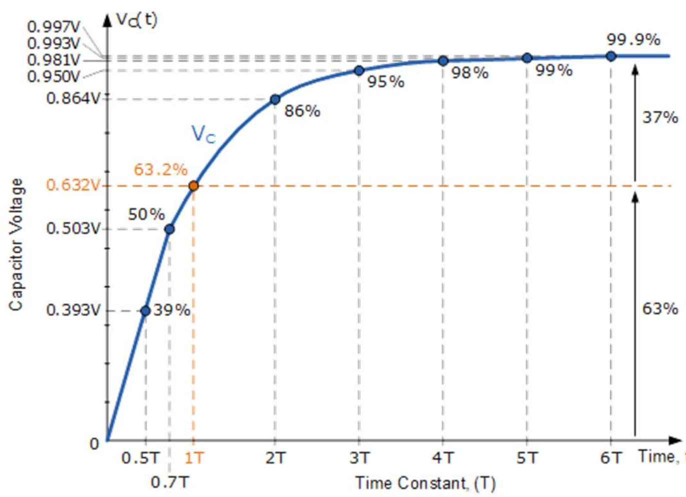
PRAG 304L: Electrical Diagnostics for Farm Machinery Lab
Instructor: Dr. Pappu Kumar Yadav
Spring 2025

Lab 4: Using Simulink for Basic Circuit Simulation and Deployment on Raspberry Pi 5 for controlling LED remotely

1. Overview

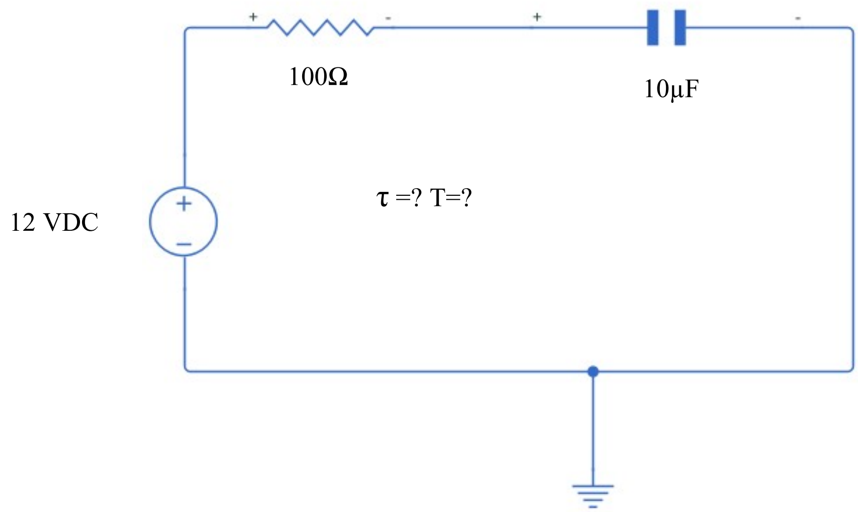
A capacitor is a device that stores electrical energy in an electric field. It consists of two conductive parallel plates separated by an insulator (dielectric). When a capacitor is connected to a battery or any voltage source, electrons start accumulating on one plate (negative plate), and an equal number of protons accumulate on the other plate (positive plate). This creates a potential difference i.e., **voltage** between the plates, which grows until it is equal to the voltage of the battery. This process is called **charging a capacitor**. The rate, i.e., how fast or slow the capacitor is charged, is controlled by using a resistor with the capacitor in a circuit.
The time taken by capacitor to reach 63.2% of its maximum value is called time constant of the capacitor (same as time constant of an inductor, Lecture 4, slide 11).



The formula to calculate time constant of a capacitor is given as:

$$\tau = RC$$

The time constant is measured in seconds (s), while R is in ohms (Ω) and C is in Farads (F). After about T= **5τ** seconds, the capacitor becomes fully charged. In the case of **discharging** a fully charged capacitor, the capacitor itself acts as a source i.e., no voltage or current source is connected in the circuit. A load (usually a **resistor**) is connected in series to discharge the capacitor. During this electrical energy is converted into heat by the resistor. A capacitor is said to be fully discharged when its voltage reaches nearly 0 volts. Like the charging case, it takes about T= **5τ** seconds for the capacitor to become fully discharged.



Determine the time constant (τ) of the capacitor during the charging process by calculation first before proceeding ahead with the simulation. Also determine approximate time (T) for fully charging the capacitor using the calculation first.

1. Objectives

i. To learn about charging and discharging of a capacitor through Simulink.
ii. To determine the time constant of a capacitor through Simulink.
iii. To determine the time taken by a capacitor to fully charge and discharge using Simulink.
iv. To control LED connected on Raspberry PI5 board remotely using a Simulink model

2. Experiment 1: Charging of a capacitor

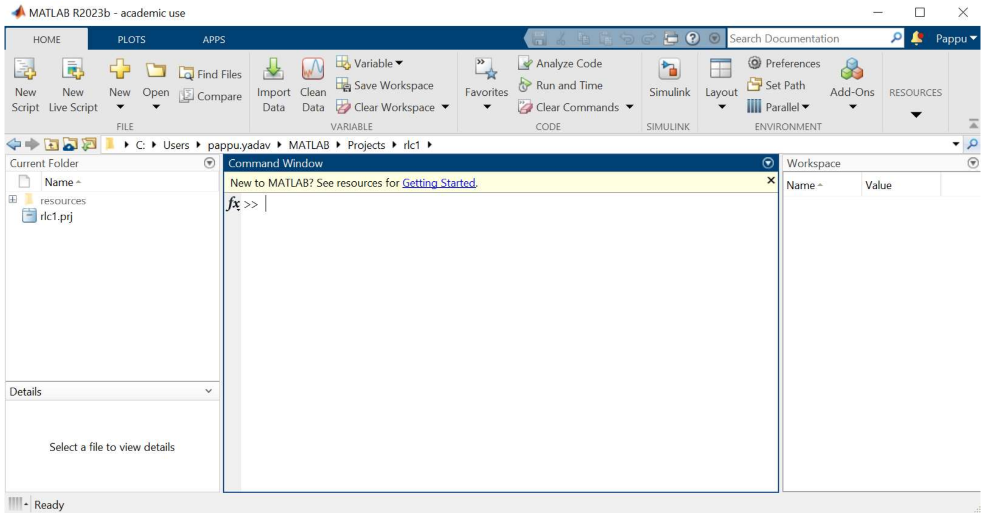
In this experiment, we will create an RC circuit with a 12 VDC power supply, a 100 Ω resistor (R) and a 10 µF capacitor as shown above. The goal is to determine the time constant (τ) of the capacitor and the time (T) taken to fully charge the capacitor by simulation in Simulink.

3. Procedure

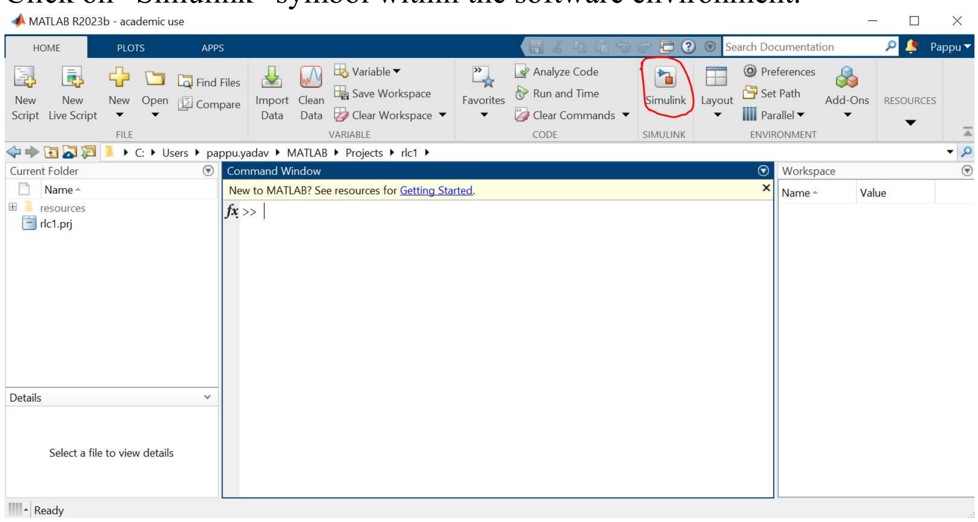
Charging

i. Open MATLAB Software on your lab computer. The symbol should appear like this.

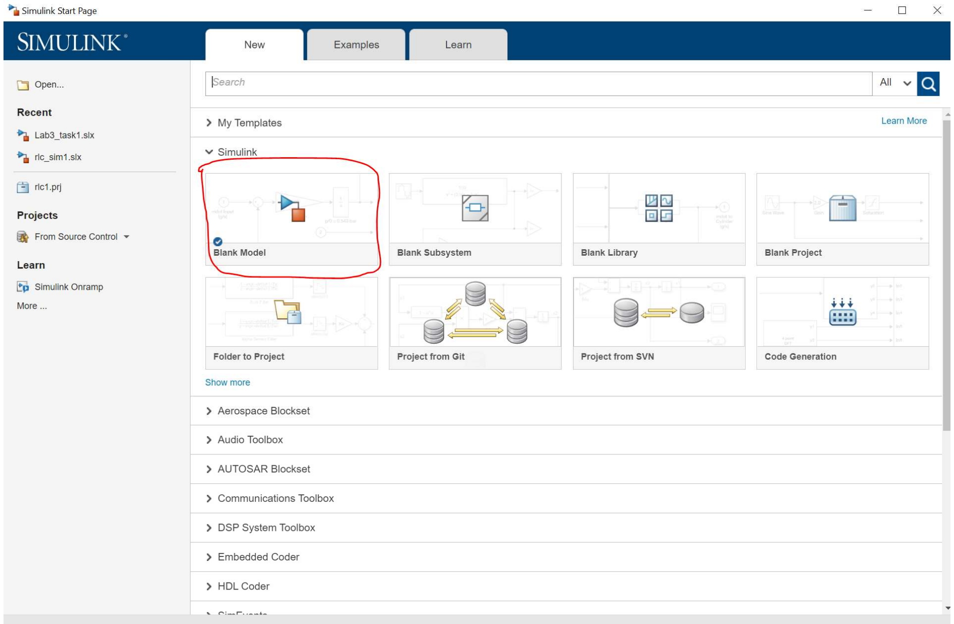
ii. When opening the first time, make sure to create an account with MATLAB using your South Dakota State University email address.
iii. You will see a window like this when MATLAB opens successfully.



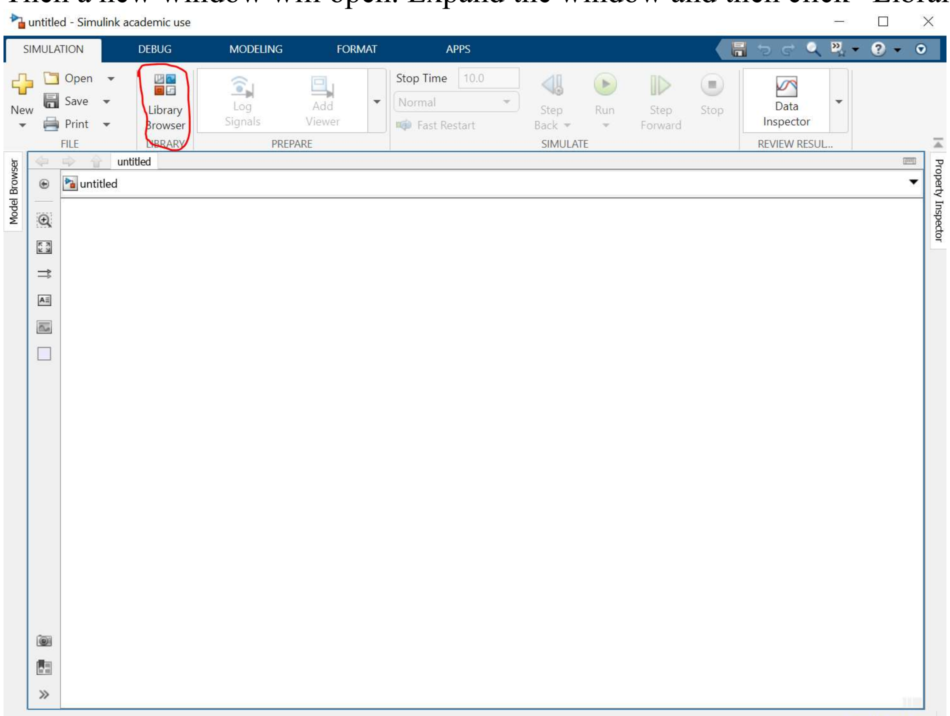
iv. Click on “Simulink” symbol within the software environment.



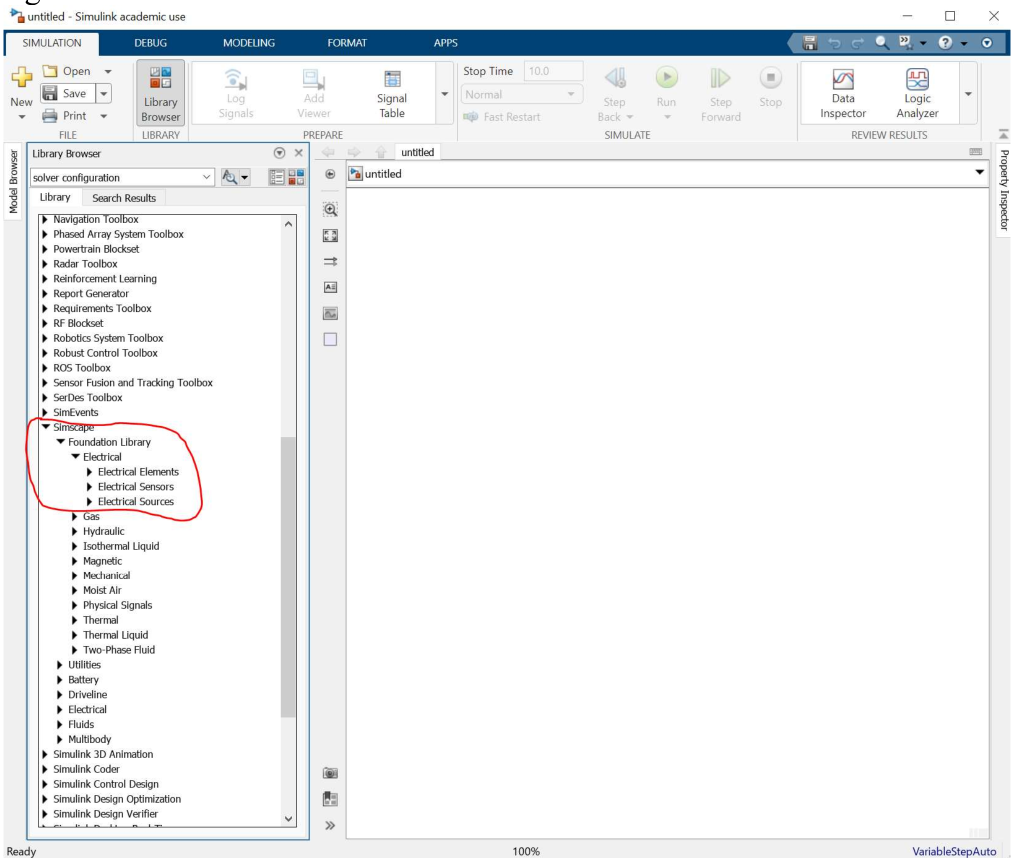
v. This will open a new window of Simulink and then hover your cursor over “Blank Model” and then click “Create Model”.



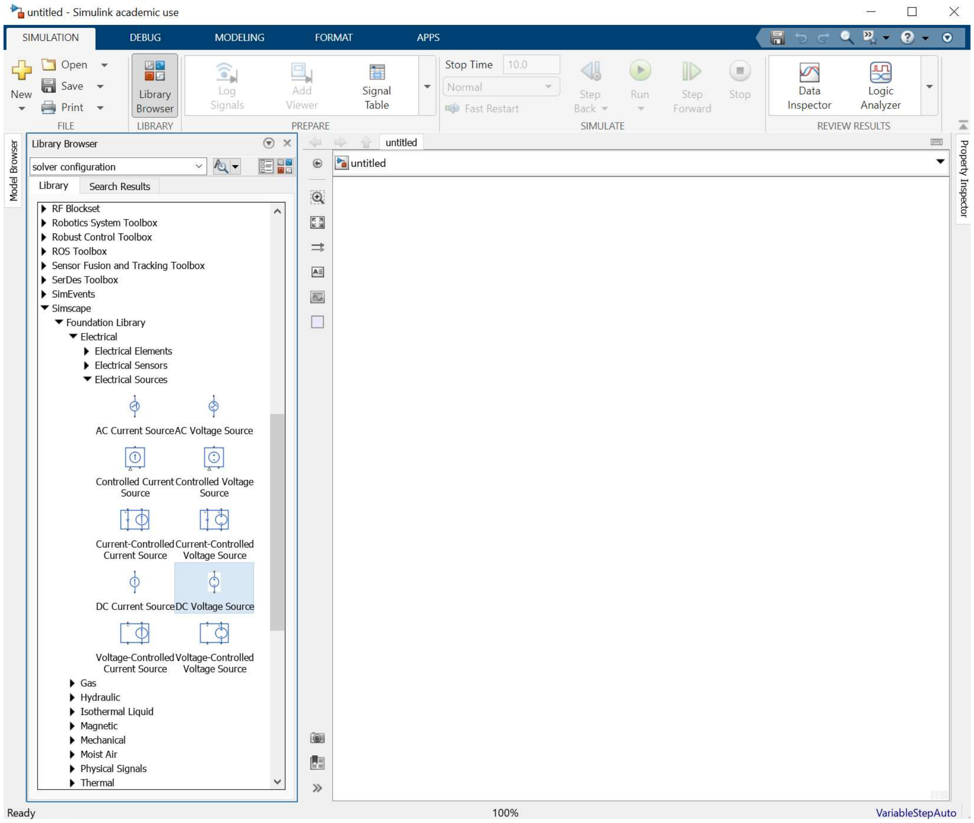
vi. Then a new window will open. Expand the window and then click "Library Browser".

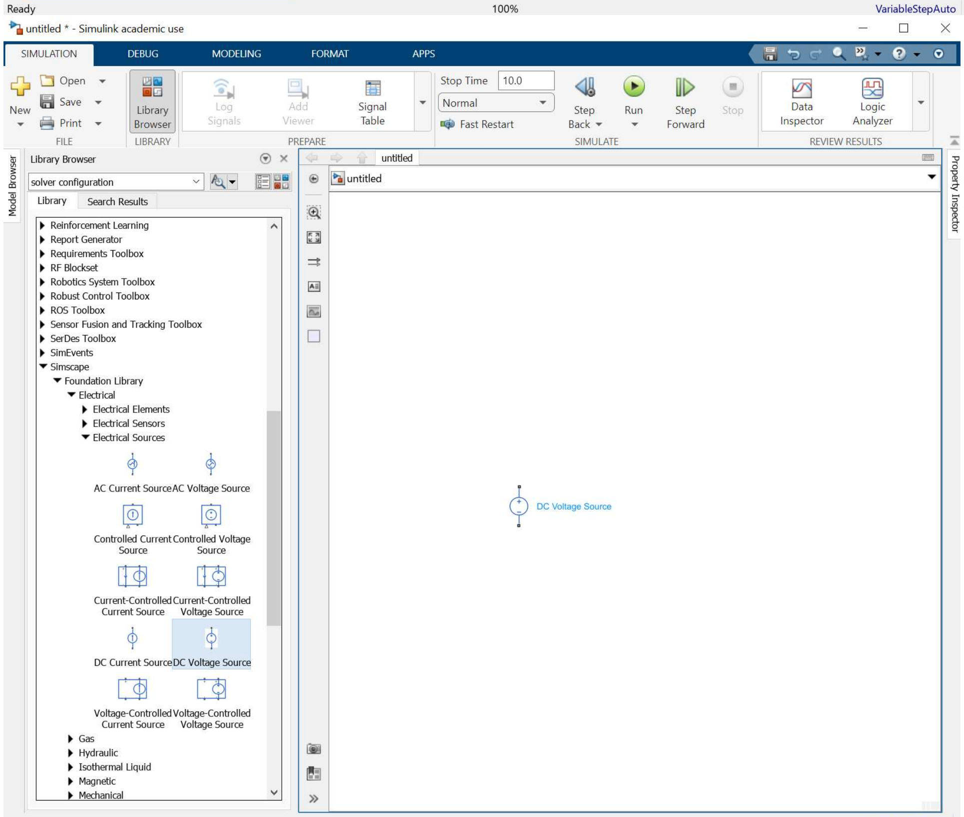


vii. This will open list of libraries available on the left side of the window. Scroll all the way down to "Simscape→Foundation Library→Electrical as shown below. This has three important groups which we will be using often through the labs.

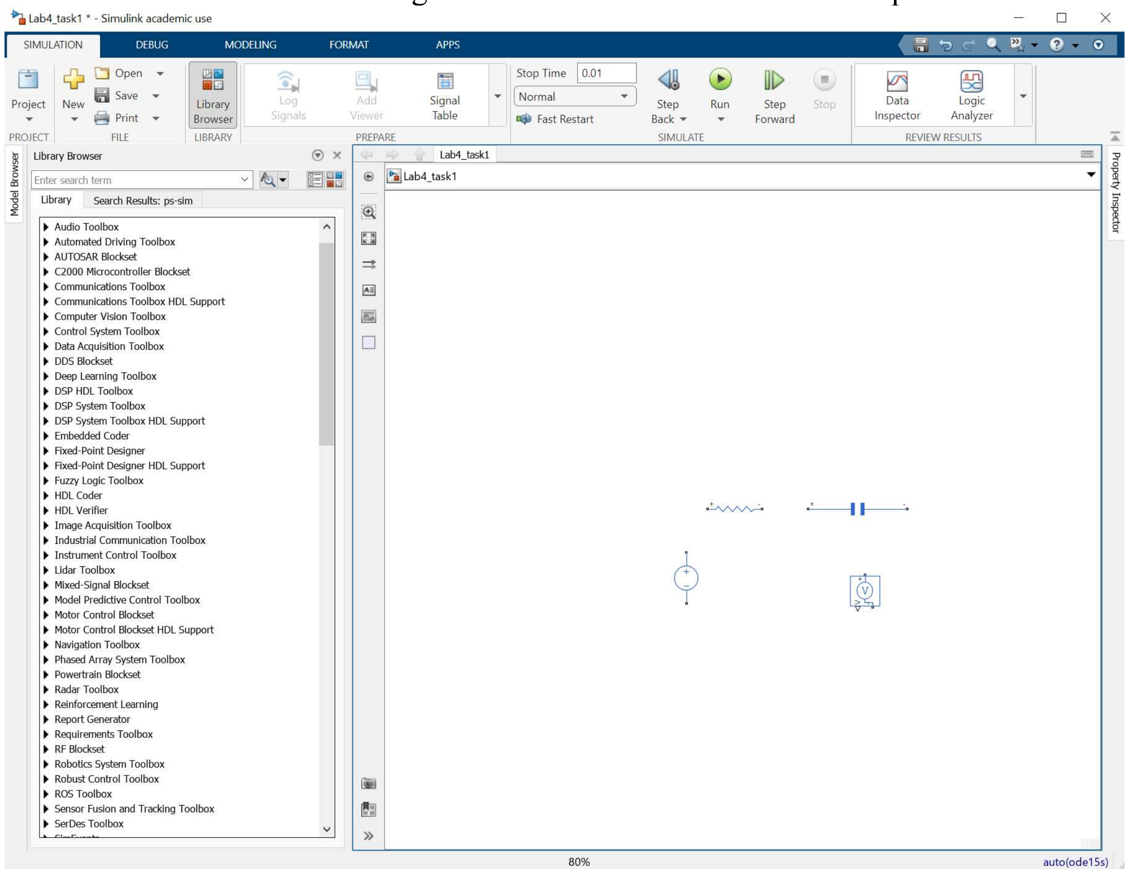


viii. We will start by first expanding "Electrical Sources" and select "DC Voltage Source" as shown below. Then right click and select "Add block to model untitled". This will add the block in the workspace on the right.

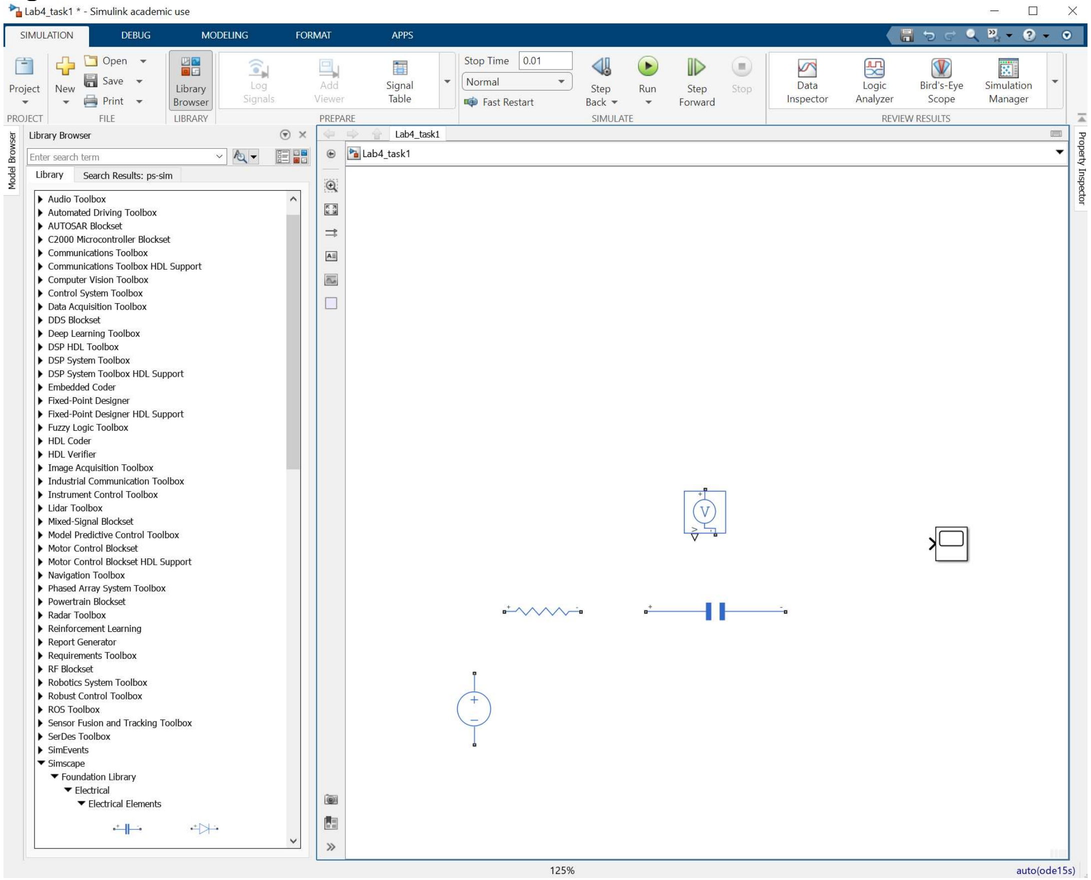

untitled - Simulink academic use
SIMULATION
DEBUG
MODELING
FORMAT
APPS
New
Open
Save
Print
FILE
Library Browser
LIBRARY
Log Signals
Add Viewer
Signal Table
PREPARE
Stop Time 10.0
Normal
Fast Restart
Step Back
Run
Step Forward
Stop
SIMULATE
Data Inspector
Logic Analyzer
REVIEW RESULTS
Library Browser
solver configuration
Library
Search Results
RF Blockset
Robotics System Toolbox
Robust Control Toolbox
ROS Toolbox
Sensor Fusion and Tracking Toolbox
SerDes Toolbox
SimEvents
Simscape
Foundation Library
Electrical
Electrical Elements
Electrical Sensors
Electrical Sources
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
Gas
Hydraulic
Isothermal Liquid
Magnetic
Mechanical
Moist Air
Physical Signals
Thermal
untitled
Model Browser
Property Inspector
Ready
100%
VariableStepAuto


untitled * - Simulink academic use
SIMULATION
DEBUG
MODELING
FORMAT
APPS
New
Open
Save
Print
FILE
Library Browser
LIBRARY
Log Signals
Add Viewer
Signal Table
PREPARE
Stop Time 10.0
Normal
Fast Restart
Step Back
Run
Step Forward
Stop
SIMULATE
Data Inspector
Logic Analyzer
REVIEW RESULTS
Library Browser
solver configuration
Library
Search Results
Reinforcement Learning
Report Generator
Requirements Toolbox
RF Blockset
Robotics System Toolbox
Robust Control Toolbox
ROS Toolbox
Sensor Fusion and Tracking Toolbox
SerDes Toolbox
SimEvents
Simscape
Foundation Library
Electrical
Electrical Elements
Electrical Sensors
Electrical Sources
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
Gas
Hydraulic
Isothermal Liquid
Magnetic
Mechanical
untitled
DC Voltage Source
Model Browser
Property Inspector
100%
VariableStepAuto

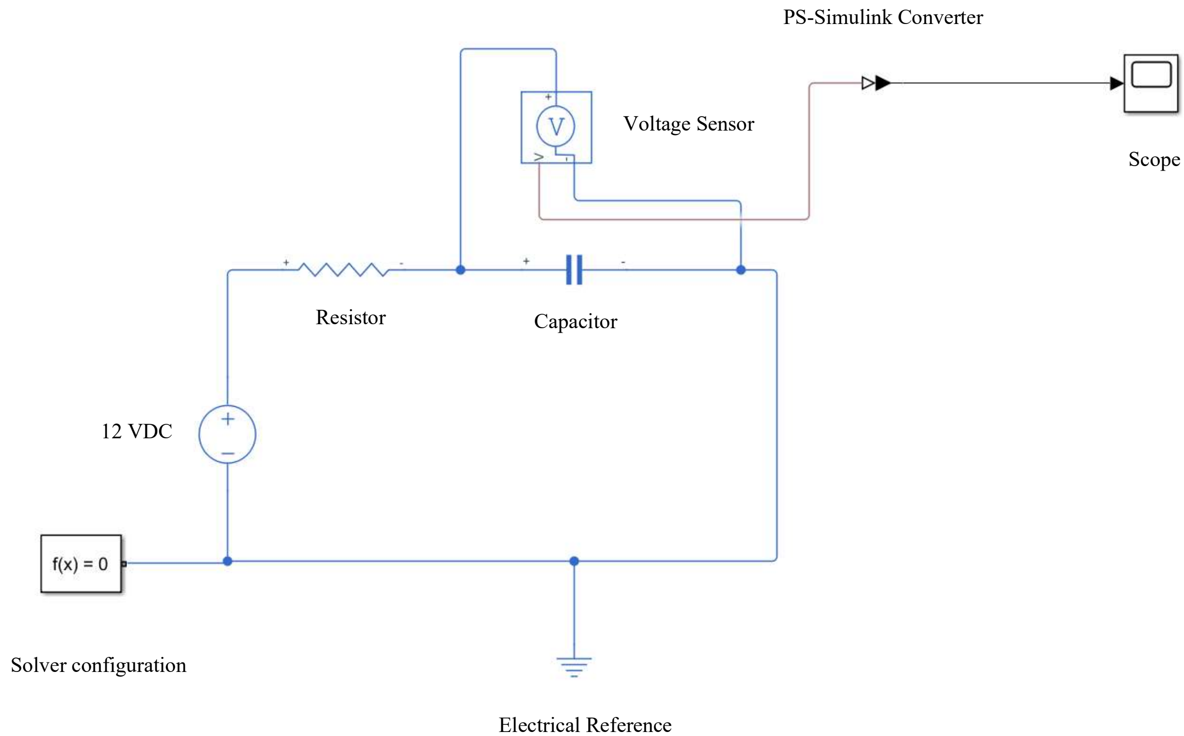
ix. In the same way, under "Electrical Elements" choose "Resistor", "Capacitor" and under "Electric sensor" choose "Voltage sensor" and add them to the workspace.



x. Besides these, we will also need two important components: Ground reference and Scope. For ground reference, add this within "Simscape→Electrical→Connectors & References" as shown below. Please note that this "Electrical" is different than the previous one. You can type "Scope" in the search bar to add it.

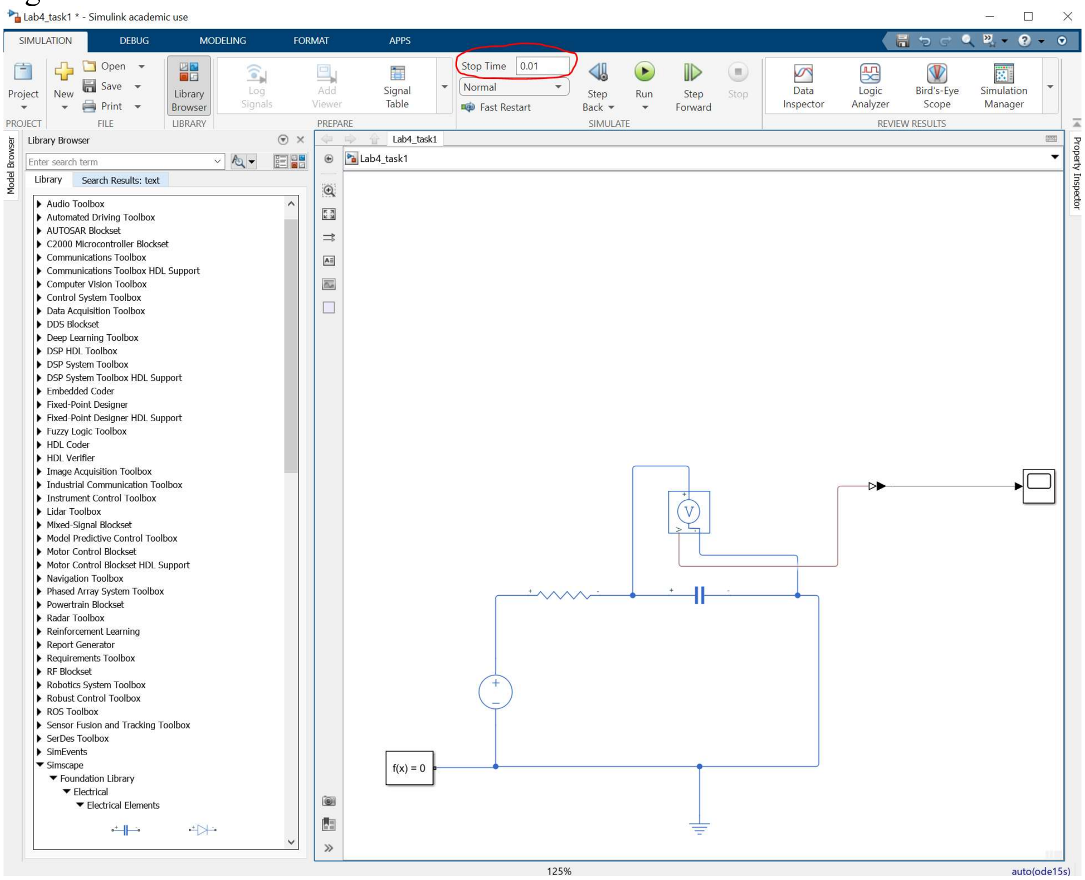


xi. Apart from all these we need two important blocks for "Simscape" components. They are "Solver Configuration" and "PS-Simulink Converter". Please do not worry about these. Just remember that "PS-Simulink Converter" block converts physical signals like voltages and currents to be displayable by Simulink software. "Solver Configuration" is almost always used for "Simscape" components and is always connected to the ground as shown. Add these by typing in the search box. Then start connecting all the blocks and you should see a complete circuit model as shown.

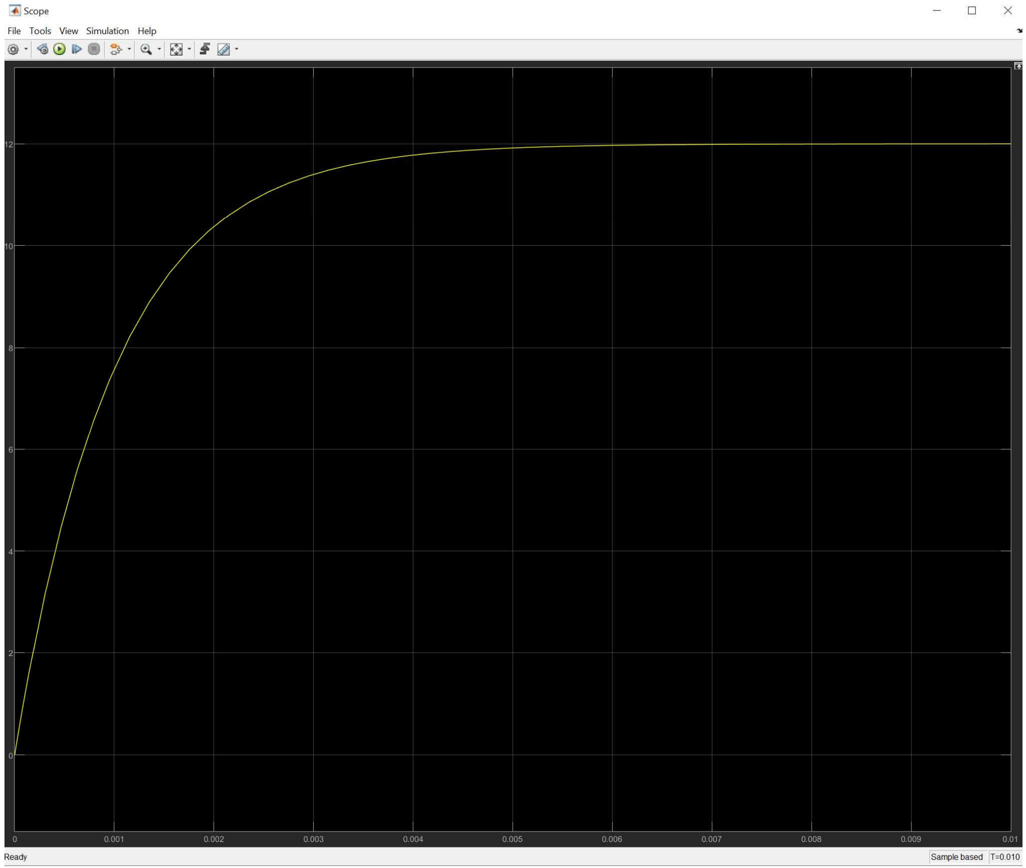


xii. Now double click each of the Resistor and Capacitor block and adjust the values as R=100 Ω and C= 10μF (i.e., $10e^{-6}$ F).

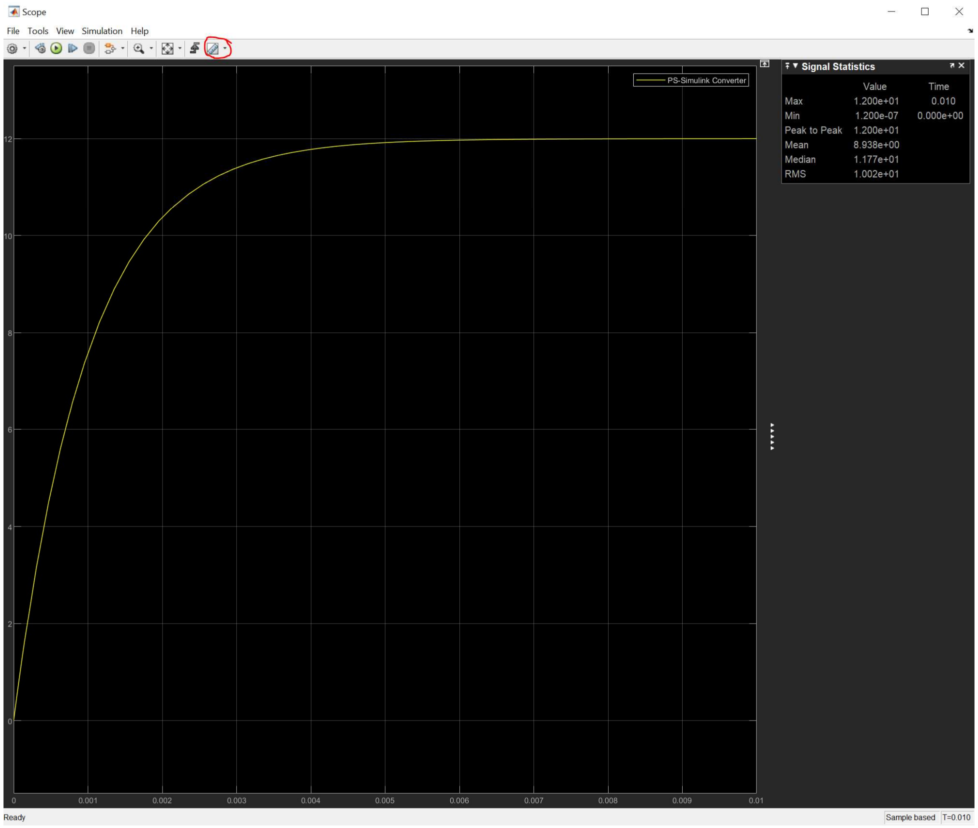
xiii. When this circuit model is ready, you can click the "Run" button to run the simulation. You can set up the time duration for how long you want to run the simulation. To generate a right looking graph, please setup "Stop Time =0.01 seconds".



xiv. After the simulation run is completed, please double click the "Scope" block and then it should open a new window with a graph as shown below.



xv. Click View-→Legend and then click the symbol within the red box shown below and choose "Signal Statistics". You will see the graph window as shown below.



Use the information within "Signal Statistics" to determine the time (T) taken by the capacitor to reach maximum voltage.

xvi. Now to determine the time constant try to calculate 63.2% of the maximum voltage of the charged capacitor i.e., 63.2% of 12V= 7.58 V. Point your cursor along Y-axis for approximate value of 7.58V and then determine the corresponding value of X-axis to determine the time in second. This time is called the time constant (τ) of the charging capacitor. Check if the total time (T) from the "Signal Statistics" is approximately equal to 5 times the determined time constant value.

Please note the values and graphs for the lab report. When you have successfully run this simulation and determined the time constant and total time values, please use the following values of Resistors and Capacitors as per your section and group and determine the time constant and total time taken to fully charge the capacitor for your lab reports.

Lab Section	**Lab Group**	**R (Ω)**	**C (µF)**
I	I	200	15
	II	150	25
	III	300	30
	IV	400	45
	V		
II	I	150	50
	II	250	60

	III	100	80
	IV	350	450
	V	350	20

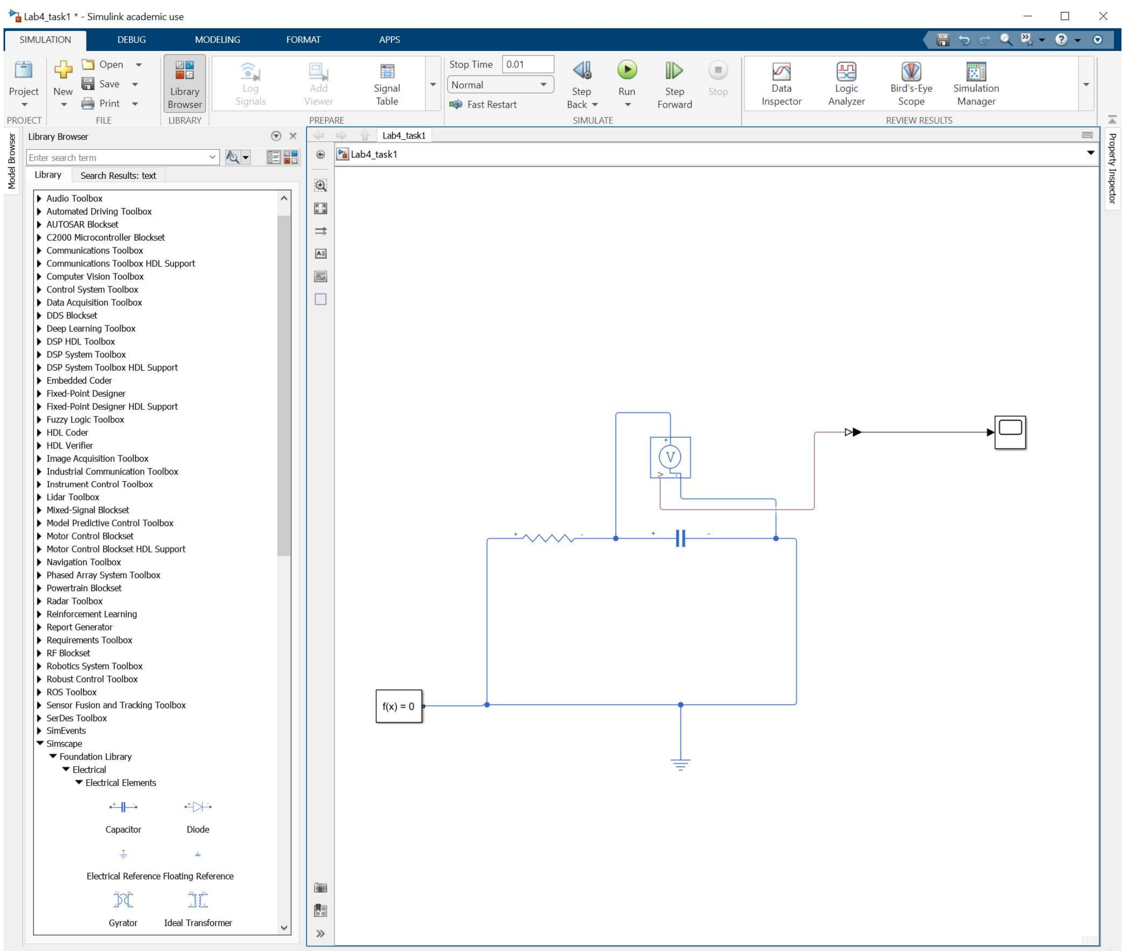
4. Experiment 2: Discharging of capacitor

In this experiment, we will create an RC circuit with no power supply, a 100 Ω resistor (R) and a 10 μF capacitor as shown below. The goal is to determine the time constant (τ) of the capacitor and the time (T) taken to fully discharge the capacitor by simulation in Simulink.

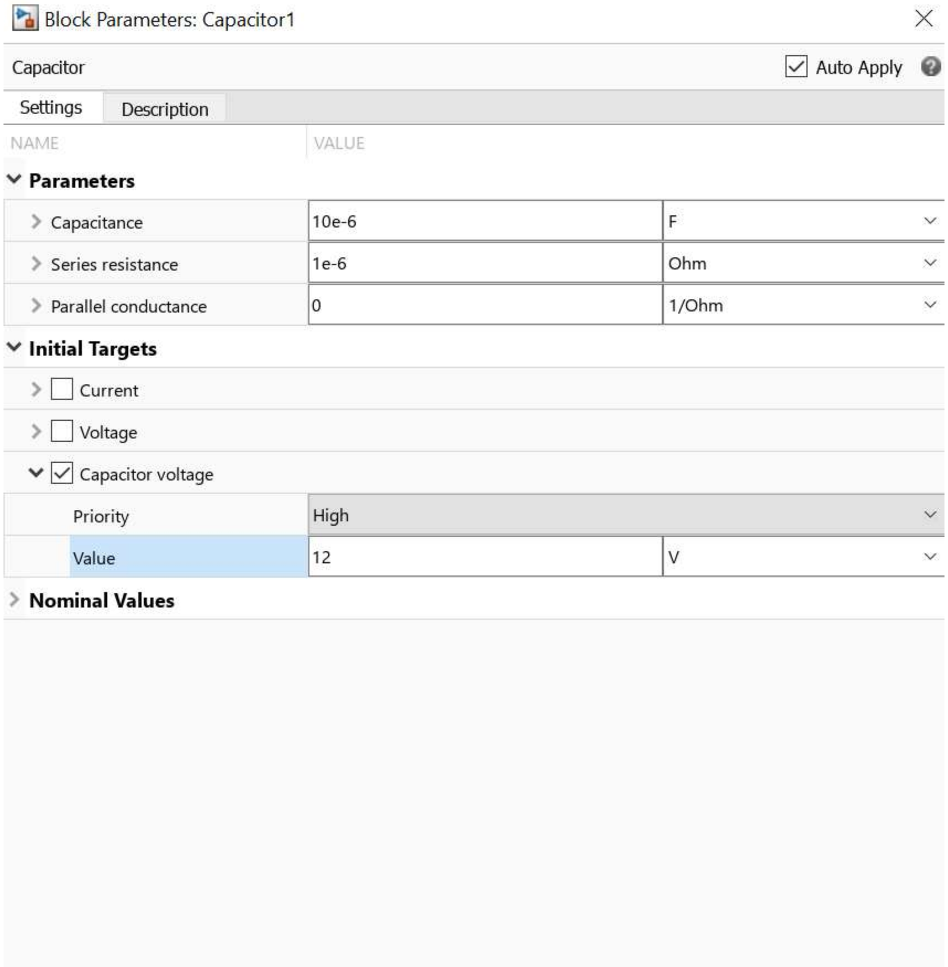
5. Procedure

Discharging

i. Repeat the steps in procedure for charging the capacitor except that no voltage source is used here. Your complete model should look like the one below.



ii. To define the capacitor fully charged to a certain voltage (here 12 VDC), double click the Capacitor block and Expand The "Initial Targets". Then check mark the "Capacitor Voltage" and set its value as 12 V as shown below.

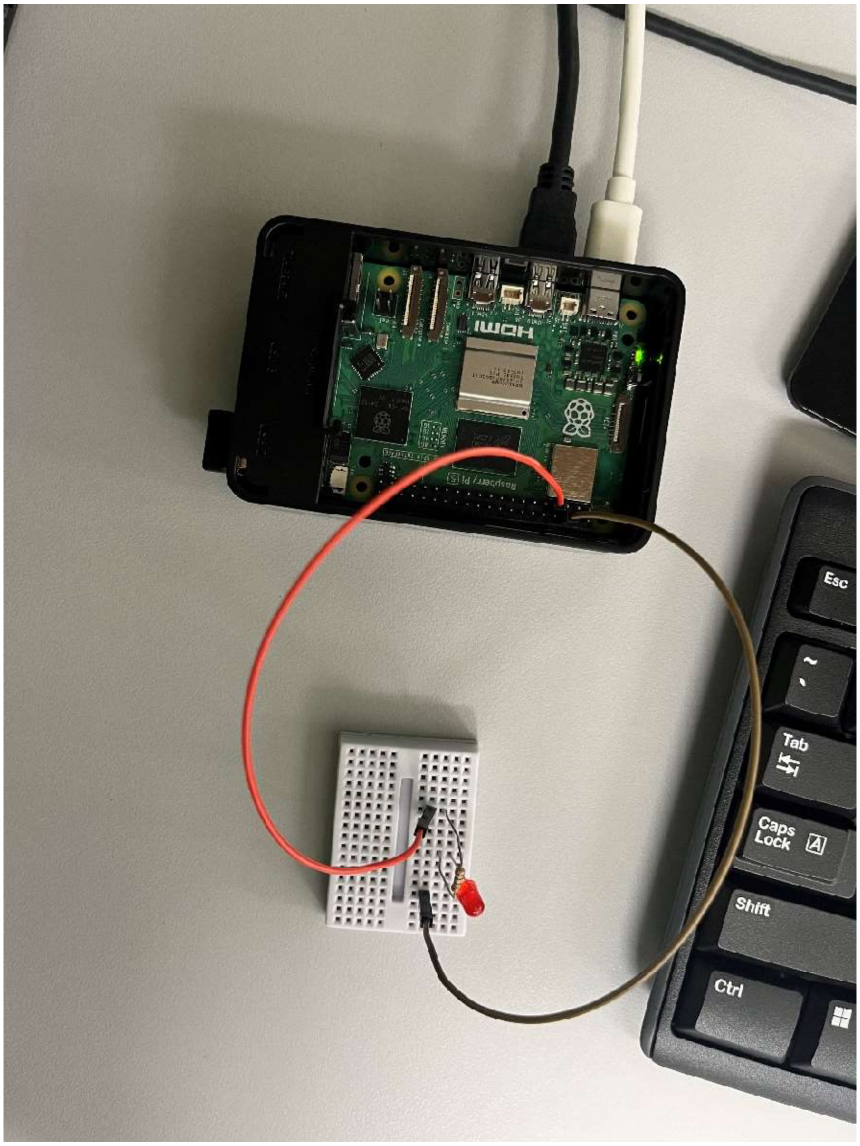


iii. Run the simulation by following steps as in the charging capacitor experiment and follow similar steps to determine the time constant of discharging capacitor and total time taken to fully discharge the capacitor. Please note down the values and graphs and include them in your lab report.

iv. Finally, use the above table to use the values of resistors and capacitors corresponding to your lab section and group to determine time constant and total time of discharging capacitor. Please make sure to include graphs generated in your lab reports.

6. **Experiment 3:** Controlling LED connected on Raspberry Pi remotely using Simulink

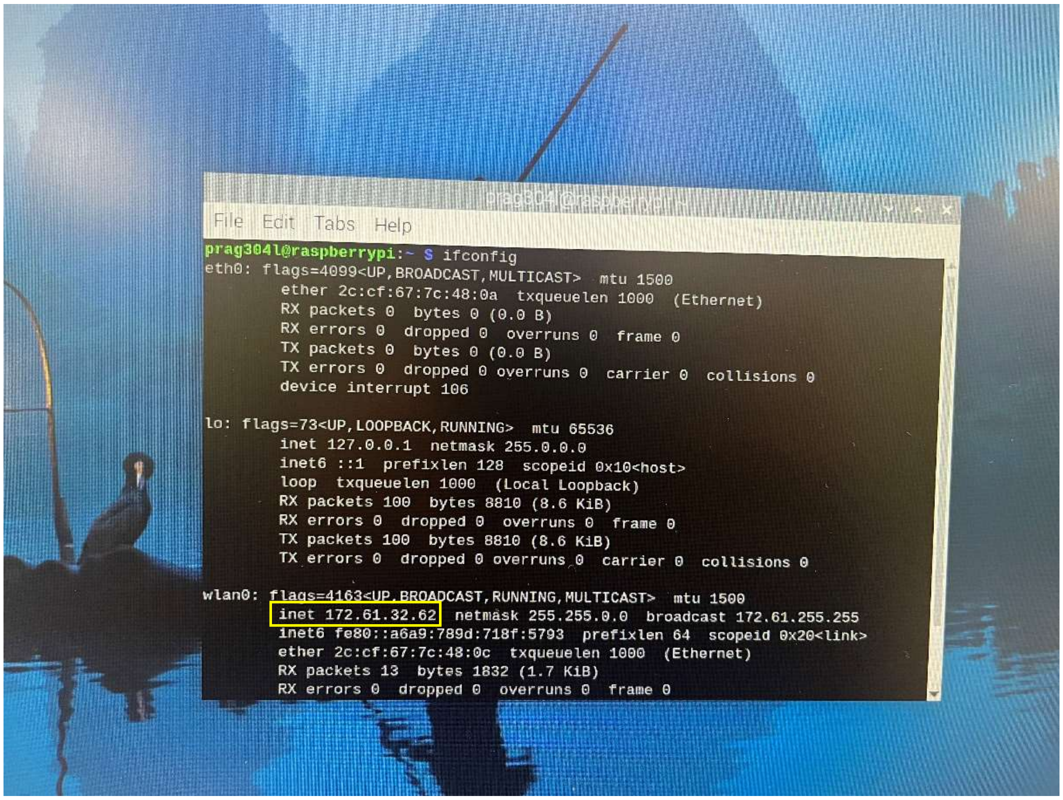
i. Connect an LED with a resistor of your choice on a breadboard as shown in the figure below. *Please note that you will be connecting the red wire to a different GPIO pin than the one shown below.*



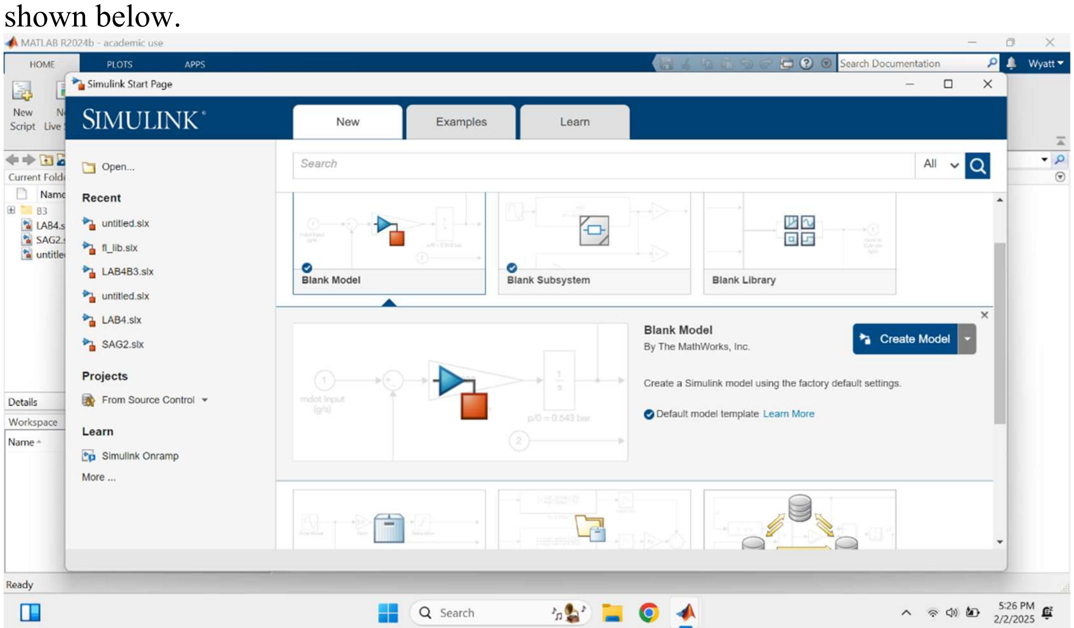
ii. Use the PIN header diagram layout provided to you as a reference and then connect your red wire to GPIO 17 and the brown/black wire to a Ground (GND) pin of your choice. (Raspberry Pi has multiple GND pins).

iii. Now power ON your Raspberry Pi following the steps from your last lab. At the same time turn on your lab's laptop/computer and open Simulink from within the MATLAB 2024 (MATLAB version earlier than 2024 doesn't support Raspberry Pi5) environment.

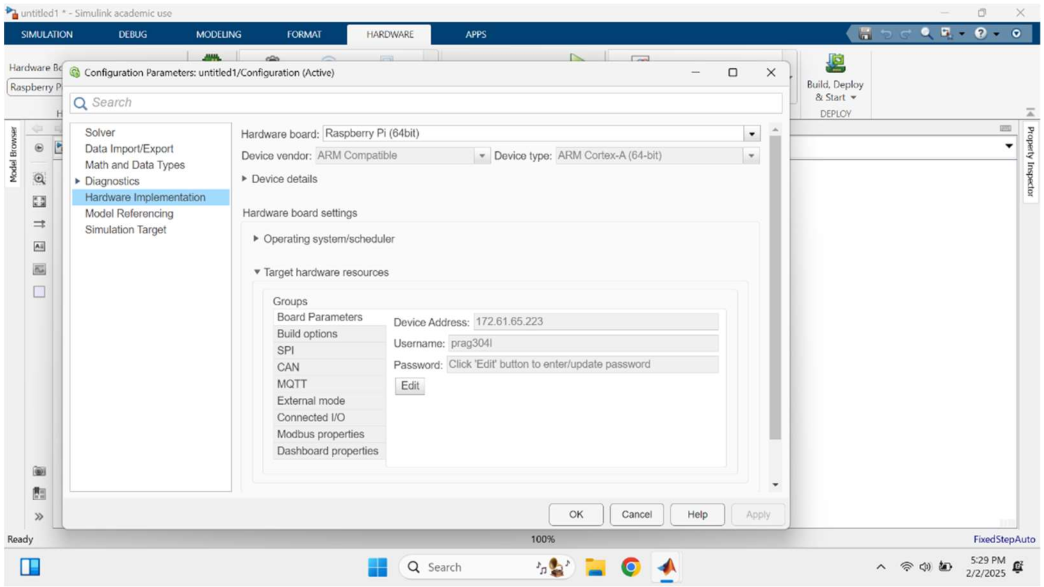
iv. On your Raspberry Pi5, open Terminal window and type "ifconfig" and hit enter on your keyboard and then find the "IP address" of your Raspberry Pi5 as shown below.



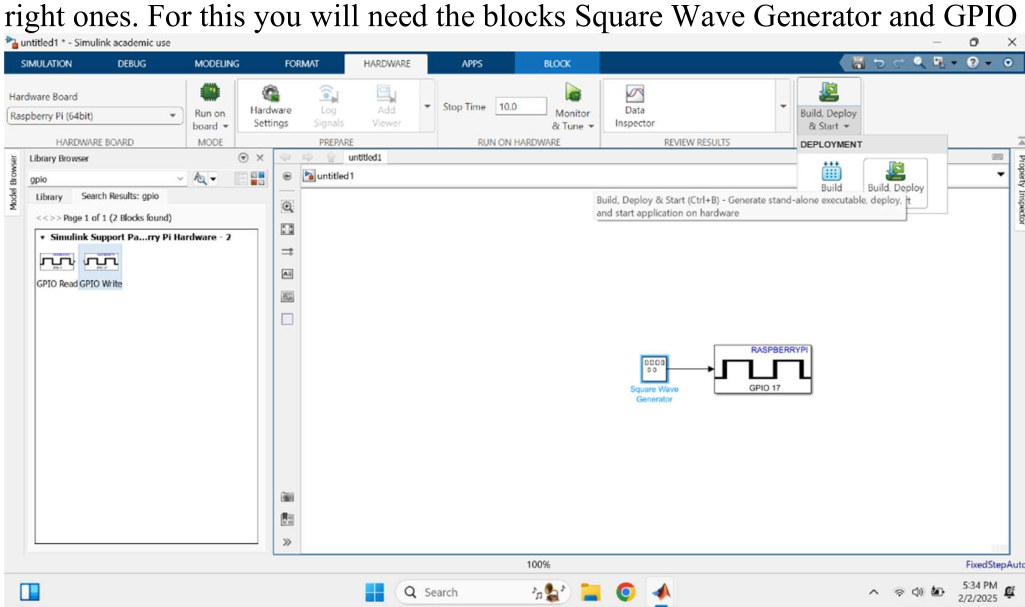
v. Now on the Simulink open a Blank Model by clicking the Blank Model option as shown below.



vi. Then setup the Hardware to Raspberry Pi5 board and connect it using the "IP Address" you got in step (iv). Use the login credentials as "username: prag304l" and password as "sdstate3" as shown below. You will need to select the "Hardware" tab and then press CTRL+E on your keyboard to open the configuration window.

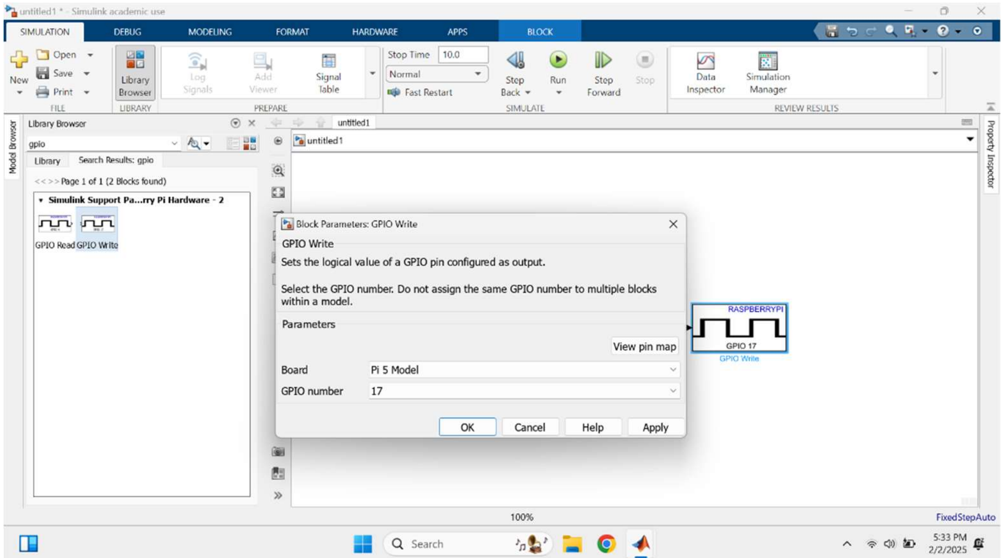


vii. Then build the Simulink model that will blink the LED on your breadboard connected to the Raspberry Pi5. The model should look like the one shown below. Note: You can simply type the names of the blocks within the Library Browser search box and press enter. This will open a list of the block related to that name. Just choose the right ones. For this you will need the blocks Square Wave Generator and GPIO Write.

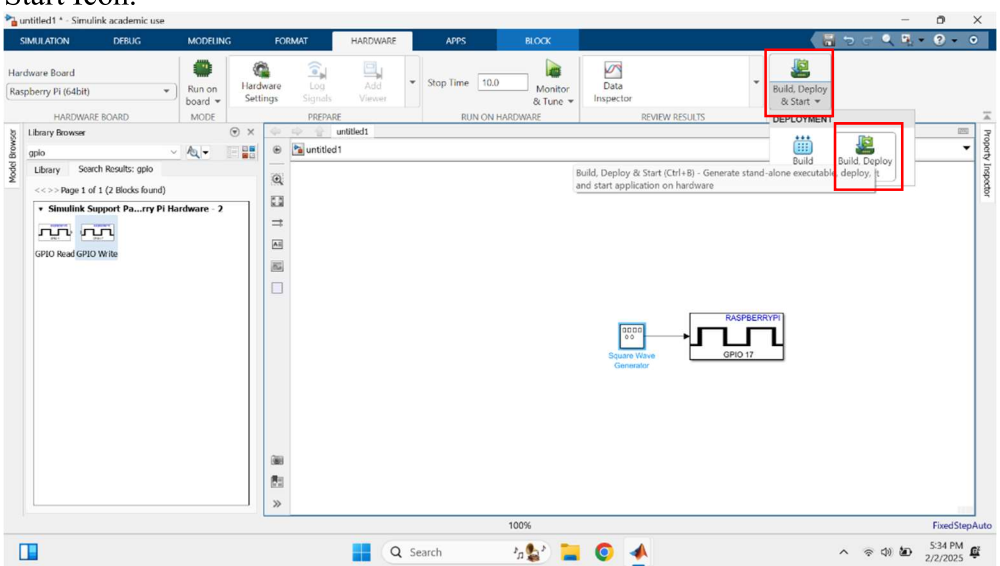


viii. Make sure to connect the two blocks with an arrow.

ix. Double click each block to make the right settings. Make sure you choose Raspberry Pi5 and GPIO 17 for the GPIO Write block and Amplitude as 1 and Freq as 1 rad/sec or Hz for the Square Wave Generator.

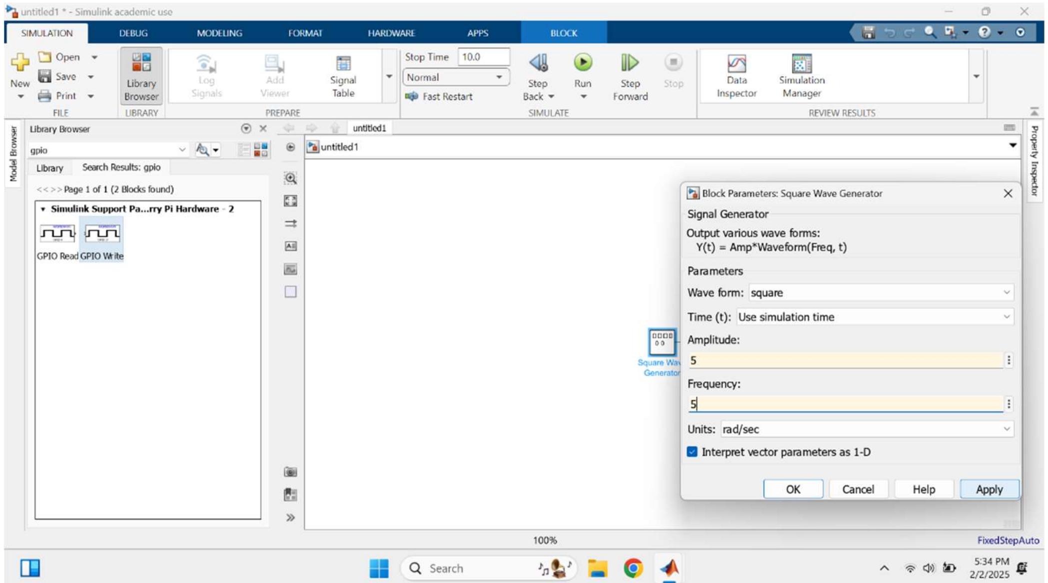


x. Now your Simulink model should be ready to build and deploy on your Raspberry Pi5 board remotely. Click the Build Deploy and Start within the Build Deploy and Start Icon.



xi. Just wait for some time and keep observing your LED. After some time, it should start blinking every second.

xii. Now repeat the experiment by changing the Amplitude to 5 and frequency to 5 rad/sec.



xiii. Again, Build Deploy and Start as you did earlier and then wait for some time and keep observing the LED.

xiv. Observe the changes and compare the changes.

xv. Take a picture of the experiment methods and results and submit lab report. Just submit pictures of experiment methods and results and explain your findings.

For Bonus of 10 points

Power off the Raspberry PI5 and take it to the High Bay and go there and connect it to a power outlet. Make sure to take the breadboard and LED setup. Split your group members and half of you can go there and the remaining half of you can stay in the lab running the Simulink. Now change the settings of Square Wave Generator (Amplitude and Frequency) and Build Deploy and Start the new model. Call your group members in the High Bay and check if the LED blinks with the new settings. When done successfully get back to the lab and include the findings in your report to claim the bonus points.