Omega = 15.3727 \ V \approx \boxed{15.373 \ V}$

d.) Final Result and Conclusion

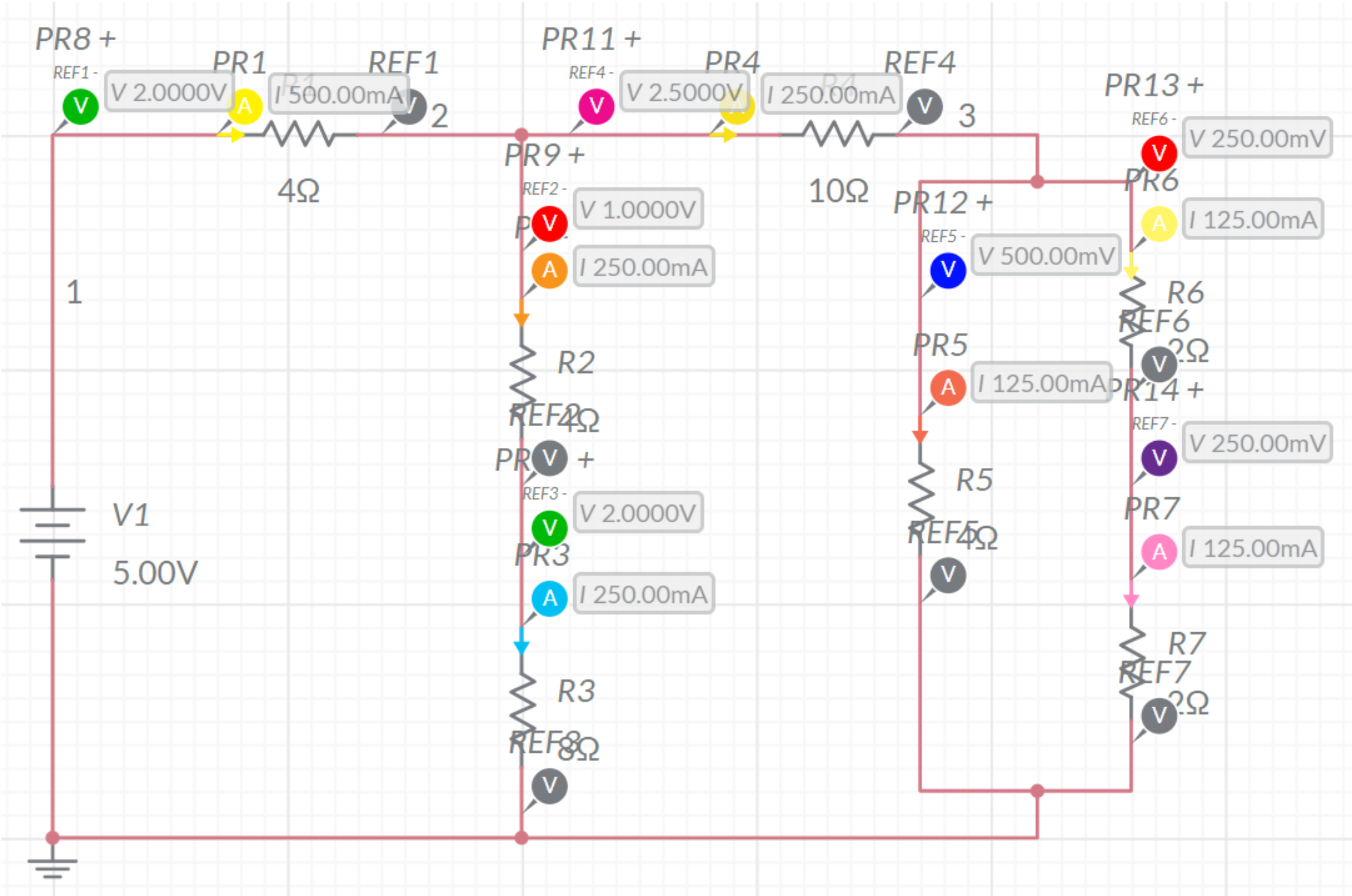
Resistor	Voltage (V)		Current (mA)	
	Multism	Theoretical	Multism	Theoretical
R1	8.6275	8.6275	86.275	86.275
R2	8.6275	8.6275	34.51	34.51
R3	15.373	15.373	43.922	43.922
R4	15.373	15.373	76.863	76.8632

Conclusion:

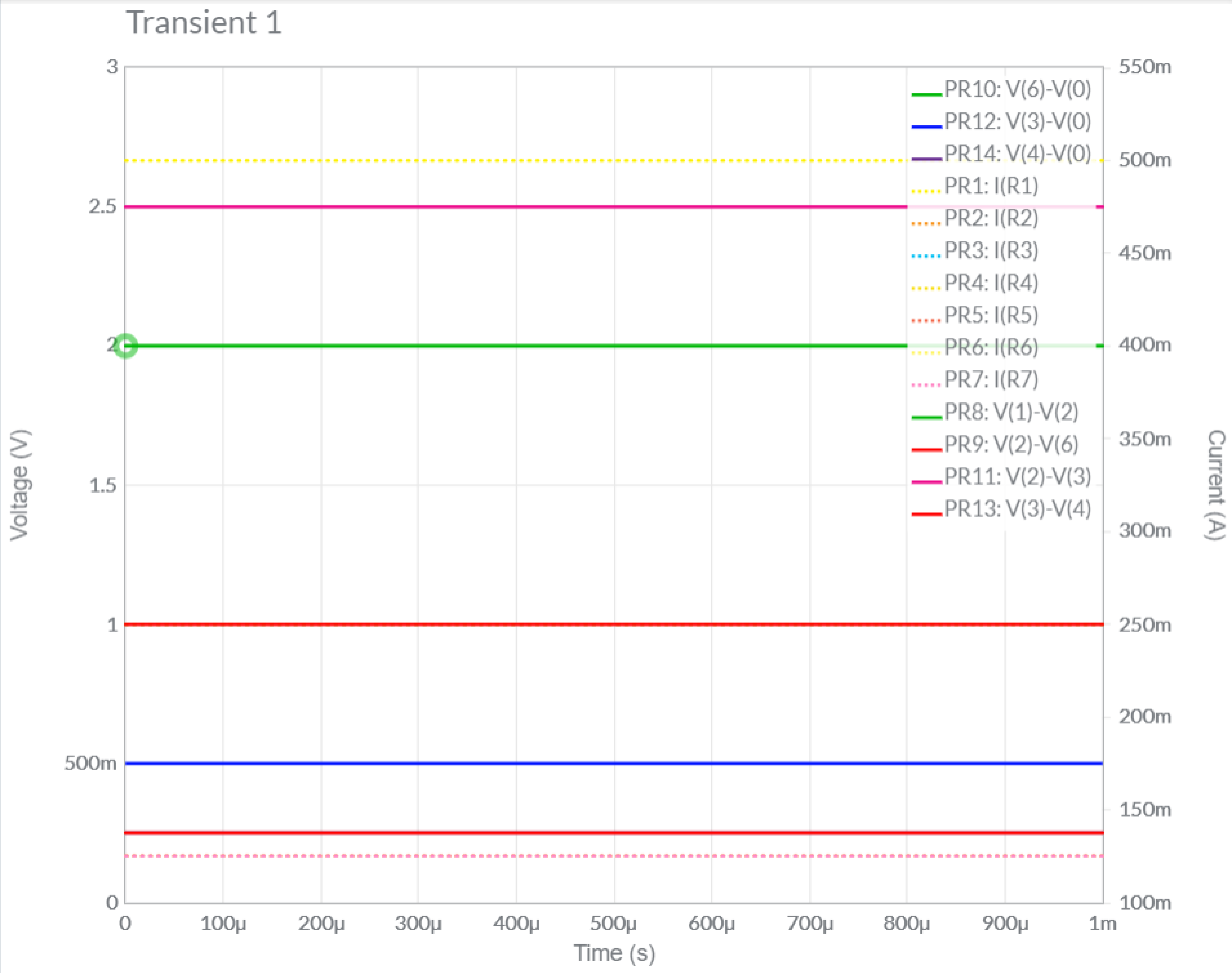
We can observe from Above Table, Both the *Theoretical* and *Multisim* Values of Current and Voltage are **Equal**.
Hence, Experiment is Performed Successfully (without any Error).

Assignment -1 Q2

a.) Implement the circuit as shown in Figure in Multisim online.



b.) Evaluate the current and voltage across each resistor using simulator.

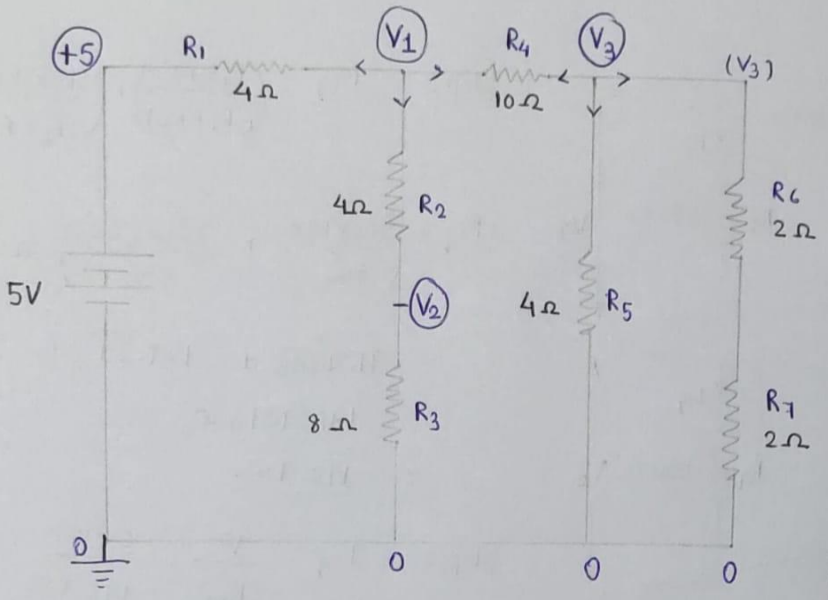


DC OP 1

Signal	Value
PR10: V(6)-V(0)	2.0000V
PR12: V(3)-V(0)	500.00mV
PR14: V(4)-V(0)	250.00mV
PR1: I(R1)	500.00mA
PR2: I(R2)	250.00mA
PR3: I(R3)	250.00mA
PR4: I(R4)	250.00mA
PR5: I(R5)	125.00mA
PR6: I(R6)	125.00mA
PR7: I(R7)	125.00mA
PR8: V(1)-V(2)	2.0000V
PR9: V(2)-V(6)	1.0000V
PR11: V(2)-V(3)	2.5000V
PR13: V(3)-V(4)	250.00mV

c.) Compare with theoretical values.

ASSIGNMENT - 2 (U19CS012)



Step 1: Solving using Node Analysis

Two Junction points as shown in figure (V1 & V3)

At Node V1

$$\frac{V_1-5}{4}+\frac{V_1}{(4+8)}+\frac{V_1-V_3}{10}=0$$

$$V_1\left(\frac{1}{4}+\frac{1}{12}+\frac{1}{10}\right)-0.1V_3=1.25$$

$$\boxed{0.433\,V_1-0.1V_3=1.25} \quad Eq^n\ 1$$

Step 2: At Node V_3:

$$\frac{V_3-V_1}{10}+\frac{V_3}{4}+\frac{V_3}{(2+2)}=0$$

$$V_3(0.6)-0.1V_1=0$$

$$\boxed{V_1=6V_3} \longrightarrow Eq^n\ 2$$

$$V_3(0.433\times 6-0.1)=1.25$$

$$V_3=0.5004\ V$$

$$\boxed{V_3 \simeq 0.5V}$$

$$\boxed{V_1=3V}$$

Across Resistor	Voltage (volt)	Current
R_1	$5-V_1=\boxed{2V}$	$I_{R_1}=\frac{V_{R_1}}{(R_1)}=\frac{2V}{4\Omega}=0.5A=\boxed{500\ mA}$
R_2	$V_{R_2}=I_{R_2}\times R_2=(0.25A)\times 4\Omega=\boxed{1V}$	$I_{R_2}=\frac{V_1}{(R_2+R_3)}=\frac{3}{12}=0.25A=\boxed{250\ mA}$
R_3	$V_{R_3}=I_{R_3}\times R_3=\underset{(A)}{0.25}\times\underset{(\Omega)}{8}=\boxed{2V}$	$I_{R_3}=I_{R_2}=0.25A=\boxed{250\ mA}$
R_4	$V_{R_4}=V_1-V_3=\boxed{2.5V}$	$I_{R_4}=\frac{V_{R_4}}{R_4}=\frac{2.5V}{10\Omega}=\boxed{250\ mA}=0.25A$
R_5	$V_{R_5}=V_3=\boxed{0.5V}$	$I_{R_5}=\frac{V_{R_5}}{R_5}=\frac{0.5}{4}=0.125A=\boxed{125 mA}$
R_6	$V_{R_6}=I_{R_6}\times R_6=\boxed{0.25V}$	$I_{R_6}=V_3/(R_6+R_7)=\frac{0.5}{4}=\boxed{125\ mA}$
R_7	$V_{R_7}=I_{R_7}\times R_7=\boxed{0.25V}$	$I_{R_7}=I_{R_6}=\boxed{125\ mA}$

d.) Final Result and Conclusion

Resistor	Voltage (V)		Current (mA)	
	Multism	Theoretical	Multism	Theoretical
R1	2	2	500	500
R2	1	1	250	250
R3	2	2	250	250
R4	2.5	2.5	250	250
R5	0.5	0.5	125	125
R6	0.25	0.25	125	125
R7	0.25	0.25	125	125

Conclusion:

We can observe from Above Table, Both the *Theoretical* and *Multisim* Values of Current and Voltage are **Equal**.
Hence, Experiment is Performed Successfully (without any Error).