SENSORS AND AUTOMATION LAB

Name: Ayush R. Bodile

Branch : Metallurgy and Material Technology

Semister: III

Batch : A

MIS : 612211013

Sensors Modelling and Simulation

PRACTICAL NO.1. Characterize the temperature sensor (RTD)

Aim: To understand the working principle of RTD.

Objective:

1. Study static and dynamic characteristics of RTD.
2. Study effect of various parameters on RTD performance.

Procedure:

1.Select the material of RTD you want to use. Temperature coefficient (α) for the same will be displayed on the screen. Note this value.Click on ' R0' to get the value of R0 for selected RTD. Note the same.Click on 'Get Temp'. The temperature for which Rt is to be found will be displayed.

2.Using formula calculate the value of Rt and enter the answer in the box provided (upto 2 decimals with rounding off). Submit the answer using submit button.If your calculation is correct, go to step 3. Repeat the procedure min 3 times.If your calculation is wrong, you will be asked to repeat the same. Please refer to GET FORMULA tab and verify your calculations.

3.After finishing minimum 3 set of correct readings, you can see the graph by clicking on 'Plot'.

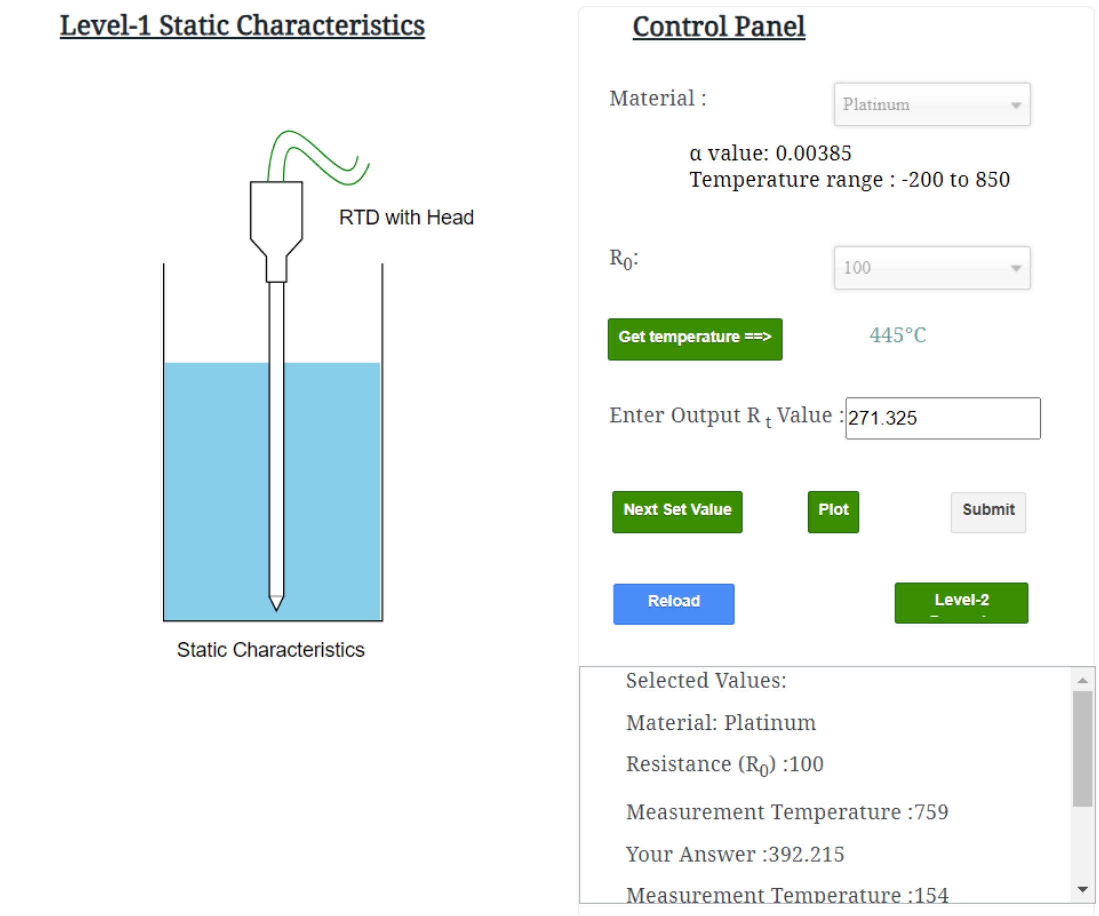
4.When the graph is displayed click Next tab to repeat the procedure with different reference resistance(R0) value and with different materials.

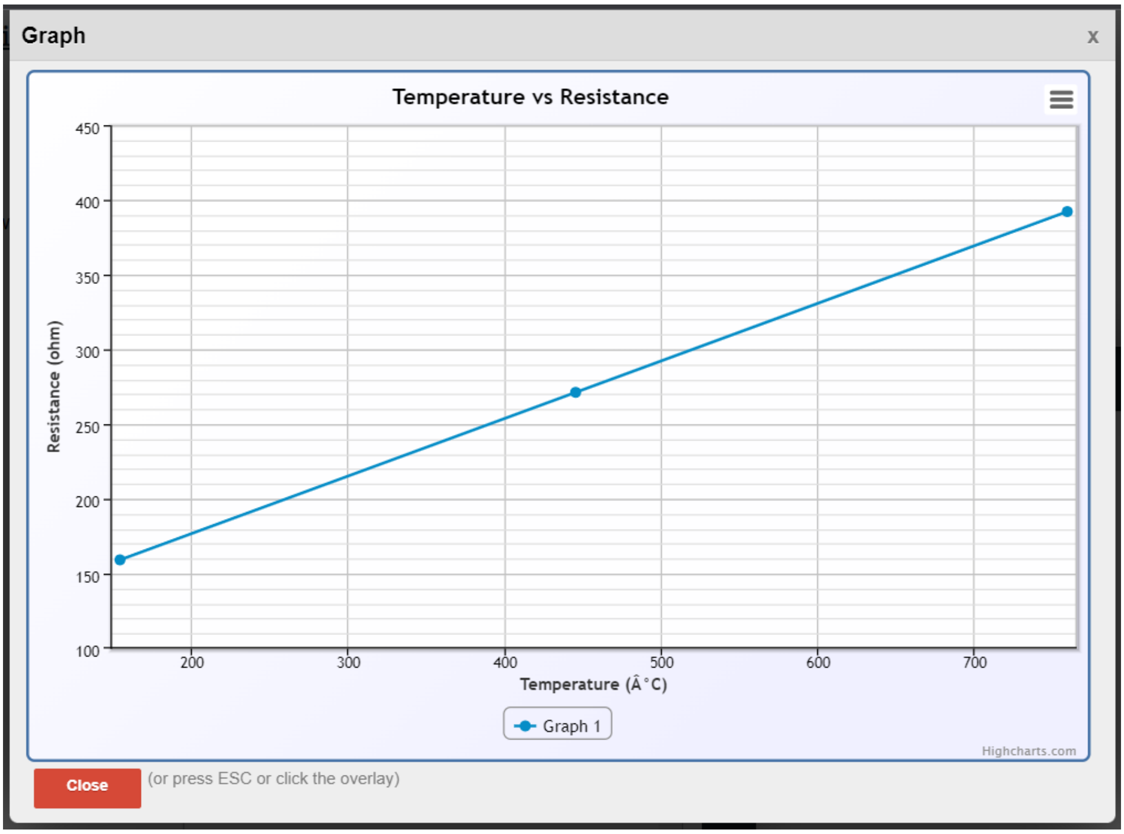
5.Minimum 3 calculations are necessary to plot the graph.

6..Study the graphs for RTD performance with different reference resistance values and different materials.

Simulation:

LEVEL 1: STATIC CHARACTERSTICS





Expt. 2: Dynamic characteristics

Aim : Study the dynamic response of RTD probe

In this experiment, Platinum RTD is used for calculating the time constant values. The response of RTD for a step change in the input is plotted. The maximum temperature the RTD attains, is same as the temperature displayed by 'GET Temp' tab.

Step by step Procedure:

1.Select the RTD as bare element (make it on). With the standard dimensions considered for bare element, the time constant value will get displayed in the output box. Verify this by using equation given under Get formula tab.

2.Now select the RTD dynamic performance with sheath. Select appropriate material and thickness value. The time constant value will get displayed in the output box. Verify this by using equation given under formula tab.

3.Observe the change in the time constant values with different materials with different thickness.

4.Now select the RTD dynamic performance with thermowell. Select appropriate material, thickness and filling material. The time constant value will get displayed in the output box when thermowell material, thickness and filling material is selected. Verify this by using equation given under formula tab.

5.Observe the change in the time constant values with different materials with different thickness. Also see the effect of change in filling material.

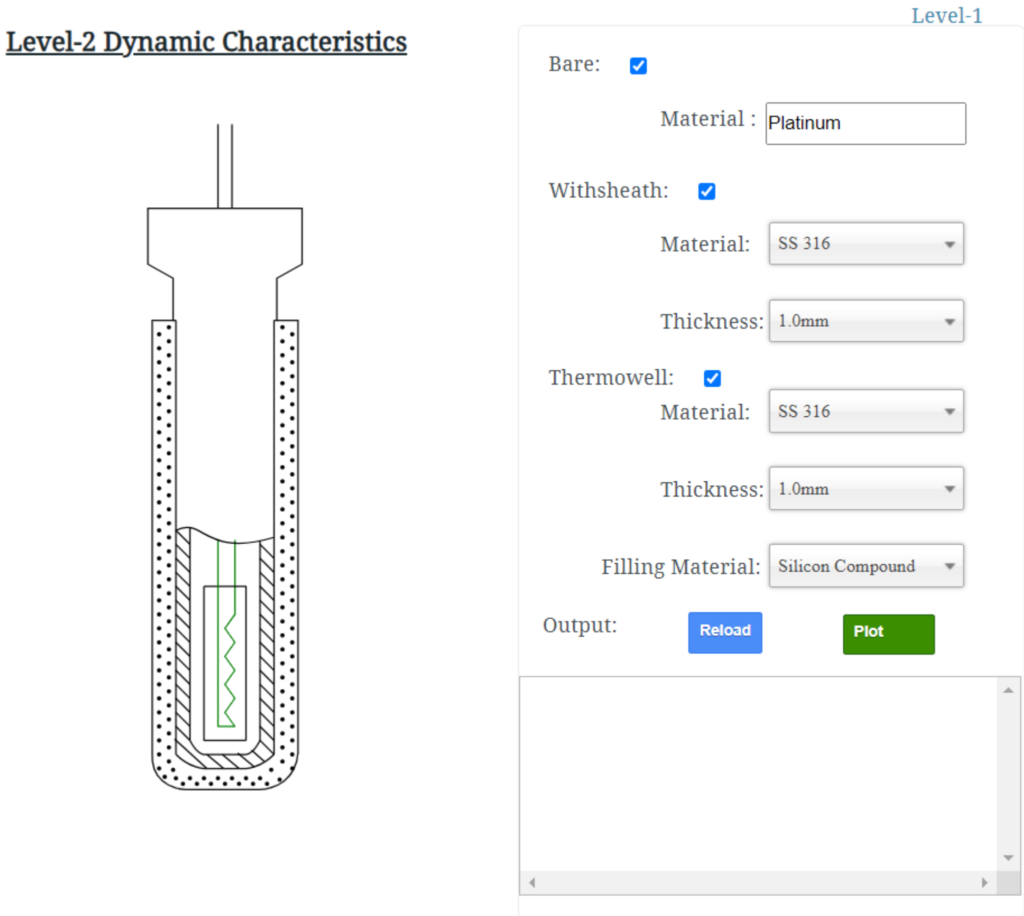
6.Click on plot button to observe the dynamic response (Time Vs Temperature).

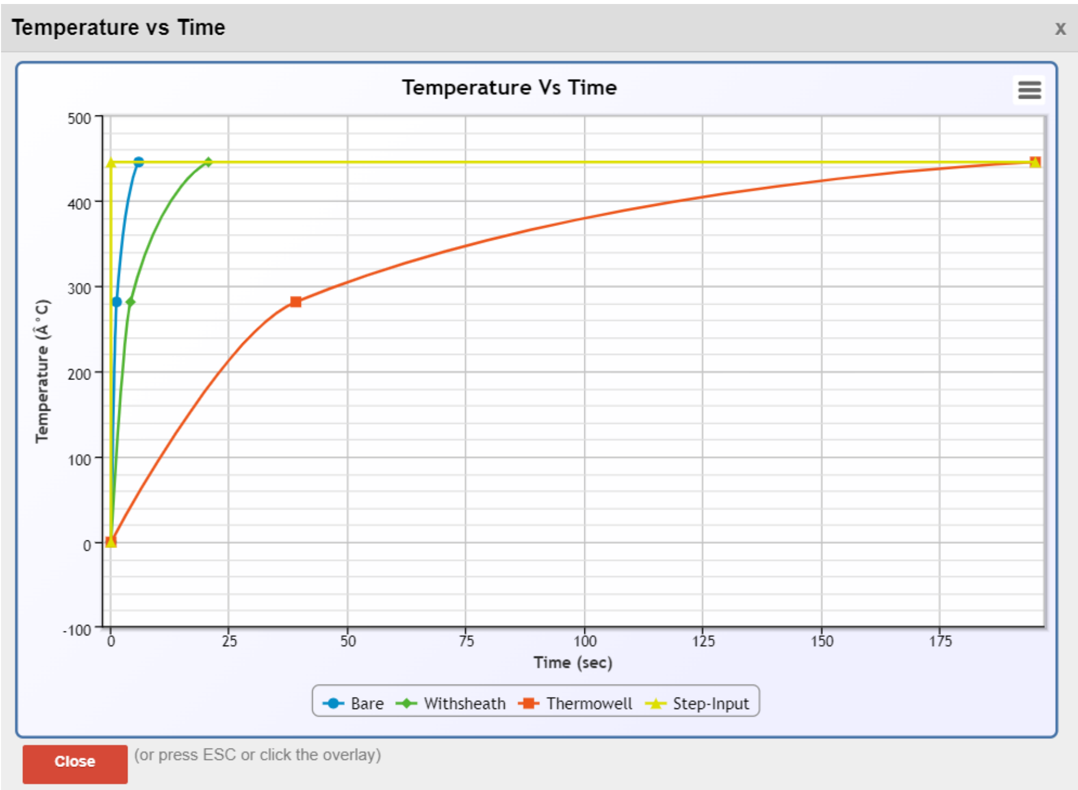
7.Observe and verify response time of RTD which is generally 5 times the time constant value.

8.After completion of both the parts, you can proceed to Post Test to find out if you have understood all aspects of the experiment.

Simulation:

LEVEL 2:DYNAMIC CHARACTERSTICS





CALCULATIONS:

* Static characteristics :-

A] Material : Platinum , $\alpha = 0.00385$, $R_0 = 100$

$\Delta T_1 = 759°C$, $\Delta T_2 = 154°C$

$\Delta T_3 = 445°C$

1) $R_{t_1} = R_0 (1 + \alpha \Delta T)$

$= 100 (1 + (0.00385) \times (759))$

$= 392.215 \ \Omega$

2) $R_{t_2} = R_0 (1 + \alpha \Delta T)$

$= 100 (1 + (0.00385) \times (154))$

$= 159.29$

3) $R_{t_3} = R_0 (1 + \alpha \Delta T)$

$= 100 (1 + (0.00385) \times (445))$

$= 271.325$

* Dynamic Characteristics :-

1) Bare :- Material : Platinum

2) Withsheath :- Material : SS 316

Thickness : 1.0 mm

Time constant : 4.11 s.

3) Thermowell :- Material : SS 316

Thickness : 1.0 mm

Filling Material : Silicon Compound

Time constant : 39.01 s.

Conclusion: The working principle of RTD has been understood.

Experiment 2: Characterize the LVDT

Aim: To understand working principle of LVDT.

Objectives:

1. Study the relation between core displacement and output of LVDT.
2. Understand the effect of change in supply frequency on LVDT performance.
3. Understand the effect of change in excitation (supply) voltage on LVDT performance.

The procedure to use the simulator is given below.

1 First you need to configure the LVDT. Click on ' Show panel' tab at the right bottom For making the circuit, drag and drop the primary coil, Armature and secondary coils at the loactions shown on left hand side.

2 Now select No of Turns, peak to peak supply volatge and frequency from the drag and drop menu, available below LVDT diagram. Click on configure block to configure LVDT.

Now click on the black rectangular core placed between primary and secondary windingDrag the core to left hand side and observe the effect on the output magnitude. This can be observed on the time vs output volatge waveform and on the Distance vs output voltage graph. The core displacement is indicated in the square box below the diagram

3.Drag the core to right hand side and observe the effect on the output magnitude. Also observe the change in the phase.

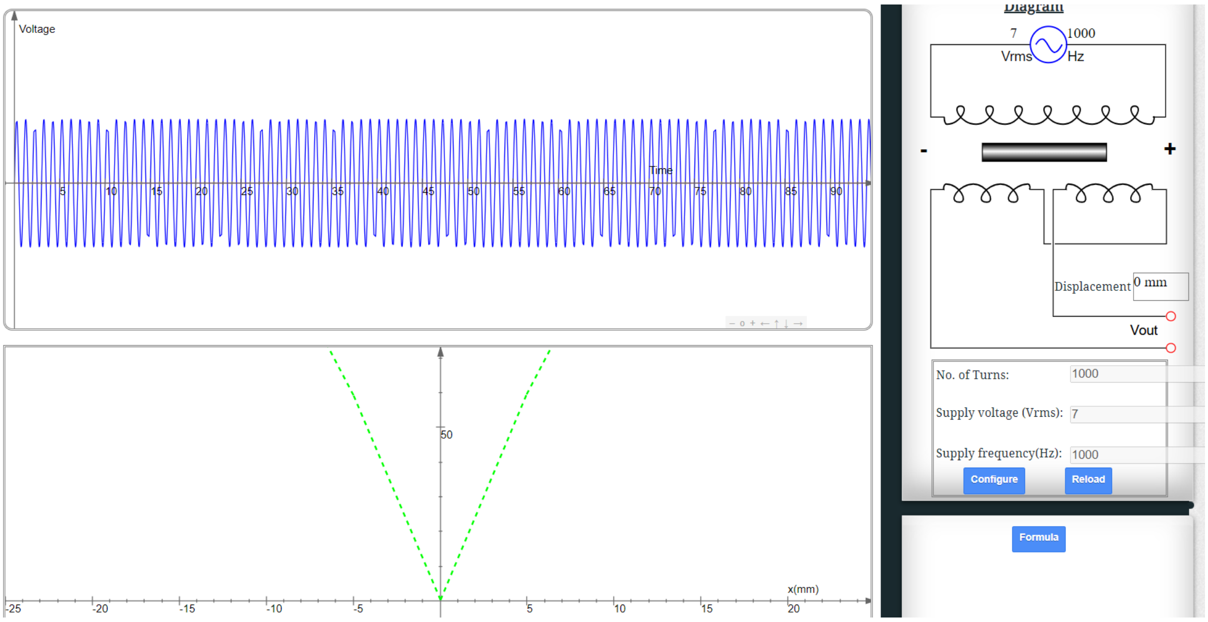
4.Repeat steps 2 to 4 by changing supply volatge keeping frequency and no of turns constant. Study the effect on the output voltage. For this click on blue color 'Configure' tab in the right side panel. You need to select required parameter value from drop down menu. After selecting the values click on green ' Configure' tab to set the parameter values.

5,Repeat steps 2 to 4 by changing supply frequency keeping and no of turns constant. Study the effect on the output voltage.

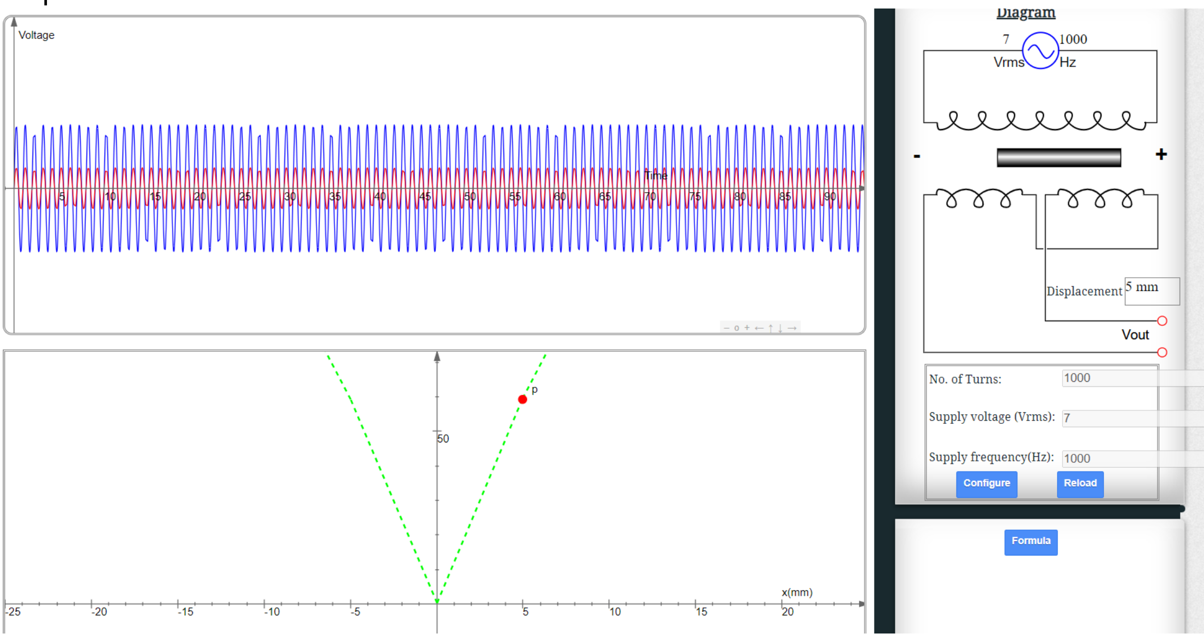
6.Now keep supply voltage and frequency constant. Change the no of turns and observe the effect on the output voltage by repeating steps 2 to 4.

Simulation Graph:

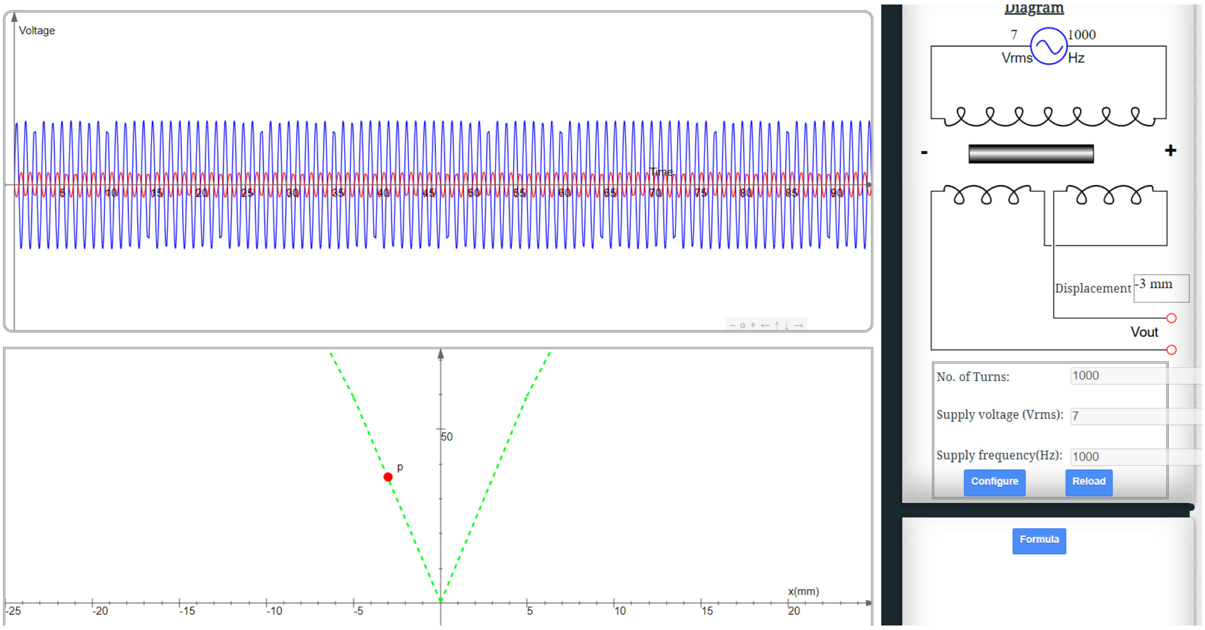
1. No of tuns: 1000
   Vrms : 7
   Supply Frequency : 1000
   Displacement : 0



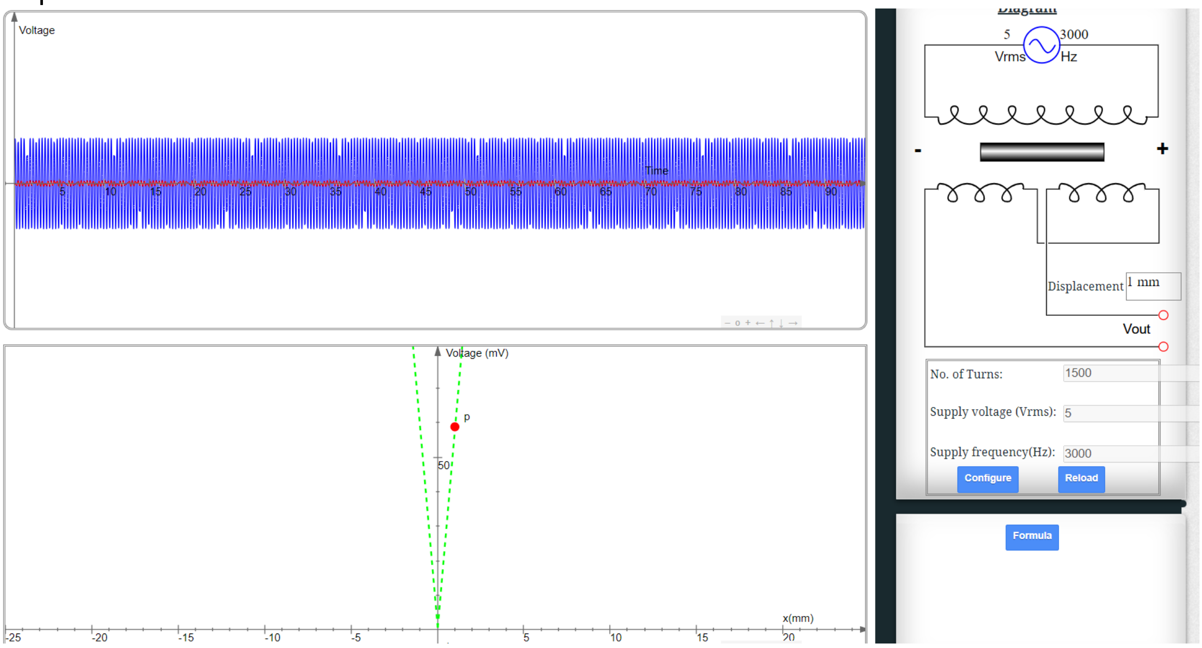
2. No of tuns: 1000
   Vrms : 7
   Supply Frequency : 1000
   Displacement : 5



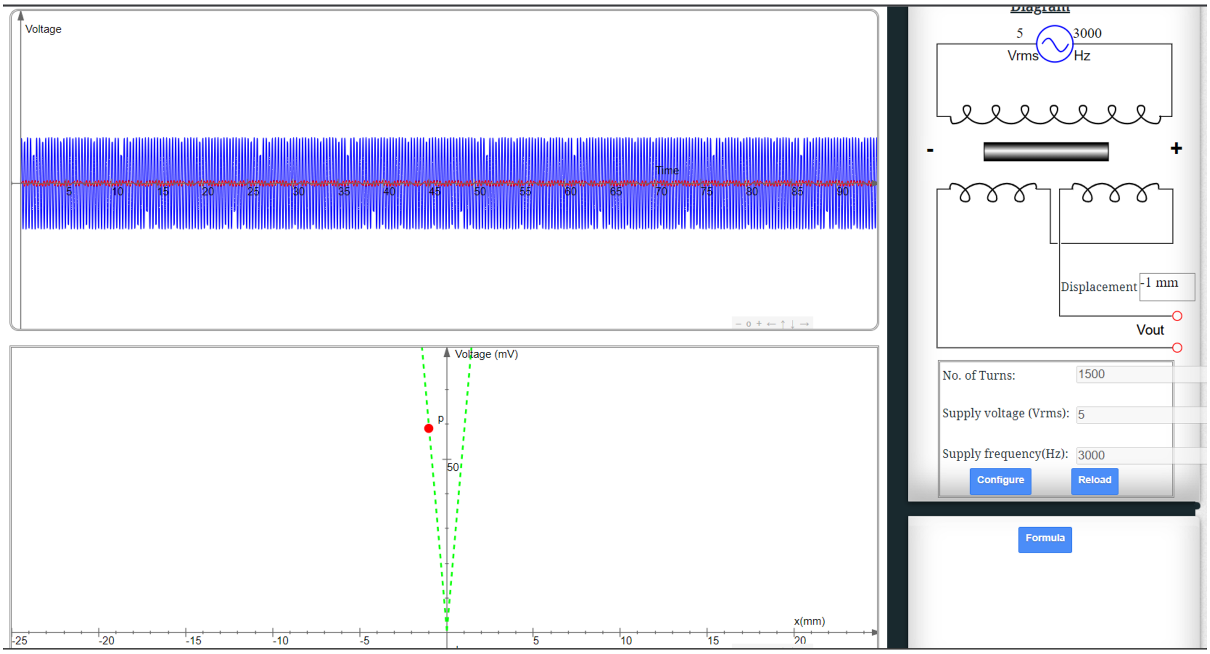
3. No of tuns: 1000
   Vrms : 7
   Supply Frequency : 1000
   Displacement : -3



4. No of tuns: 1500
   Vrms   :   5
   Supply Frequency : 3000
   Displacement : 1



5. No of tuns: 1500
   Vrms   :   5
   Supply Frequency : 3000
   Displacement : -1



Conclusion:
Therefore, we have simulated a LVDT and observed the effect of different parameters and their differing values.

Experiment 3: Simulate the performance of a chemical sensor

Aim: To understand the working principle of chemical sensors

Objectives :

1. Study the working principle of pH and conductivity.
2. Calibrate the pH sensor.
3. Study the effect of temperature on pH measurement.
4. Study effect of temperature and effect of contamination on conductivity measurement.

Level 1: Conductivity Measurement

Aim : To study the measurement of the conductivity of the sample.

Step by step Procedure:

1.Study the schematic diagram.

2.Select the sample from drop down menu.

3.Select the concentration of the sample.

4.Cell constant will be fixed which is displayed as 1 on screen.

5.After entering data the PPM Concentration, Equivalent weight and Density at 25°C is displayed in the right side panel.

6.Enter the calculated specific conductance. For calculations of specific conductance click on Formula tab.

7.Using the formula, calculate the value of the conductivity for the selected sample and enter the answer in the box provided (upto 2 decimals with rounding off). Submit the answer using submit button.

8.If your calculation is correct it will be displayed on the screen. Otherwise you need to recalculate.

9.Repeat the procedure for different concentrations of a same sample.

10.Minimum three calculations are necessary to plot the graph and after three calculations the plot tab will be activated.

Level 2: Effect of Temperature.

Aim :Study the effect of temperature change on the measurement of conductivity.

In this experiment, by using the formula one can change the value of the temperature and calculate the conductivity.

Step by step Procedure:

1.Study the schematic diagram.

2.The sample selected is by default the user has selected for level 1.

3.Select the concentration of the substance. Selected concentration will be displayed on the screen.

4.Click on Get Temp. The temperature value and its temperature coefficient will be displayed on the right hand side on the screen.

5.Enter the calculated conductivity at a temperature of interest. For calculations of conductivity click on FORMULA tab.

6.Using formula, calculate the value of the conductivity for the selected sample and enter the answer in the box provided (upto 2 decimals with rounding off). Submit the answer using submit button.

7.If your calculation is correct, the output conductivity at the selected temperature will be displayed on the screen.

8.Repeat the procedure for different temperatures of the same substance and concentration.

9.Minimum three calculations are necessary to plot the graph and after three calculations the plot tab will be activated.

10.After study hide the graph and repeat the experiment with varying substance and concentration.

Level 3: Effect of Contamination

Aim : Study the effect of contamination on measurement of pH.

1.In this experiment, one can change the value of the contamination and obser its effect on the conductivity of the substance.

Step by step Procedure:

1.Study the diagram completely.

2.Select the substance .Selected substance will be displayed on the screen.

3.Select the concentration of the substance. Selected concentration will be displayed on the screen.

4.Cell constant will be fixed which is displayed as 1 on screen.

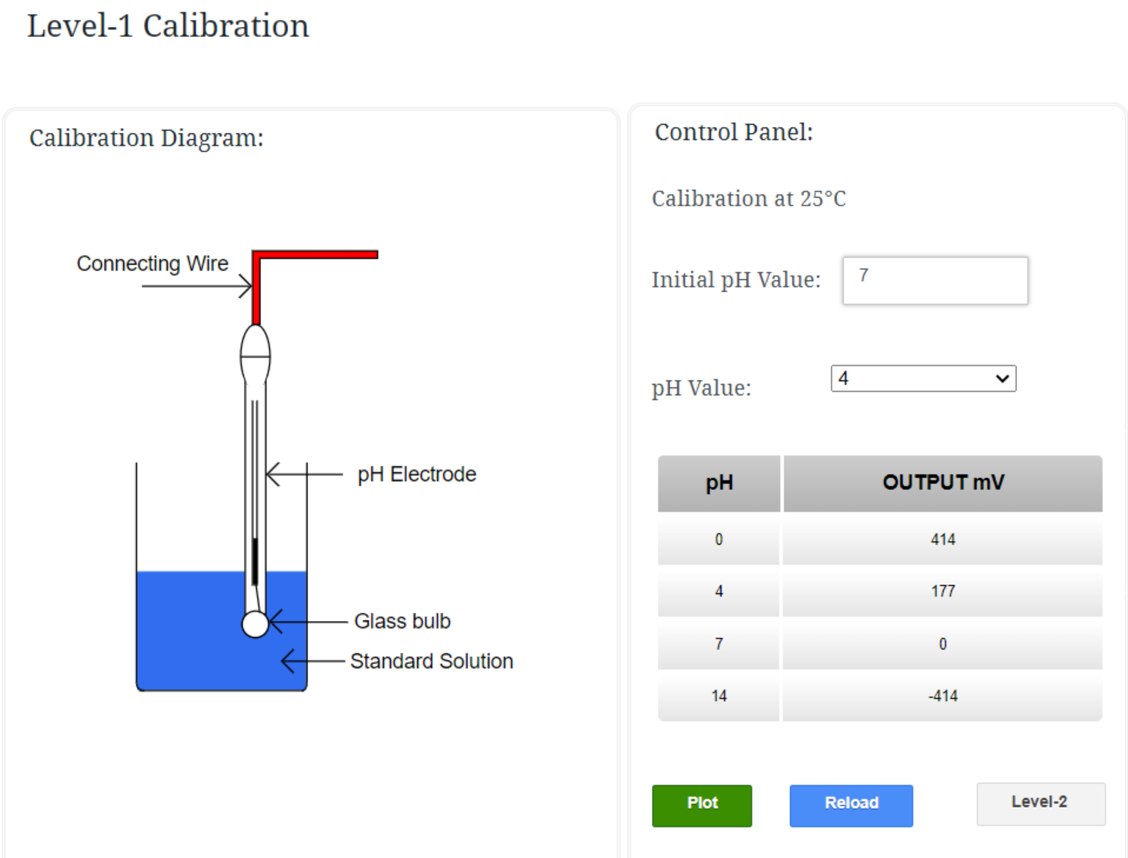
5.Select the contamination in mm.

After selecting the contamination the Specific conductance, Modified Cell Constant ,Modified specific conductance value is displayed on the screen.

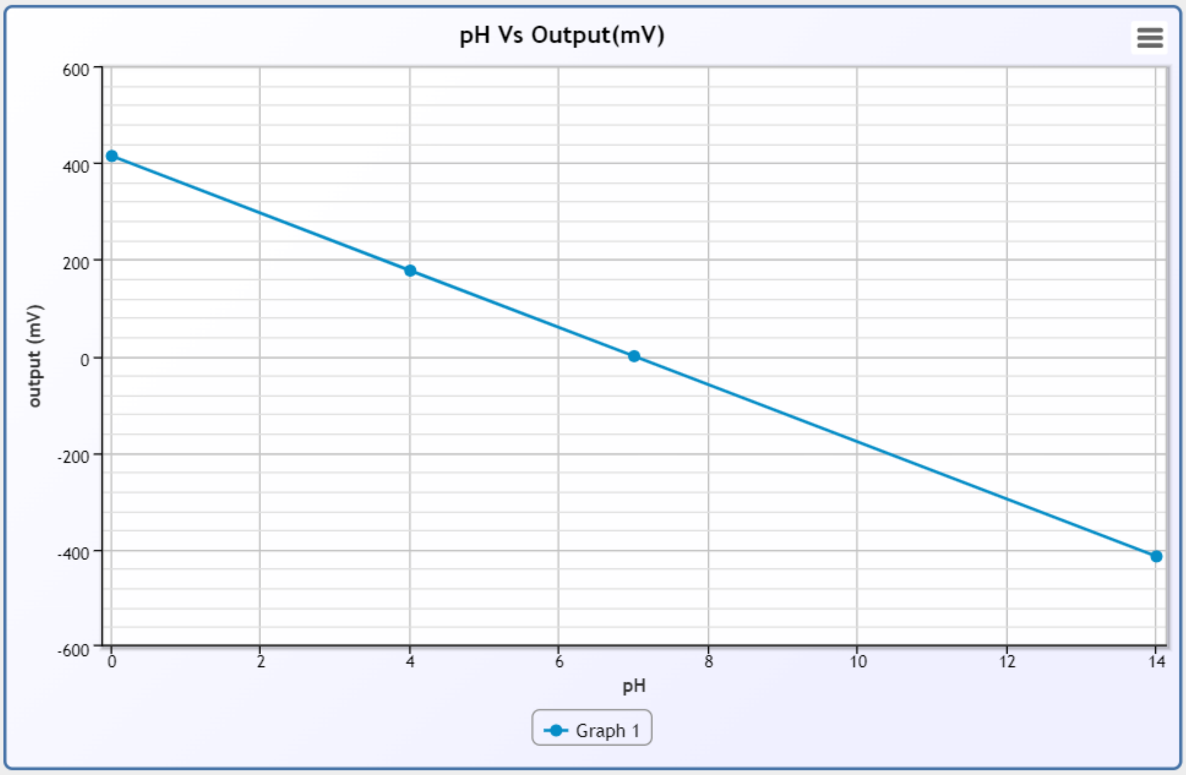
6.By comparing the Specific conductance, Modified specific conductance one can see the effect of contamination on substance conductivity.

7.Change the substance, concentration and observe the result.

pH CALIBRATION SIMULATOR:-



pH Vs Output(mV) Graph

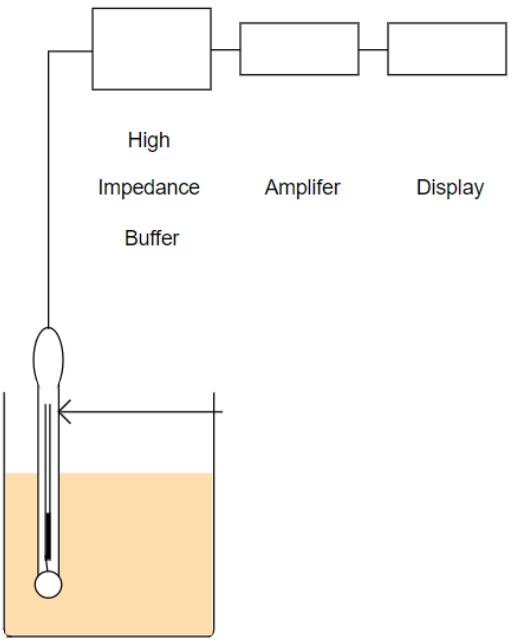


Close (or press ESC or click the overlay)

Level-2 Measurement

<--Level-1

Measurement Diagram:



Control Panel:

Temperature at 25°C

Sample: Baking Soda

Enter Output Voltage: (in mV)

Submit

Lemon Juice	2.2	283.97
Boric Acid	5	118.32
Baking Soda	9	-118.32

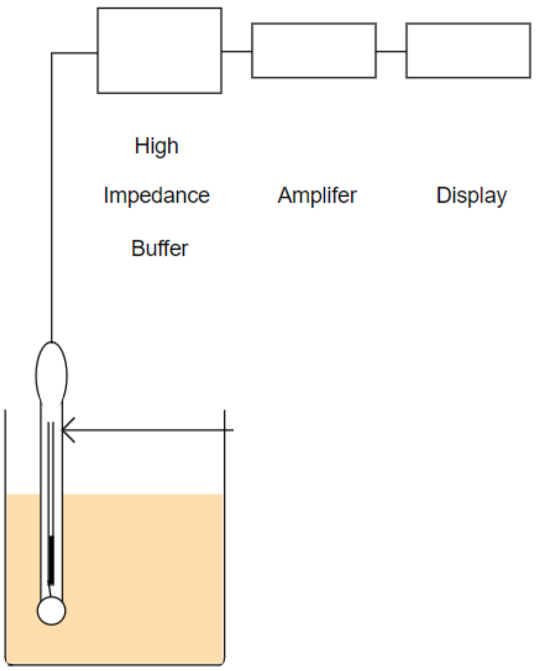
Reload

Level-3

Level-3 Temperature

<--Level-2

Temperature Diagram:



Control Panel:

Sample: Baking Soda

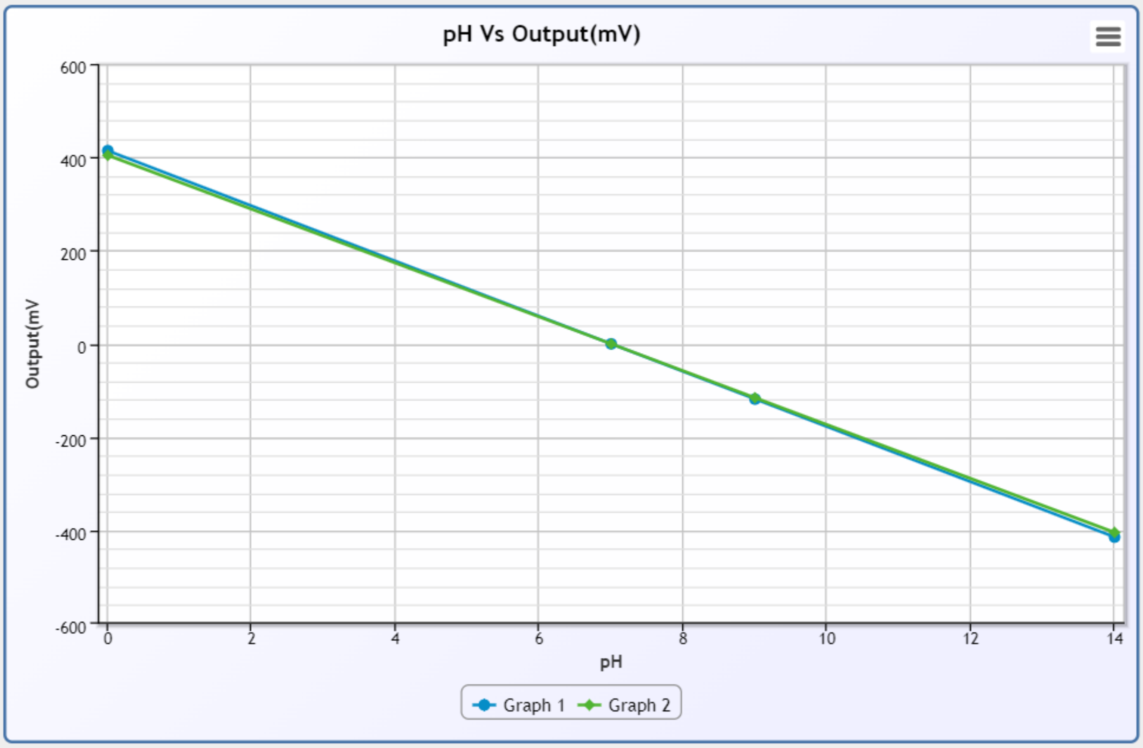
At 25°C Output mV =-118.32

Get Temperature ==> 18°C

Temperature °C	pH	OUTPUT mV
18	9	-115.54

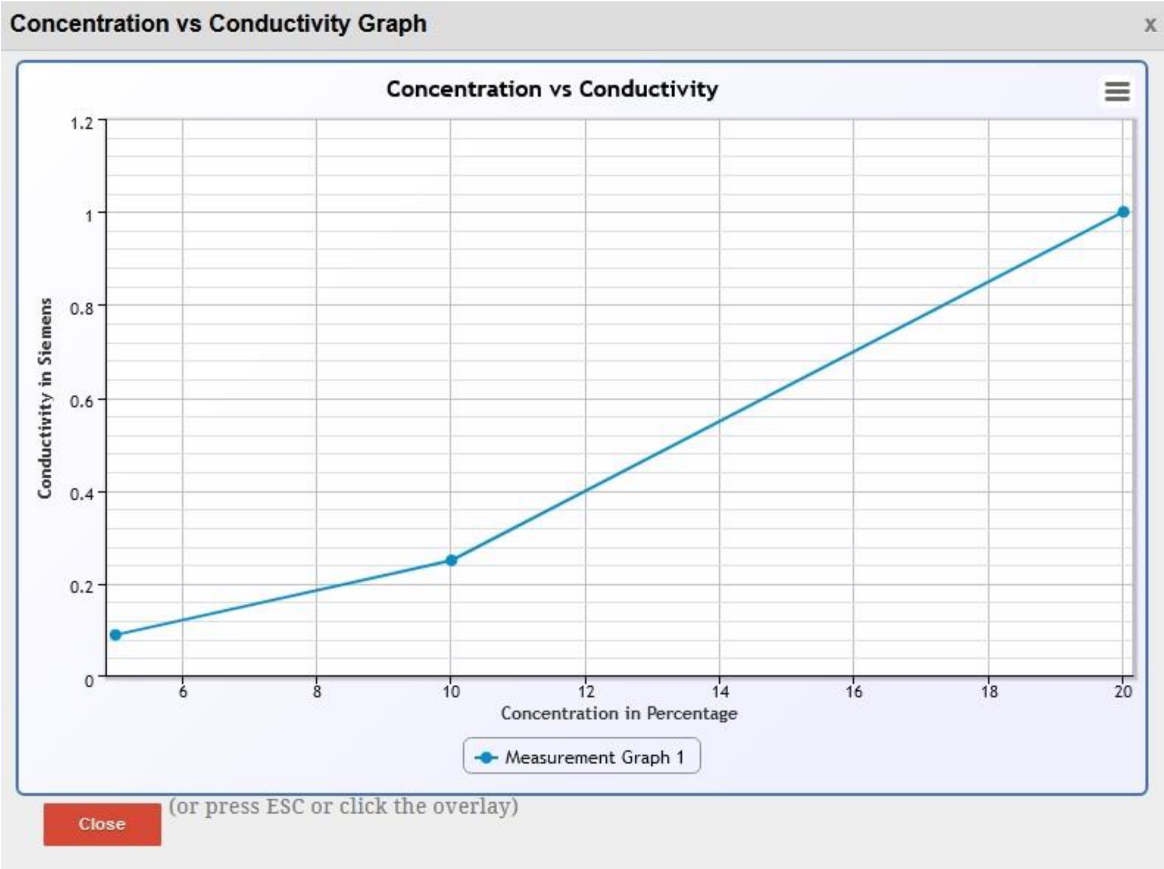
Plot Reload

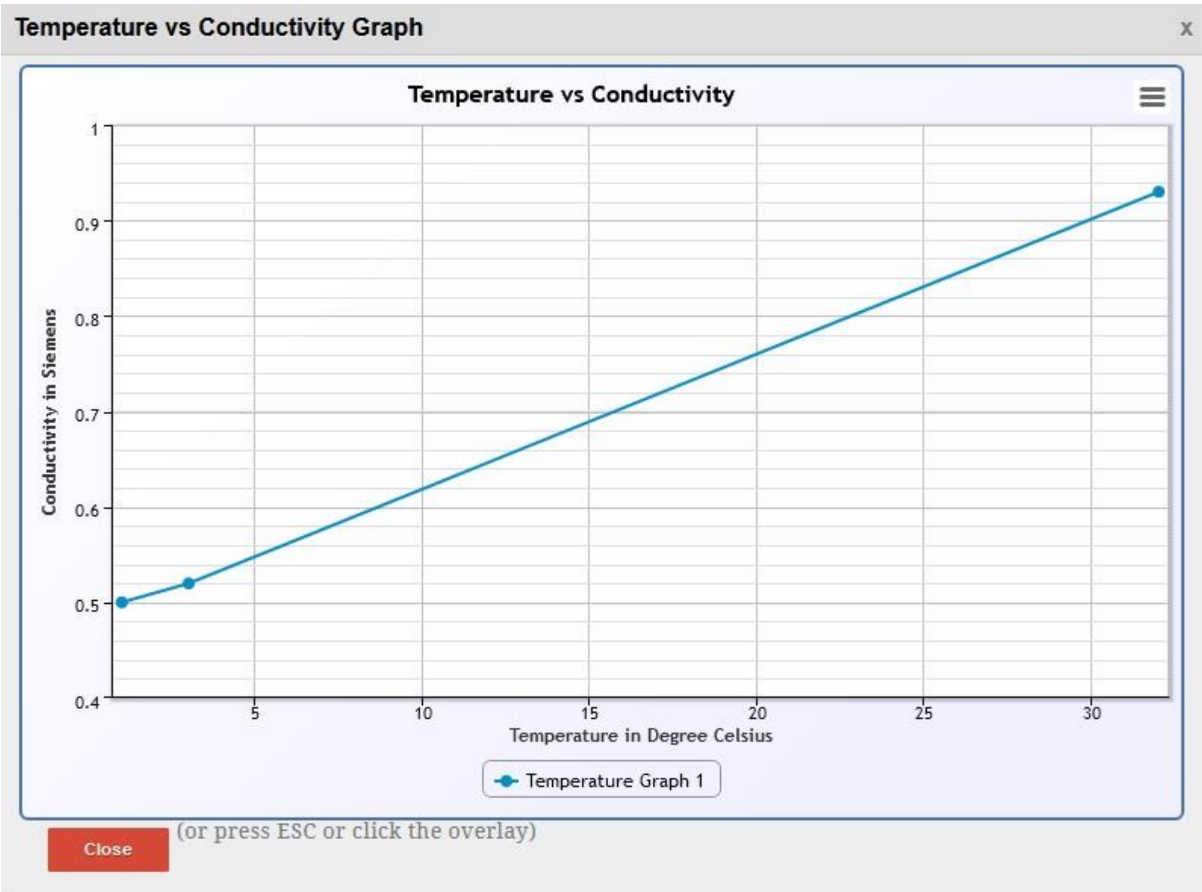
pH Vs Output(mV) Graph x

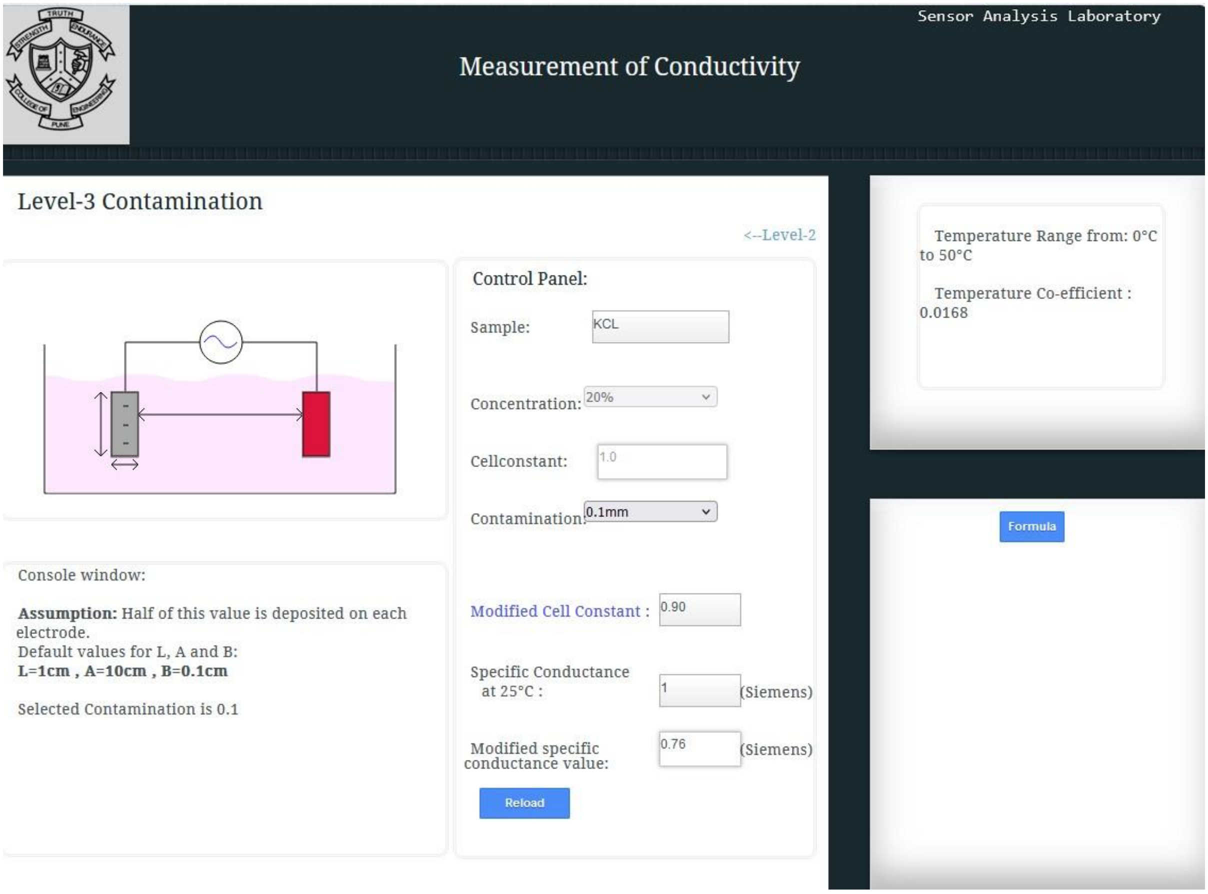


Close (or press ESC or click the overlay)

CONDUCTIVITY SIMULATOR:-







Calculations:

* Calculations of Exp. 3 :-

Formula : $E = \left(E_0 - 2.3026\left(\frac{RT}{F}\right) pH_c\right) \times 1000$

Given : $E_0 = 0$, $R = 8.3144\ J/K$,
$T = 25°C = 298.15\ K$
$F = 269485\ C/mole$

Sol^n :-

1) For baking soda, $pH_c = 9 - 7 = 2$
∴ $E = \left(0 - 2.3026\left(\frac{8.3144 \times 298.15 \times 2}{269485}\right)\right) \times 1000$
$E = -118.32\ mV$

2) For boric acid, $pH_c = 5 - 7 = -2$
∴ $E = 118.32\ mV$

3) For Lemon juice, $pH_c = 2.2 - 7 = -4.8$
∴ $E = 283.97$

Conclusion:
Therefore, we have simulated chemical sensors. We have calibrated the pH sensor and compared the effect of temperature difference on the same sample.

Experiment 4:. Measurement of level in tank using capacitive type level probe

Aim: To measure tank level using capacitance level probe.

Objectives:

1. Review various methods of level measurement.
2. Understand working of capacitance level transmitter.

Procedure:

1.Study the given diagram completely.

2.Select the height of the tank in centimetres.

3.The value of radius of outer cylinder/pipe for pipe in pipe type probe r2 is fixed 2.5cm.

4.Select the value of radius of inner cylinder/pipe for pipe in pipe type probe r1 in centimetres.

5.Span value will get displayed. Span is 90% of the height of the tank.

6.Select the service for which capacitance probe is to be used.

7.Click on 'Configure System'. It will ask for confirmation.

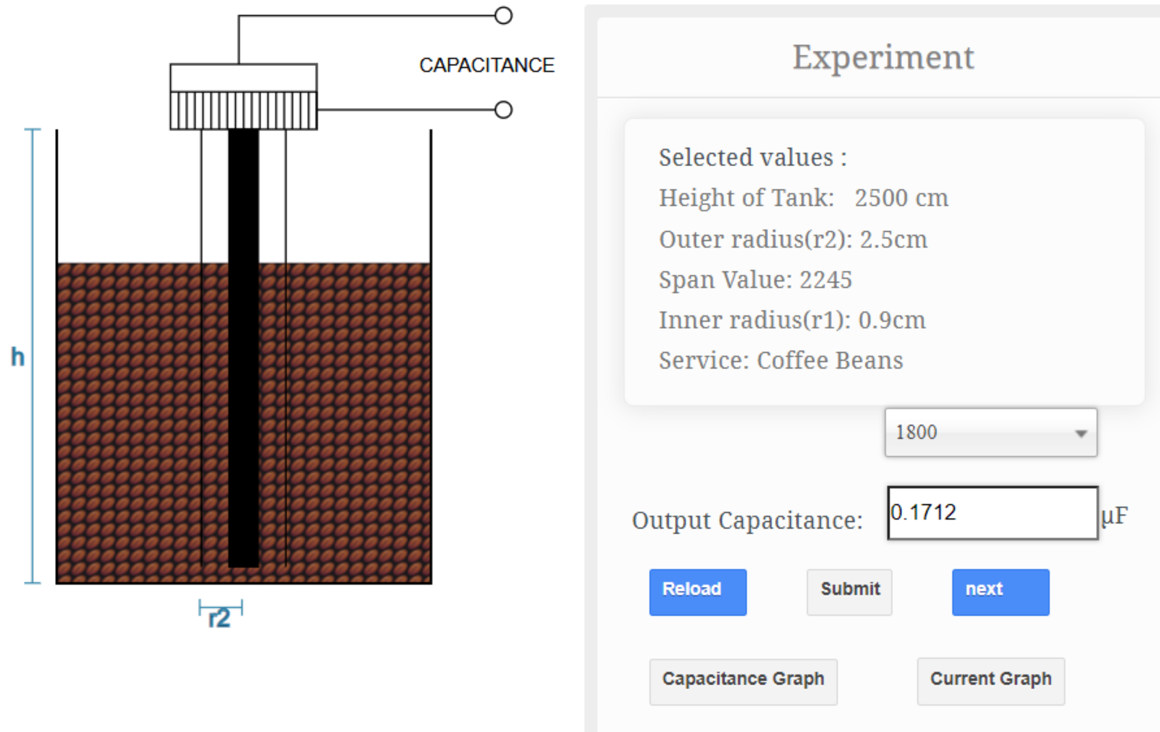
8.After confirming, select the fluid level from drop down menu for selected service.

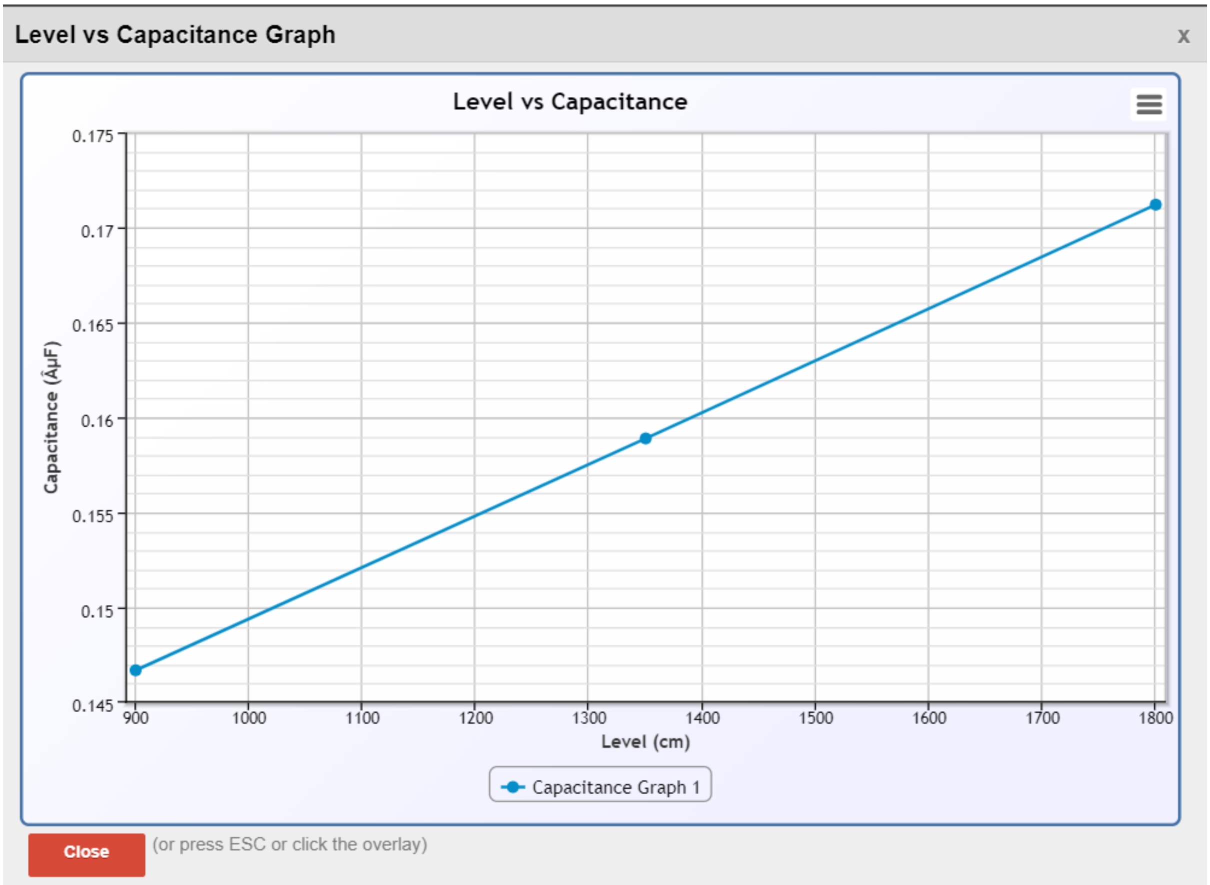
9.Enter the calculated user output capacitance in μF. For calculations of output capacitance click on GET FORMULA tab. Using formula calculate the value of the output capacitance for the corresponding level and enter the answer in the box provided (upto 2 decimals with rounding off). Submit the answer using submit button.

10.If your calculation is correct it will be displayed on the screen. Minimum three calculations are necessary to plot the graph. Click on level vs. capacitance graph and level vs. current graph and study them.

11. Click on next set of values. Change the value of r1and repeat steps 7 to 10 Click on graphs and study them .

SIMULATION:





CALCULATION:

* Calculations of Exp. 4 :-

Given: Tank height (h) = 2500 cm
Outer radius (r_2) = 2.5 cm
Inner radius (r_1) = 0.9 cm
Span value = 2245
Service : Coffee Beans

1) Coffee Beans level height, h_2 = 1350 cm. $\Rightarrow h_1 = 895$ cm

$$\therefore C = \frac{2\pi\varepsilon_0(\varepsilon_1 h_1 + \varepsilon_2 h_2)}{\ln\left(\frac{r_2}{r_1}\right)}$$

$$= \frac{2\pi\varepsilon_0(8.45 + 1.5(13.5))}{\ln\left(\frac{2.5}{0.9}\right)}$$

$$= 0.1589\ \mu F$$

2) $h_2 = 900$ cm $\Rightarrow h_1 = 1345$ cm

$$\therefore C = \frac{2\pi\varepsilon_0(\varepsilon_1 h_1 + \varepsilon_2 h_2)}{\ln\left(\frac{2.5}{0.9}\right)}$$

$$= \frac{2\pi\varepsilon_0(1345 + 1.5 \times 900)}{\ln(2.5/0.9)}$$

$$= 0.147\ \mu F.$$

3) $h_2 = 1800$ cm $\Rightarrow h_1 = 445$ cm

$$\therefore C = \frac{2\pi\varepsilon_0(445 + 1.5 \times 1800)}{\ln(2.5/0.9)}$$

$$= 0.1712\ \mu F$$

Conclusion:
Therefore, we simulated capacitive level sensor and compared the capacitance of the sensor
against different levels of liquid.

Programmable Logic Controller

Experiment 5: Study hardware and software used in PLC.

Aim To study hardware and software associated with PLC

Objectives 1.Learn the basics and hardware components of PLC

2.Understand configuration of PLC system

3. Study various building blocks of PLC

Procedure: Hardware and Software Study Procedure (Short Version):

Hardware: Gather Resources: Obtain a PLC system or simulator and access relevant documentation.

1.Identify Components: Learn about key hardware components, including the CPU, input/output modules, power supply, and communication interfaces.

2.Wiring and Connections: Understand how to wire input/output devices and connect Install the PLC to power.

Software:

3.Select Language: Choose a programming language (e.g., ladder logic) for the PLC.

Software: Install and configure PLC programming software on your computer.

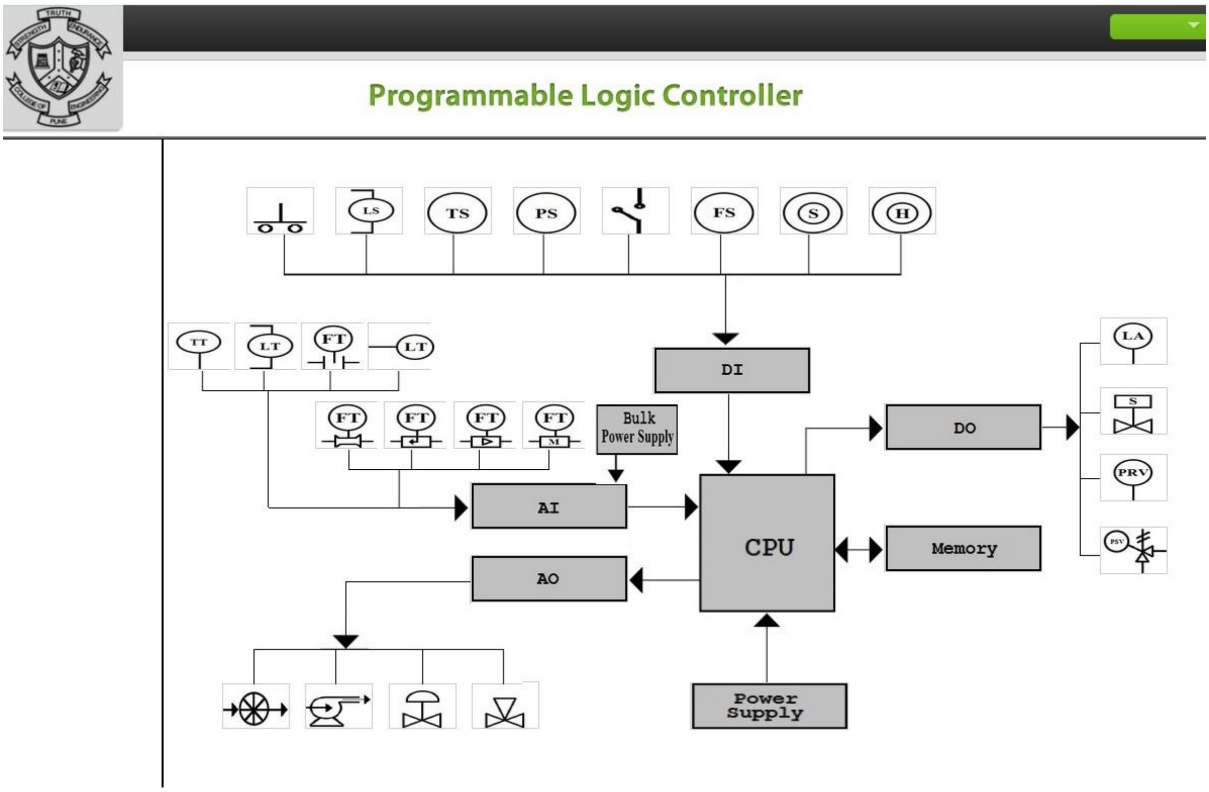
Create Program: Develop a simple control program, download it to the PLC, and run it.

4.Monitor and Debug: Use the software's monitoring and debugging tools to test and troubleshoot the program.

Explore Advanced Features: Learn about data logging, HMI integration, and security settings.

5.Documentation and Practice:Consult Manuals: Refer to PLC hardware and software manuals for specific details.Use Online Resources: Access online tutorials and seek guidance from experienced professionals.Practice: Apply your knowledge through projects and simulations.Following this procedure will help you gain a solid understanding of PLC hardware and software.

SIMULATION:



Conclusion:
Thus we have studied PLC hardware and software used in implementing industrial automation
applications.

EXPERIMENT NO 6: IMPLEMENTATION OF LOGIC GATES

Aim: -

To understand Simple Ladder program.

Objectives: -

1.Develop a ladder using standard procedure.

2.Solve the problem using ladder programming.

Procedure for Creating and Testing Ladder Logic Diagrams: -

1. Preparatory Steps:

a. Understand the problem statement, e.g., testing logic for OR gates, AND gates, etc.

b. Develop and validate the logic on paper by considering various cases.

c. Prepare a truth table and test the logic with all valid cases.

d. Open the PLC simulator by clicking the "Simulator" button.

2. Creating the Ladder Diagram:

a. In the simulator workspace, you'll see the first window.

b. Place contacts and coils as needed by selecting them from the left panel.

c. Double-click on contacts and coils to assign tag names like "start," "stop," "motor," "lamp," etc. Ensure that tag names match process connections.

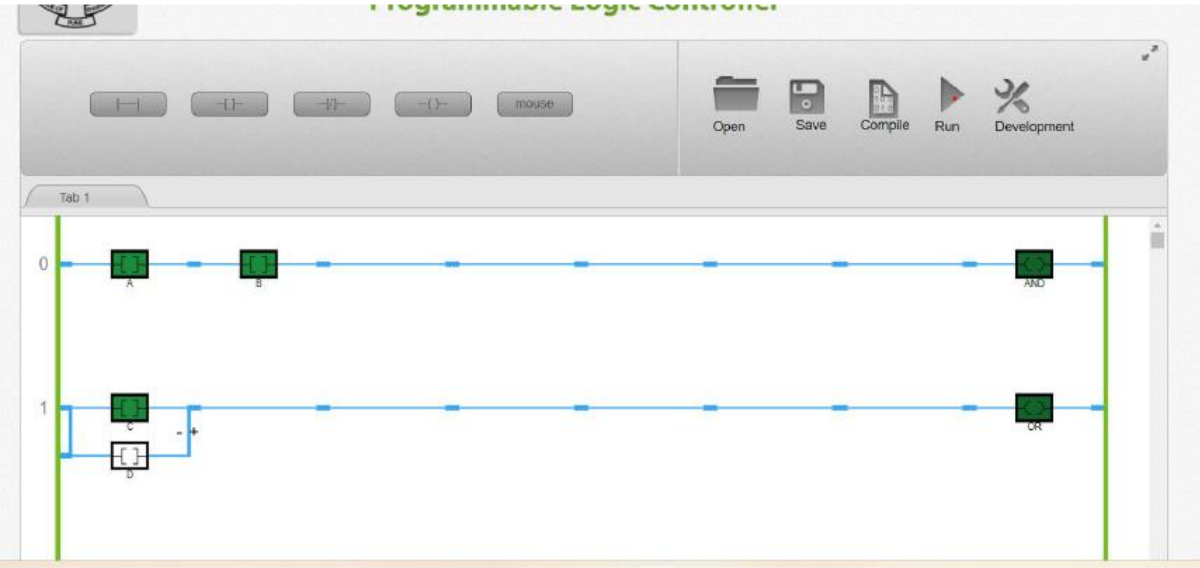
d. Note that tag names are case-sensitive, so use the correct case for bit-level usage.

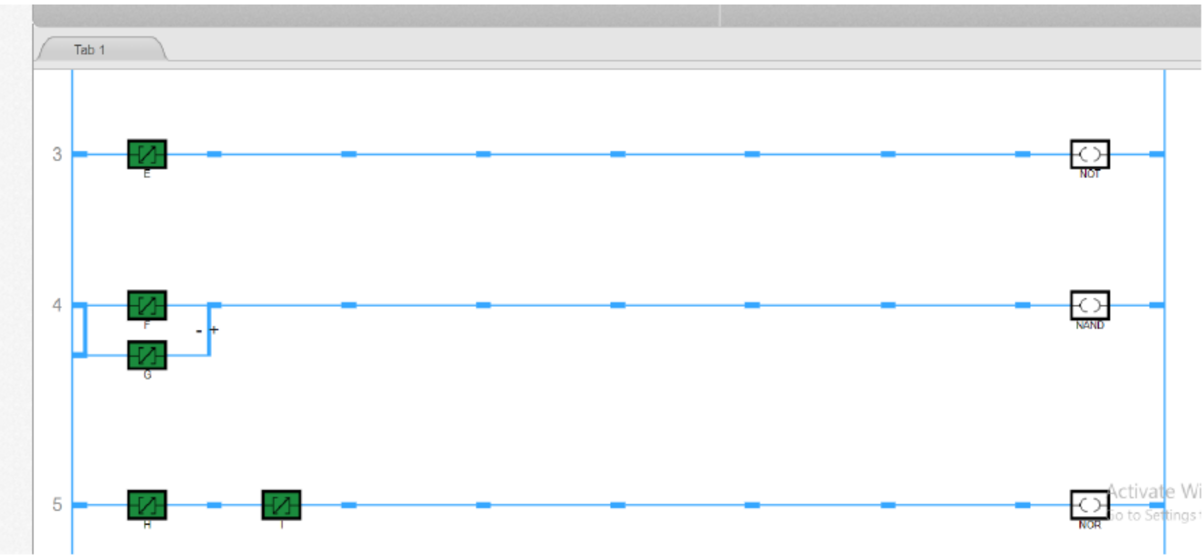
e. Click the "Run" button on the left side to enter the run mode. Both sides of the rung will turn green, indicating it's ready for testing. In run mode, you can't make changes; switch to design/program mode for modifications.

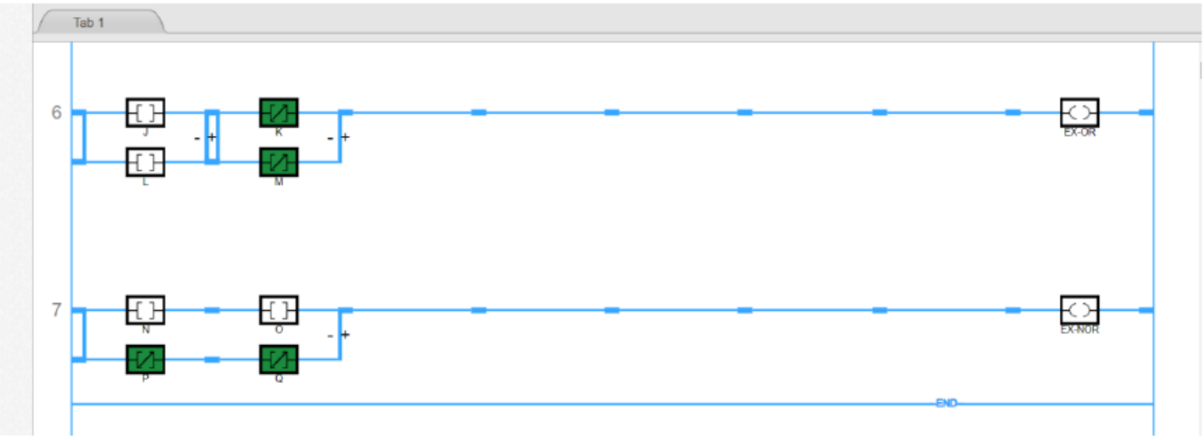
f. Double-click on contacts to toggle their state. This will change the output contact's status.

g. Keep in mind that ladder contacts are initially in a de-energized state, meaning they are not active.

Simulation:







Conclusion:
Thus, we have implemented and verified logic gates as per their individual truth tables.

EXPERIMENT : 7 .Develop an application using On-Delay timer

Aim
To develop an application using On-Delay timer.

Objectives

1. Study the timing diagram of On Delay Timer
2. Solve the assignment of Ton timer

Procedure: In this experiment, an on-delay timer is being tested using a simulator. The following timer bits are observed:

1. "q" bit: The initializing bit.
2. "EN" bit: The enable bit.
3. "DN" bit: The done bit.
4. "TT" bit: The timer timing bit.

1.The user needs to enter a preset value for the timer. To reset the timer counter, a "Reset" command is used with a tag, which should match the counter's tag, like "CU1."

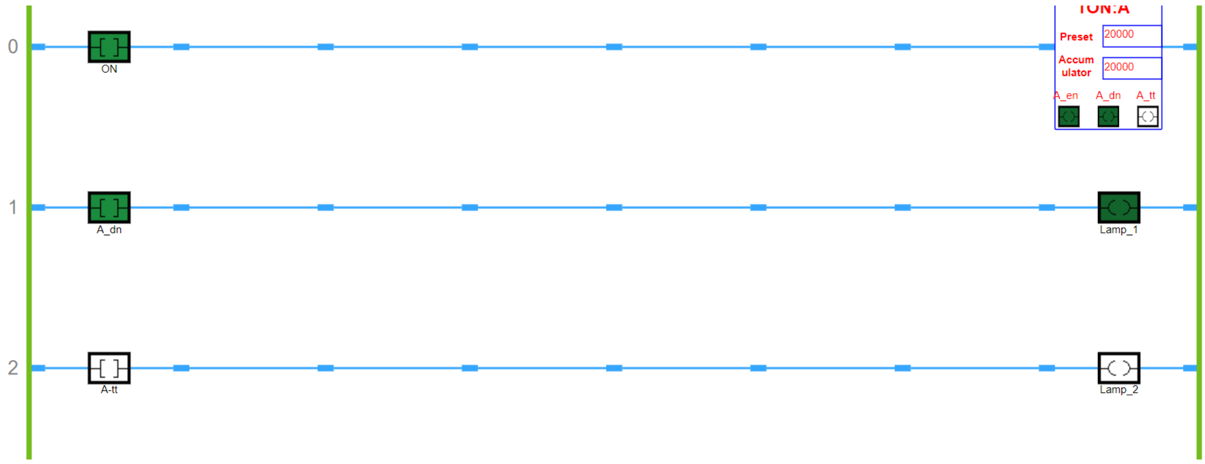
2.The timer can be configured with a time base of either 1 millisecond (1 mS) or 10 milliseconds (10 mS) based on the application's needs.

3.To test the "EN," "DN," and "TT" bits, you can double-click on them and assign a tag name. The same tag name is used in a new rung to check the status or activate the timer's output. These bits can also be used for cascading timers.

4.In run mode, you can observe the bit status when input "a" is toggled. When the delay is over, you can check the output bit status.

5.To reset the "DN" counter, you use the "Reset" command with a tag matching the counter's tag, like "CD1." This allows the counter to be configured for new counts without reloading the page.

Simulation:



Conclusion:

An On-Delay timer application offers precise control over the timing of actions, enhancing efficiency and safety in various domains, including manufacturing, automation, and energy management. It provides a valuable tool for automating processes and tasks, ultimately improving productivity and resource management.

EXPERIMENT 8: Develop an application using Off-Delay timer

Aim:
To Develop an application using OFF Delay Timer

Objectives:

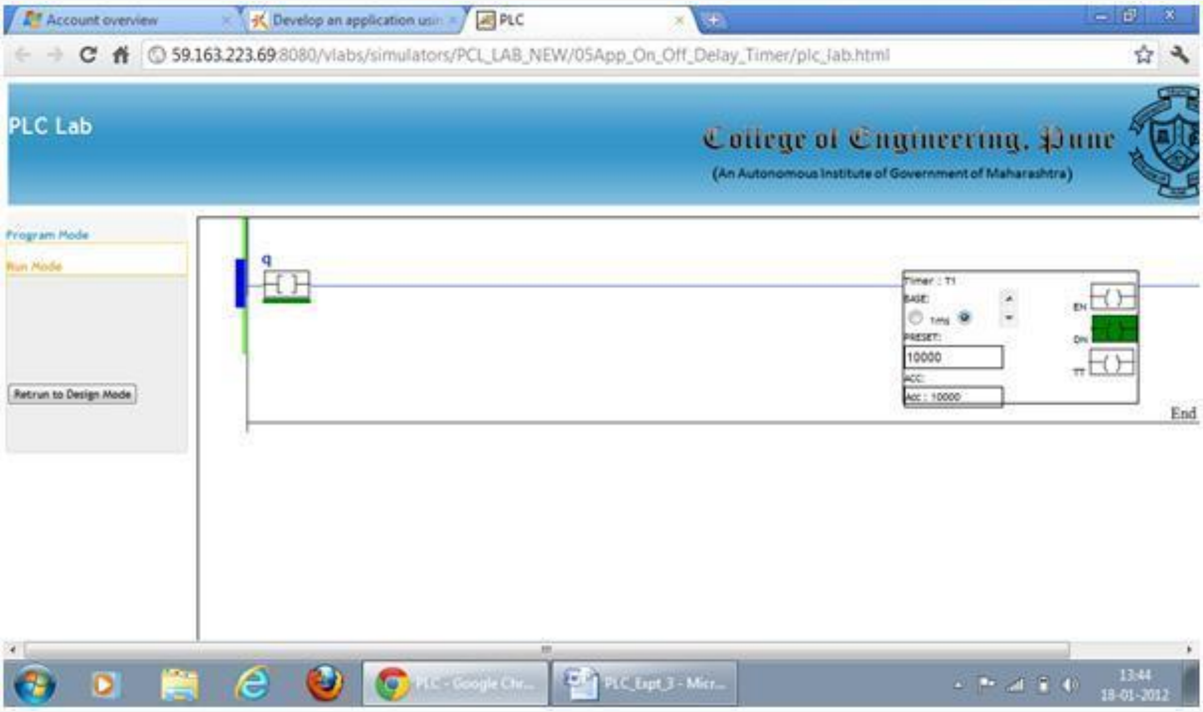
1. Study the timing diagram of OFF Delay Timer
2. Solve the assignment of Toff timer

PROCEDURE:

The configuration of off delay timer is same as 'on delay timer'.

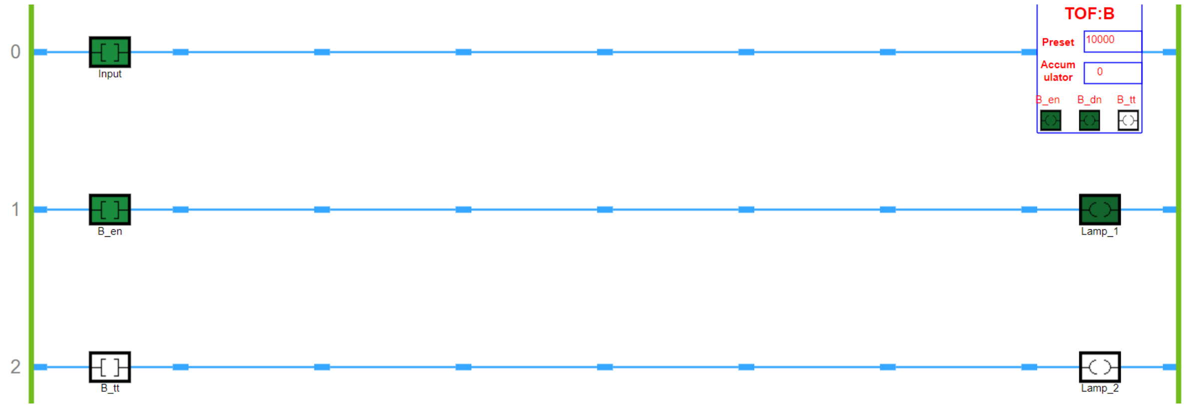
A typical difference can be observed in the operation (in Run mode) .

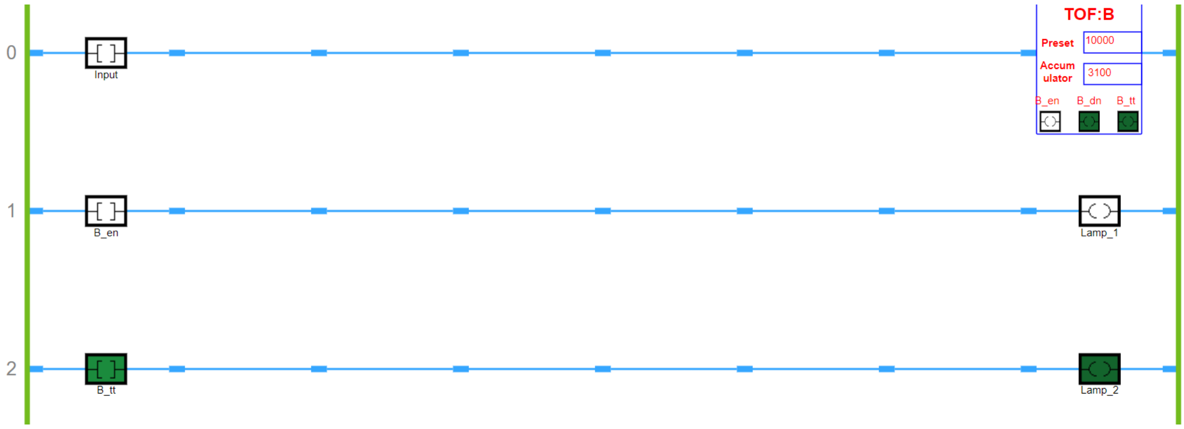
When the q bit is energised the output DN bit goes high. The timer starts only after toggling the initialisation bit again.

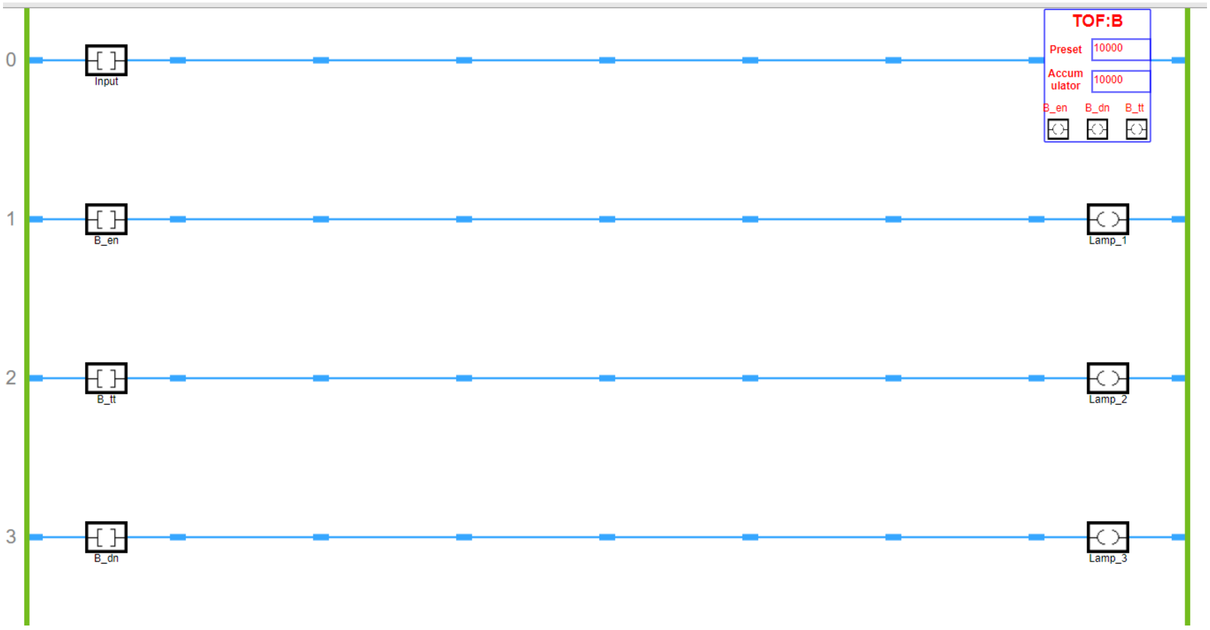


To test the EN, DN, and TT bits; double click on the bit and give tag name to the bit. The same tag name is to be used in the new rung to test the status or to energies the output. You can also test the cascading of the timer using these bits.

Simulation:







Conclusion:

The application has been developed using Off-Delay Timer.

Experiment 9: Develop an application using up/down counter

Aim

To Develop an application using UP/DOWN counter

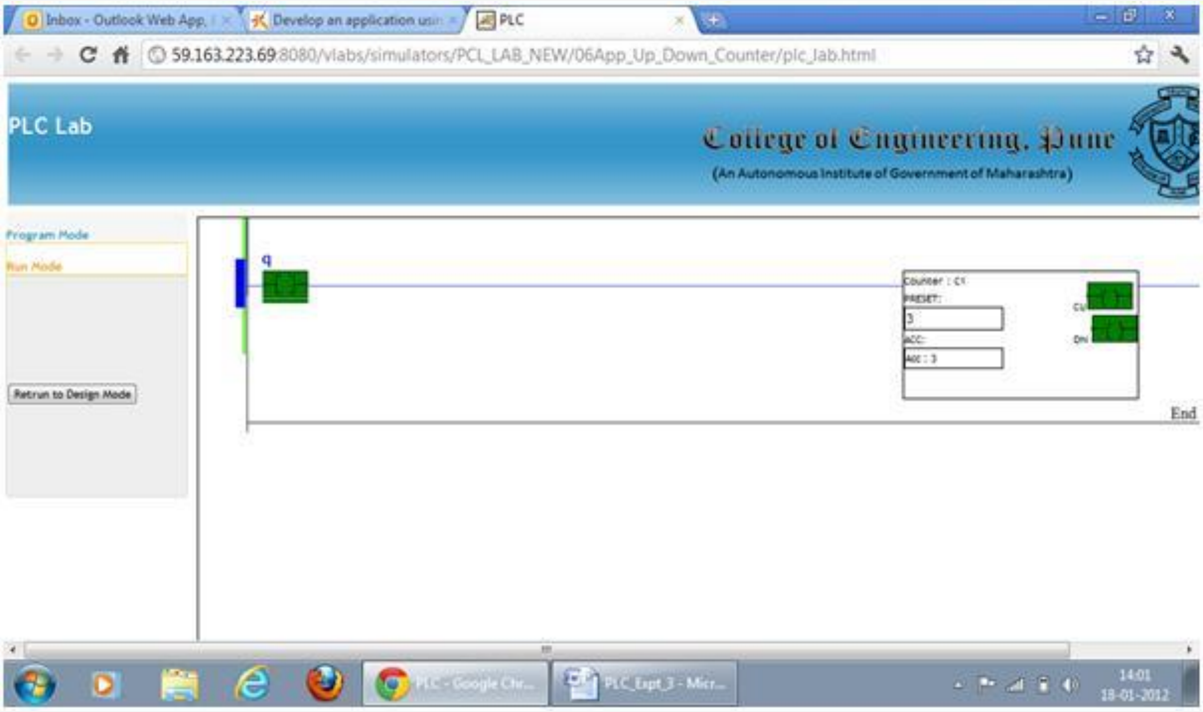
Objectives

1. Study Counter timing diagram.
2. Develop an application specific ladder program using counters.

Procedure:

The counter counts the pulses received at input. The pulses can be given by toggling the input bit "q" in this case. The counter will keep on counting till it reaches the preset value set by the user. Once the accumulator is equal to preset the DN bit will be energised. After this instant if next pulse is detected the accumulator will increment without changing the status of DN bit.

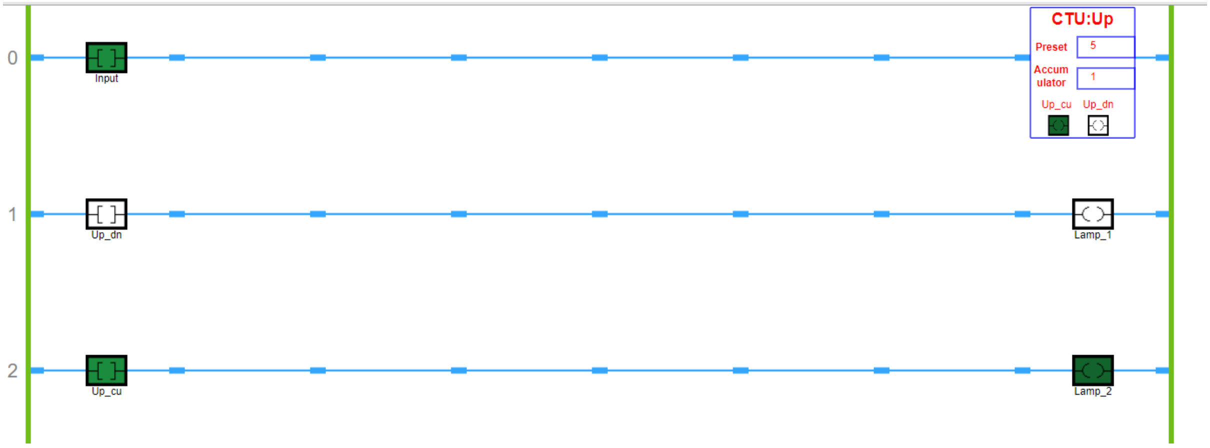
To reset the counter use "Reset" command so that the counter can be configured for new counts without reloading the page. Please note the tag of the reset bit must be the tag of counter e.g."CU1". The screen shot will appear as shown below.



In case of down counter the entire procedure will remain same. Only the number of counts are to be entered in the accumulator tab. The preset value is zero. When the input contact closes, the accumulator will go on decrementing, will reach to zero '0' value and the status of done bit will change.

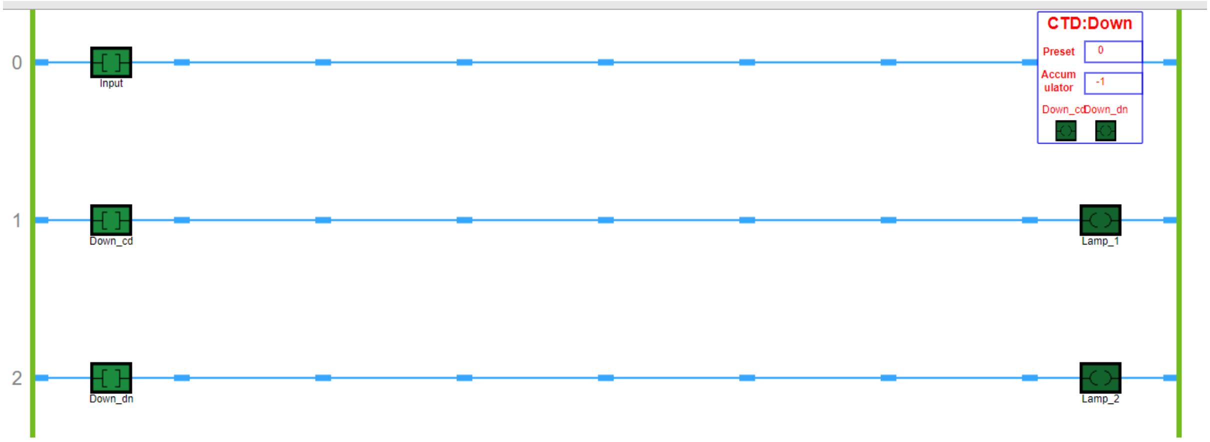
To reset the DN counter use "Reset" command so that the counter can be configured for new counts without reloading the page. Please note the tag of the reset bit must be the tag of counter e.g. "CD1".

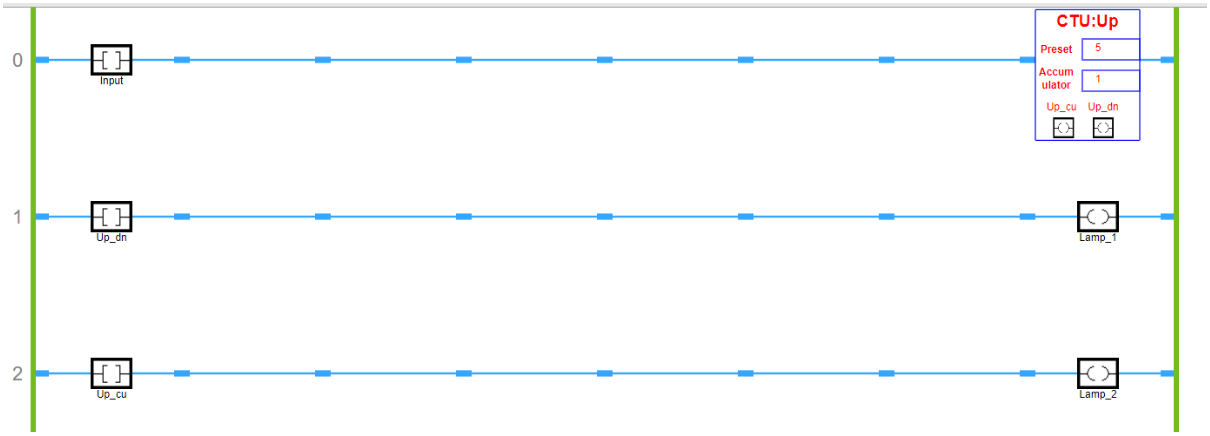
Simulation:

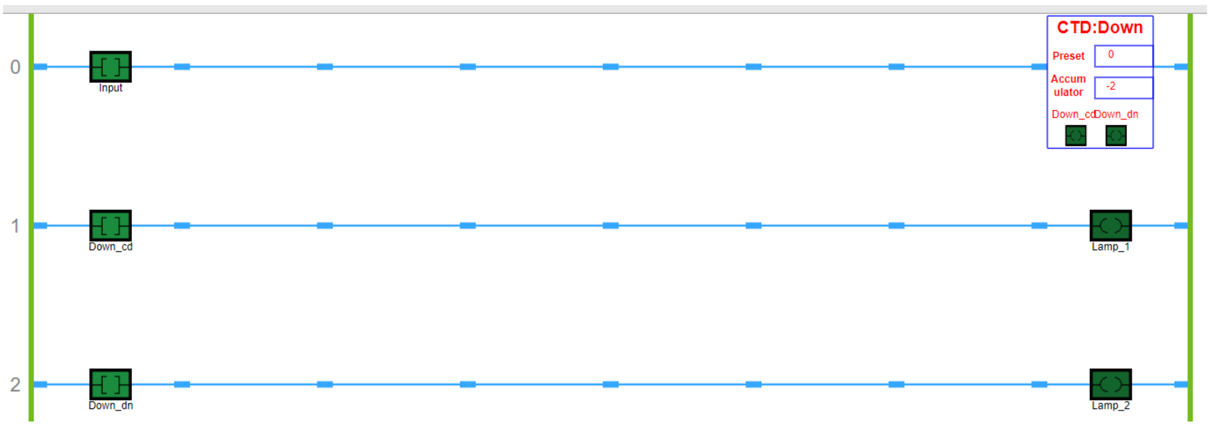

0
Input
CTU:Up
Preset
5
Accum
ulator
1
Up_cu
Up_dn
1
Up_dn
Lamp_1
2
Up_cu
Lamp_2


0
Input
CTU:Up
Preset
5
Accum
ulator
5
Up_cu
Up_dn
1
Up_dn
Lamp_1
2
Up_cu
Lamp_2


0
Input
CTD:Down
Preset
0
Accum
ulator
0
Down_cdDown_dn
1
Down_cd
Lamp_1
2
Down_dn
Lamp_2







Conclusion:

The application has been developed using Up/Down Counter.