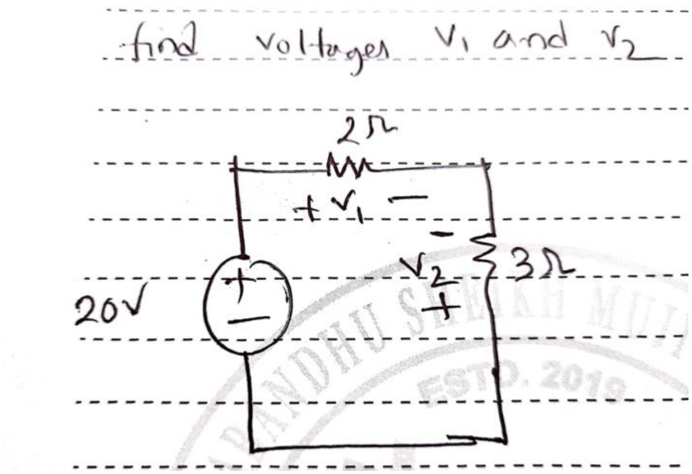
Basic Electric Circuits Simulation

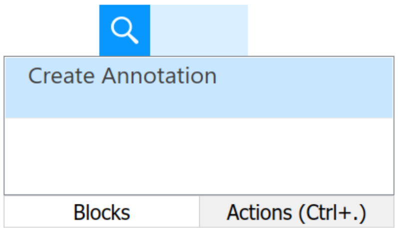
Example 01:



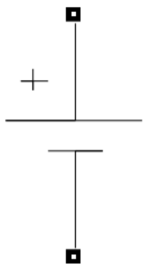
New -> Simulink Model

Select Blank Model

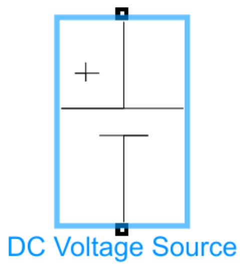
You double click on the platform, a search bar appear and you write the component you want.

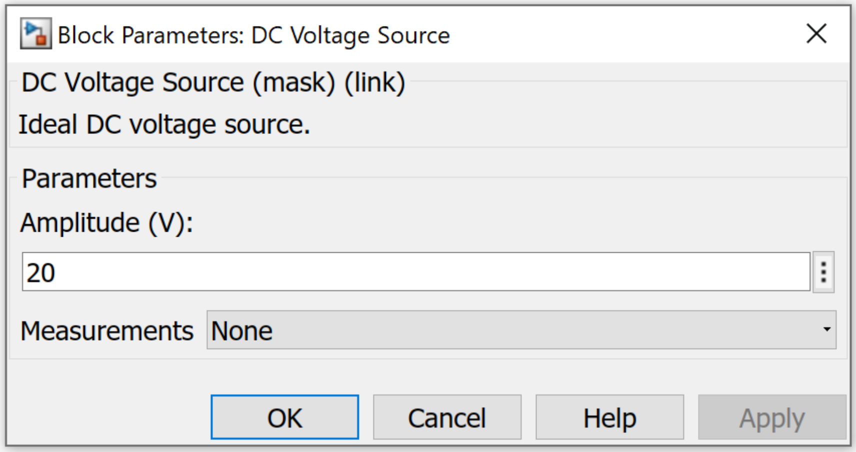


We select 'DC Voltage Source (Electrical/Specialized Power Systems/Sources)'



You get something like this. Double click on this to set value.

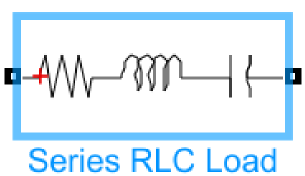




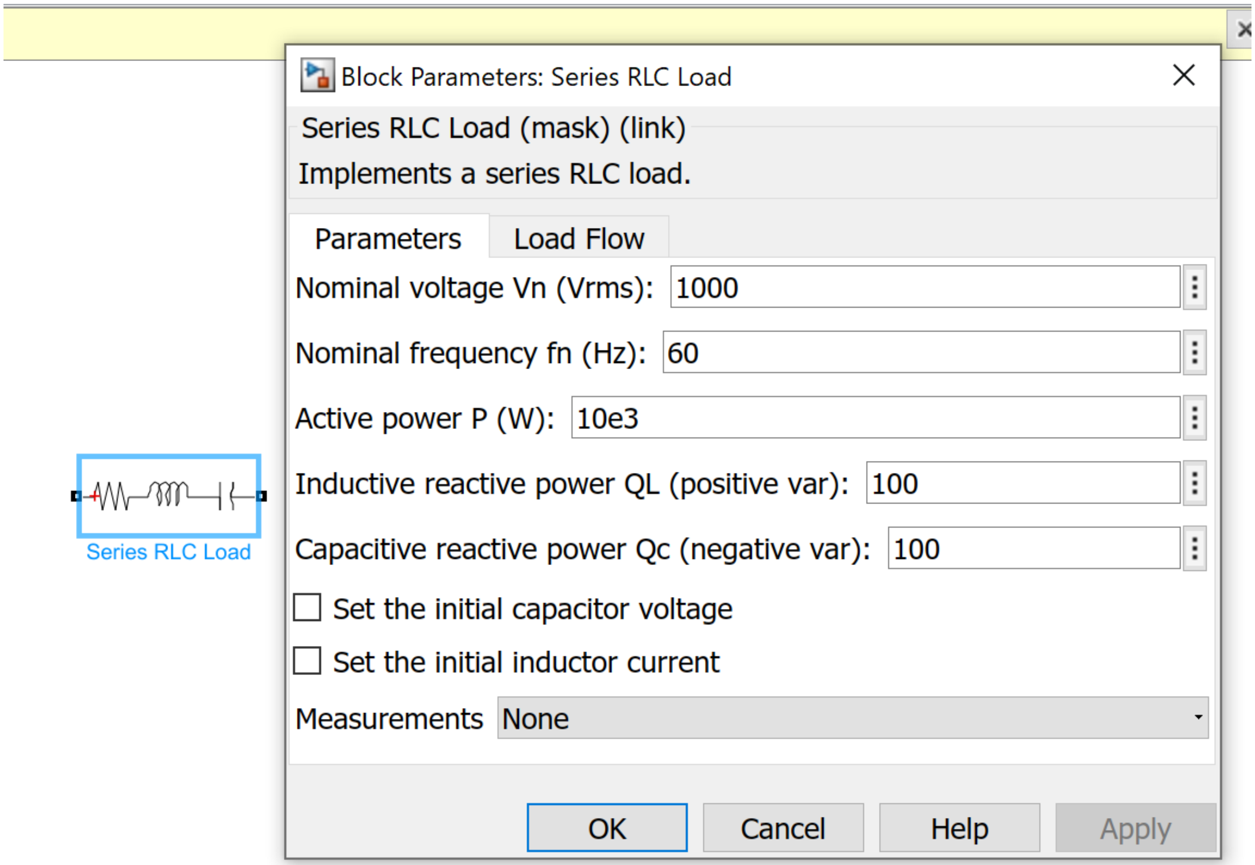
Setting the value 20 V.

Now, same way in the bar write rlc, you will get several options.

This is Series RLC Load

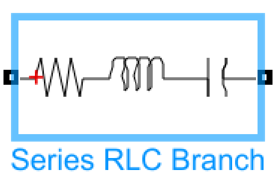


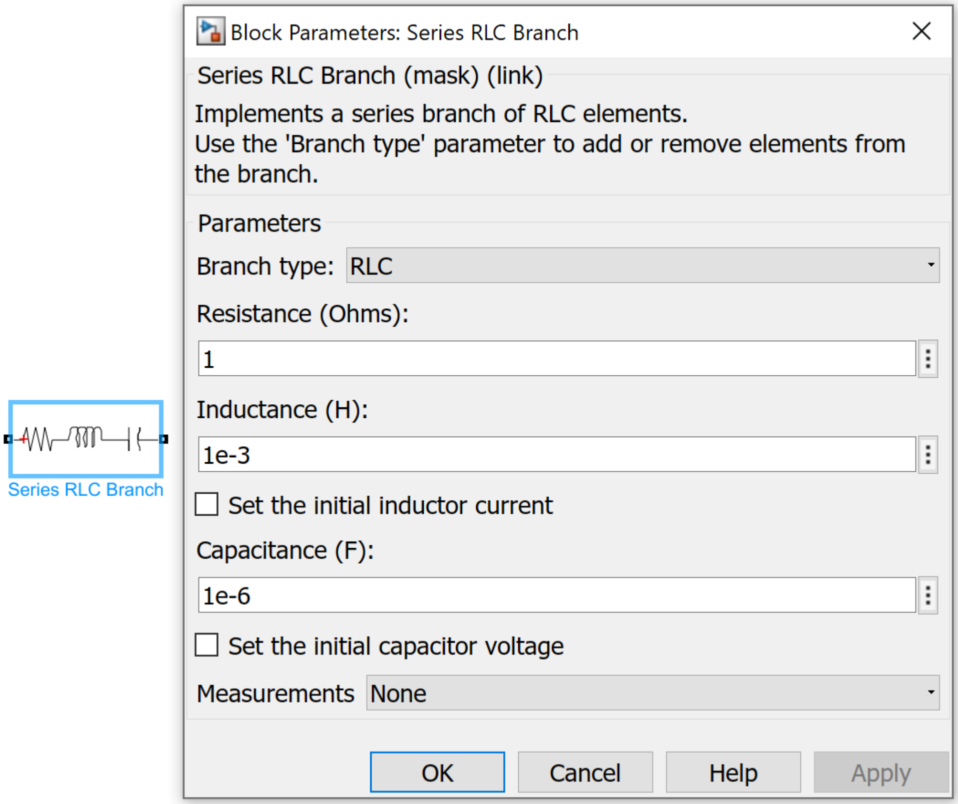
Double click on it:



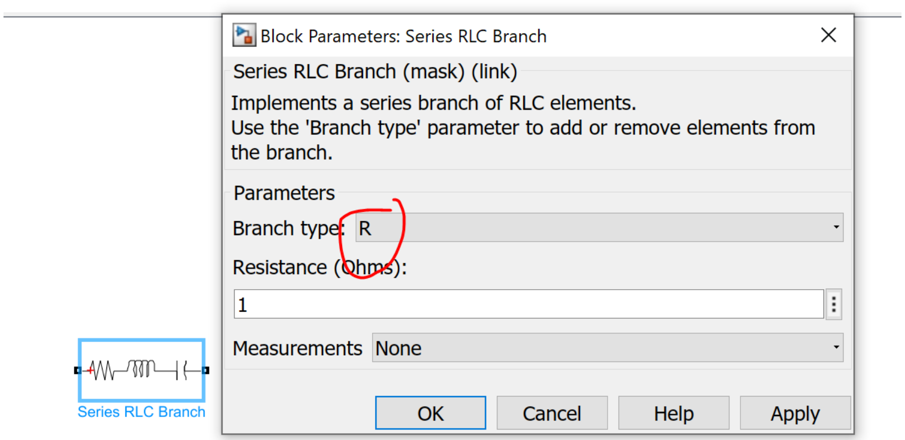
Now, let's bring Series RLC Branch

And double click on it.

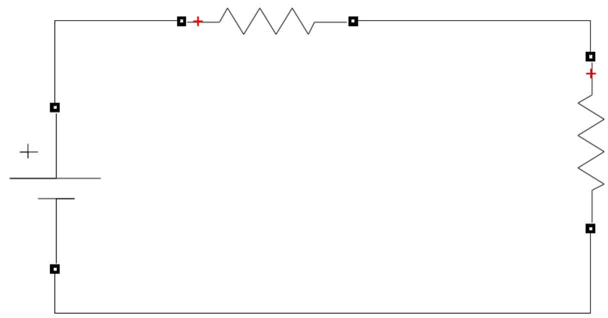




They may look the same, but they are not. For our circuit, as we want to just set values for the passive elements (R), we will consider the Series RLC Branch. Click on Branch Type and select R.

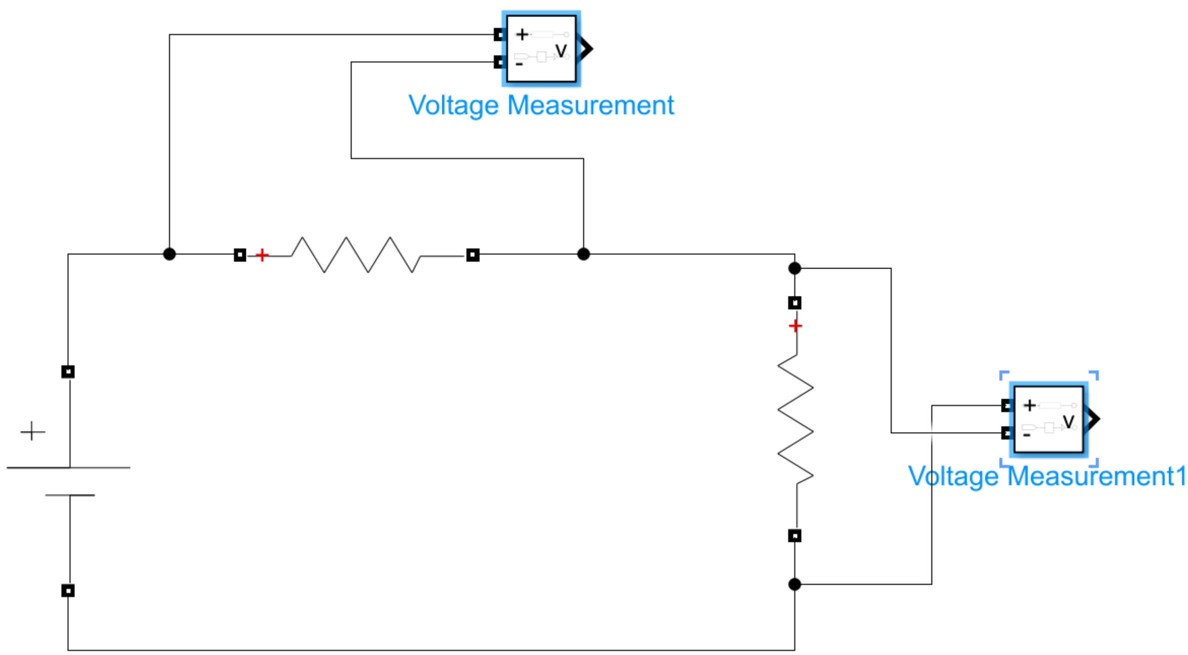


We set the value 2 ohm. You can just copy the element by selecting it and Clt+C and Clt + V. Right click on it to find options for Rotate and Flip. You can also use Clt + R



We want to find voltages across these resistance

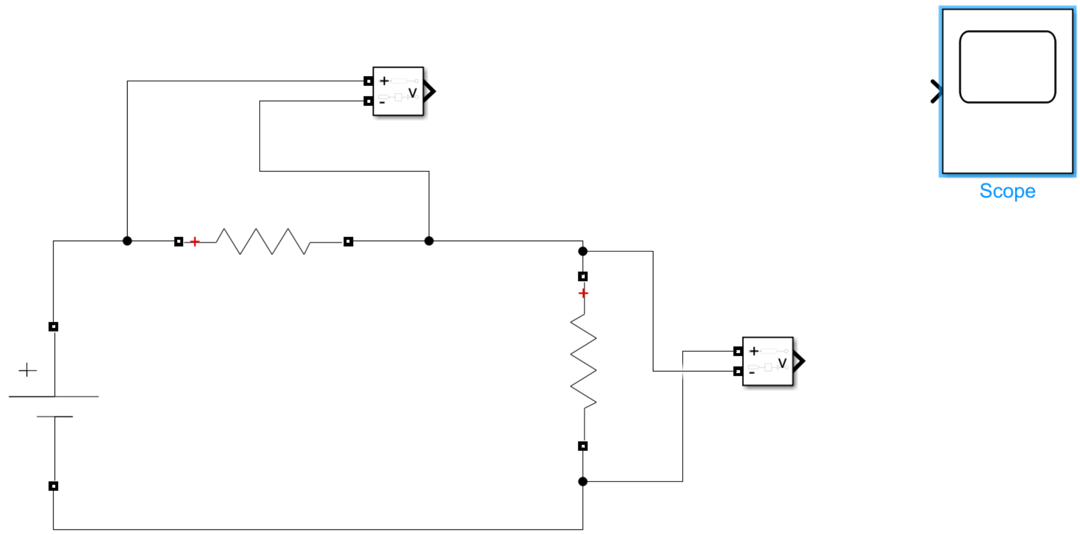
So, we now bring voltage measurement



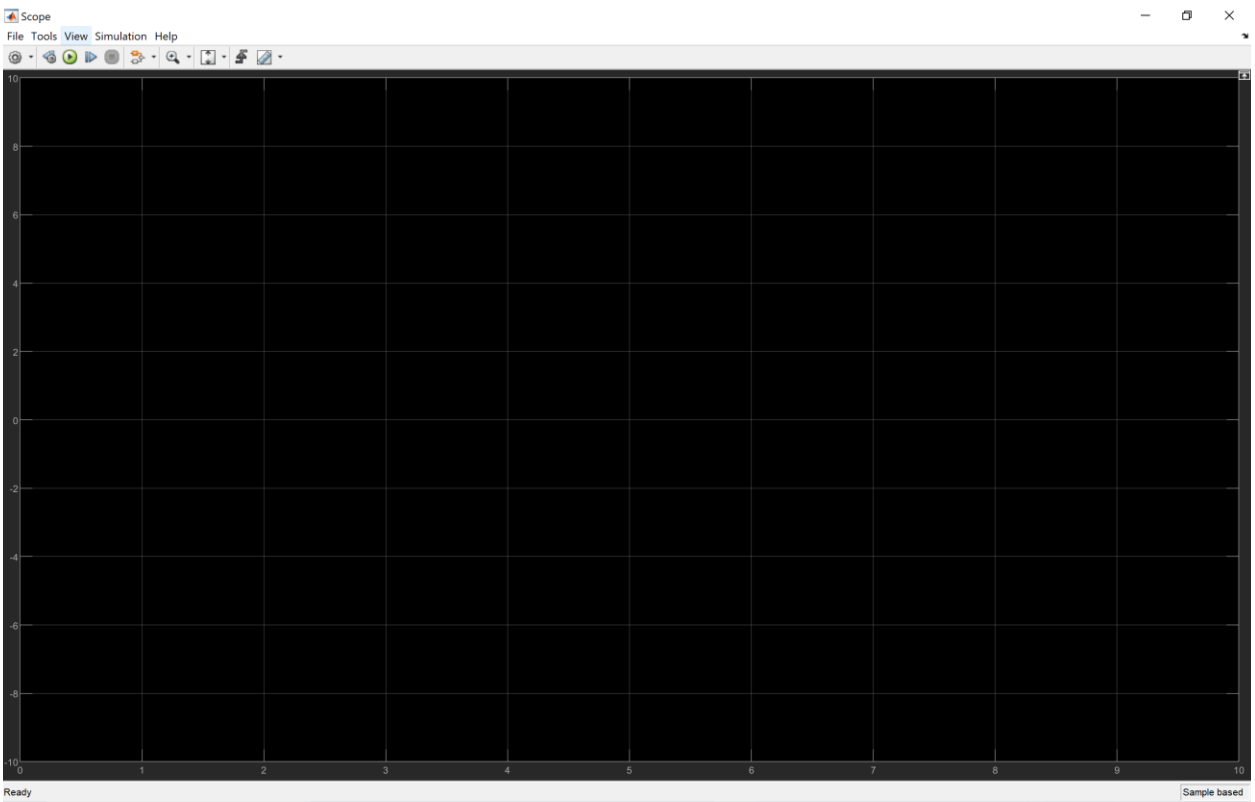
See, where we connect the measuring device. See the provided circuit. On the wire, not with the elements directly. Also, see that + will go with the + and – will go with the -.

For v2, see that – point is in the above side. You will have to connect accordingly.

Now we bring scope, which help us to observe output voltage form.

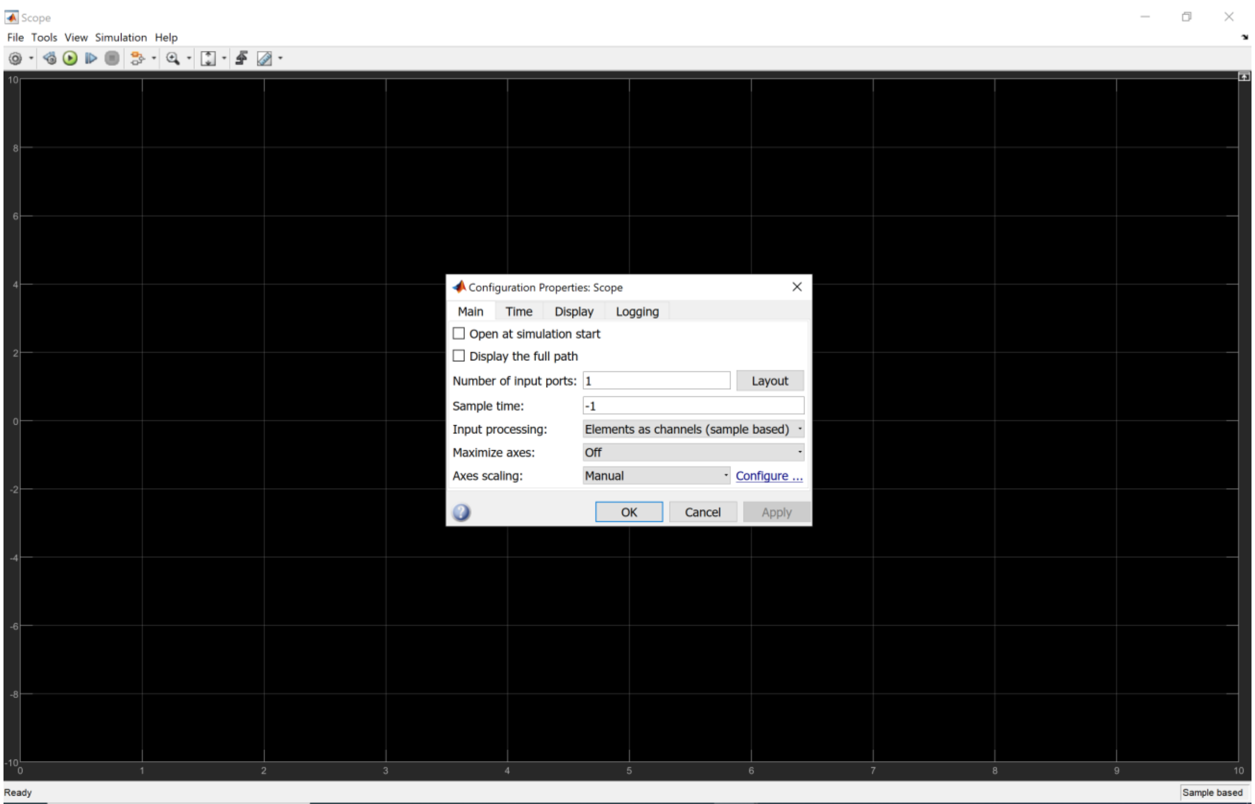


Double click on the scope. A window will appear.



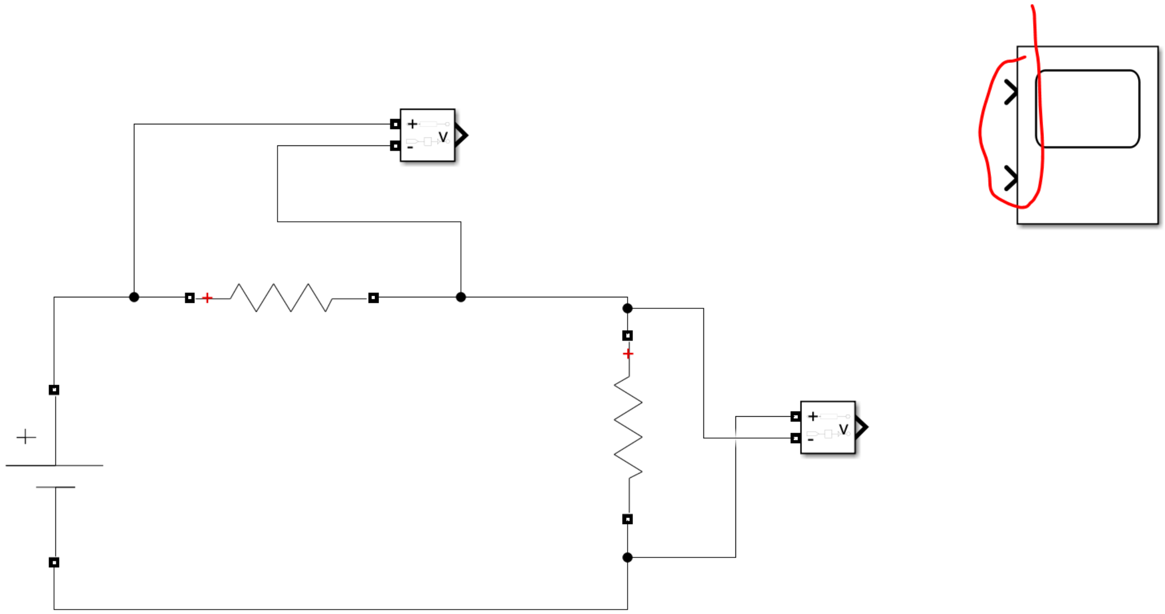
Click on the following:

You get this:

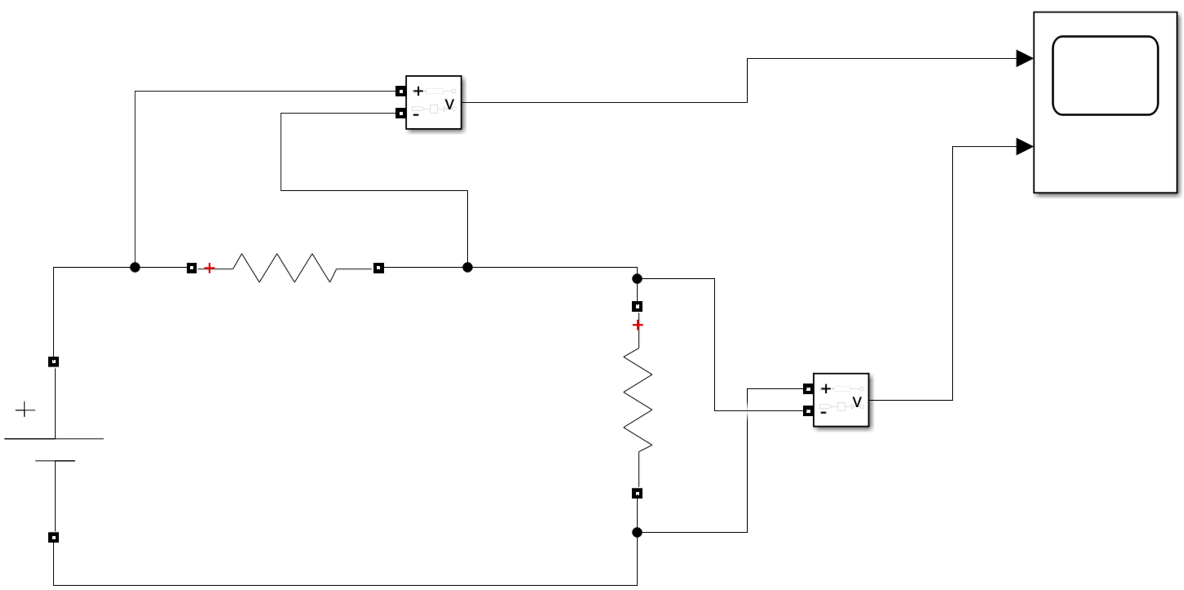


As we are measuring two quantities, we have two inputs. Write 2. And then ok.

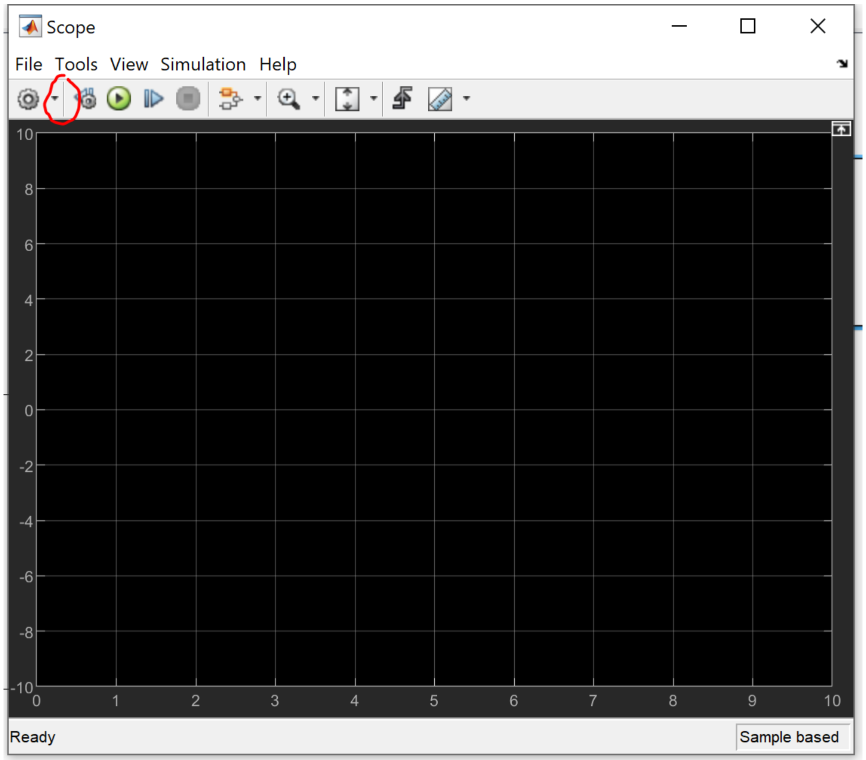
We have two input now in the scope



So, we now have our system:

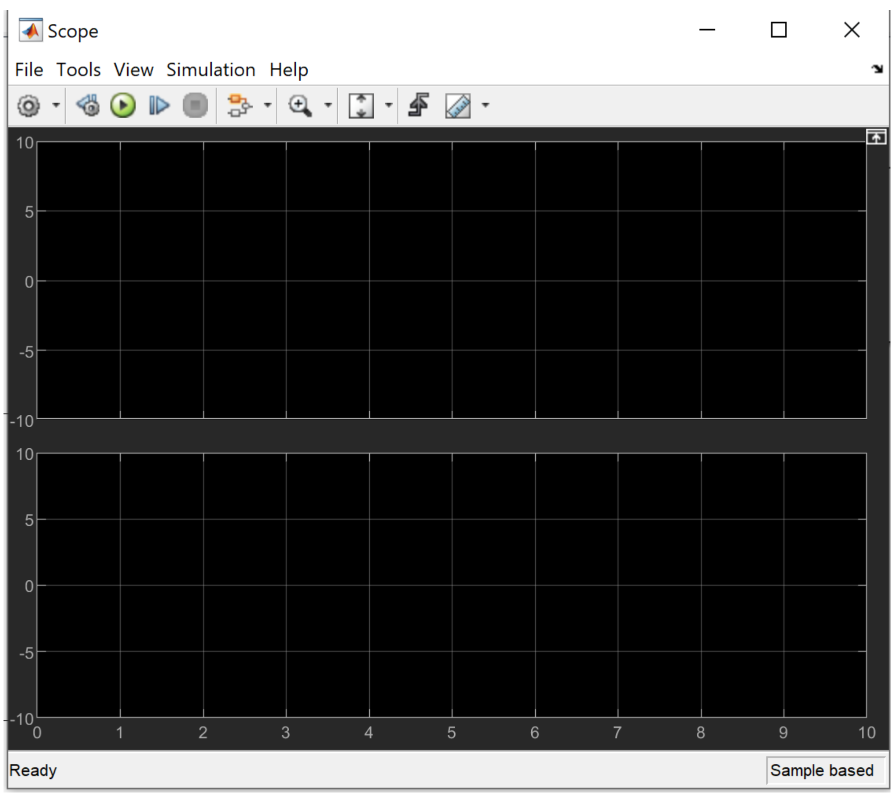


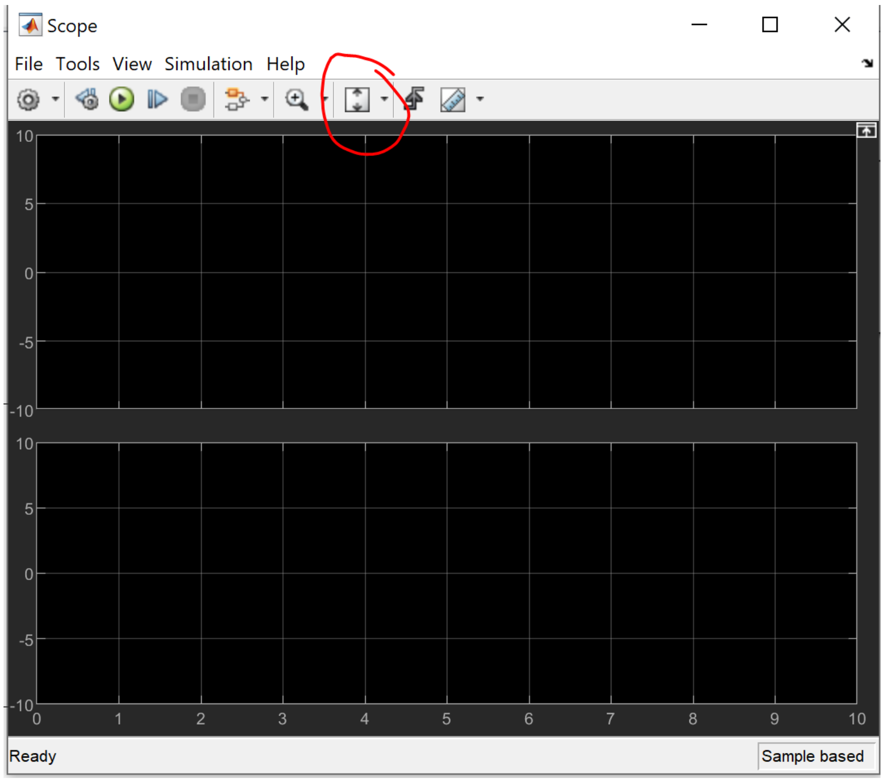
Now again, we will click on the scope. And click on this:



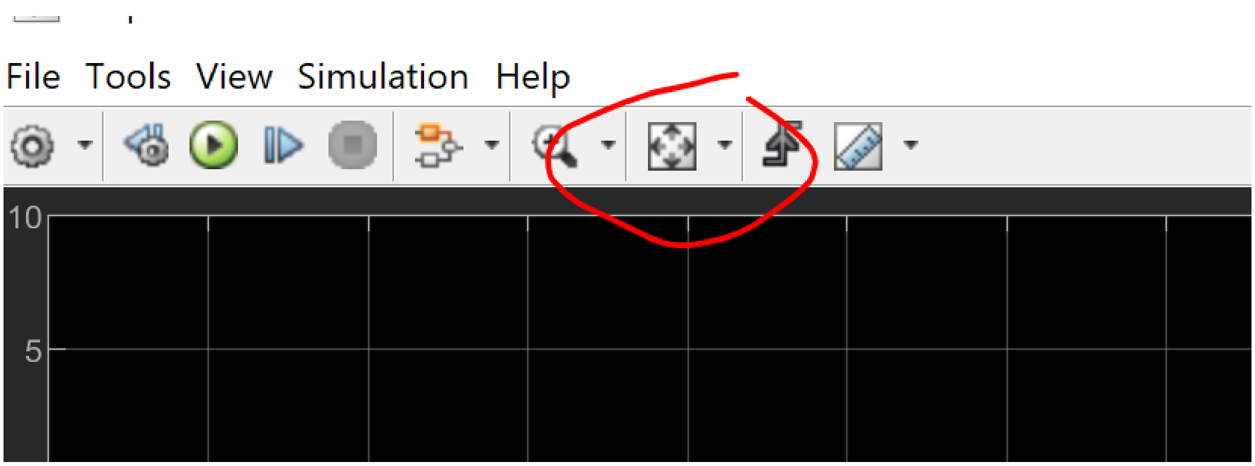
You will find four options, one is the 3rd one, that is layout. Select two squares from first column.

It now becomes like this:



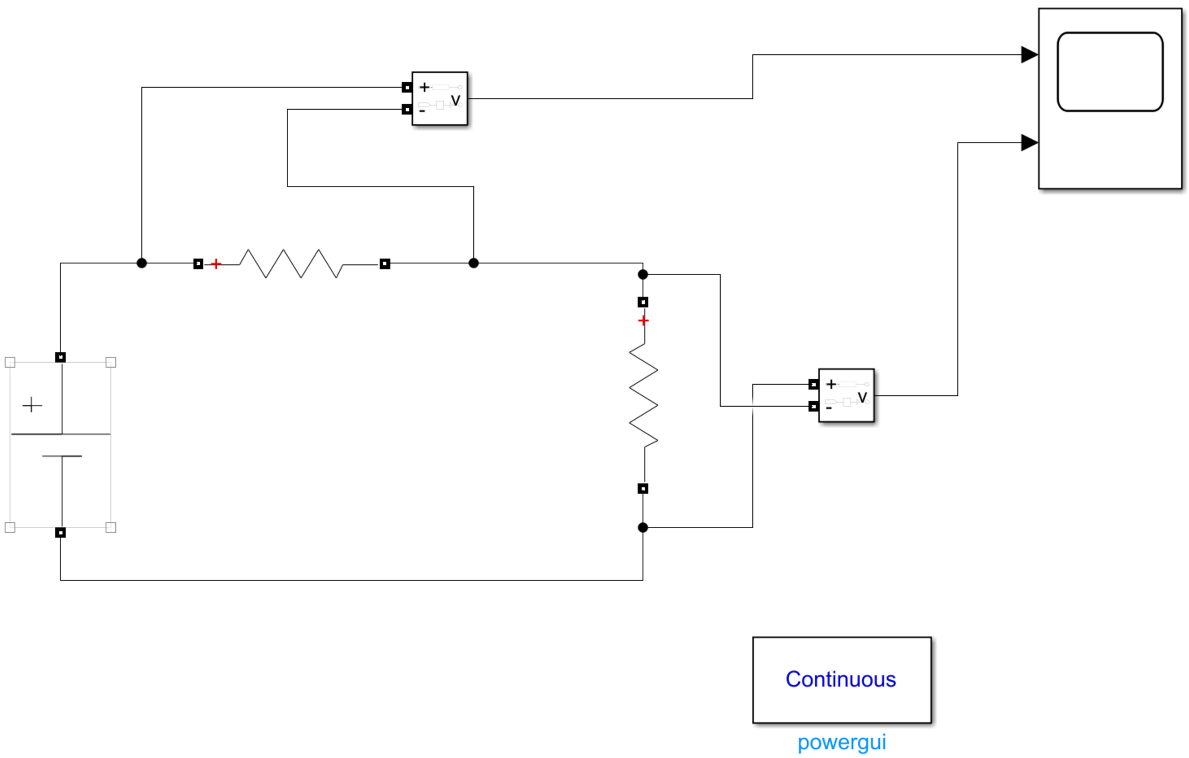


Now, change this one to consider both x and y

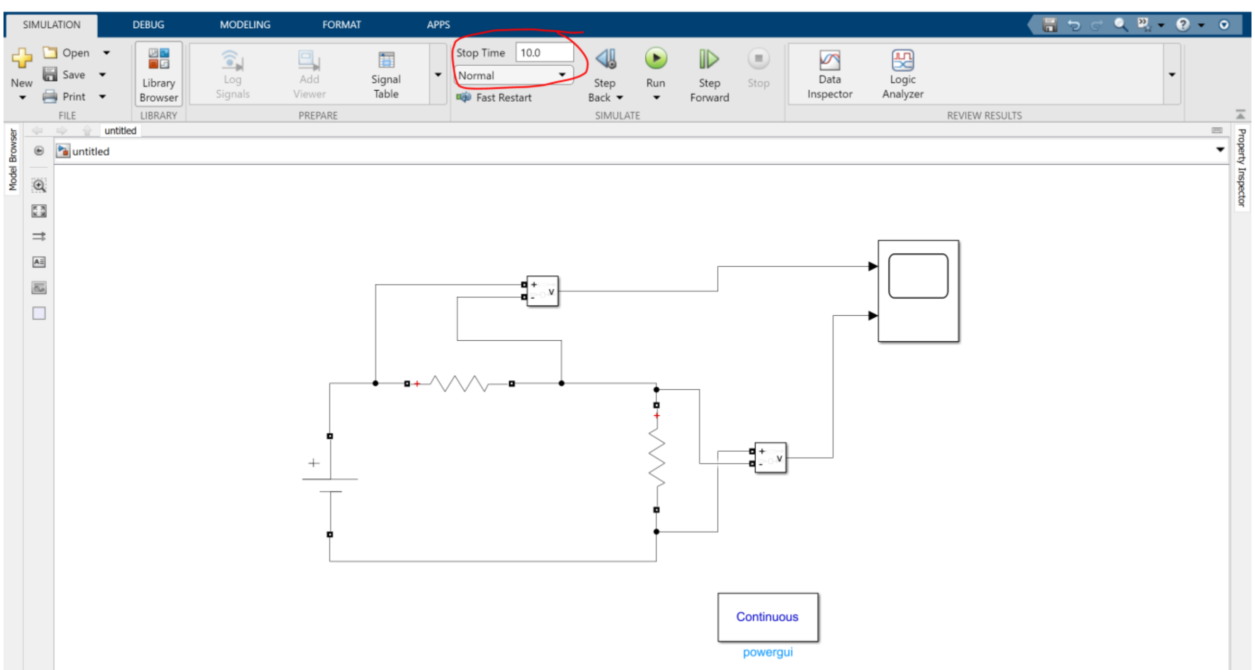


Now bring powergui. This one is used for power system simulation.

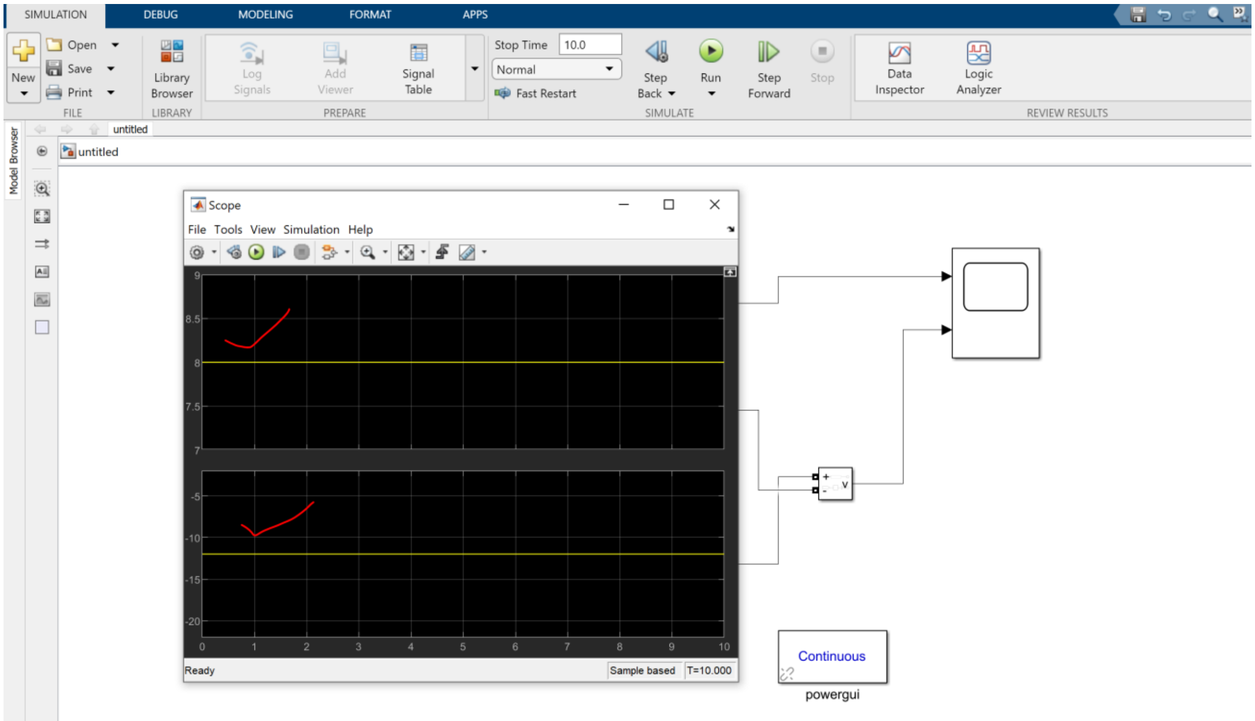
This is my who system:



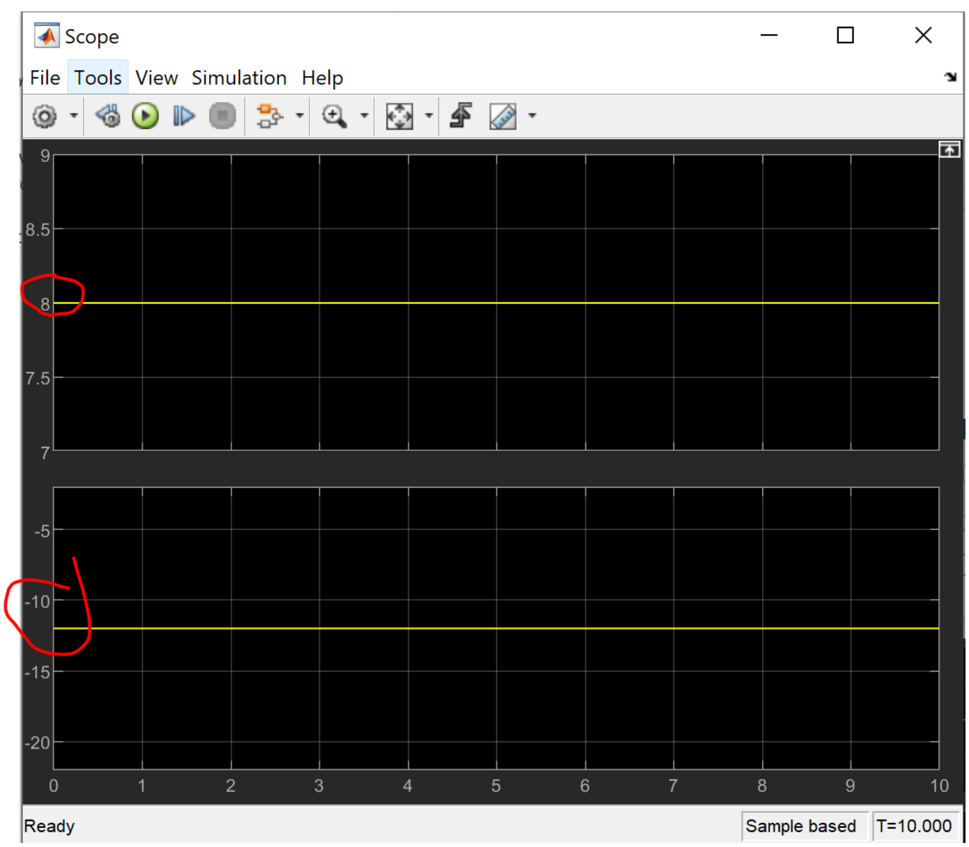
Now, I set time = 10, meaning the simulation will run for 10 seconds.



Now, click run.

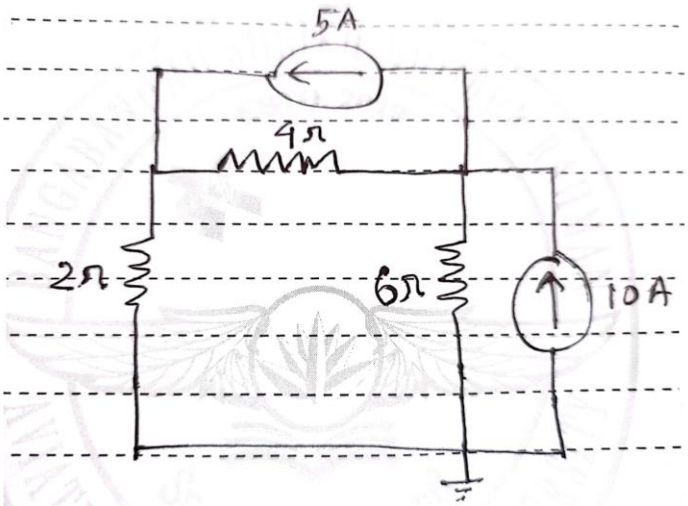


We get v1 = 8V and v2 = -12



Example 02:

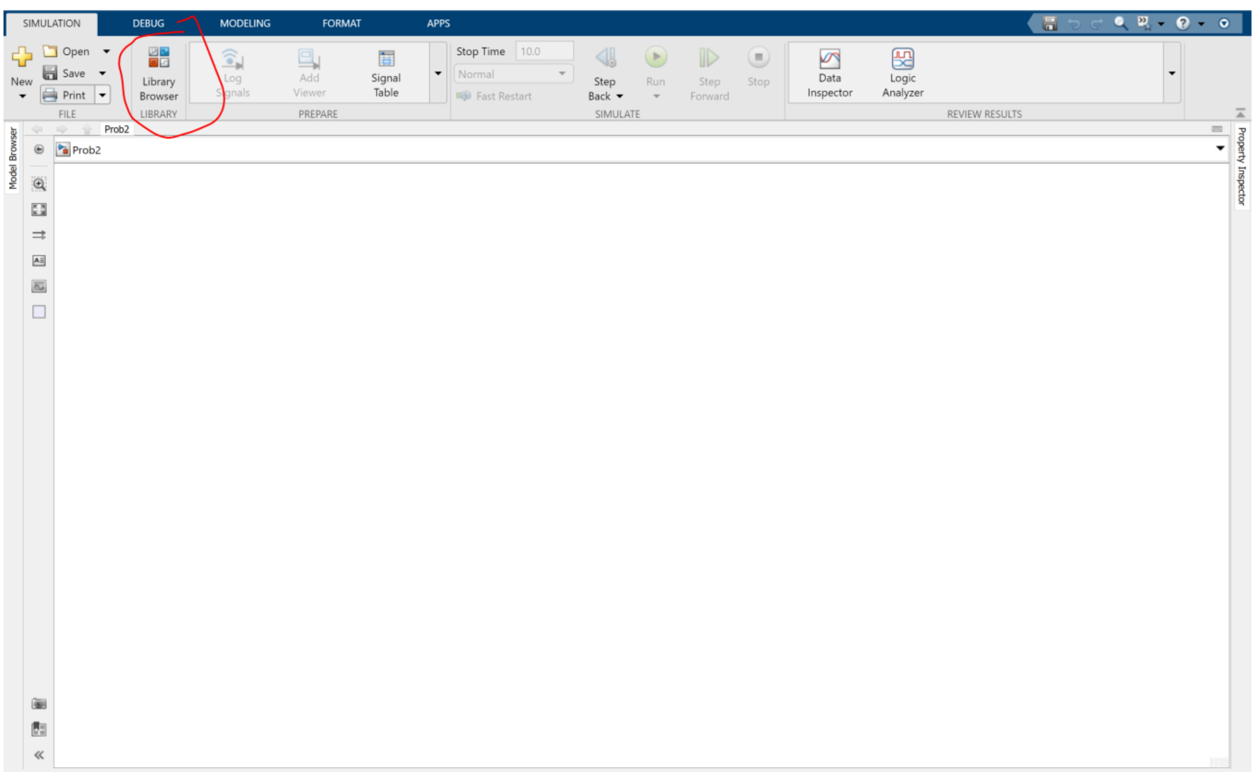
Calculate the node voltages in the following circuit:

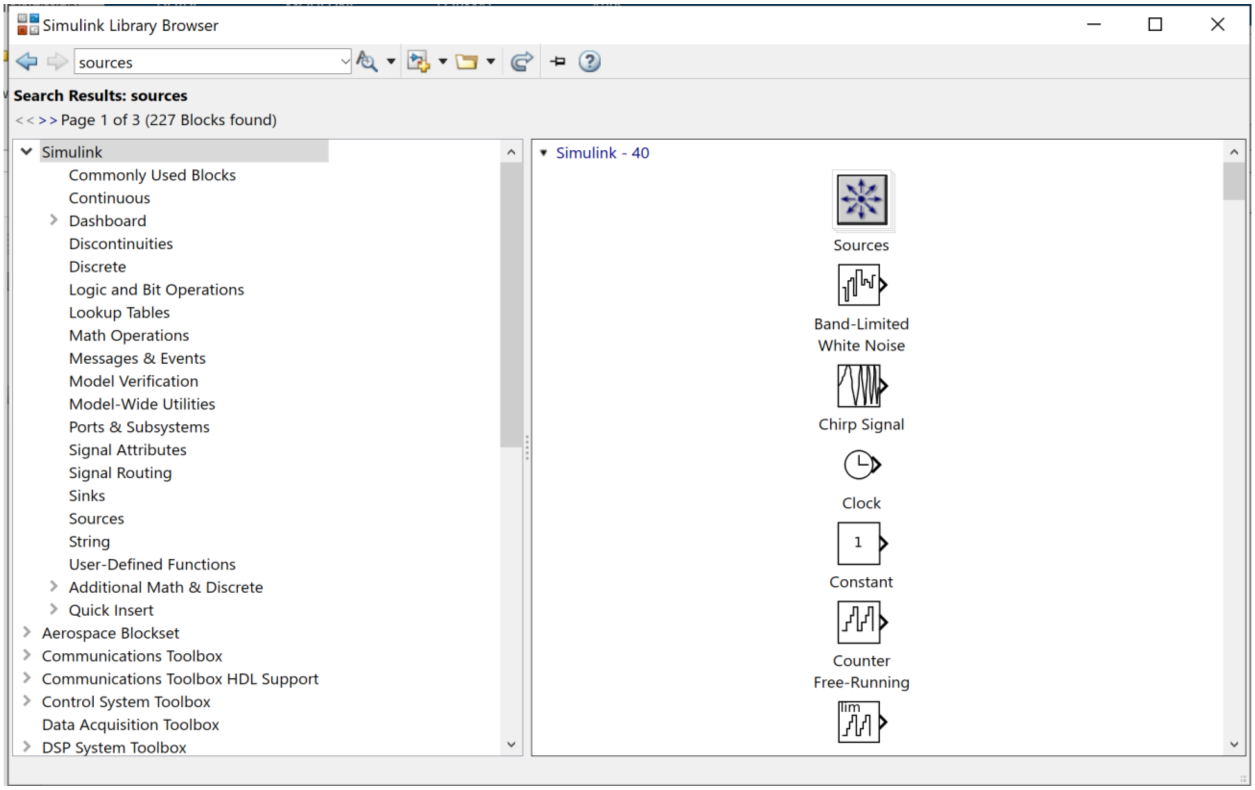


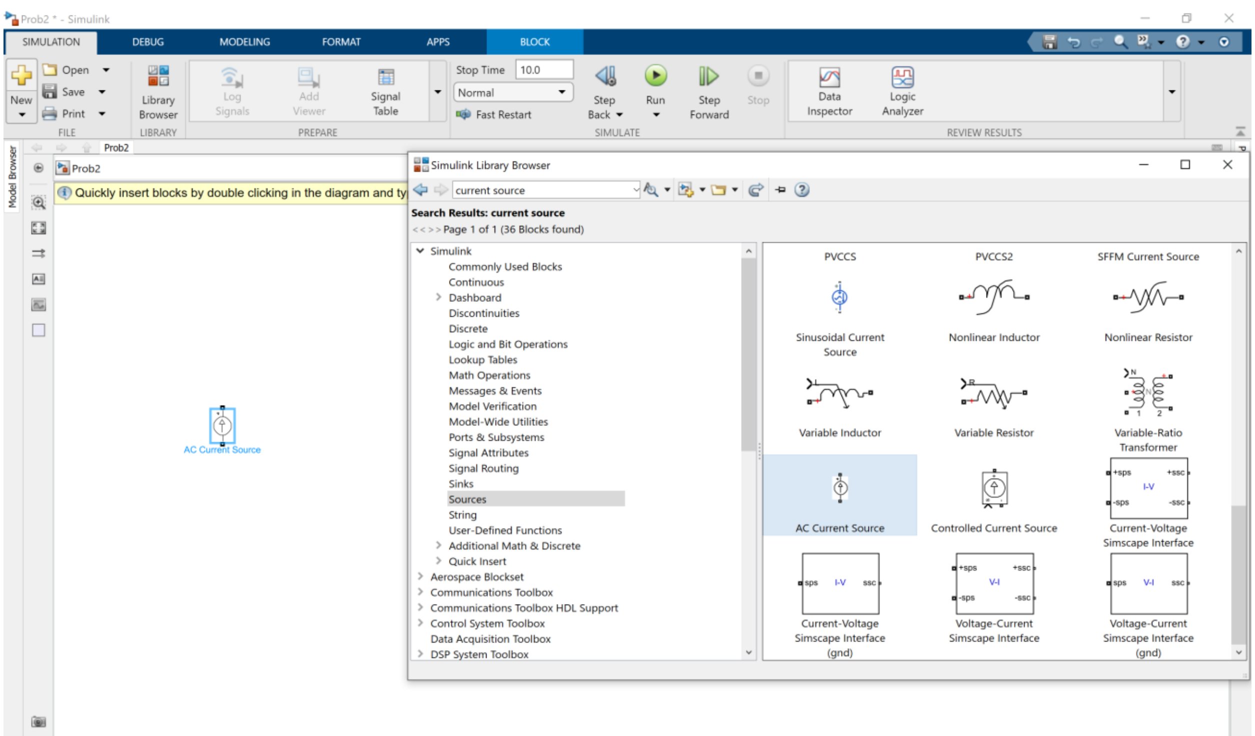
Now, let's build this in the Simulink.

MATLAB does not have direct icon for DC Current Source. So let's try this way where, we bring AC current source. But we make the frequency zero.

Here, you can browse components:

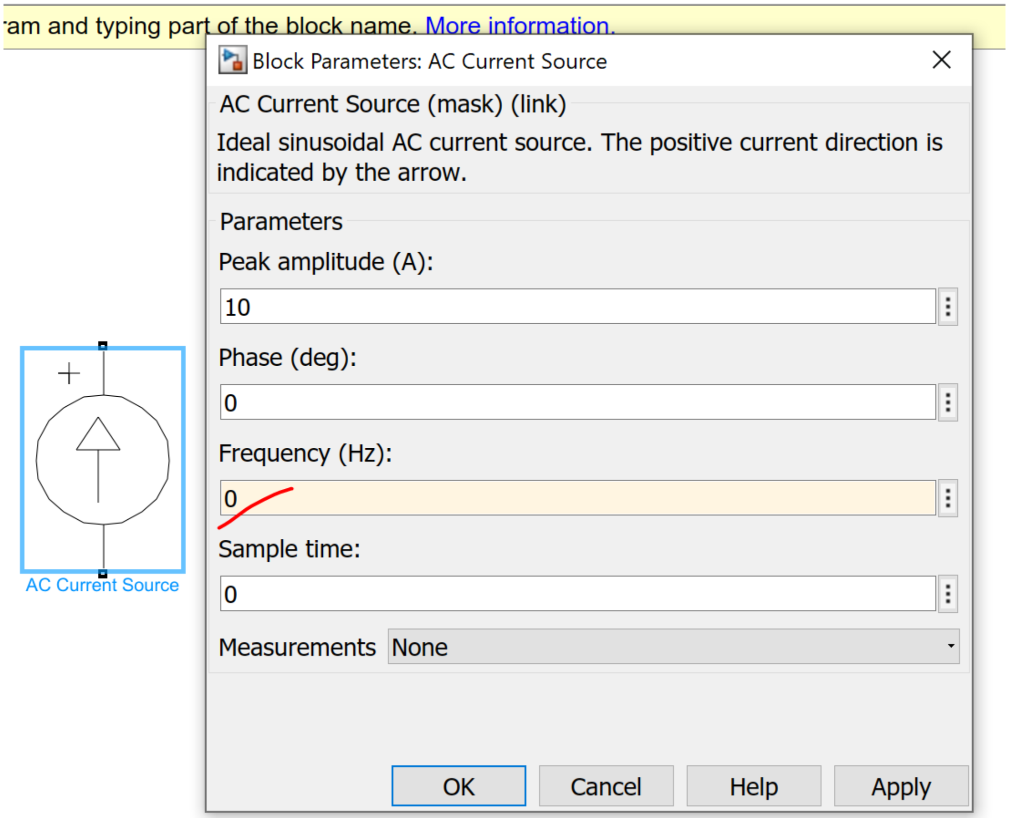




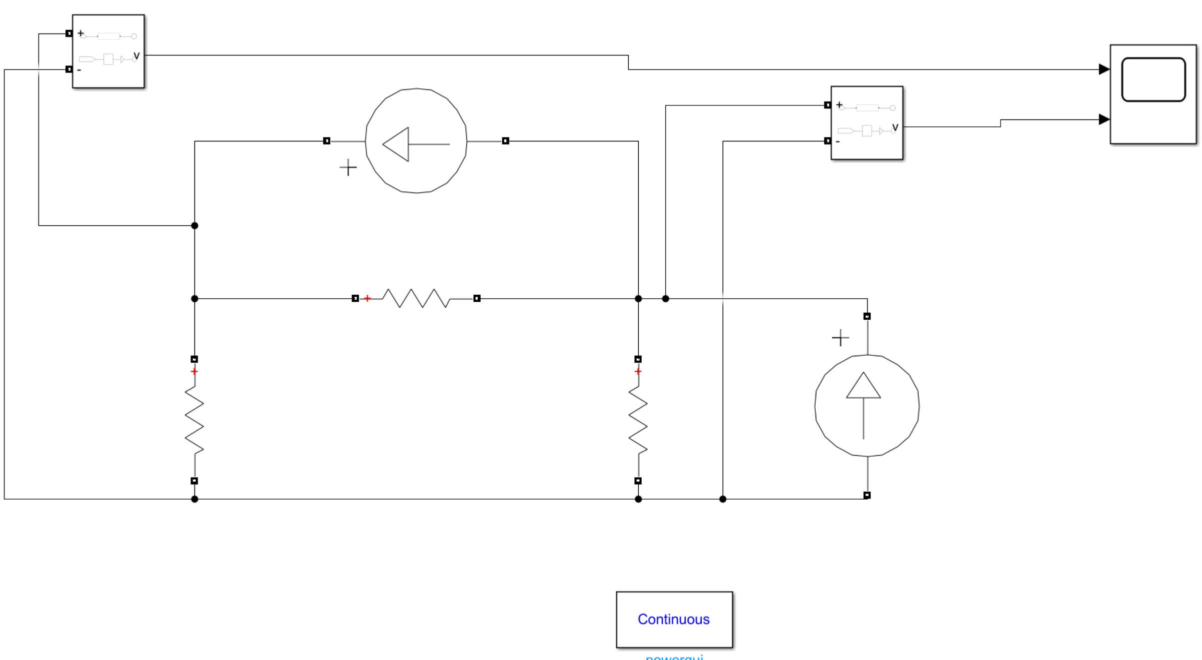


You can just drag the component in the model from the library.

Now, making the frequency zero, it becomes DC

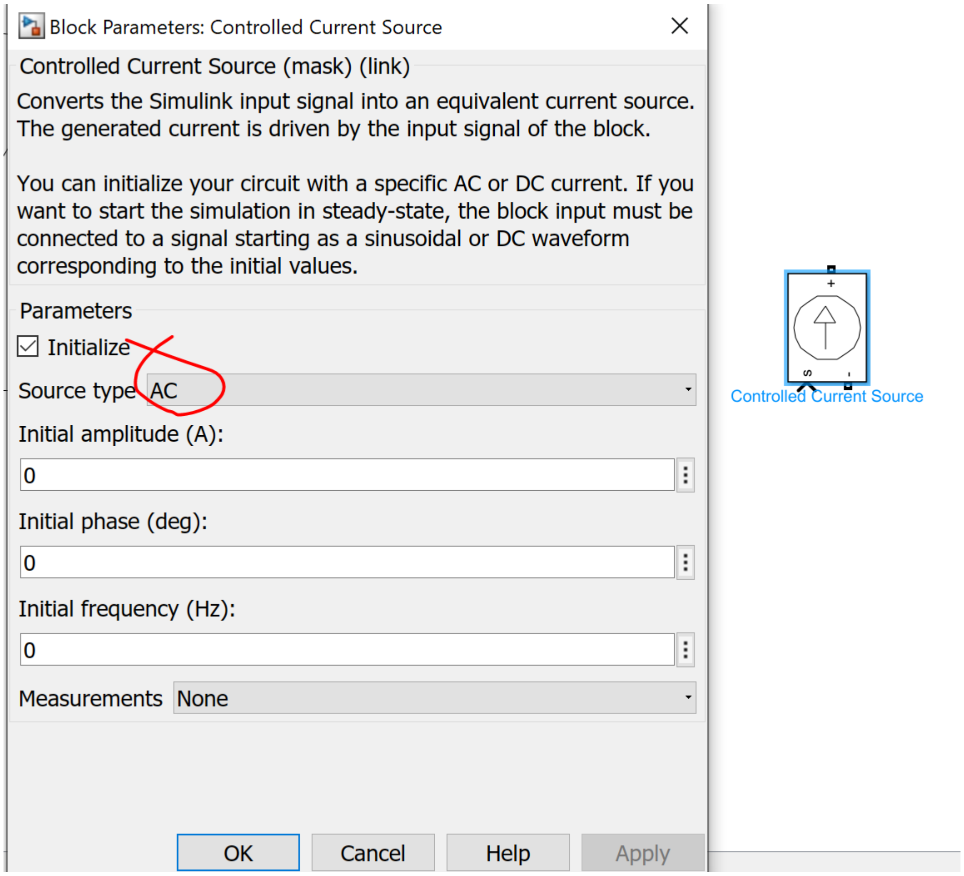


Select the component and doing CLT + R, you can rotate the component.

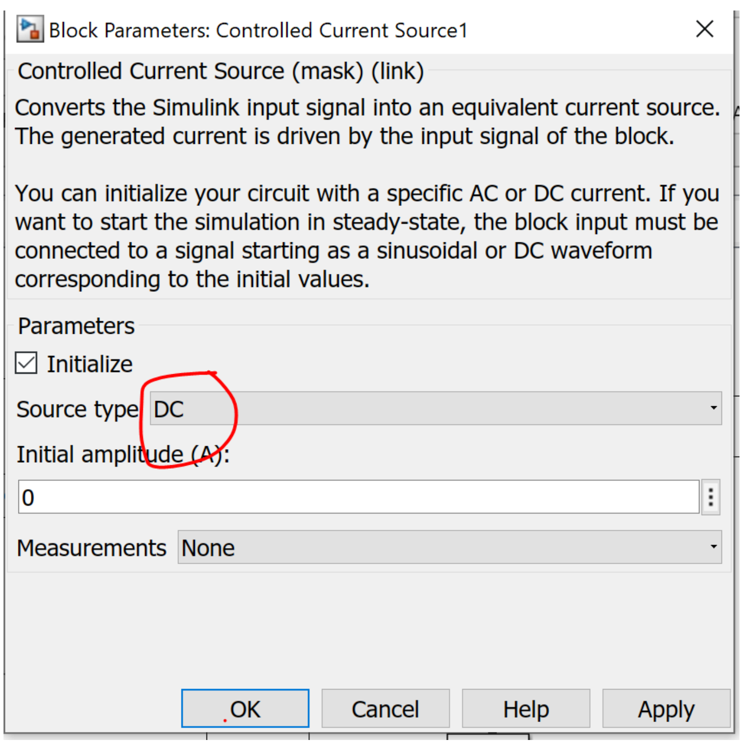


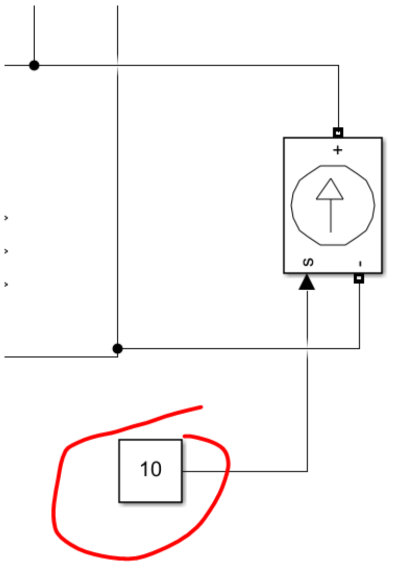
It seems the result is zero. But that should not happen. So, it seems, using AC Current Source with zero frequency do not work.

Let’s try another current source. This is called controlled current source (Electrical/ Specialized Power Systems/ Sources)



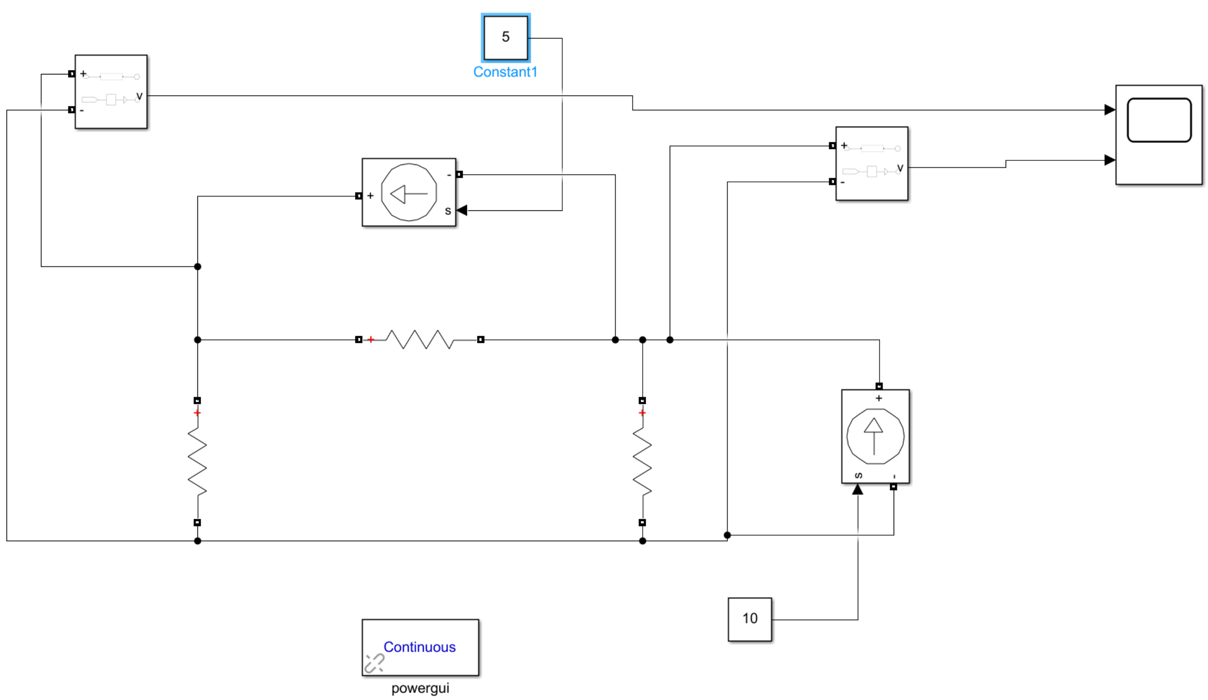
See that it can be AC or DC. Select DC:



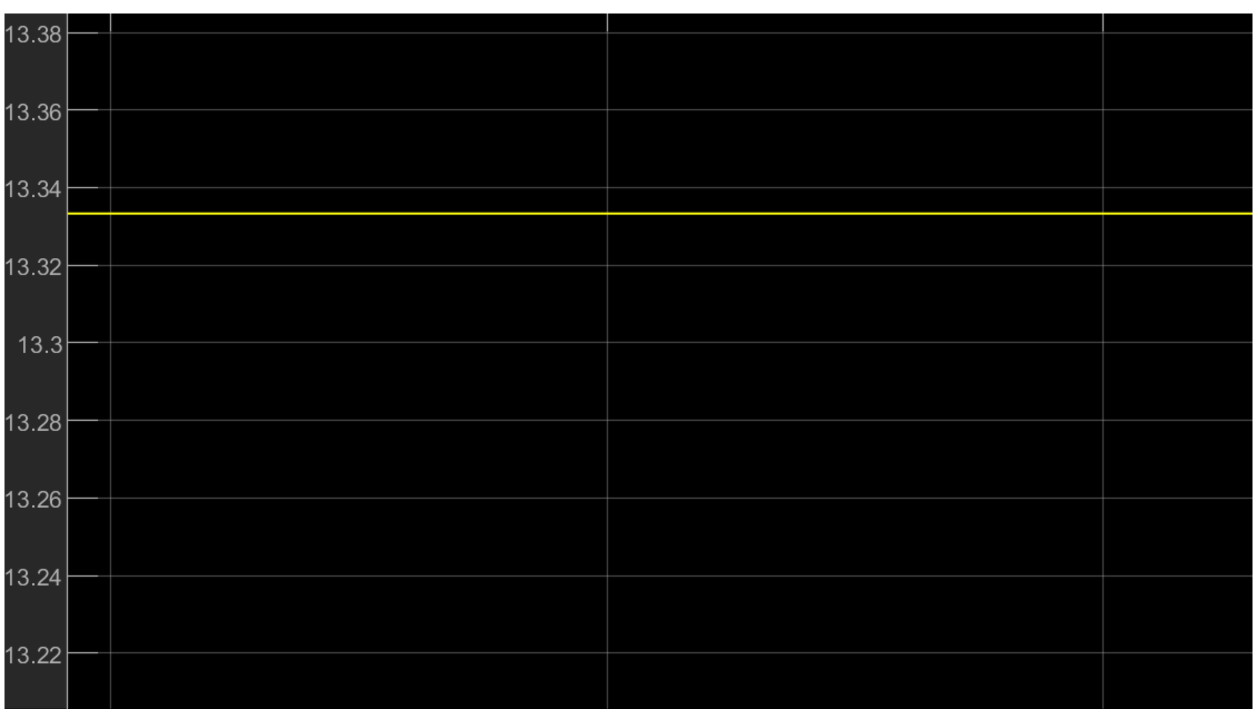


We bring this constant to set value for our controlled source.

Now, here is the whole set up:

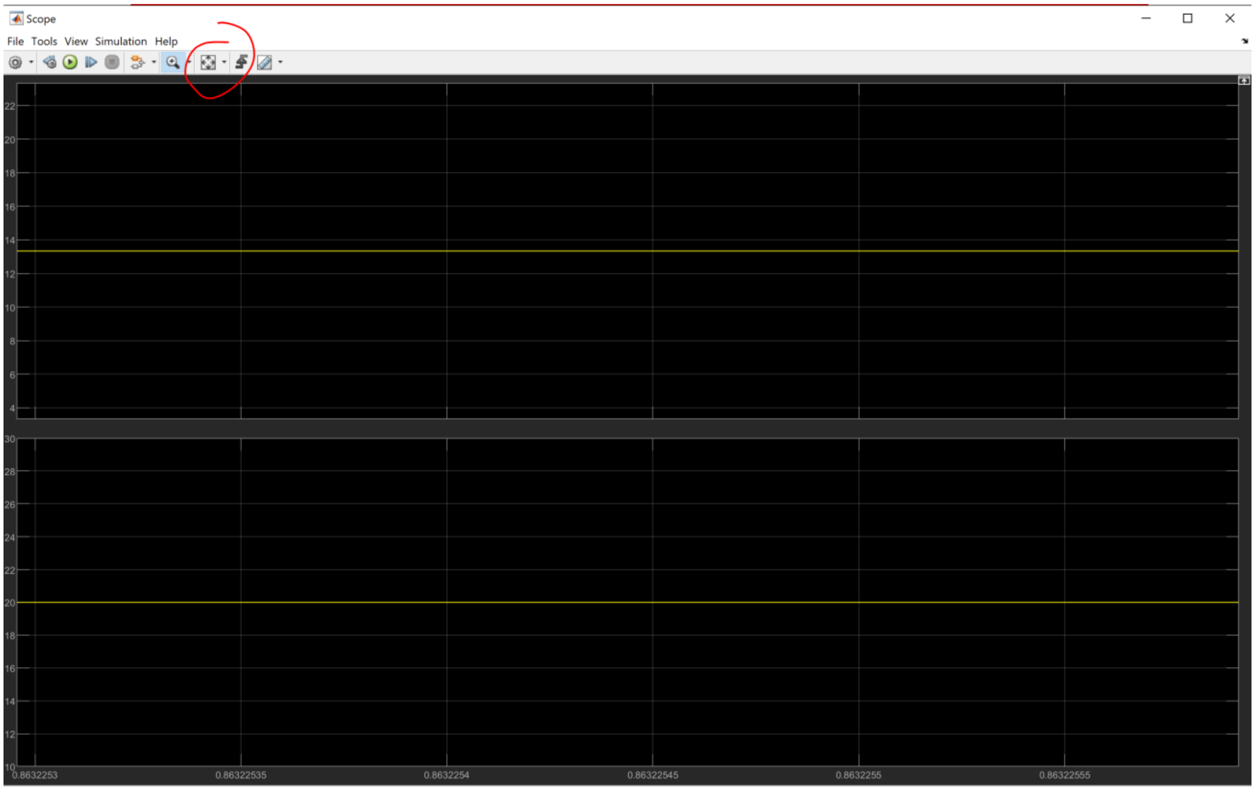


V2 = 20 V. For V1, yu can zoom in on the line to get the exact value.

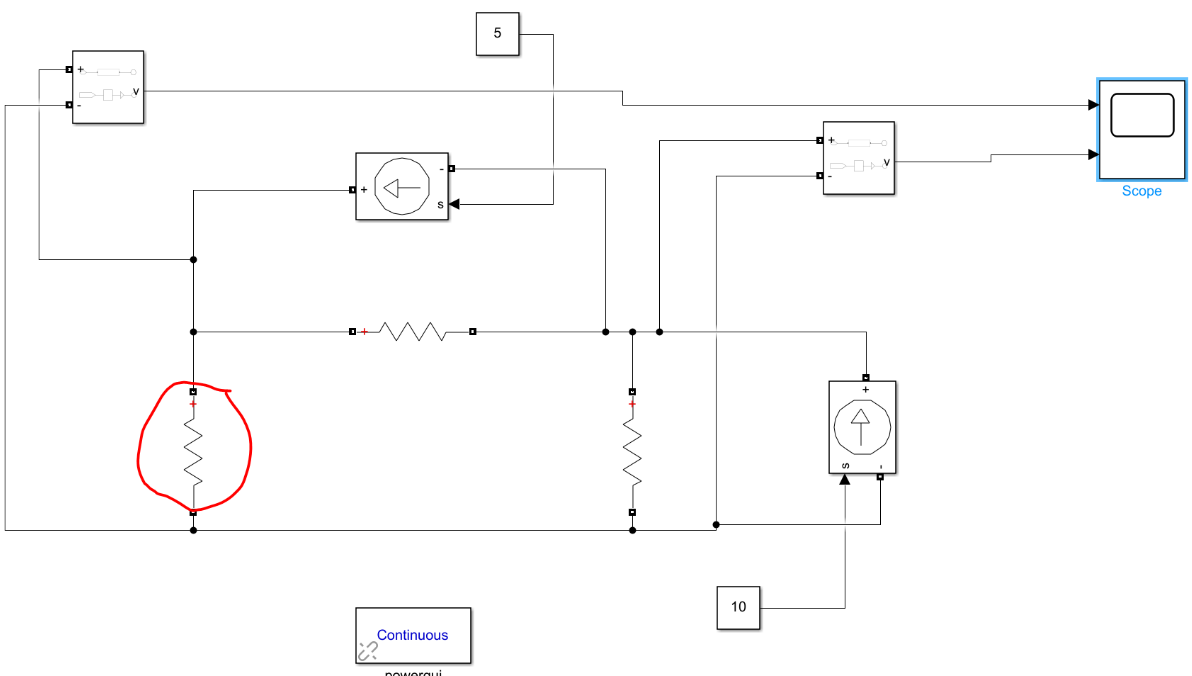


V1 = 13.33 V

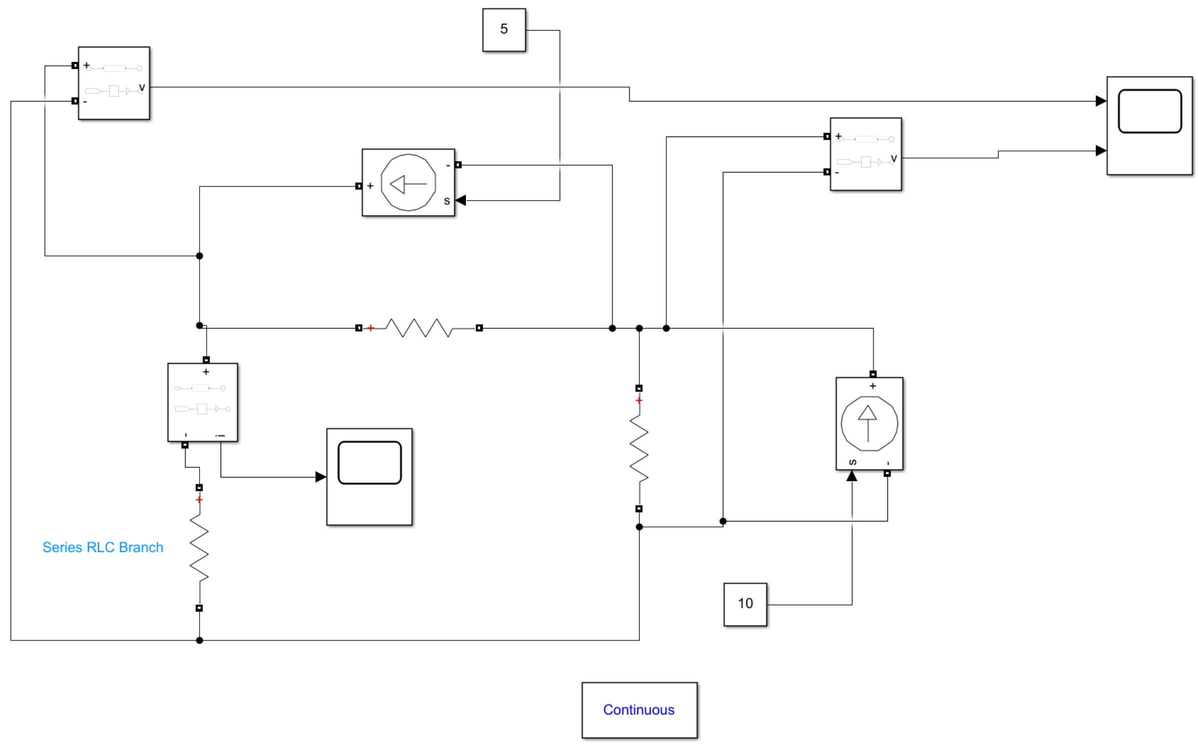
To zoom out click on this:



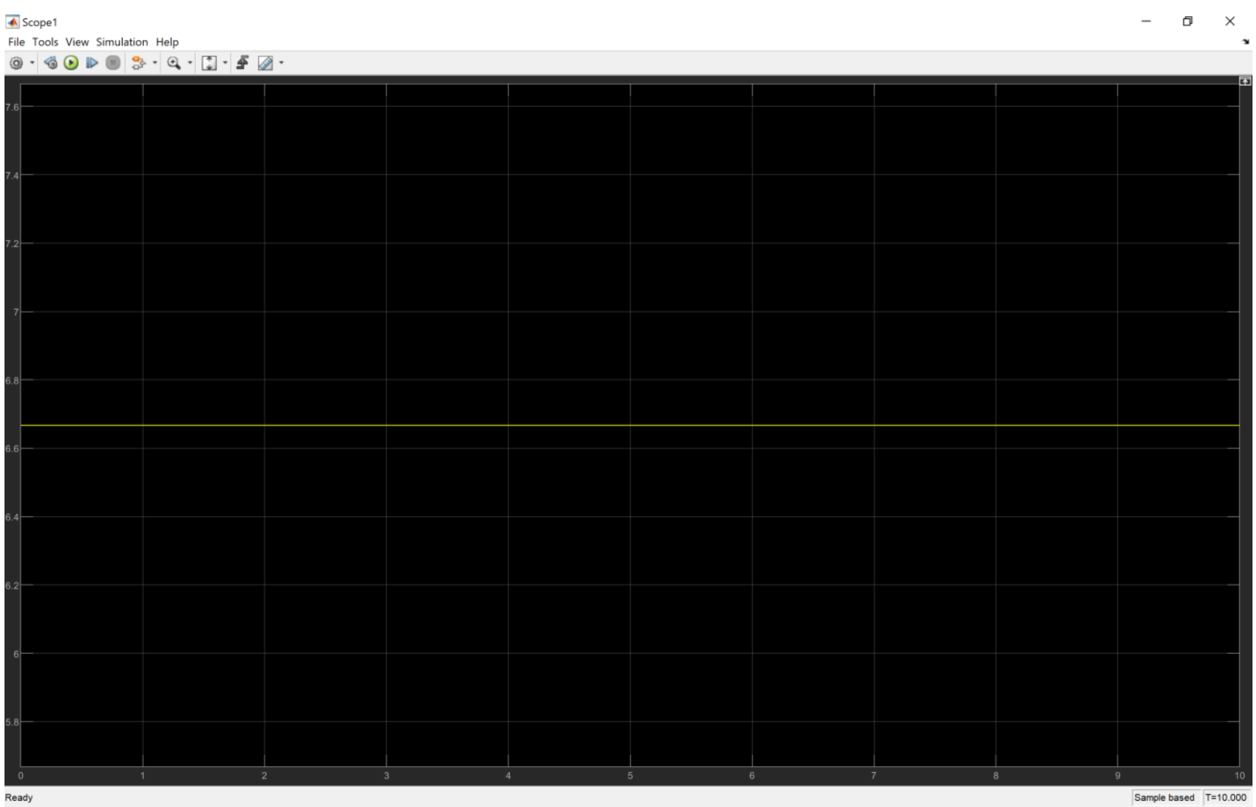
Now let's say you want to measure current in the following branch:



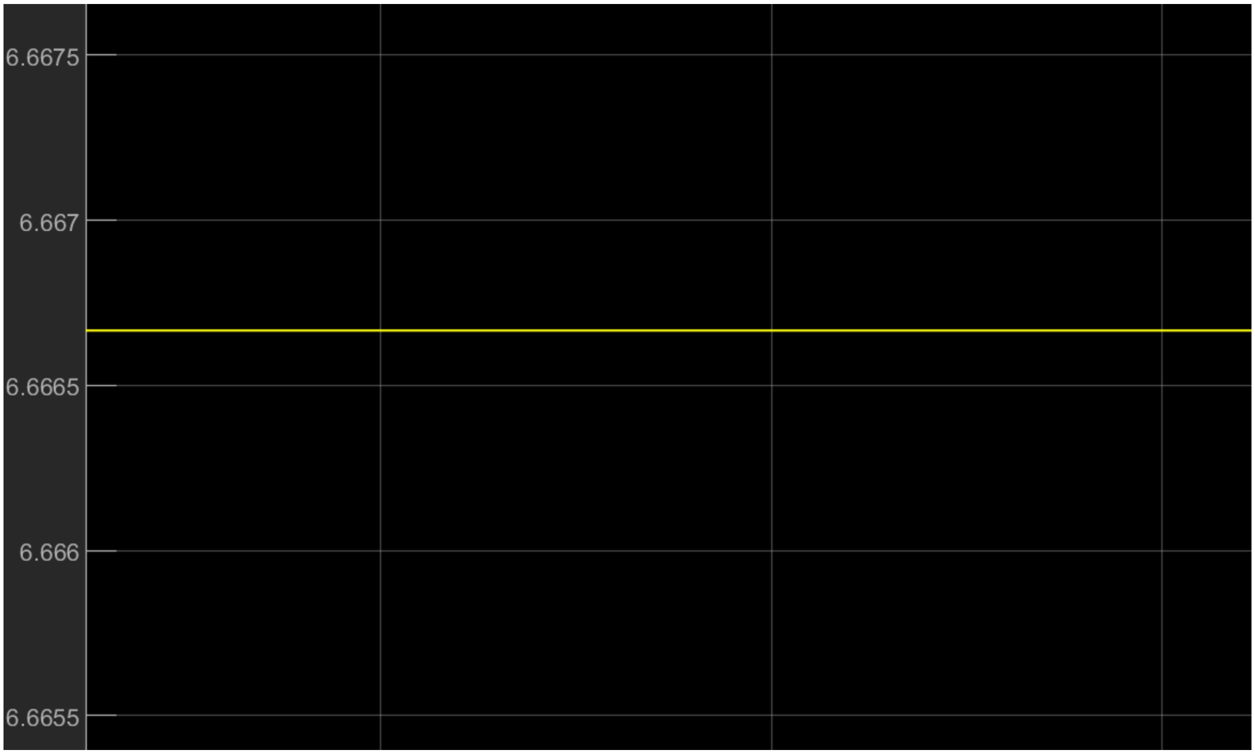
We bring current measurement block and add it in series.



After run for 10 sec. you get this:



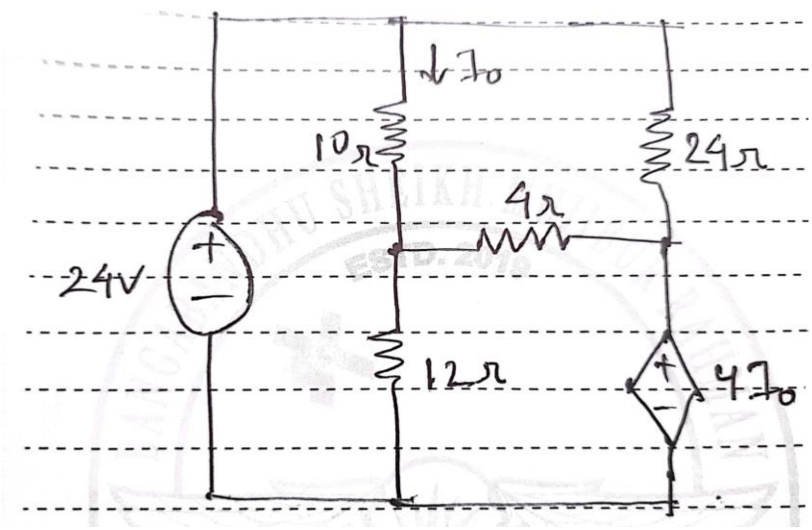
Zoom in:



The current is 6.6666

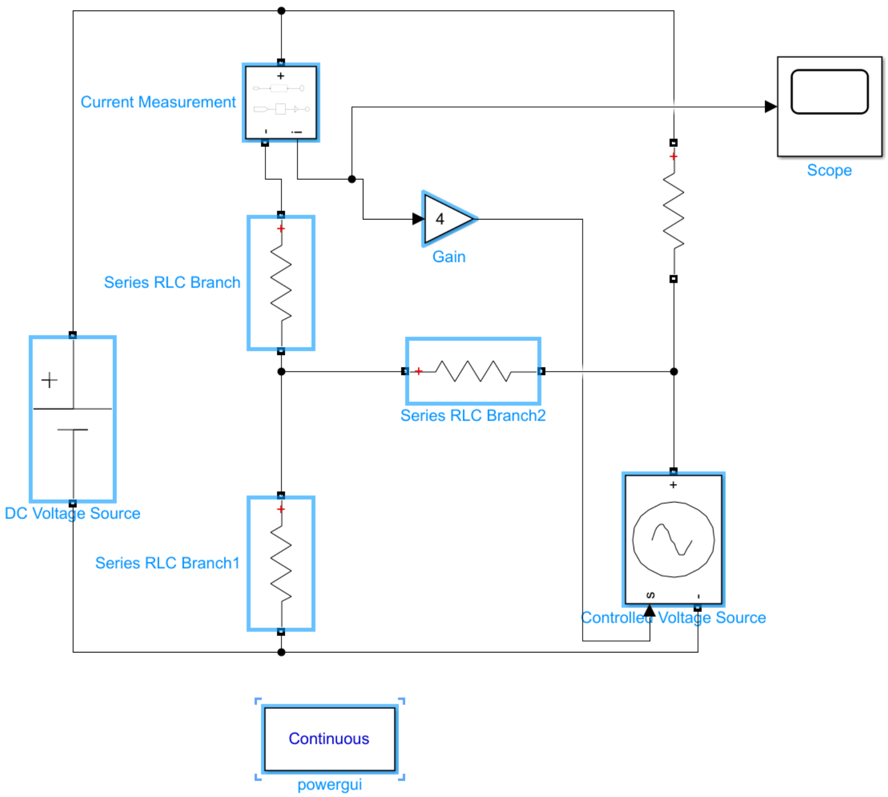
Example 03:

Find the current I_0 in the following circuit:

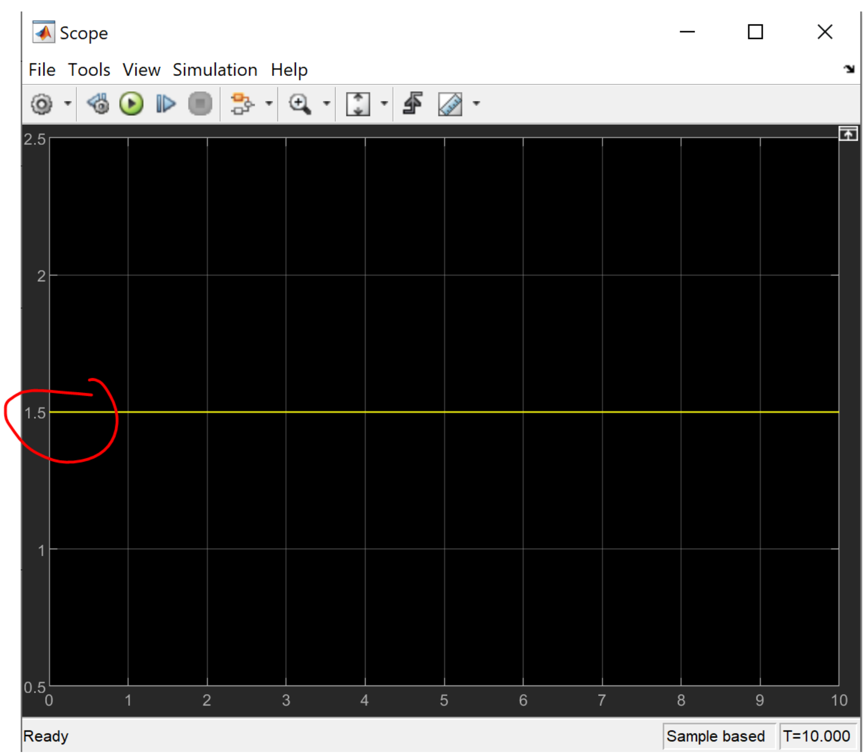


We have a dependent source here.

Here you bring ‘Controlled Voltage Source (Electrical/Specialized Power Systems/Sources)’

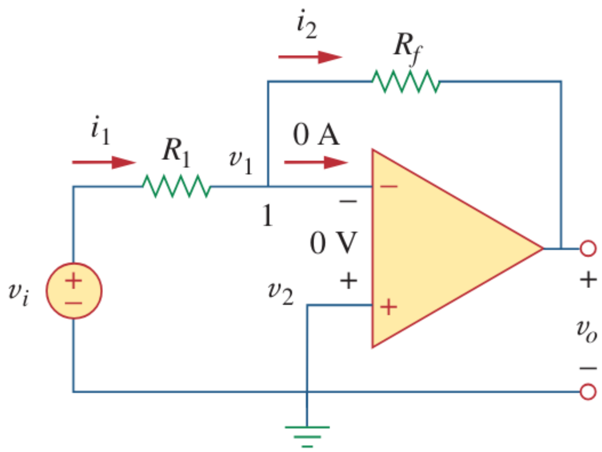


With current measurement we take the current I_0 then we use gain to make 4 I_0 which then is the input for the dependent voltage source. From scope we can see the current is 1.5 A



Operational Amplifiers Simulations

Inverting Amplifier:



A key feature of the inverting amplifier is that both the input signal and the feedback are applied at the inverting terminal of the op amp.

An inverting amplifier reverses the polarity of the input signal while amplifying it.

In this circuit, the noninverting input is grounded, v_i is connected to the inverting input through R_1, and the feedback resistor R_f Rf is connected between the inverting input and output.

Applying KCL at node 1,

$$i_1 = i_2 \quad \Rightarrow \quad \frac{v_i - v_1}{R_1} = \frac{v_1 - v_o}{R_f}$$

But $v_1 = v_2 = 0$ for an ideal op amp, since the noninverting terminal is grounded. Hence,

$$\frac{v_i}{R_1} = -\frac{v_o}{R_f}$$

$$\boxed{v_o = -\frac{R_f}{R_1} v_i}$$

The voltage gain is $A_v = v_o/v_i = -R_f/R_1$.

Example 5.3

Refer to the op amp in Fig. 5.12. If $v_i = 0.5$ V, calculate: (a) the output voltage v_o, and (b) the current in the 10-kΩ resistor.

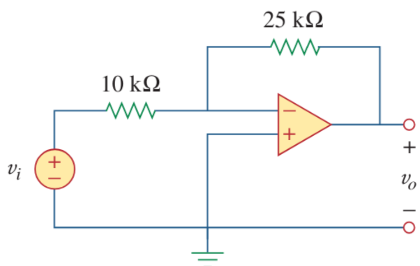


Figure 5.12
For Example 5.3.

Solution:

(a) Using Eq. (5.9),

$$\frac{v_o}{v_i} = -\frac{R_f}{R_1} = -\frac{25}{10} = -2.5$$

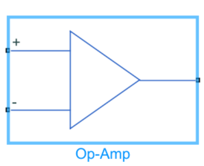
$$v_o = -2.5v_i = -2.5(0.5) = -1.25 \text{ V}$$

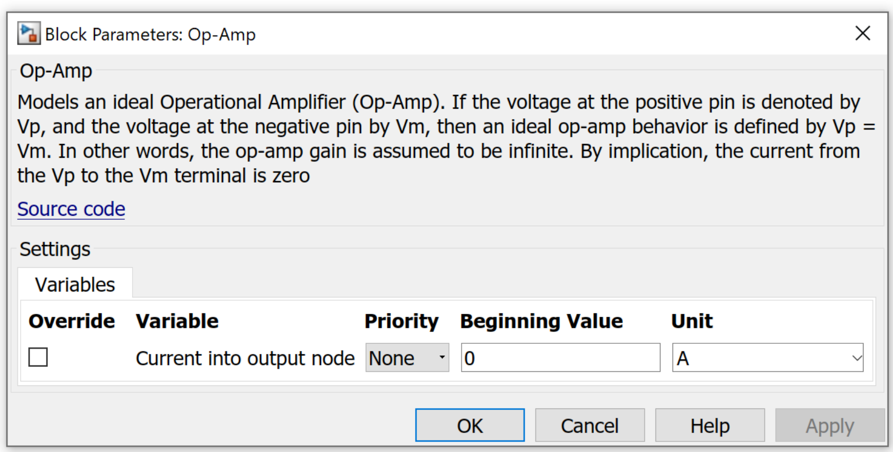
(b) The current through the 10-kΩ resistor is

$$i = \frac{v_i - 0}{R_1} = \frac{0.5 - 0}{10 \times 10^3} = 50\ \mu\text{A}$$

Let’s simulate this in MATLAB Simulink

This is the only kind of op-amp in the MATLAB Simulink. You just type op-amp.





In our circuit the above terminal is negative terminal. So, what we do,

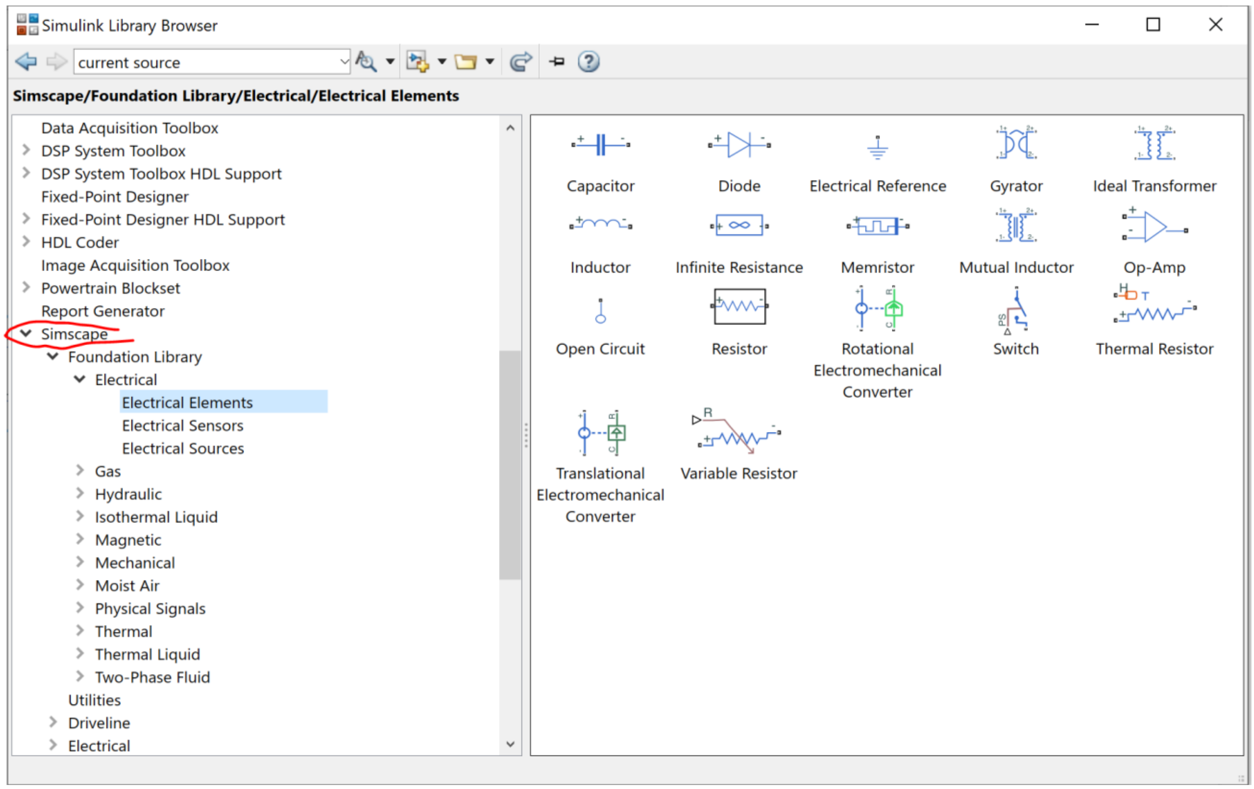
Right click -> Rotate and Flip -> Flip Block -> Up-Down

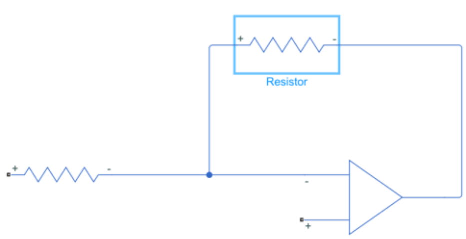
Another issue, see that this element color is blue. But the previous element we used, were black.

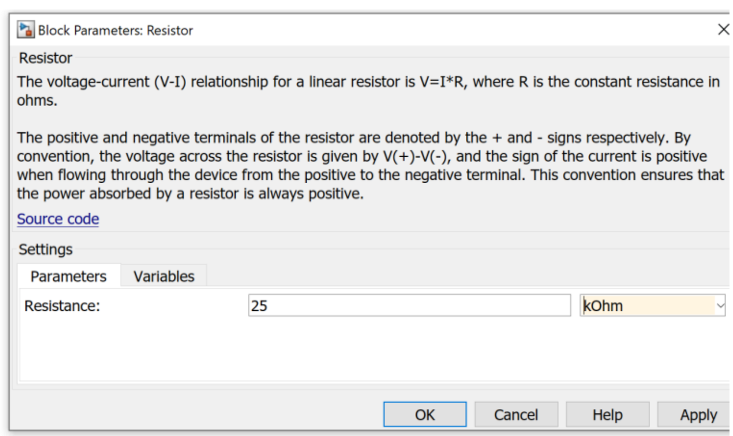
And if you try to connect blue element with black, that does not happen. One is Simulink library and another one is simscape

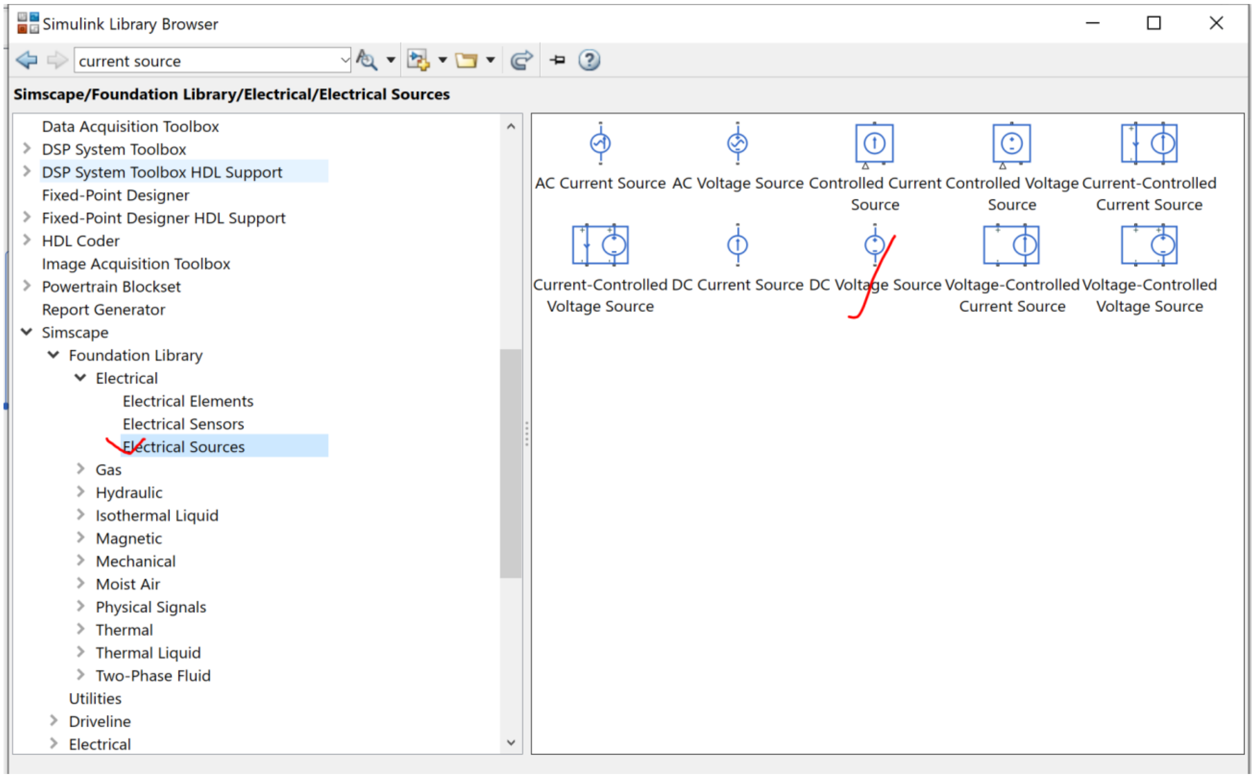
Basically the black ones are Simulink element where the blue ones are considered physical element.

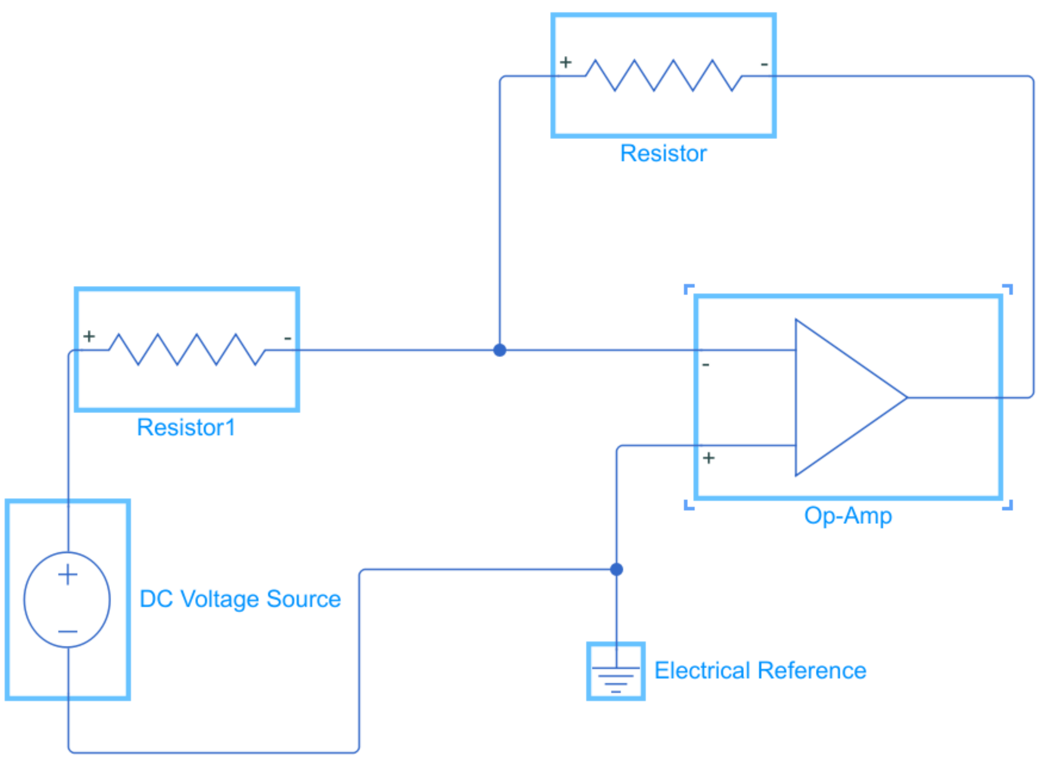
In the library, in the simscape we will get all the physical elements.



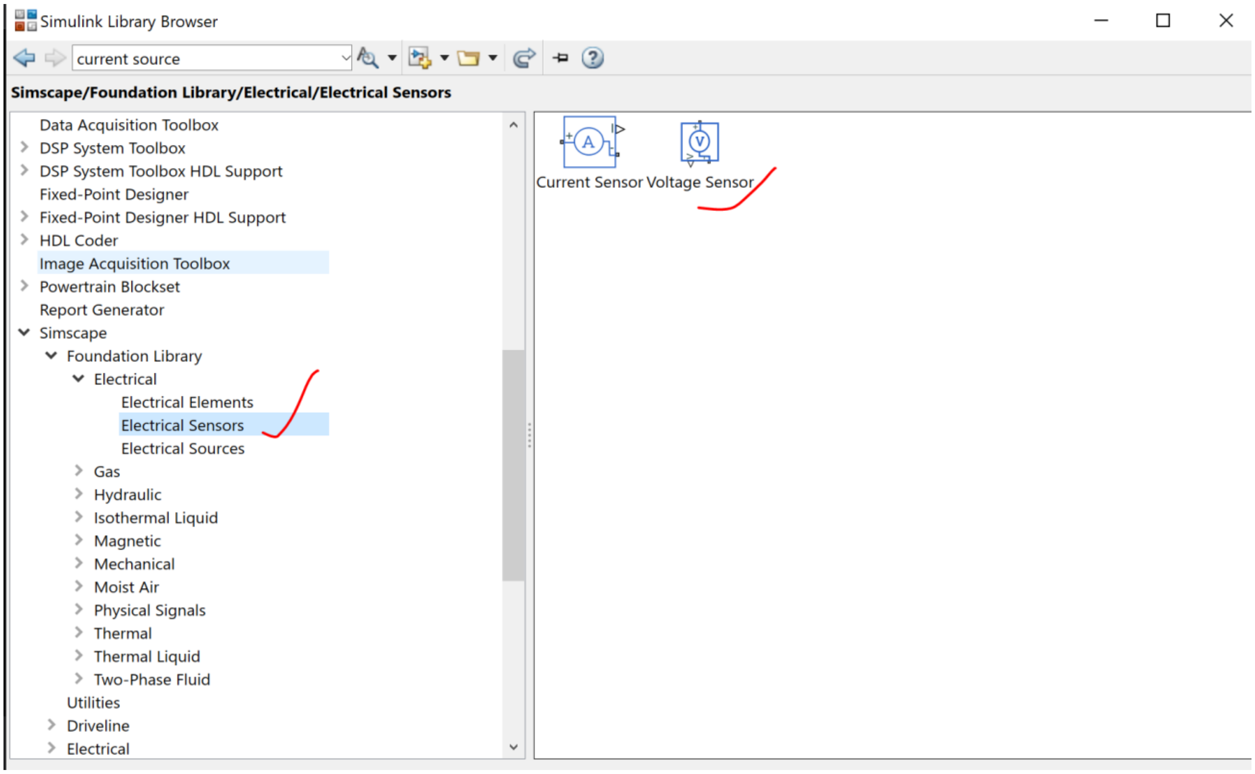


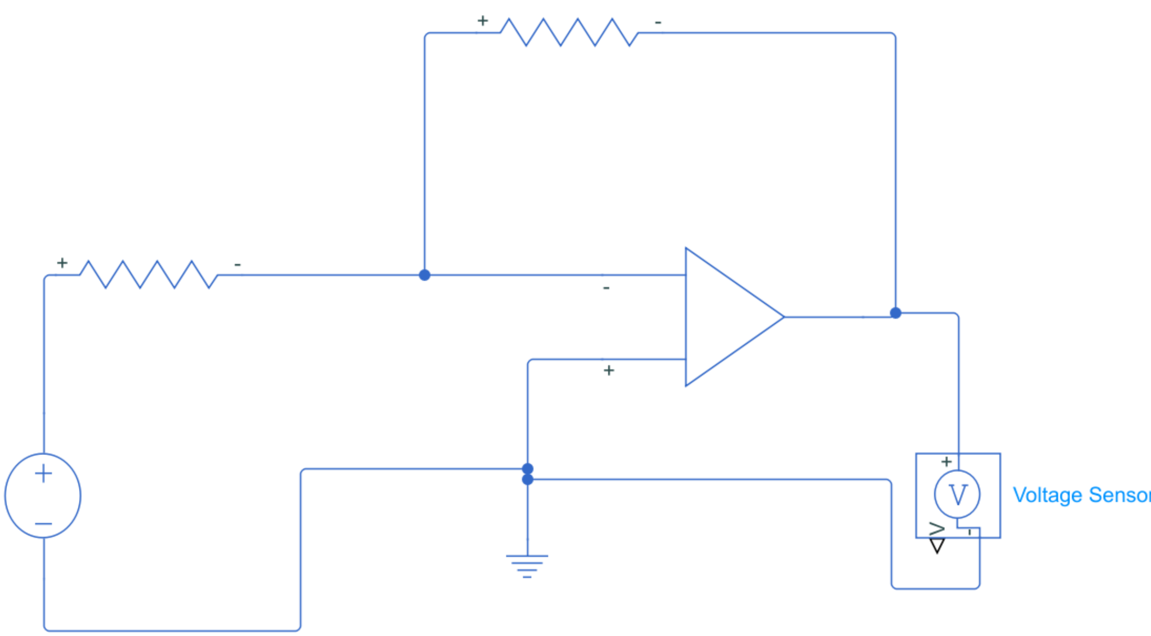



Simulink Library Browser
current source
Simscape/Foundation Library/Electrical/Electrical Sources
Data Acquisition Toolbox
DSP System Toolbox
DSP System Toolbox HDL Support
Fixed-Point Designer
Fixed-Point Designer HDL Support
HDL Coder
Image Acquisition Toolbox
Powertrain Blockset
Report Generator
Simscape
Foundation Library
Electrical
Electrical Elements
Electrical Sensors
Electrical Sources
Gas
Hydraulic
Isothermal Liquid
Magnetic
Mechanical
Moist Air
Physical Signals
Thermal
Thermal Liquid
Two-Phase Fluid
Utilities
Driveline
Electrical
AC Current Source
AC Voltage Source
Controlled Current Source
Controlled Voltage Source
Current-Controlled Current Source
Current-Controlled Voltage Source
DC Current Source
DC Voltage Source
Voltage-Controlled Current Source
Voltage-Controlled Voltage Source


Resistor
Resistor1
Op-Amp
DC Voltage Source
Electrical Reference

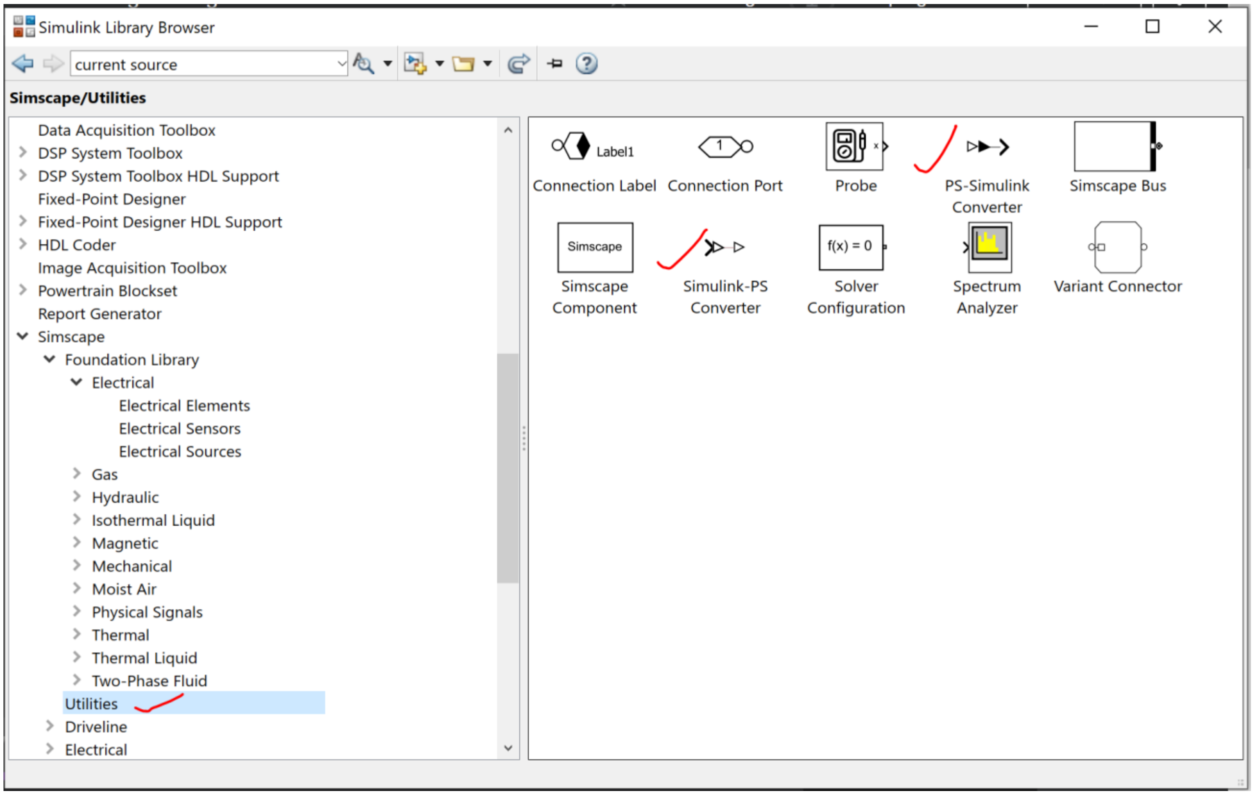
For measurement:

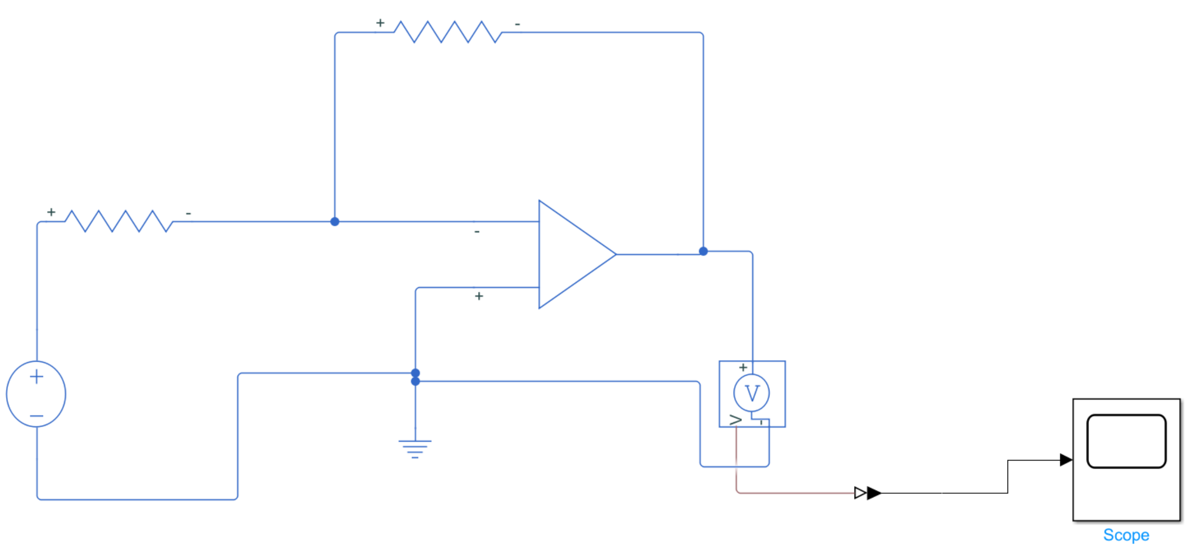




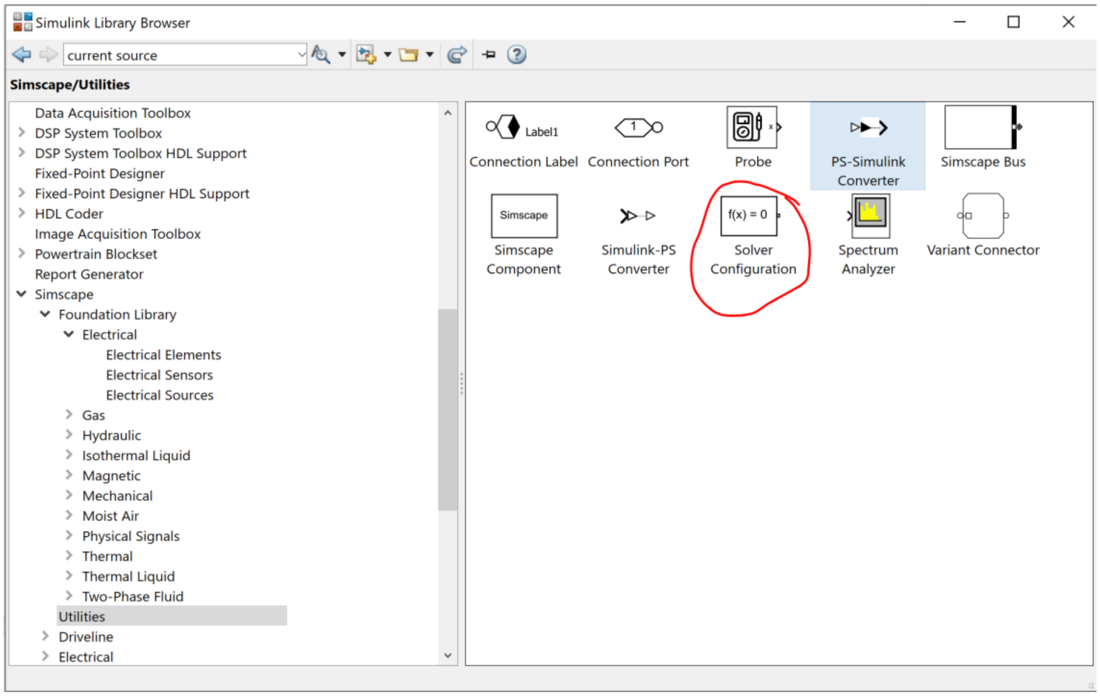
Now like previously you can’t add scope due to Simulink library issue. So, what we do:

Utilities -> PS-Simulink Converter (Converts physical signal to Simulink signal)



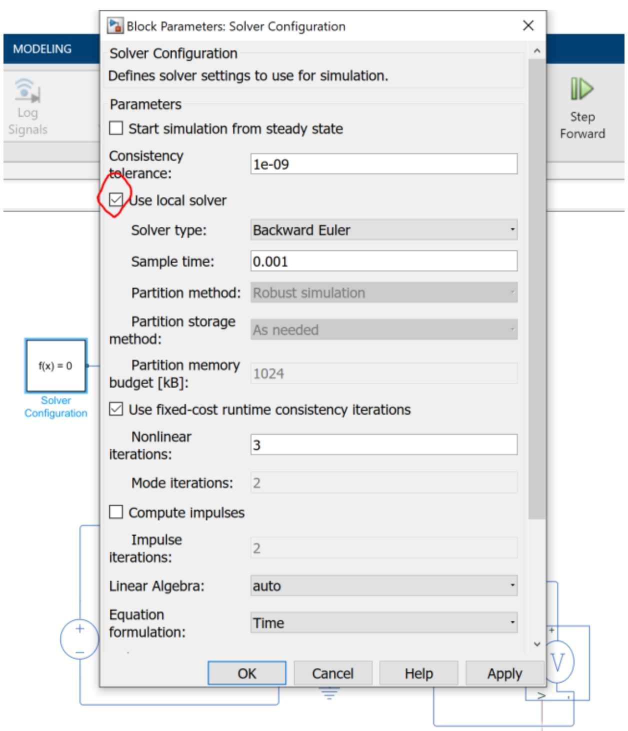


Now, previously we required powergui. Here we will use solver configuration which will solve the governing equations of this physical system and give us the output.



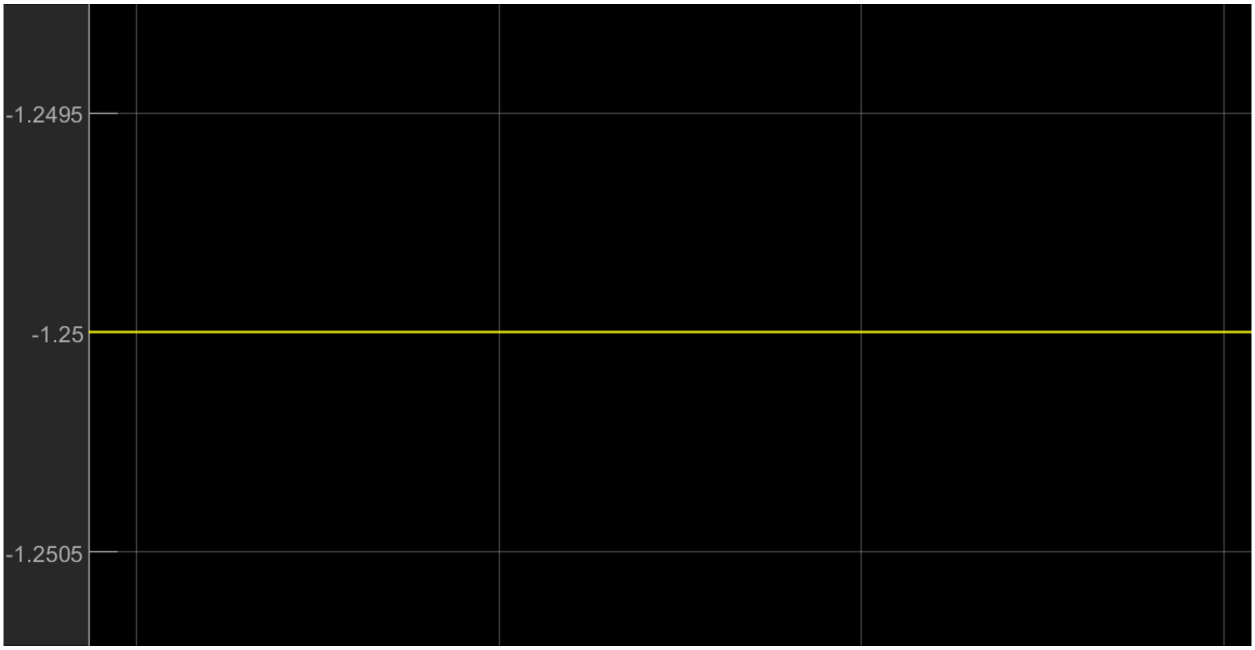
You will have to connect the solver in any point of the physical system

Tick on this:

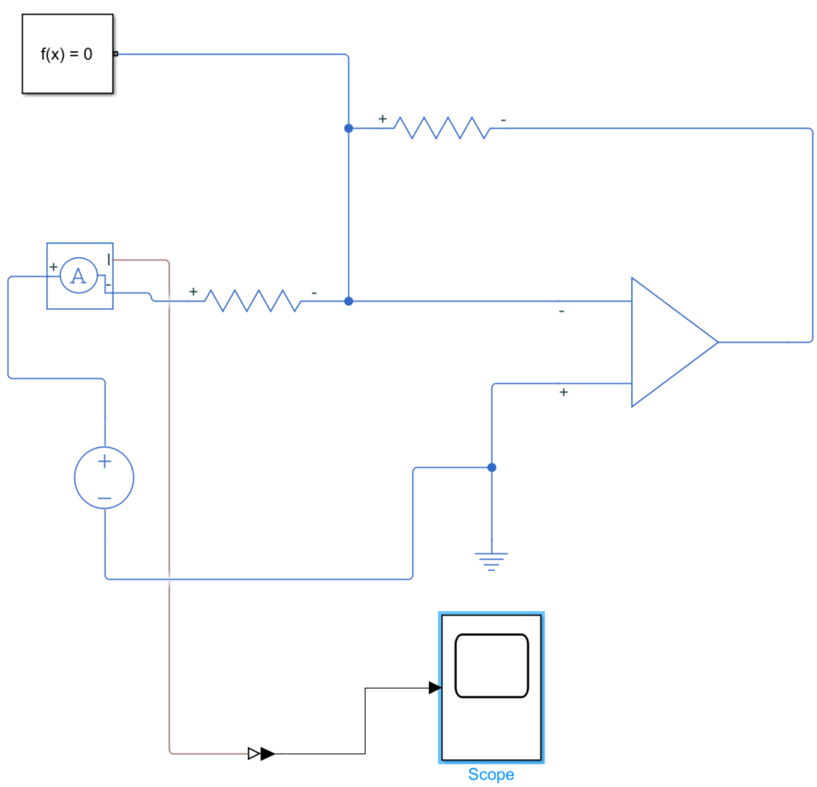


Now, run the simulation and see the result on the scope

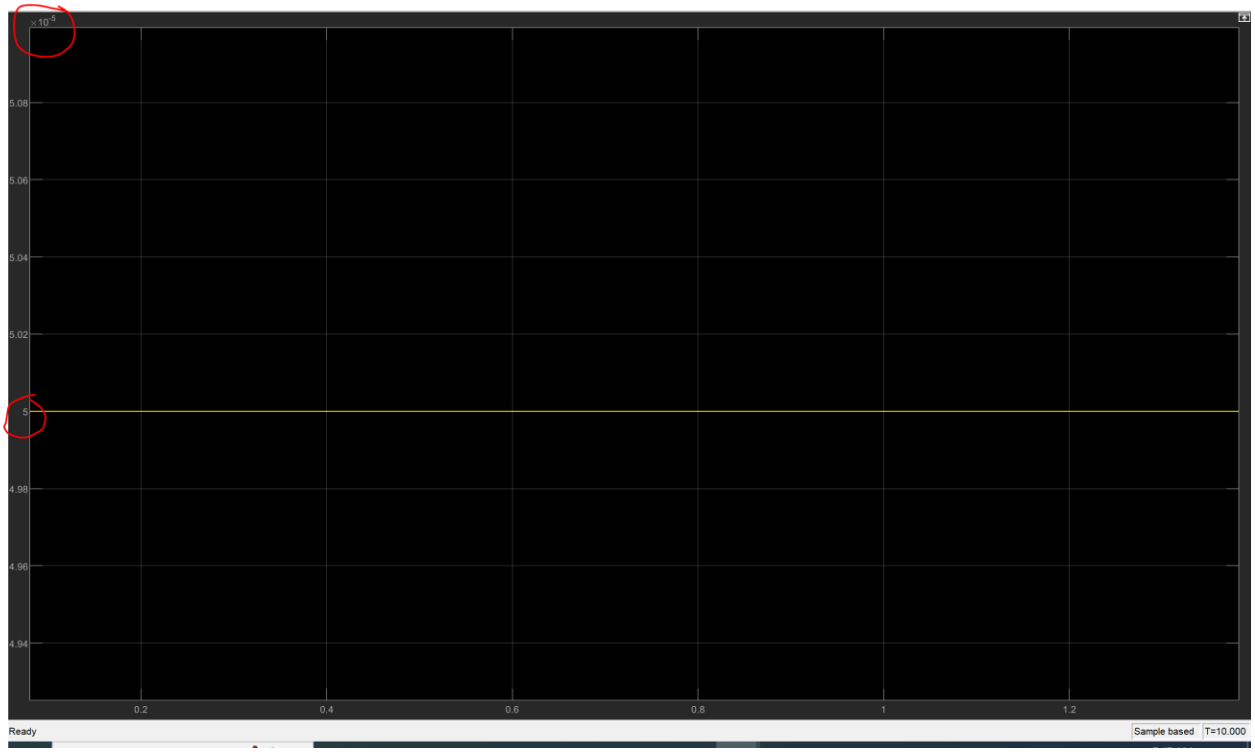
If you zoom in you will see the ans -1.25 V



Now to find current through the 10Kohm register, we bring current sensor



The ans is 5 * 10 ^ -5



Non-Inverting Amplifier:

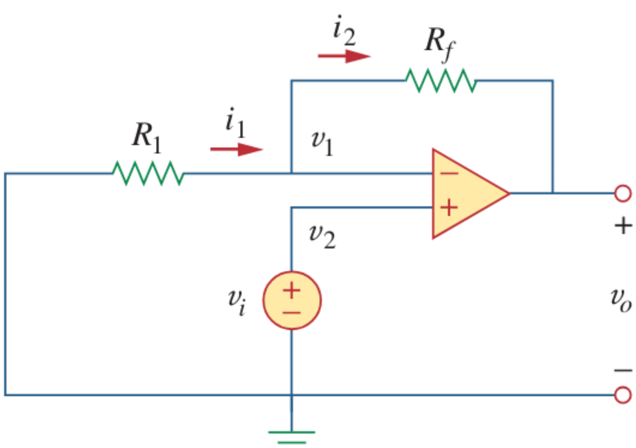


Figure 5.16
The noninverting amplifier.

Another important application of the op amp is the noninverting amplifier shown in Fig. 5.16. In this case, the input voltage v_i is applied directly at the noninverting input terminal, and resistor R_1 is connected

between the ground and the inverting terminal. We are interested in the output voltage and the voltage gain. Application of KCL at the inverting terminal gives

$$i_1 = i_2 \quad \Rightarrow \quad \frac{0 - v_1}{R_1} = \frac{v_1 - v_o}{R_f} \tag{5.10}$$

But $v_1 = v_2 = v_i$. Equation (5.10) becomes

$$\frac{-v_i}{R_1} = \frac{v_i - v_o}{R_f}$$

or

$$\boxed{v_o = \left(1 + \frac{R_f}{R_1}\right)v_i} \tag{5.11}$$

The voltage gain is $A_v = v_o/v_i = 1 + R_f/R_1$, which does not have a negative sign. Thus, the output has the same polarity as the input.

A noninverting amplifier is an op amp circuit designed to provide a positive voltage gain.

Notice that if feedback resistor $R_f = 0$ (short circuit) or $R_1 = \infty$ (open circuit) or both, the gain becomes 1. Under these conditions ($R_f = 0$ and $R_1 = \infty$), the circuit in Fig. 5.16 becomes that shown in Fig. 5.17, which is called a *voltage follower* (or *unity gain amplifier*) because the output follows the input. Thus, for a voltage follower

$$\boxed{v_o = v_i} \tag{5.12}$$

Such a circuit has a very high input impedance and is therefore useful as an intermediate-stage (or buffer) amplifier to isolate one circuit from another, as portrayed in Fig. 5.18. The voltage follower minimizes interaction between the two stages and eliminates interstage loading.

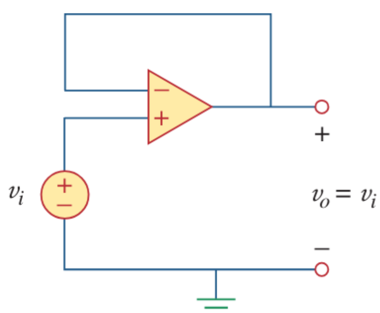


Figure 5.17
The voltage follower.

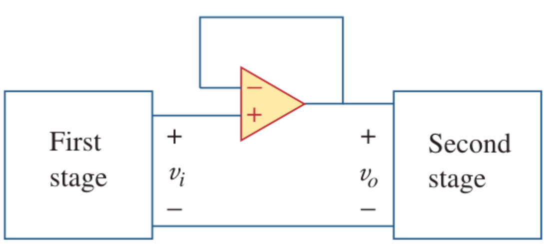


Figure 5.18
A voltage follower used to isolate two cascaded stages of a circuit.

As the R1 will be infinity, it will have a very hight input impedance, that isolate the two circuit

For the op amp circuit in Fig. 5.19, calculate the output voltage v_o.

Solution:

We may solve this in two ways: using superposition and using nodal analysis.

METHOD 1 Using superposition, we let

$$v_o = v_{o1} + v_{o2}$$

where v_{o1} is due to the 6-V voltage source, and v_{o2} is due to the 4-V input. To get v_{o1}, we set the 4-V source equal to zero. Under this condition, the circuit becomes an inverter. Hence Eq. (5.9) gives

$$v_{o1} = -\frac{10}{4}(6) = -15 \text{ V}$$

To get v_{o2}, we set the 6-V source equal to zero. The circuit becomes a noninverting amplifier so that Eq. (5.11) applies.

$$v_{o2} = \left(1 + \frac{10}{4}\right)4 = 14 \text{ V}$$

Thus,

$$v_o = v_{o1} + v_{o2} = -15 + 14 = -1 \text{ V}$$

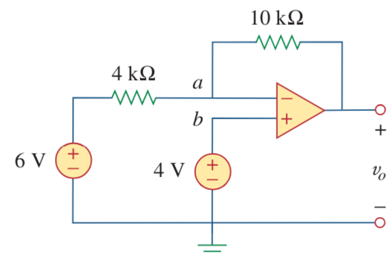


Figure 5.19
For Example 5.5.

METHOD 2 Applying KCL at node a,

$$\frac{6 - v_a}{4} = \frac{v_a - v_o}{10}$$

But $v_a = v_b = 4$, and so

$$\frac{6 - 4}{4} = \frac{4 - v_o}{10} \quad \Rightarrow \quad 5 = 4 - v_o$$

or $v_o = -1$ V, as before.

5.6 Summing Amplifier

Besides amplification, the op amp can perform addition and subtraction. The addition is performed by the summing amplifier covered in this section; the subtraction is performed by the difference amplifier covered in the next section.

> A summing amplifier is an op amp circuit that combines several inputs and produces an output that is the weighted sum of the inputs.

The summing amplifier, shown in Fig. 5.21, is a variation of the inverting amplifier. It takes advantage of the fact that the inverting configuration can handle many inputs at the same time. We keep in mind

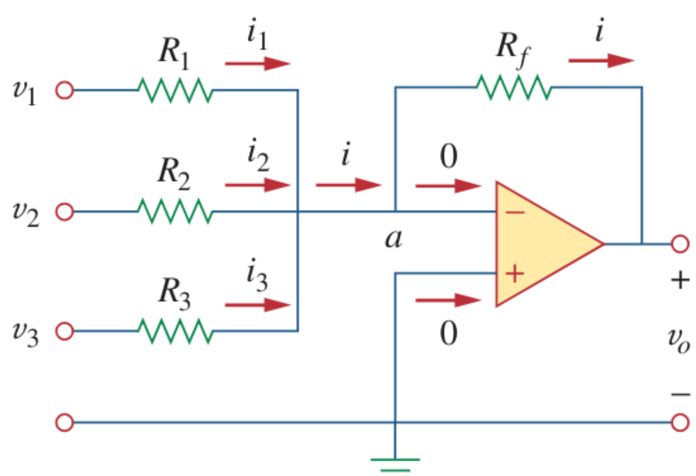


Figure 5.21
The summing amplifier.

that the current entering each op amp input is zero. Applying KCL at node a gives

$$i = i_1 + i_2 + i_3 \tag{5.13}$$

But

$$i_1 = \frac{v_1 - v_a}{R_1}, \quad i_2 = \frac{v_2 - v_a}{R_2}$$
$$i_3 = \frac{v_3 - v_a}{R_3}, \quad i = \frac{v_a - v_o}{R_f} \tag{5.14}$$

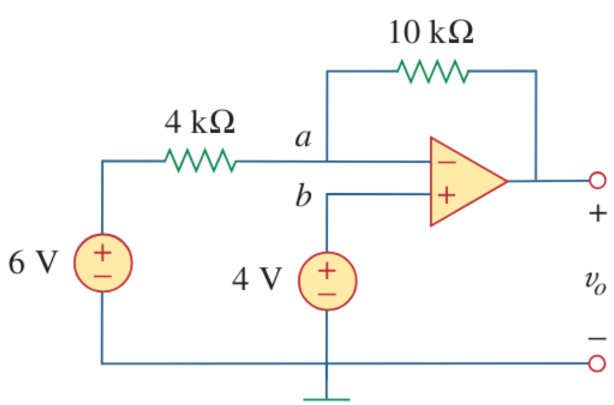
We note that $v_a = 0$ and substitute Eq. (5.14) into Eq. (5.13). We get

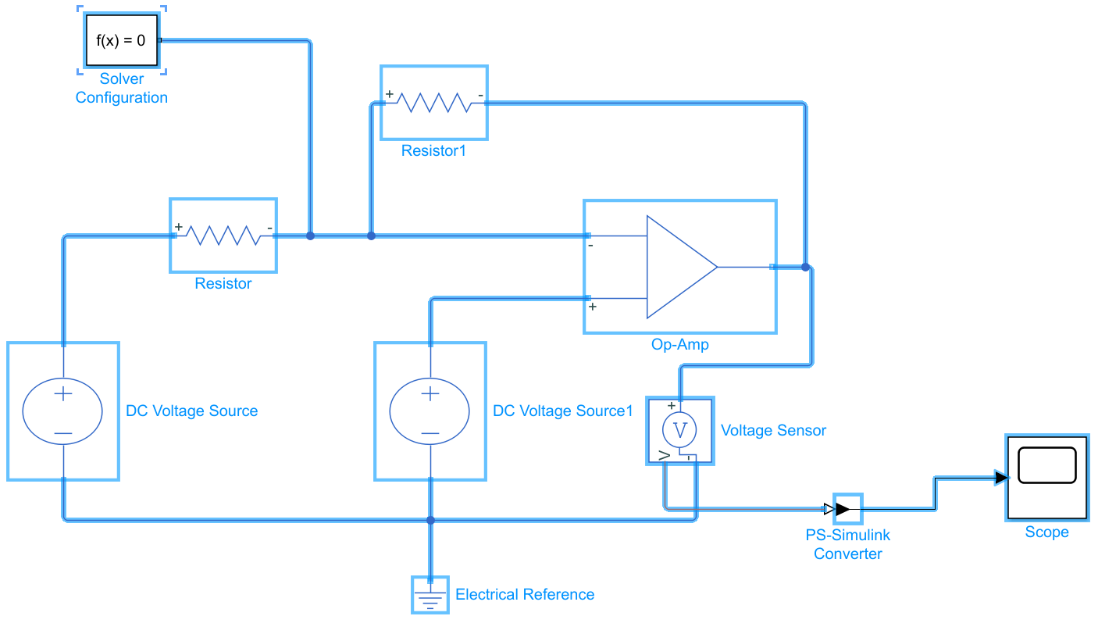
$$\boxed{v_o = -\left(\frac{R_f}{R_1}v_1 + \frac{R_f}{R_2}v_2 + \frac{R_f}{R_3}v_3\right)} \tag{5.15}$$

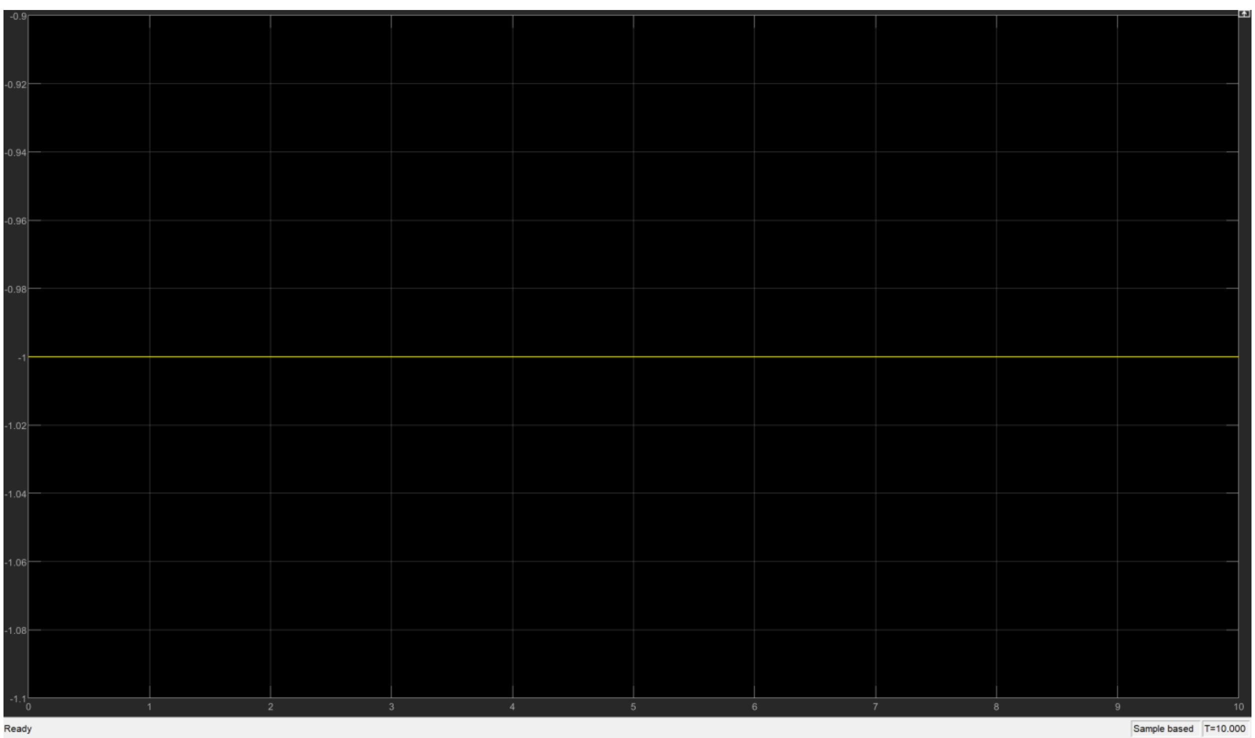
indicating that the output voltage is a weighted sum of the inputs. For this reason, the circuit in Fig. 5.21 is called a *summer*. Needless to say, the summer can have more than three inputs.

Now, Simulation of Non-Inverting Amplifier:

For the op amp circuit in Fig. 5.19, calculate the output voltage v_o.







See that the answer is -1 V

Calculate v_o and i_o in the op amp circuit in Fig. 5.22.

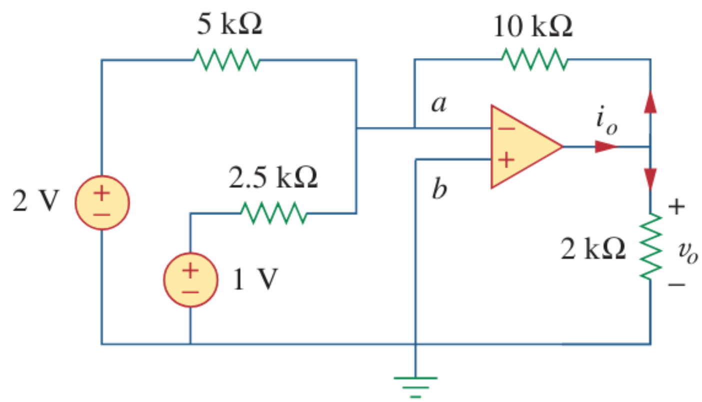


Figure 5.22
For Example 5.6.

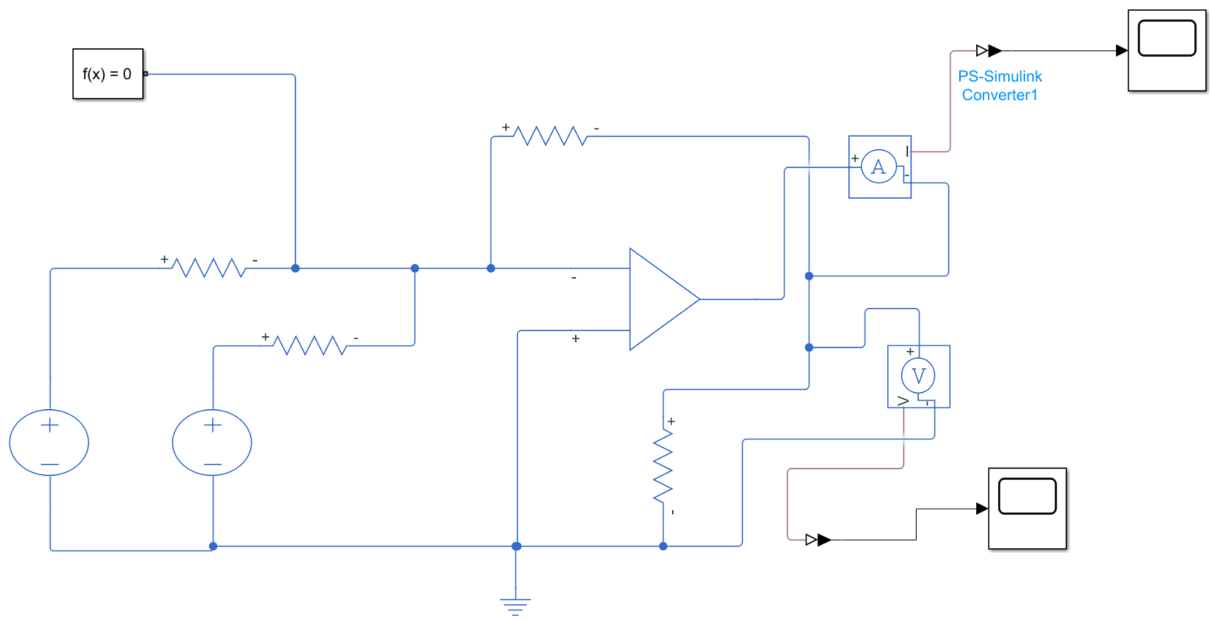
Solution:

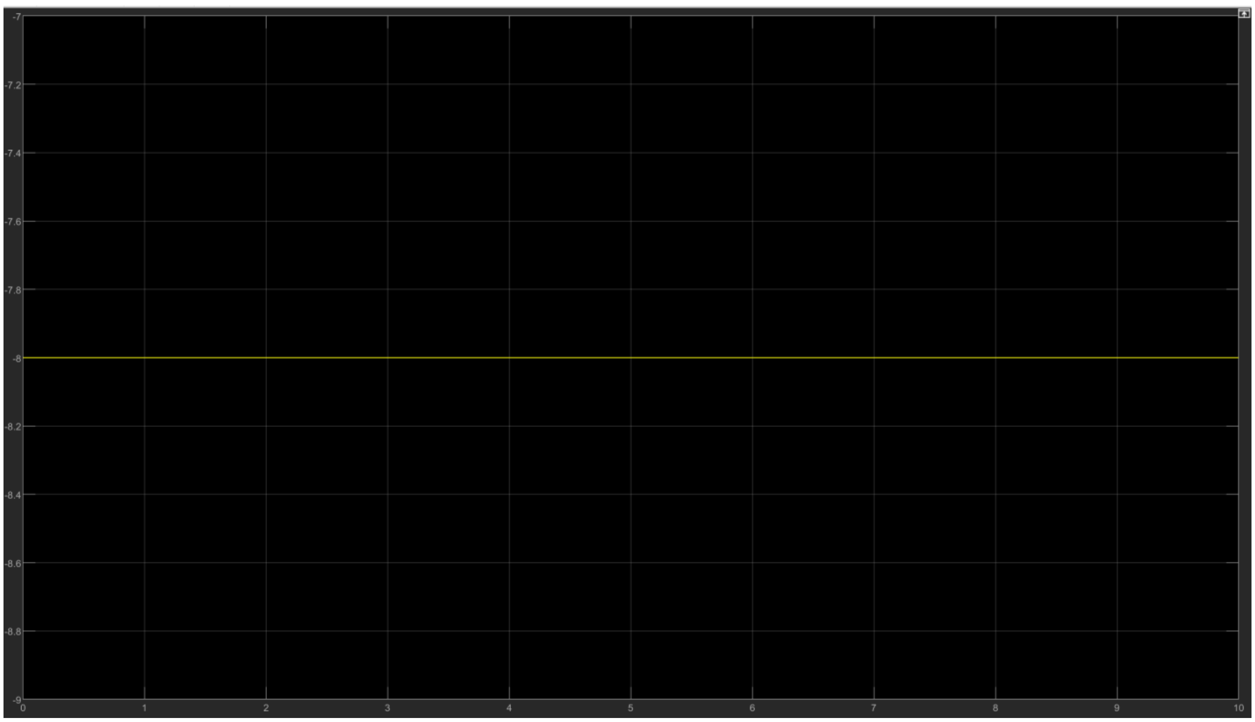
This is a summer with two inputs. Using Eq. (5.15) gives

$$v_o = -\left[\frac{10}{5}(2) + \frac{10}{2.5}(1)\right] = -(4 + 4) = -8 \text{ V}$$

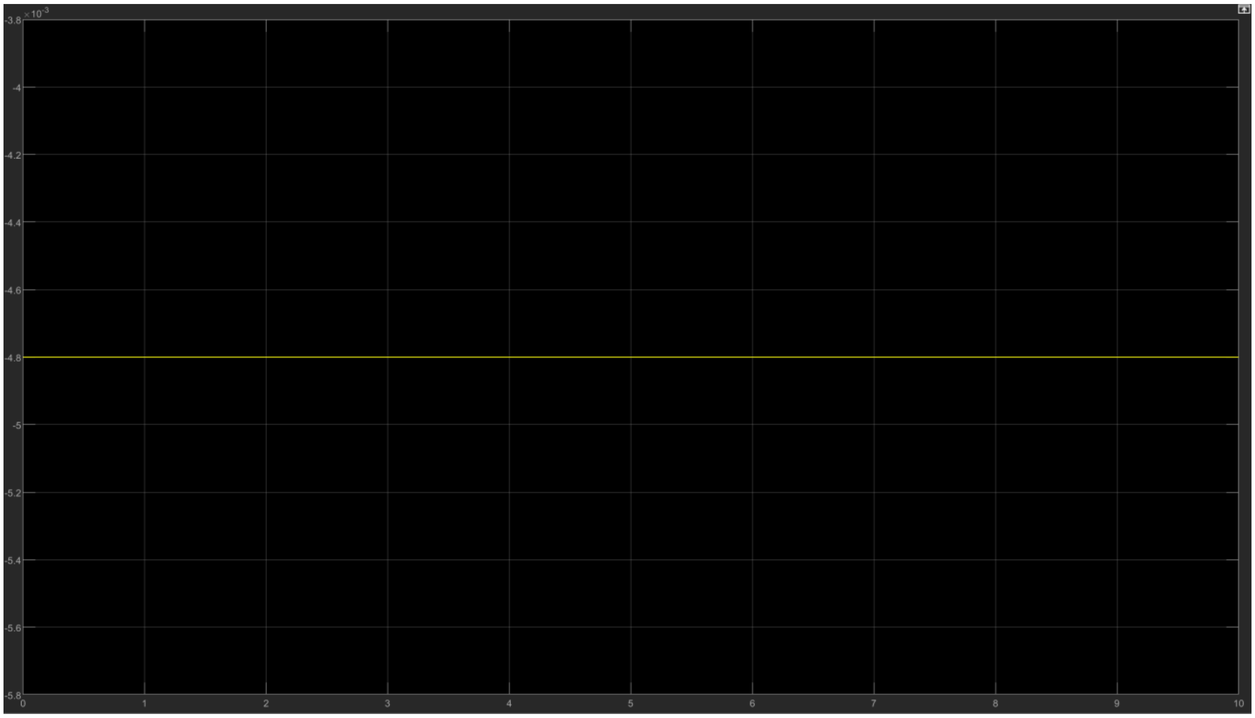
The current i_o is the sum of the currents through the 10-kΩ and 2-kΩ resistors. Both of these resistors have voltage $v_o = -8$ V across them, since $v_a = v_b = 0$. Hence,

$$i_o = \frac{v_o - 0}{10} + \frac{v_o - 0}{2} \text{mA} = -0.8 - 4 = -4.8 \text{ mA}$$



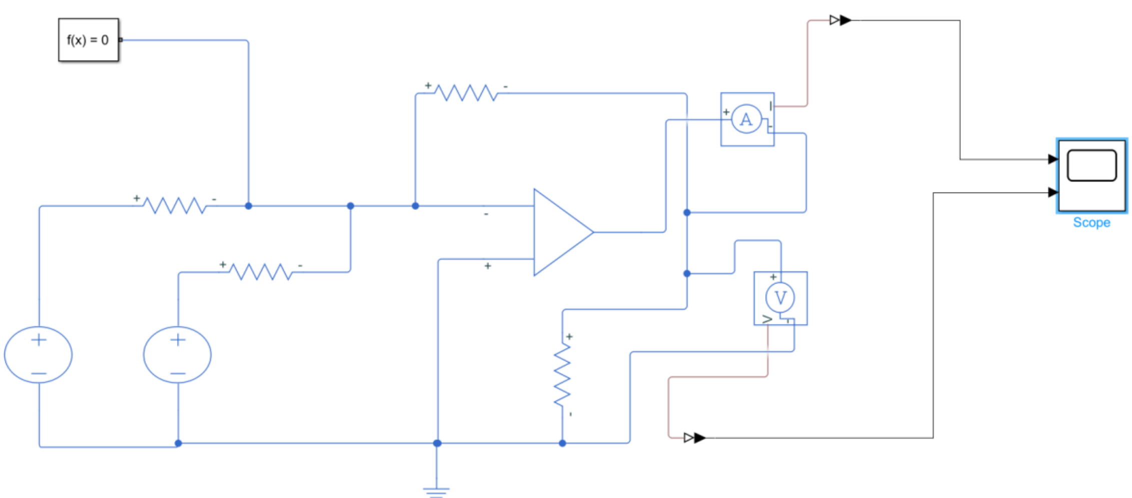


Voltage -8 V



Current -4.8

You can try this circuit with only one scope and see the result together.



5.7 Difference Amplifier

Difference (or differential) amplifiers are used in various applications where there is a need to amplify the difference between two input signals. They are first cousins of the *instrumentation amplifier*, the most useful and popular amplifier, which we will discuss in Section 5.10.

The difference amplifier is also known as the *subtractor*, for reasons to be shown later.

> A difference amplifier is a device that amplifies the difference between two inputs but rejects any signals common to the two inputs.

Consider the op amp circuit shown in Fig. 5.24. Keep in mind that zero currents enter the op amp terminals. Applying KCL to node *a*,

$$\frac{v_1 - v_a}{R_1} = \frac{v_a - v_o}{R_2}$$

or

$$v_o = \left(\frac{R_2}{R_1} + 1\right)v_a - \frac{R_2}{R_1}v_1 \tag{5.16}$$

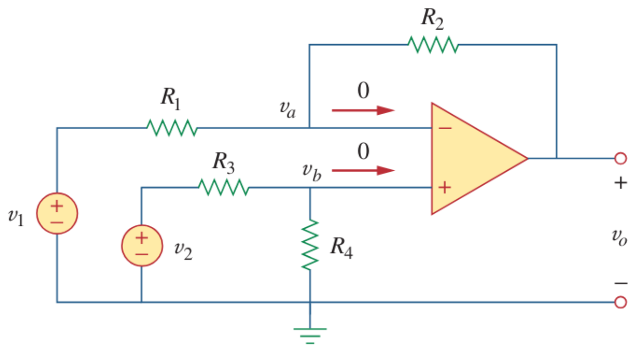


Figure 5.24
Difference amplifier.

Applying KCL to node b,

$$\frac{v_2 - v_b}{R_3} = \frac{v_b - 0}{R_4}$$

or

$$v_b = \frac{R_4}{R_3 + R_4} v_2 \tag{5.17}$$

But $v_a = v_b$. Substituting Eq. (5.17) into Eq. (5.16) yields

$$v_o = \left(\frac{R_2}{R_1} + 1\right)\frac{R_4}{R_3 + R_4} v_2 - \frac{R_2}{R_1} v_1$$

or

$$\boxed{v_o = \frac{R_2(1 + R_1/R_2)}{R_1(1 + R_3/R_4)} v_2 - \frac{R_2}{R_1} v_1} \tag{5.18}$$

Since a difference amplifier must reject a signal common to the two inputs, the amplifier must have the property that $v_o = 0$ when $v_1 = v_2$. This property exists when

$$\frac{R_1}{R_2} = \frac{R_3}{R_4} \tag{5.19}$$

Thus, when the op amp circuit is a difference amplifier, Eq. (5.18) becomes

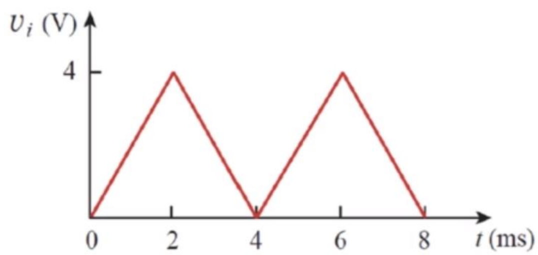
$$v_o = \frac{R_2}{R_1}(v_2 - v_1) \tag{5.20}$$

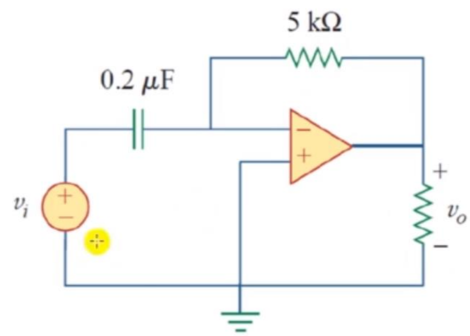
If $R_2 = R_1$ and $R_3 = R_4$, the difference amplifier becomes a *subtractor*, with the output

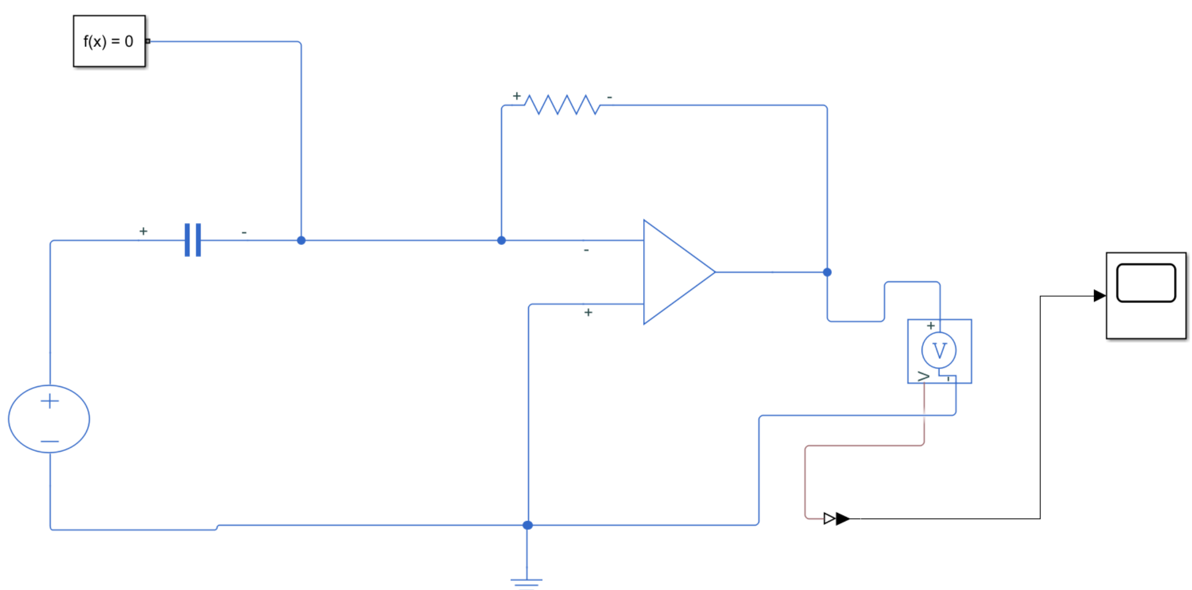
$$v_o = v_2 - v_1 \tag{5.21}$$

Example of Differentiator:

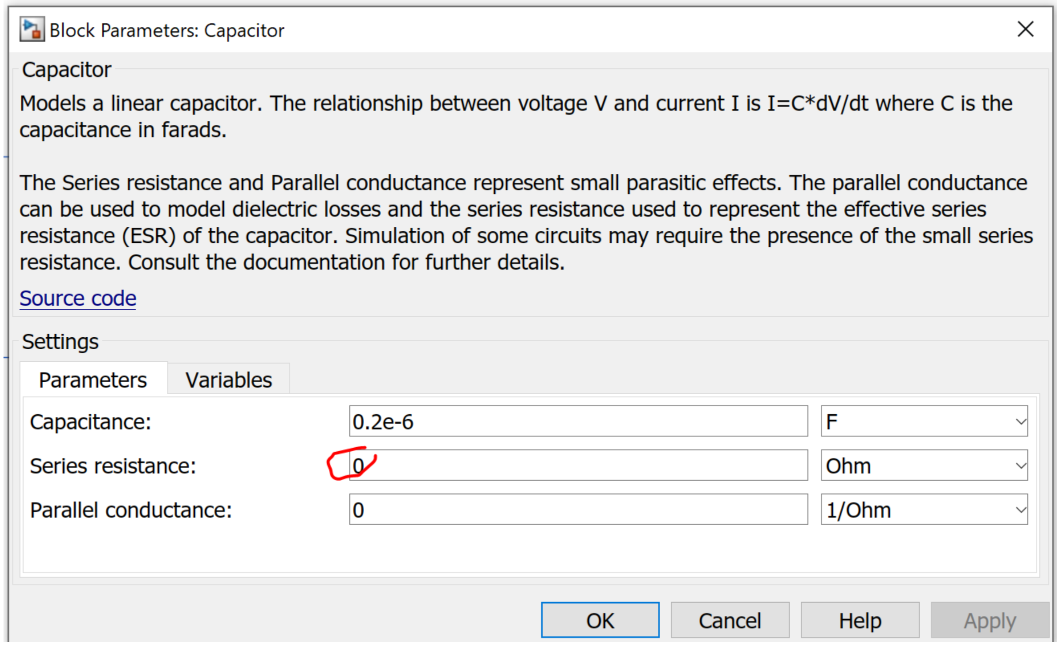
Sketch the output voltage for the circuit , given the input voltage . Take $v_o = 0$ at $t = 0$.





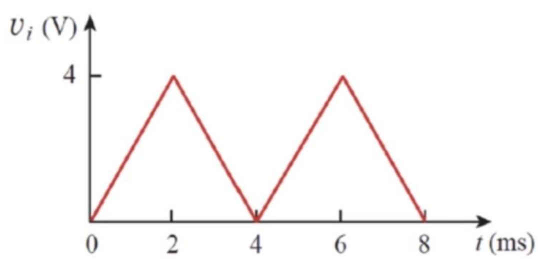


For capacitor, we bring capacitor.

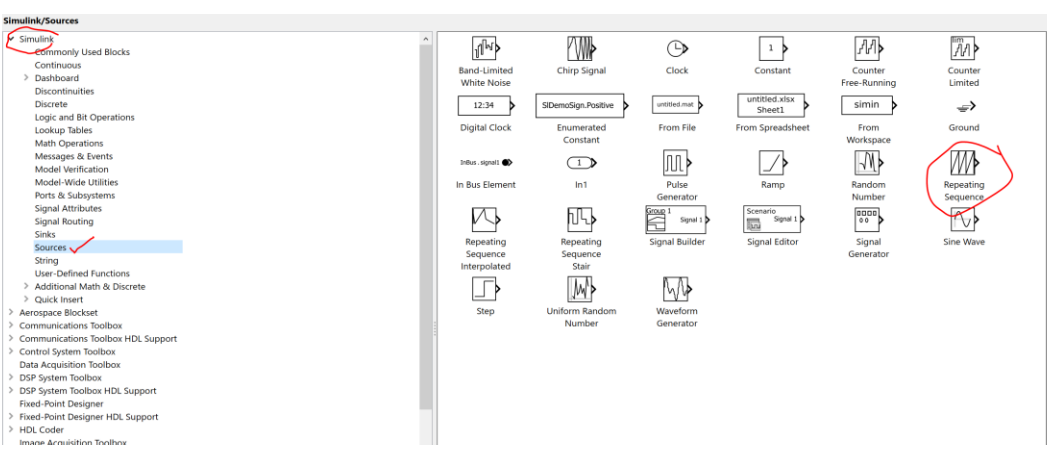


Making it zero because ideal element.

Now how to give the input?

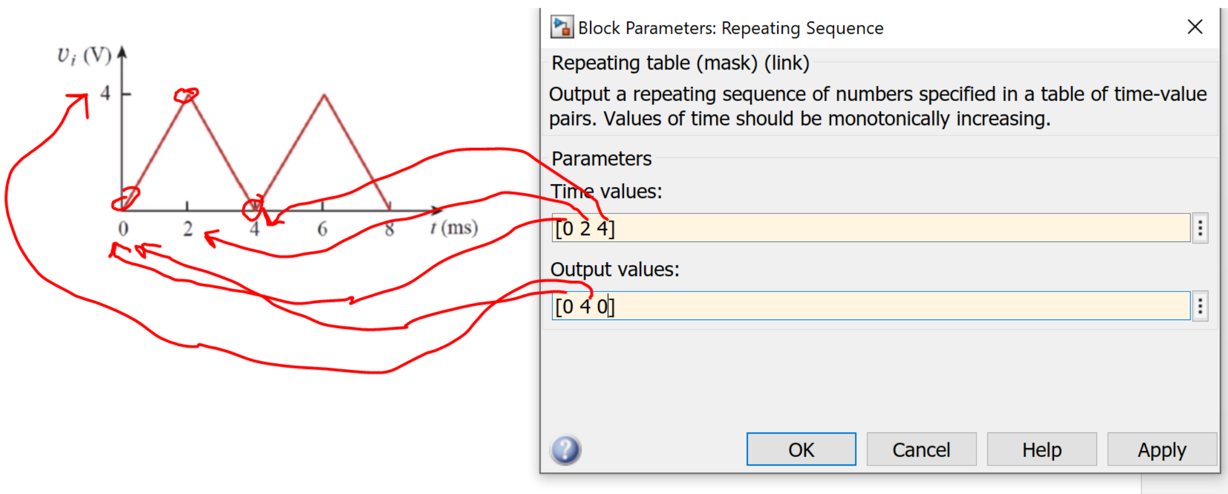


To produce the input above, we use a controlled voltage source



Now in the library we go to Simulink -> Sources -> Repeating Sequence

We bring repeating sequence.



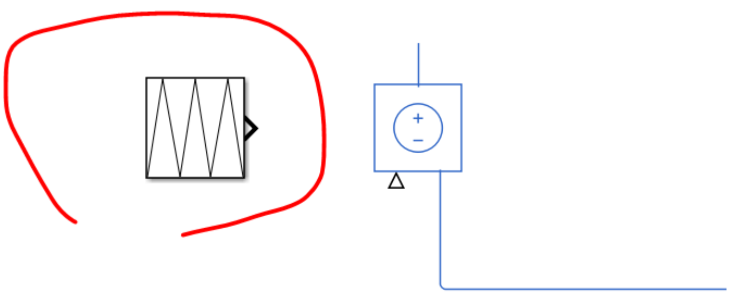
When time = 0, v = 0

Time = 2, v = 4

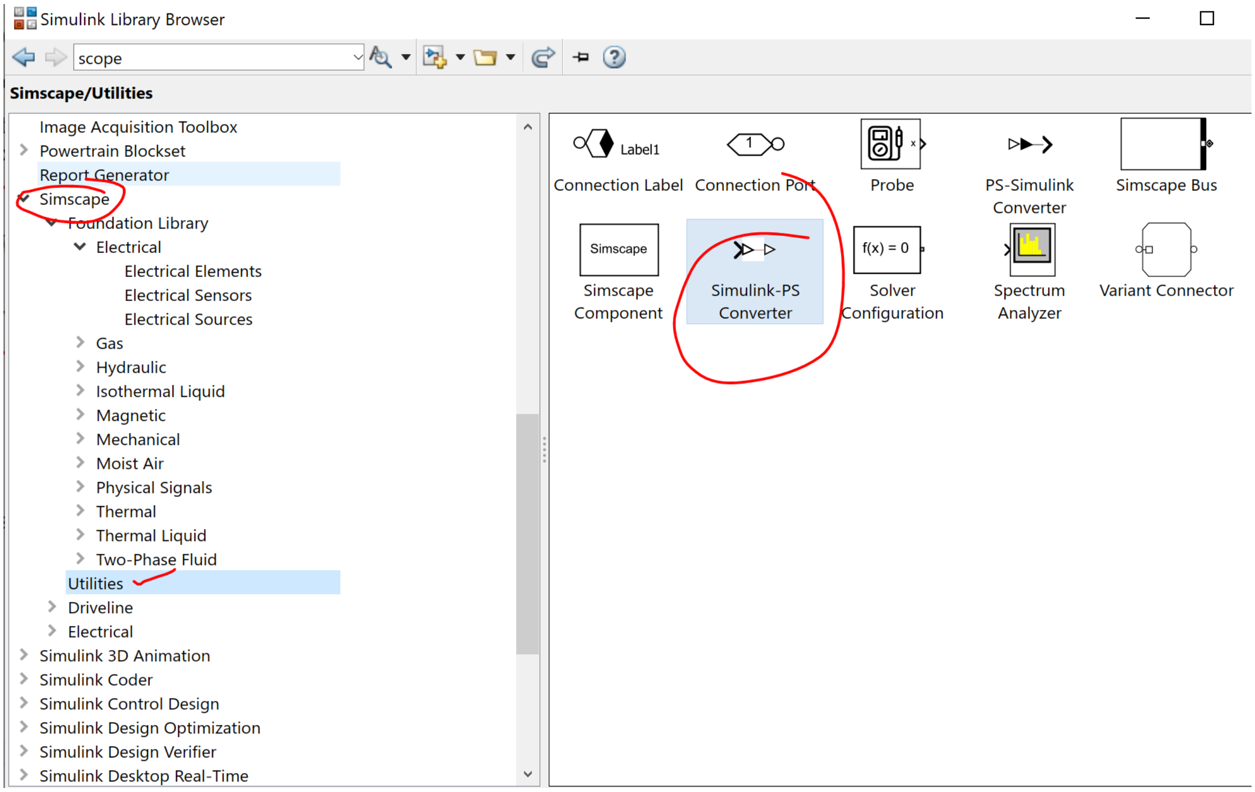
Time = 4, v = 0

This is the pattern that is repeating.

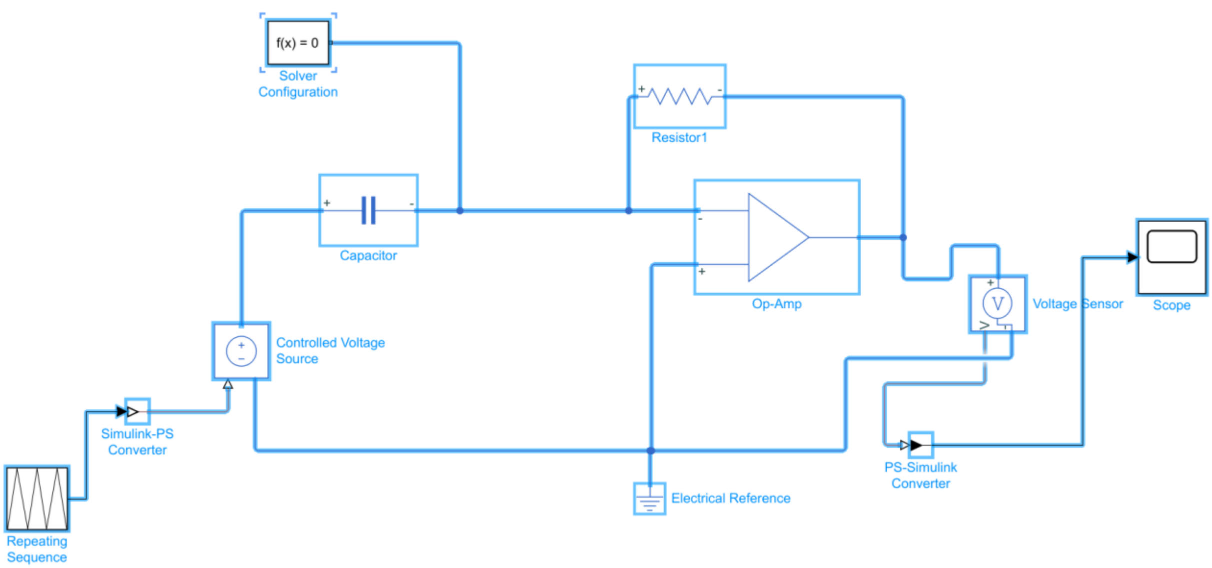
When you put the value the icon of repeating sequence get changed. It becomes like the one we want in our circuit.



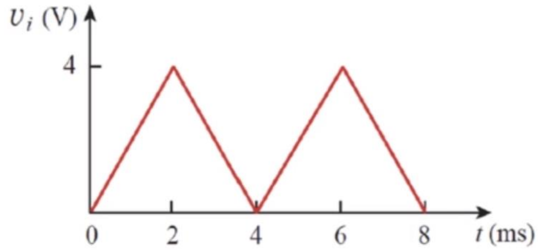
To connect this with the circuit we need Simulink – PS converter. Can be found in utilities under simscape.



Here is the circuit situation:

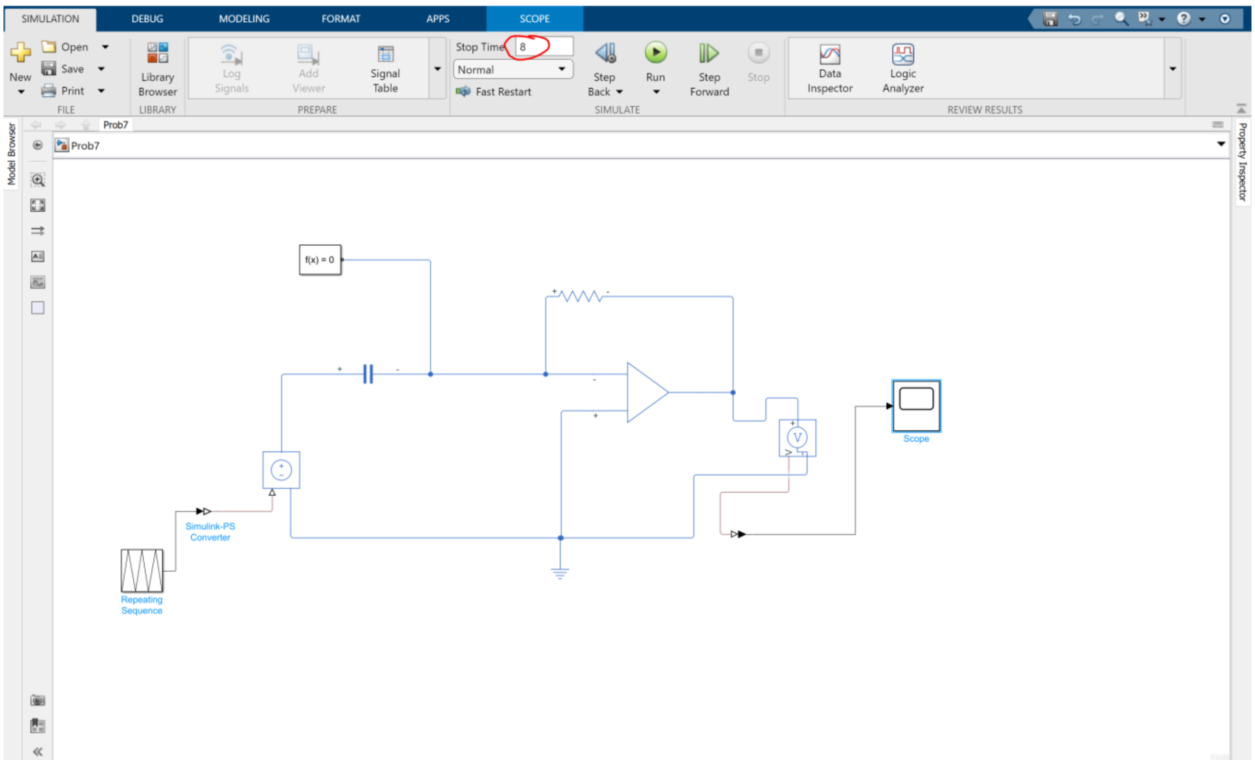


Now let's see the input:



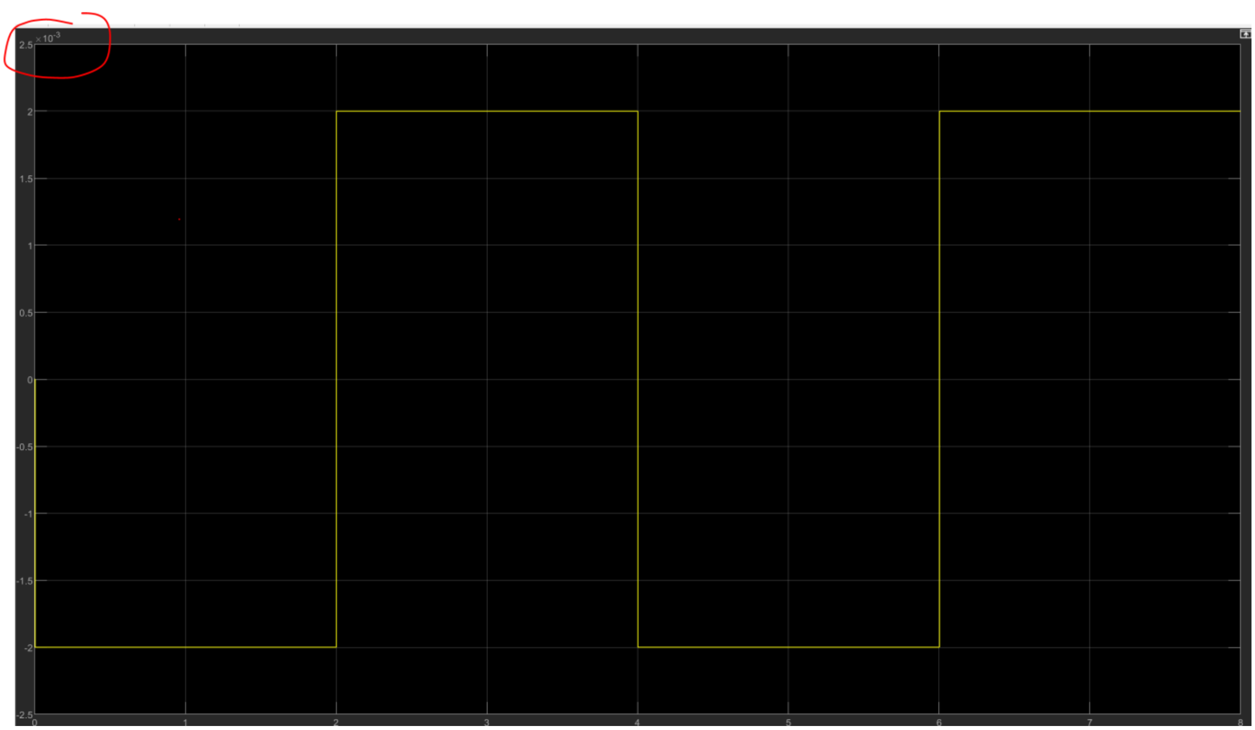
It ran for only 8ms

So run only that time.

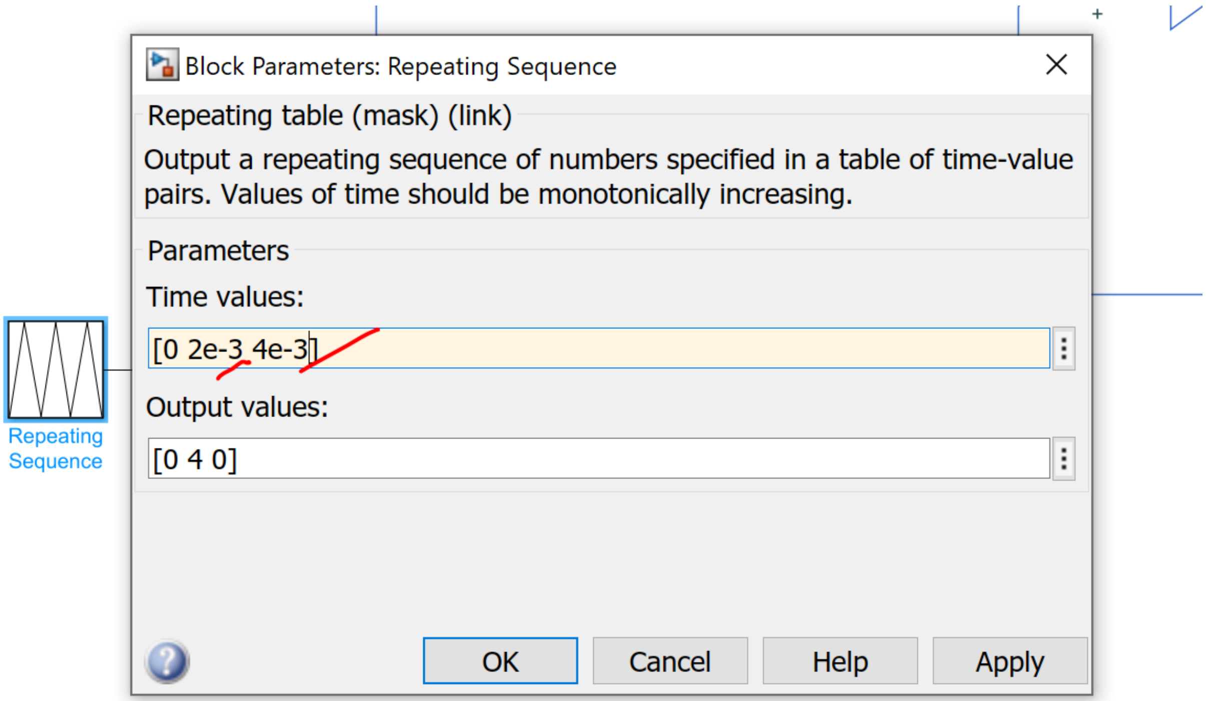


Here is the answer:

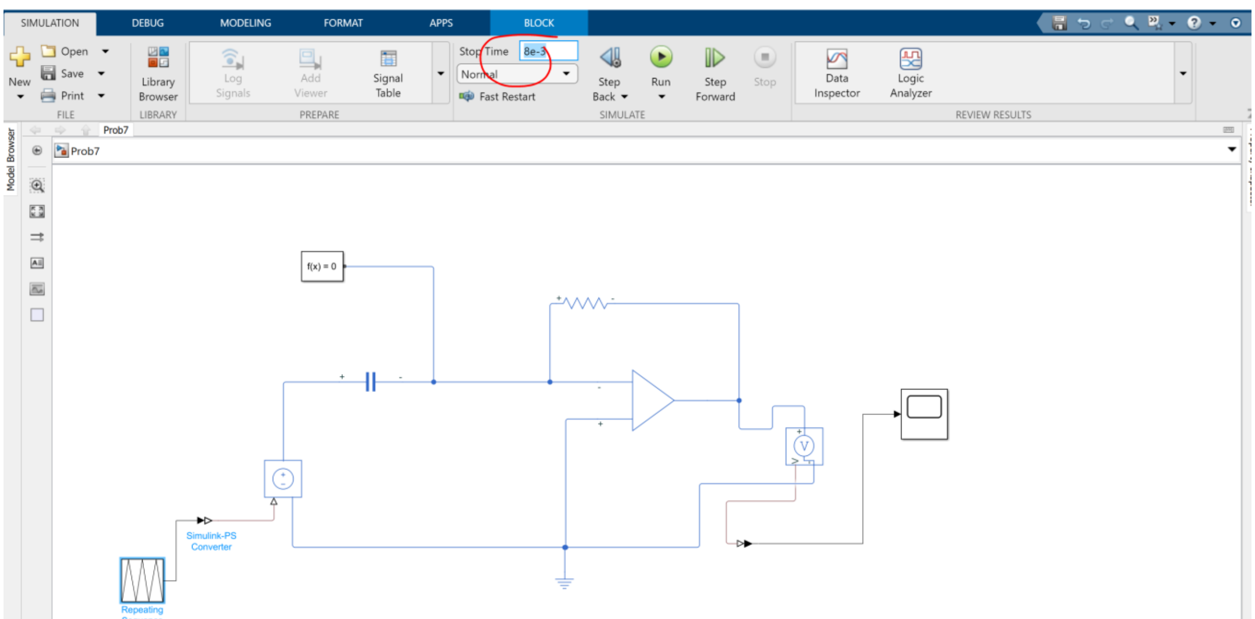
Now, in out answer it is 2 V. but here it is showing 2 mV. This is because, our time input has been in second, not mili second.

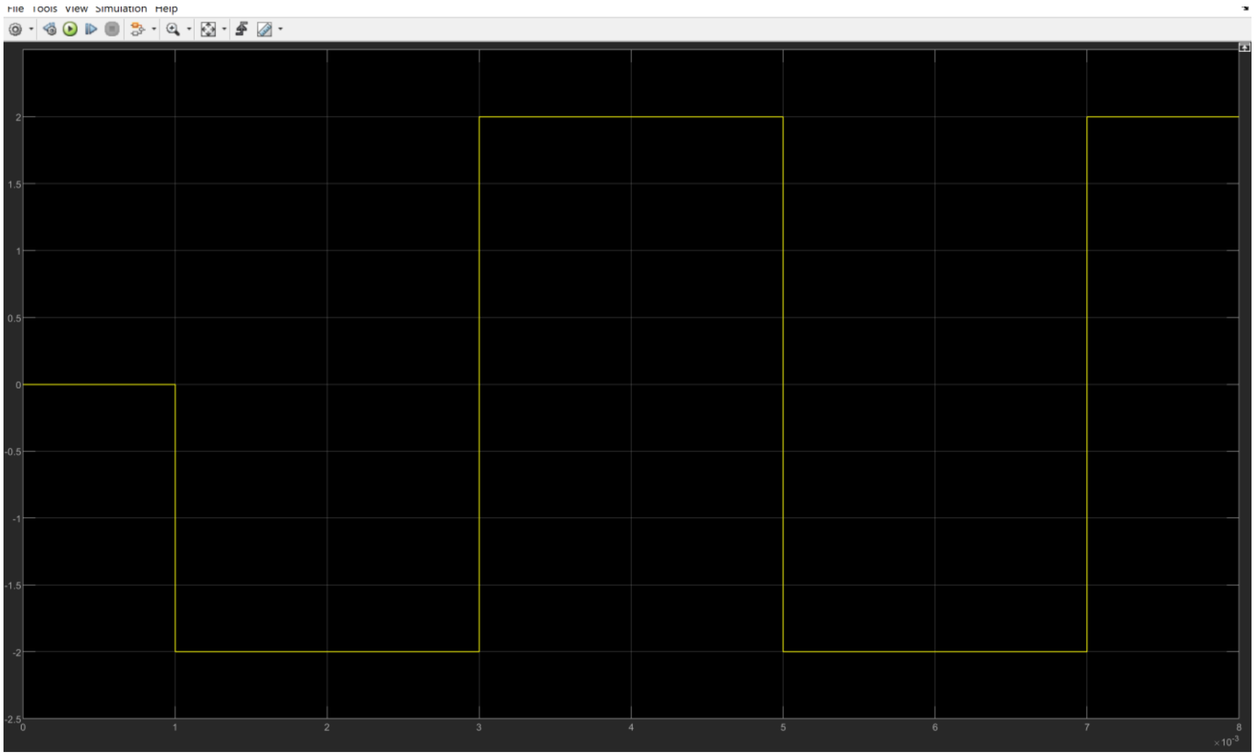


So we update this accordingly



We brought mili now in the time.





See, that time is in the mili (x10^-3), not voltage.