SENSORS AND AUTOMATION LAB

Name: DHAIRYAPRASAD BHOSALE

Class: SY METALLURGY

Batch: A

MIS No.: 612211012

Sensors Modelling and Simulation

1. Characterize the temperature sensor (RTD)
2. Characterize the LVDT
3. Simulate the performance of a chemical sensor
4. Measurement of level in tank using capacitive type level probe

Programmable Logic Controller

1. Study hardware and software used in PLC
2. Implementation of Logic gates
3. Develop an application using On-Delay timer
4. Develop an application using Off-Delay timer
5. Develop an application using up/down counter

Experiment 1: Characterize the Temperature Sensor (RTD)

Objectives:

1. Aim: To understand the working principle of RTD

Study static and dynamic characteristics of RTD

2. Study effect of various parameters on RTD performance

Procedure:

Part 1: Static characteristics

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.
2. Click on ' R0' to get the value of R0 for selected RTD. Note the same.
3. Click on 'Get Temp'. The temperature for which Rt is to be found will be displayed.
4. Using formula calculate the value of Rt and enter the answer in the box provided (up to 2 decimals with rounding off). Submit the answer using submit button.
5. If your calculation is correct, go to step 3. Repeat the procedure min 3 times.
6. If your calculation is wrong, you will be asked to repeat the same. Please refer to GET FORMULA tab and verify your calculations.
7. After finishing minimum 3 set of correct readings, you can see the graph by clicking on 'Plot'.
8. When the graph is displayed click Next tab to repeat the

procedure with different reference resistance(R0) value and with different materials.

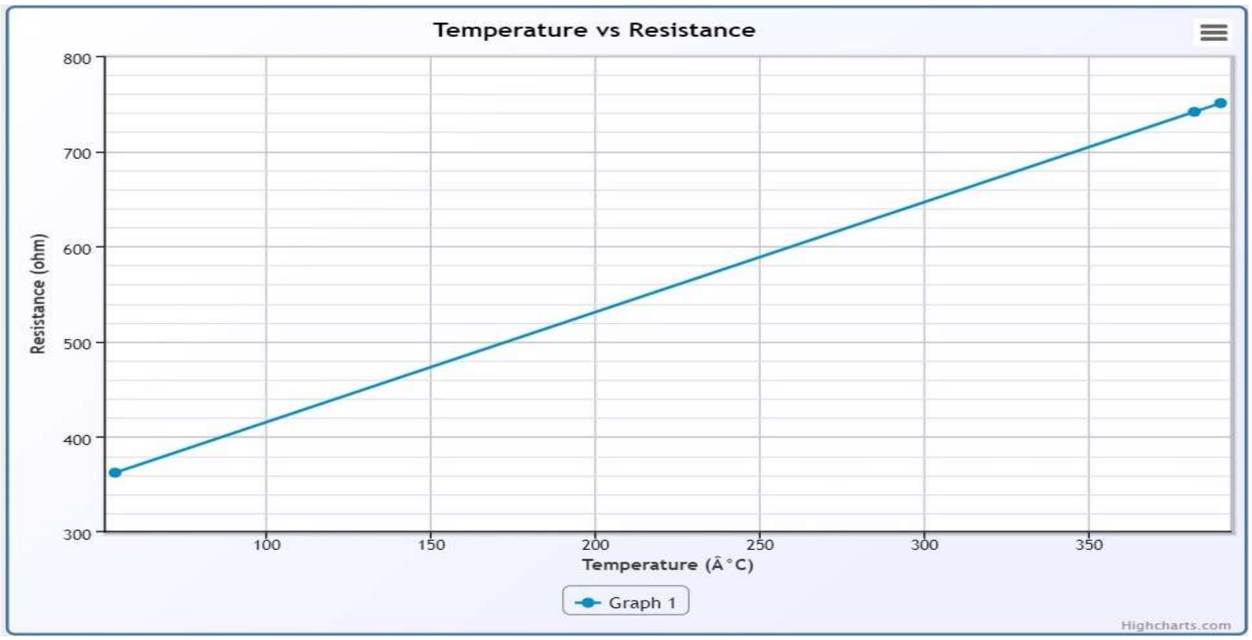
9. Minimum 3 calculations are necessary to plot the graph.
10. Study the graphs for RTD performance with different reference resistance values and different materials

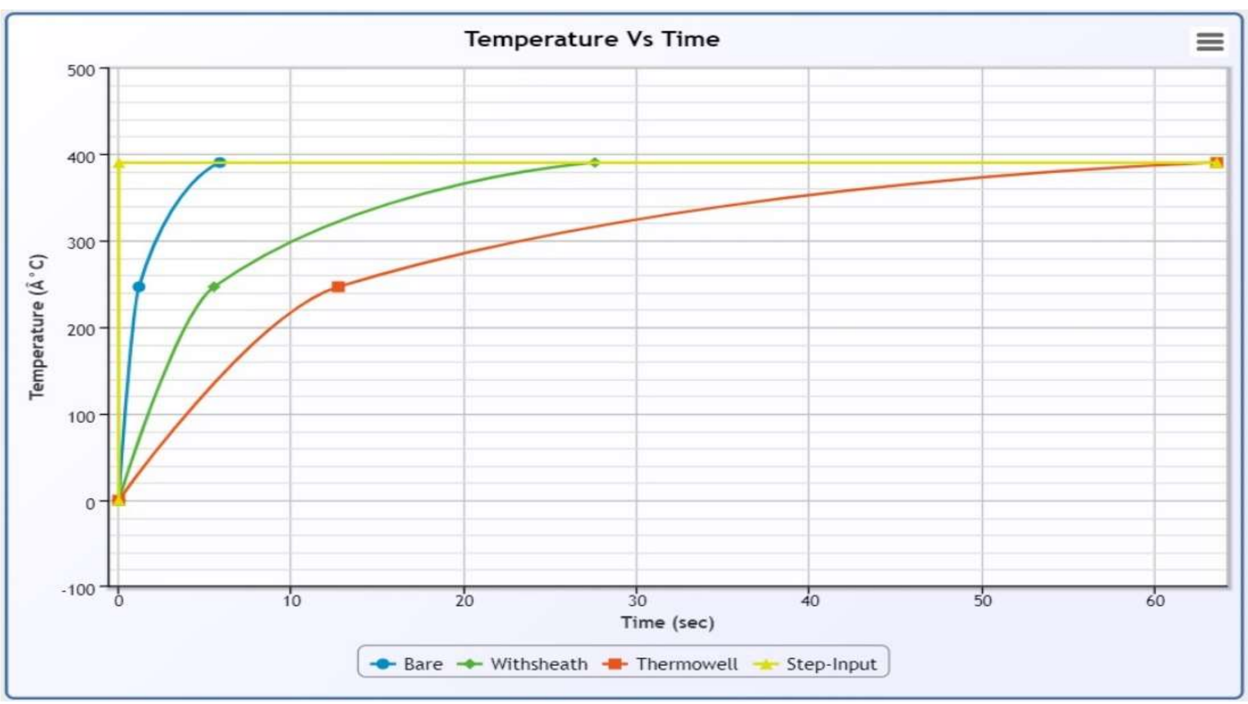
Part 2: Dynamic Characteristics:

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.

Observation:





Temp. coefficient	Ro	Temperature (0C)	Rt
0.00385	300	54	362.37
0.00385	300	382	741.21
0.00385	300	390	750.45

Material Platinum.

$R_0 - 300\ \Omega$

1. $T_1 = 54°C$

$R_T = 300(1 + 0.00385 \times 54)$

$= 362.37\ \Omega$

2. $T_2 = 382°C$

$R_T = 300(1+0.00385 \times 382)$

$= 741.21\ \Omega$

3. $T_3 = 390°C$

$R_T = 300(1 + 0.00385 \times 390)$

$= 750.45\ \Omega$

Conclusion:

1. Resistance of RTD probe depends on the temperature and material as different materials have different values of temperature coefficients.
2. The value of time constant increases when RTD is placed inside a sheath and it increases further when placed inside a thermowell.

Experiment 2: Characterize the LVDT

Aim: To understand working principle of LVDT

Objectives:

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

Procedure:

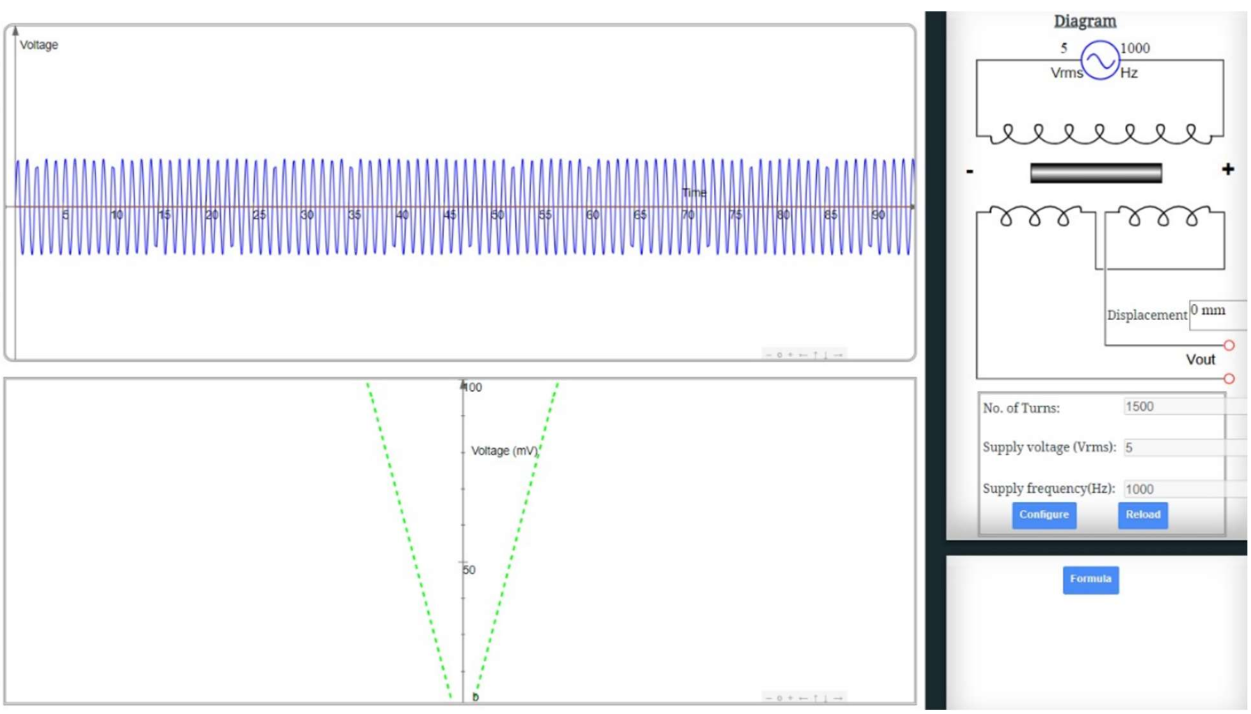
1. First you need to configure the LVDT. Click on ' Show panel' tab at the right bottom. For making the circuit, drag and drop the primary coil, Armature and secondary coils at the locations shown on left hand side.
2. Now select No of Turns, peak to peak supply voltage and frequency from the drag and drop menu, available below LVDT diagram. Click on configure block to configure LVDT.
3. Now click on the black rectangular core placed between primary and secondary windings.
4. Drag the core to left hand side and observe the effect on the output magnitude. This can be observed on the time vs output voltage waveform and on the Distance vs output voltage graph. The core displacement is indicated in the square box below the diagram

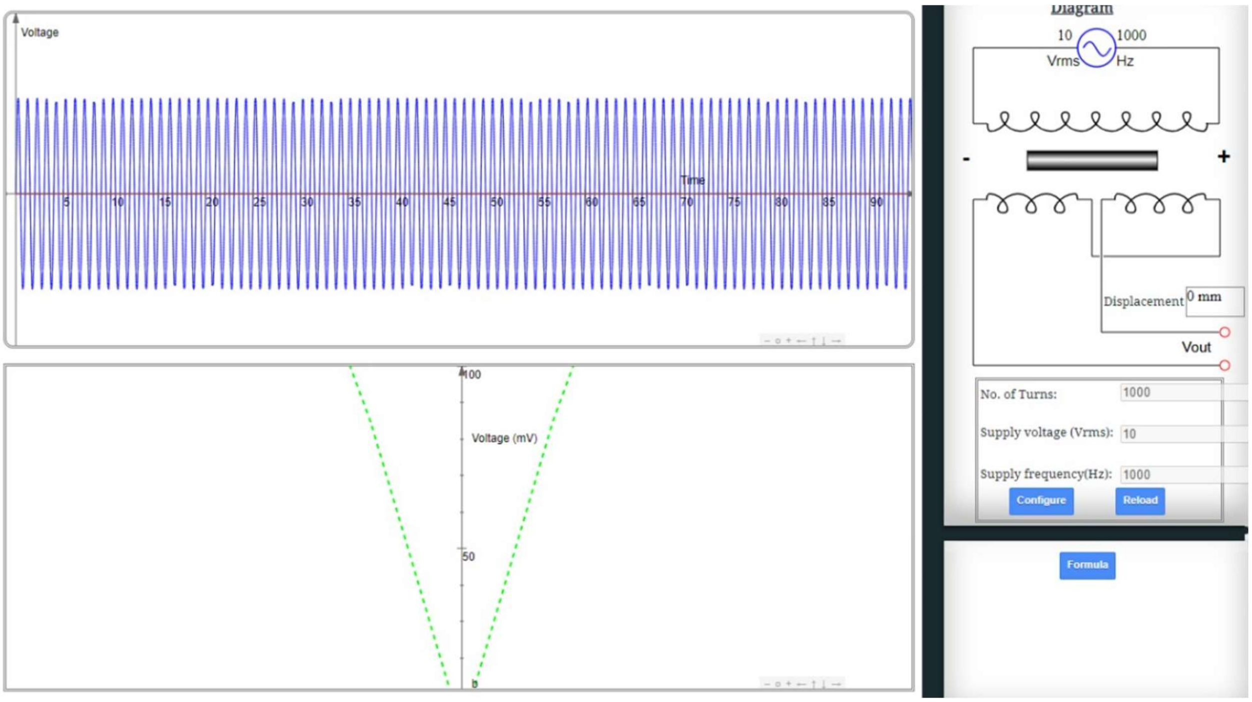
5. Drag the core to right hand side and observe the effect on the output magnitude. Also observe the change in the phase.

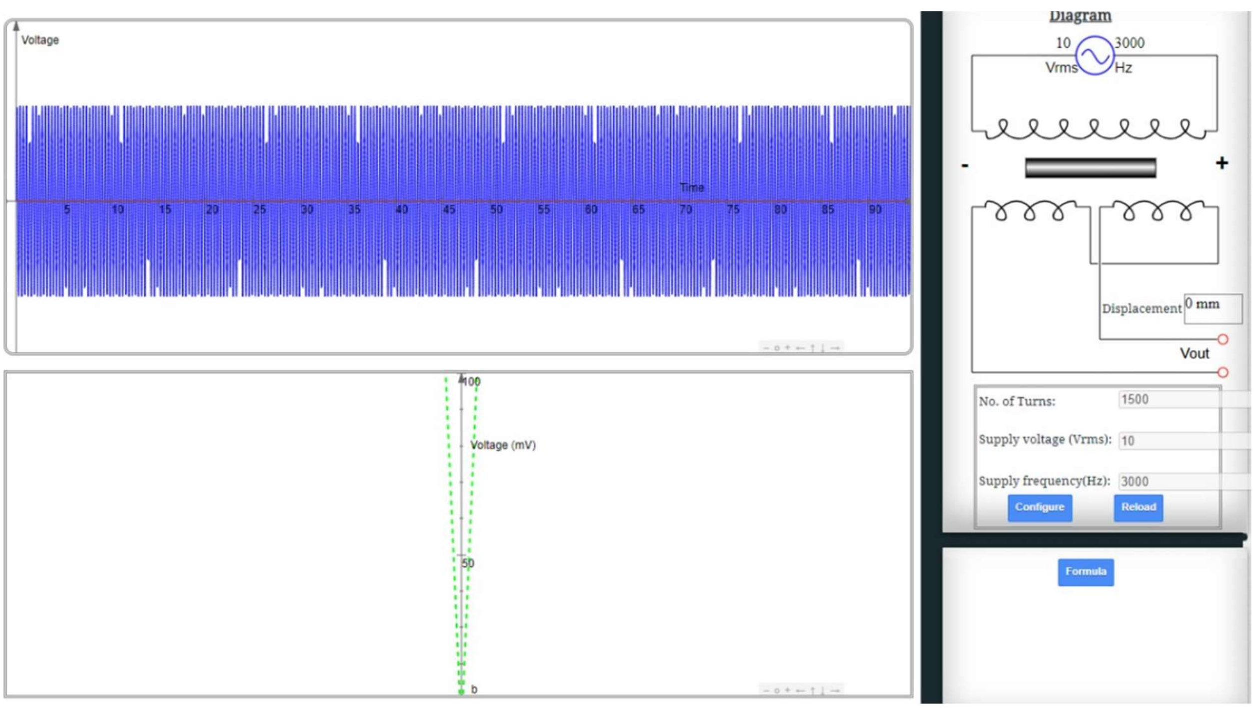
6. Repeat steps 2 to 4 by changing supply voltage keeping frequency and no of turns constant. Study the effect on the output voltage. For this click on blue colour 'Configure' tab in the right-side panel. You need to select required parameter value from drop down menu. After selecting the values click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

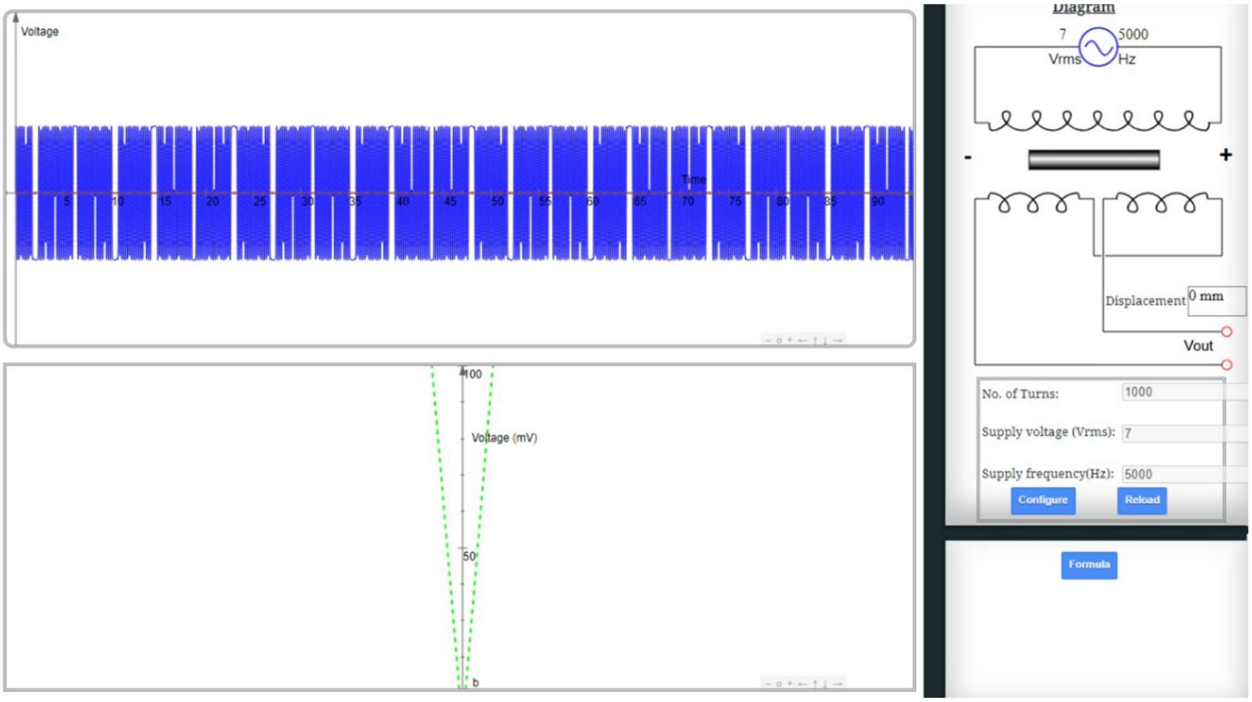
7. Repeat steps 2 to 4 by changing supply frequency keeping and no of turns constant. Study the effect on the output voltage.

8. 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.




Voltage
Time
5 10 15 20 25 30 35 40 45 50 55 60 65 70 75 80 85 90
100
Voltage (mV)
50
0
Diagram
10
Vrms
1000
Hz
-
+
Displacement
0 mm
Vout
No. of Turns: 1000
Supply voltage (Vrms): 10
Supply frequency(Hz): 1000
Configure
Reload
Formula


Voltage
Time
5 10 15 20 25 30 35 40 45 50 55 60 65 70 75 80 85 90
100
Voltage (mV)
50
0
Diagram
10
Vrms
3000
Hz
-
+
Displacement
0 mm
Vout
No. of Turns: 1500
Supply voltage (Vrms): 10
Supply frequency(Hz): 3000
Configure
Reload
Formula



Conclusion:

Waveforms change as we change the number of turns, supply voltage and supply frequency.

Experiment 3: Simulate the performance of a chemical sensor

Aim: To understand the working principles of a chemical sensor.

Objectives:

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

Procedure:

Level 1: pH probe calibration

Aim: Study calibration of a pH probe.

Step by step Procedure:

1. When the simulator window opens, initial pH value is displayed in the control panel on the screen. Select the pH value for which the probe is to be calibrated.
2. After entering pH value, the pH verses output mV values are displayed.
3. Click on 'Plot'. The graph of pH verses millivolt will be displayed.
4. After getting the graph, hide the graph and switch on to the next experiment i.e. Level 2

Level 2: Measurement of pH

Aim: Study of Measurement of pH.

In this experiment, by using the formula one can calculate the pH and understand the pH measurement system.

Step by step Procedure:

1. Study the measurement diagram completely.
2. Select the sample. After selecting the sample, pH value will be displayed on the screen.
3. Enter the calculated millivolt value. For calculations of millivolt click on FORMULA tab.
4. Using the formula, calculate the value of the millivolt output for the sample. In the formula, value of pHc is the pH value deviated from 7. Subtract the value of pH from 7 and enter the answer in the box provided (upto 2 decimals with rounding off). Submit the answer using submit button.
5. If your calculation is correct, the output millivolt value will be displayed on the screen.
6. Repeat the procedure if necessary for different samples.

Level 3: Effect of Temperature

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

In this experiment, by using the formula one can calculate the pH and understand the pH measurement system.

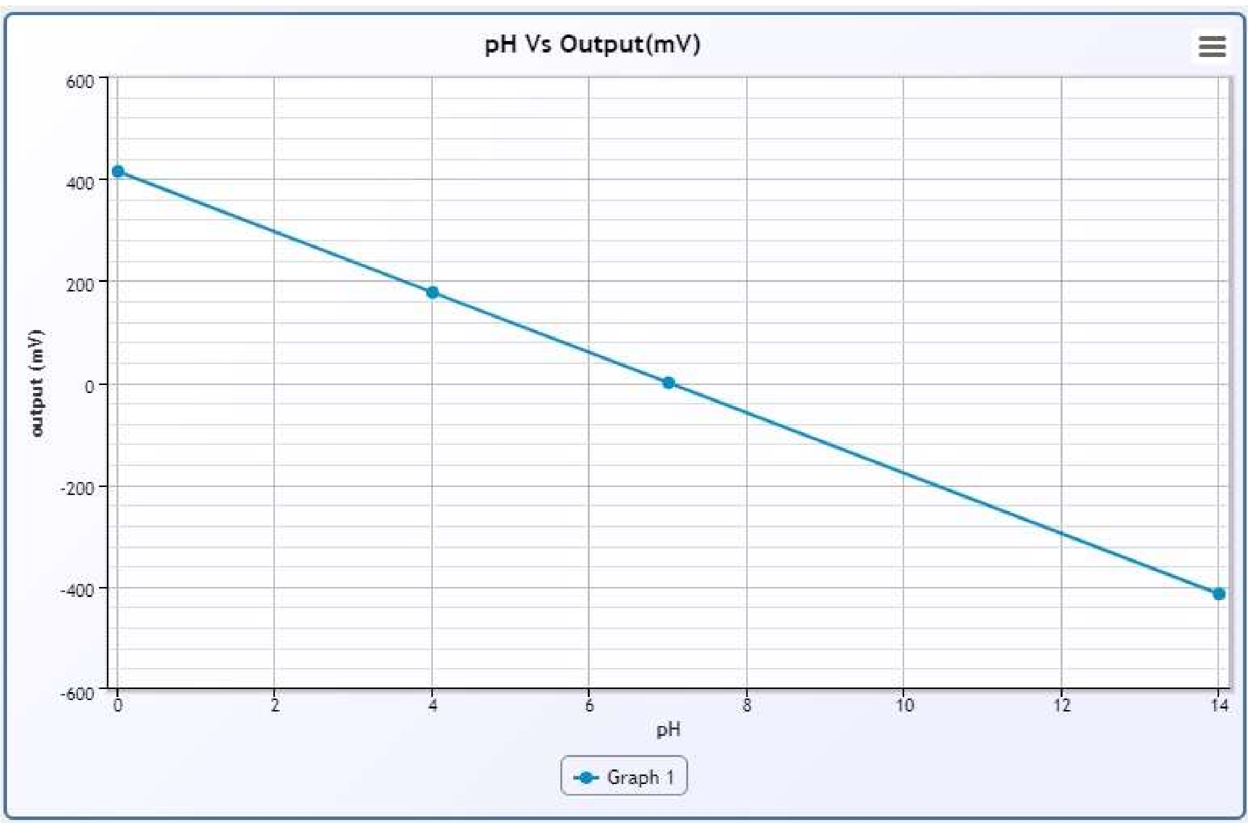
Step by step Procedure:

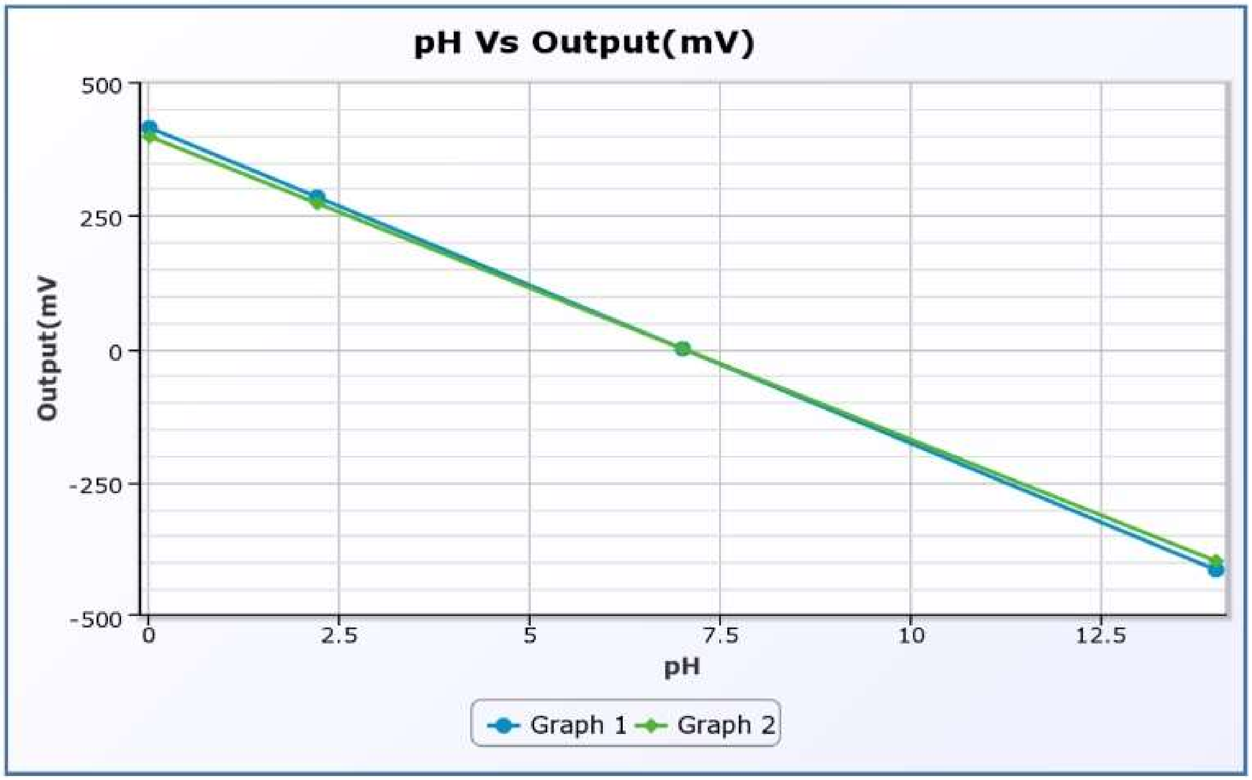
1. In this level the sample is same as selected in level 2. This is default. The pH of the sample is displayed in the control panel.

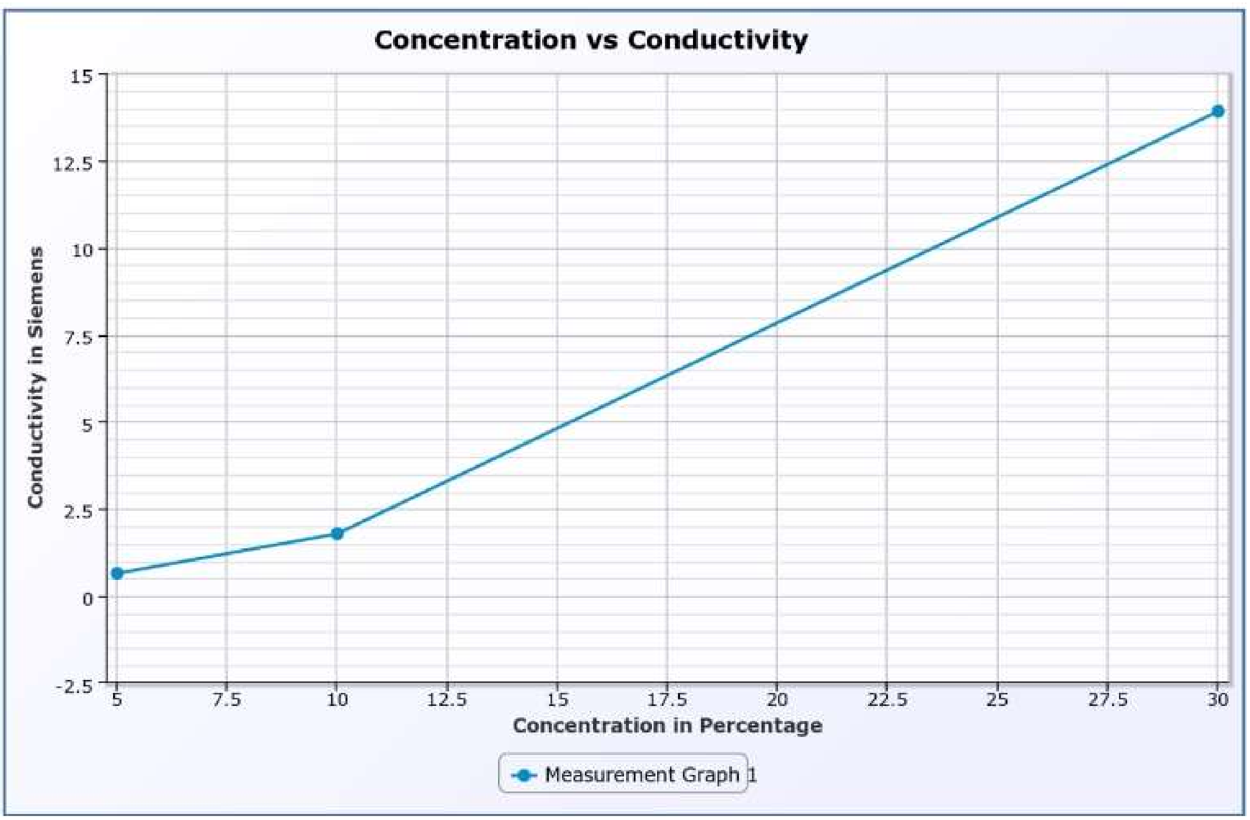
2. Click on Get Temperature. The temperature of sample and the output milli volt will be displayed on the screen now. Compare this value with the previous level output value. The reference temperature considered for the previous levels is 25°C.

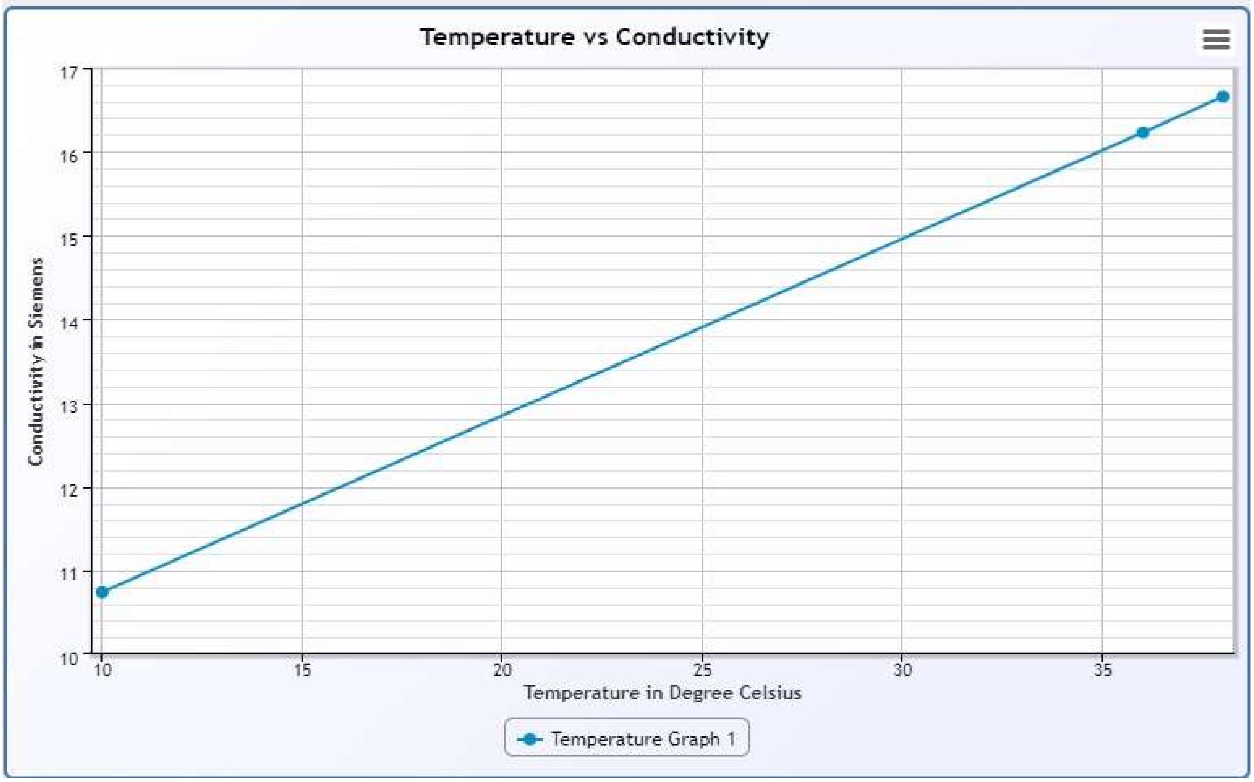
3. Click on Plot Click on Plot tab to plot the graph.

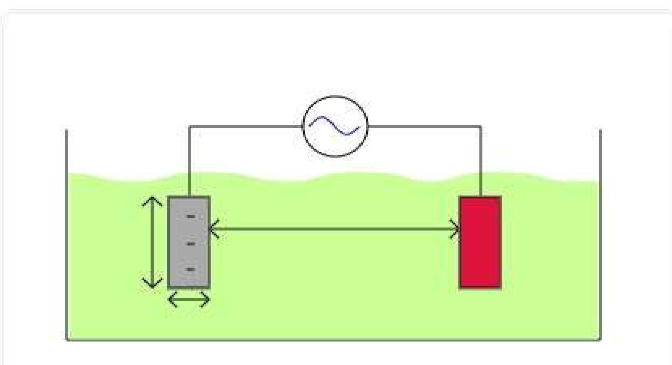
4. You can verify the answer by putting the values in a formula. Only value of the temperature (in Kelvin) will change.




pH Vs Output(mV)
Output(mV
500
250
0
-250
-500
0
2.5
5
7.5
10
12.5
pH
Graph 1
Graph 2


Concentration vs Conductivity
Conductivity in Siemens
15
12.5
10
7.5
5
2.5
0
-2.5
5
7.5
10
12.5
15
17.5
20
22.5
25
27.5
30
Concentration in Percentage
Measurement Graph 1





Console window:

Assumption: Half of this value is deposited on each electrode.
Default values for L, A and B:
L=1cm , A=10cm , B=0.1cm

Selected Contamination is 0.1

Control Panel:

Sample: HCL

Concentration: 30%

Cellconstant: 1.0

Contamination: 0.1mm

Modified Cell Constant : 0.90

Specific Conductance at 25°C : 13.91 (Siemens)

Modified specific conductance value: 12.52 (Siemens)

Reload

Calculation:

$$E = \left(0 - 2.3026 \times \left(\frac{8.3144 \times 298.15}{96485}\right)(2.2-7)\right) \times 1000$$

$$= 283.966$$

30% HCL

38°C $C_T = 13.91\,[1 + 0.0152 \times (38-25)] = 16.66$

10°C $C_T = 13.91\,[1 + 0.0152 \times (10-25)] = 10.74$

36°C $C_T = 13.91\,[1 + 0.0152 \times (36-25)] = 16.23$

For Conductivity Measurement

5%

$$C = \frac{50000 \times 1.023}{1000 \times 36.46} = 1.403$$

$$SC = \frac{1.403 \times 426 \times (1 - 0.37\sqrt{1.403} + 0.38 \times 1.403)}{1000} = 0.65$$

10%

$$C = \frac{100000 \times 1.048}{1000 \times 36.46} = 2.874$$

$$SC = \frac{2.874 \times 426 \times (1 - 0.37\sqrt{2.874} + 0.38 \times 2.874)}{1000} = 1.79$$

30%

$$C = \frac{300000 \times 1.149}{1000 \times 36.46} = 9.454$$

$$SC = \frac{9.454 \times 426 \times (1 - 0.37\sqrt{9.454} + 0.38 \times 9.454)}{1000} = 13.91$$

Conclusion:

1. pH value changes as temperature is varied.
2. Specific conductance increases as concentration is increased.
3. Temperature also has and effect on specific conductance.
4. Specific conductance decreases with increasing amount of contamination.

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

Aim: To measure tank level using capacitive probes.

Objectives:

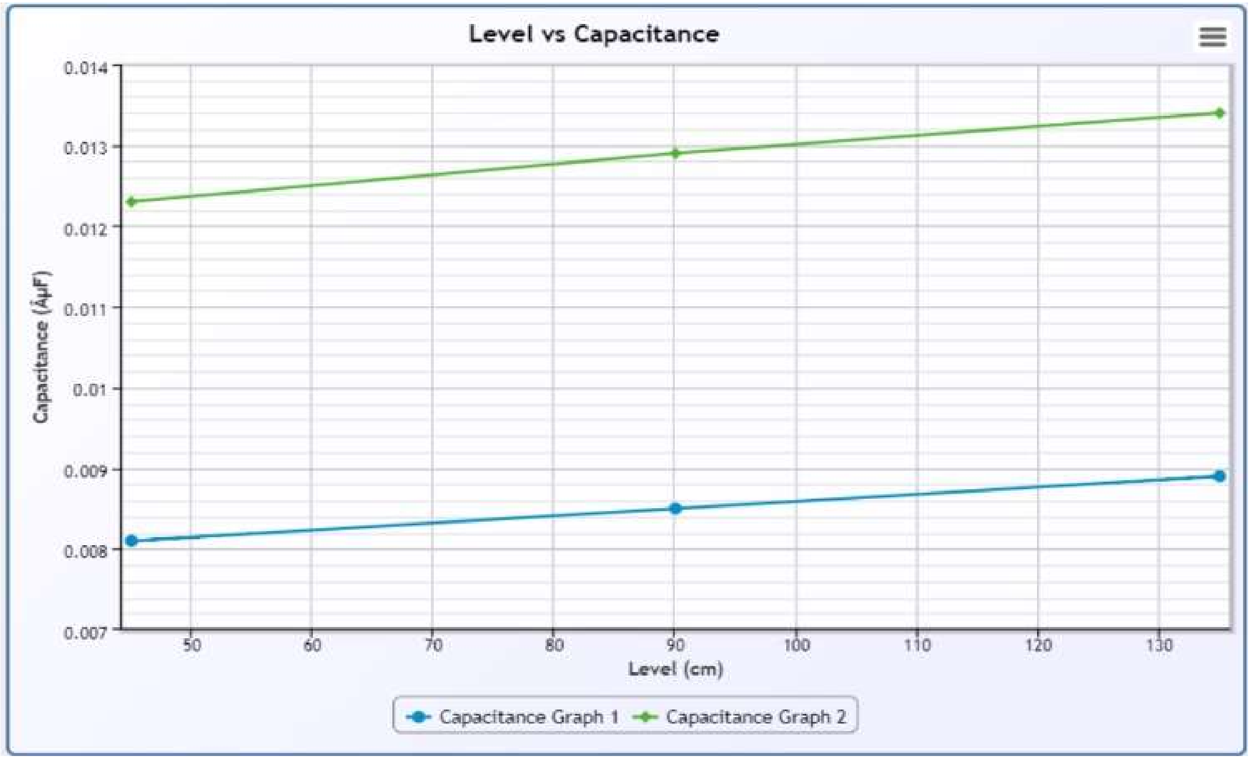
1. Review various methods of level measurement.
2. Understand working of capacitance level transmitter.

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.

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 (up to 2 decimals with rounding off). Submit the answer using submit button.

Observation:



Height of Tank: 500 cm

Outer radius(r2): 2.5cm

Span Value: 445

Service: Coffee Beans

r1	h1	h2	C
0.1	400	45	0.0081
0.1	355	90	0.0085
0.1	310	135	0.0089
0.3	400	45	0.0012
0.3	355	90	0.0013
0.3	310	135	0.00134

Calculations:

45 $\quad C = \dfrac{2\pi \varepsilon_0 (400 + 1.5 \times 45)}{\ln(2.5/0.1)} = 0.0081\ \mu F$

90 $\quad C = \dfrac{2\pi \varepsilon_0 (355 + 1.5 \times 90)}{\ln(2.5/0.1)} = 0.0085\ \mu F$

135 $\quad C = \dfrac{2\pi \varepsilon_0 (310 + 1.5 \times 135)}{\ln(2.5/0.1)} = 0.0089\ \mu F.$

$r_1 = 0.3$

45 $\quad C = \dfrac{2\pi \varepsilon_0 (400 + 1.5 \times 45)}{\ln(2.5/0.3)} = 0.0012\ \mu F$

90 $\quad C = \dfrac{2\pi \varepsilon_0 (355 + 1.5 \times 90)}{\ln(2.5/0.3)} = 0.00128\ \mu F$

135 $\quad C = \dfrac{2\pi \varepsilon_0 (310 + 1.5 \times 135)}{\ln(2.5/0.3)} = 0.00134\ \mu F$

Conclusion:

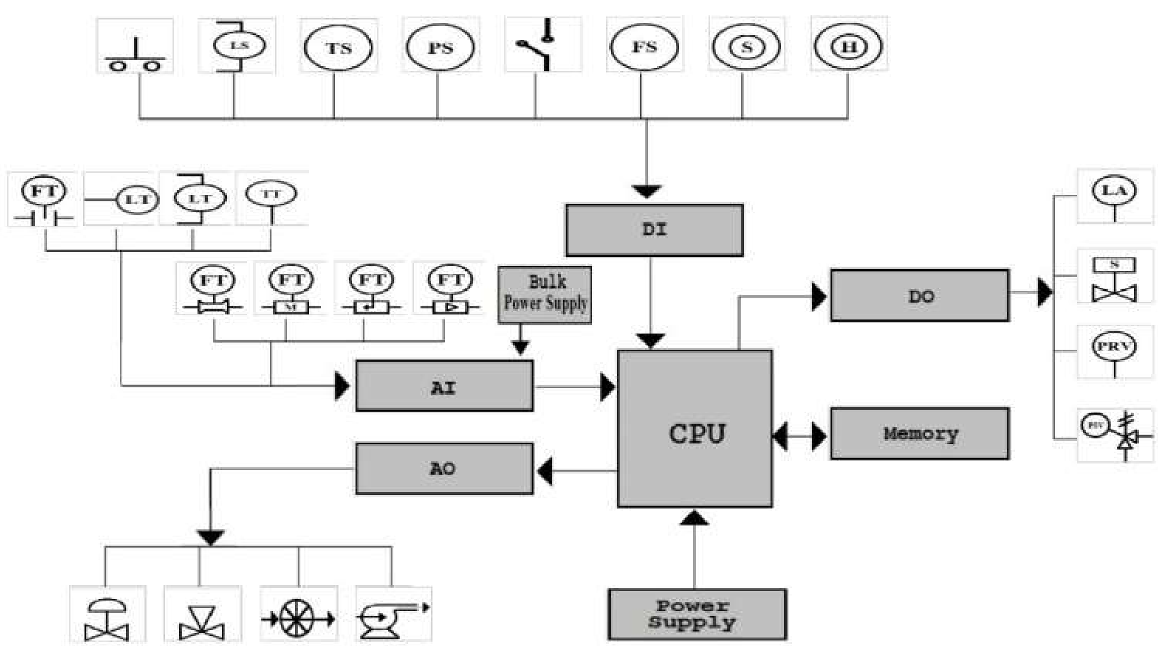
The Level vs Capacitance graph changes as we vary the inner radius, but no change is observed in the Level vs Current graph.

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



Experiment 6: Implementation of Logic gates

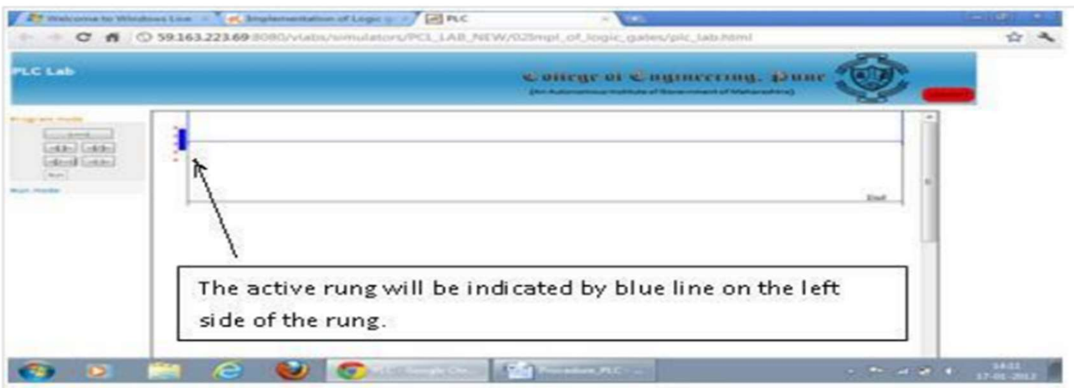
Aim: To understand Simple Ladder program

Objectives:

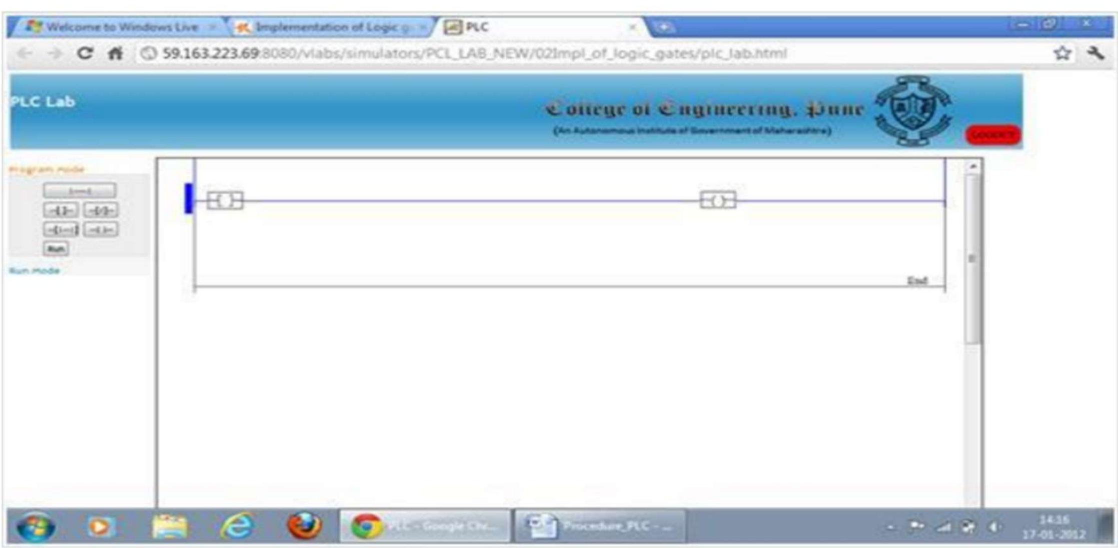
1. Develop a ladder using standard procedure.
2. Solve the problem using ladder programming.

Procedure:

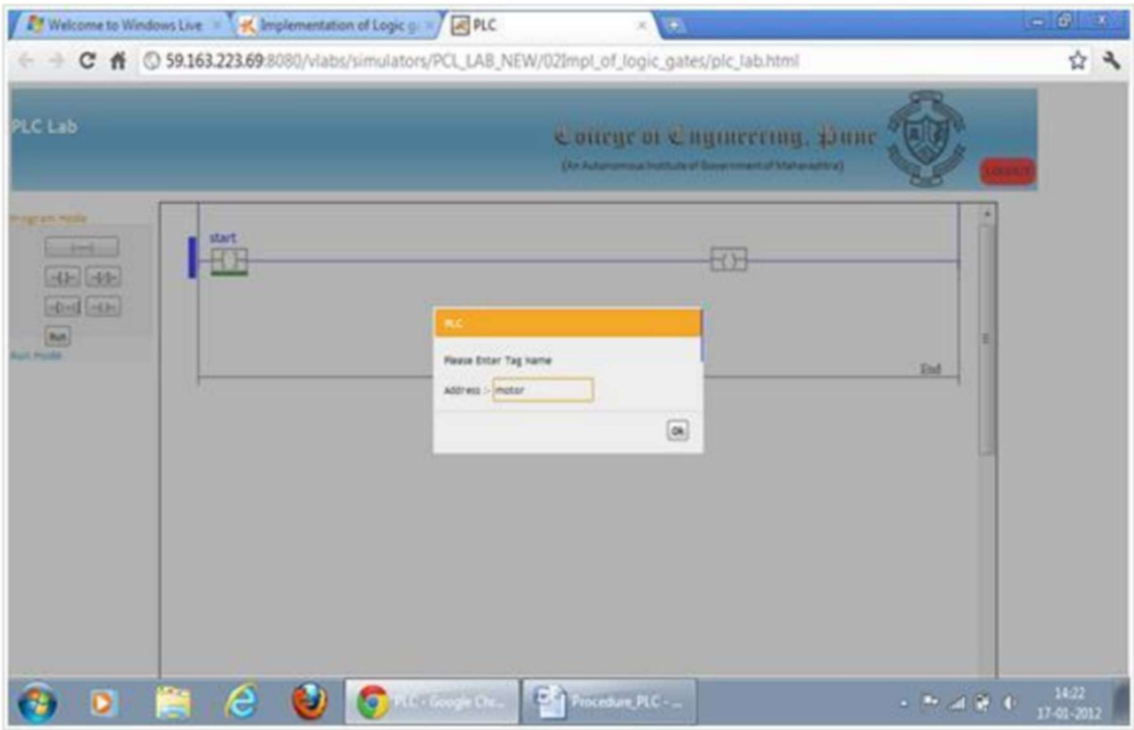
1. Prior to starting of the development of ladder diagram following steps needs to be understood:
   a. Understand the problem statement like test the logic for OR gate, AND Gate etc.
   b. Develop the logic on paper and validate the logic by considering various cases
   c. Prepare the truth table and test the logic using all valid cases
   d. Go to simulator icon and click on the "Simulator" button
   e. The PLC simulator will be opened in new window
2. The procedure for writing the ladder diagram in the work space is as follows
   a. The screen shot for the first window will appear like this.



b. Place the contacts as per the requirement by clicking the appropriate contact shown in the left pan of the window. In the example demonstrated below one normally contact and one coil is placed as shown the figure.



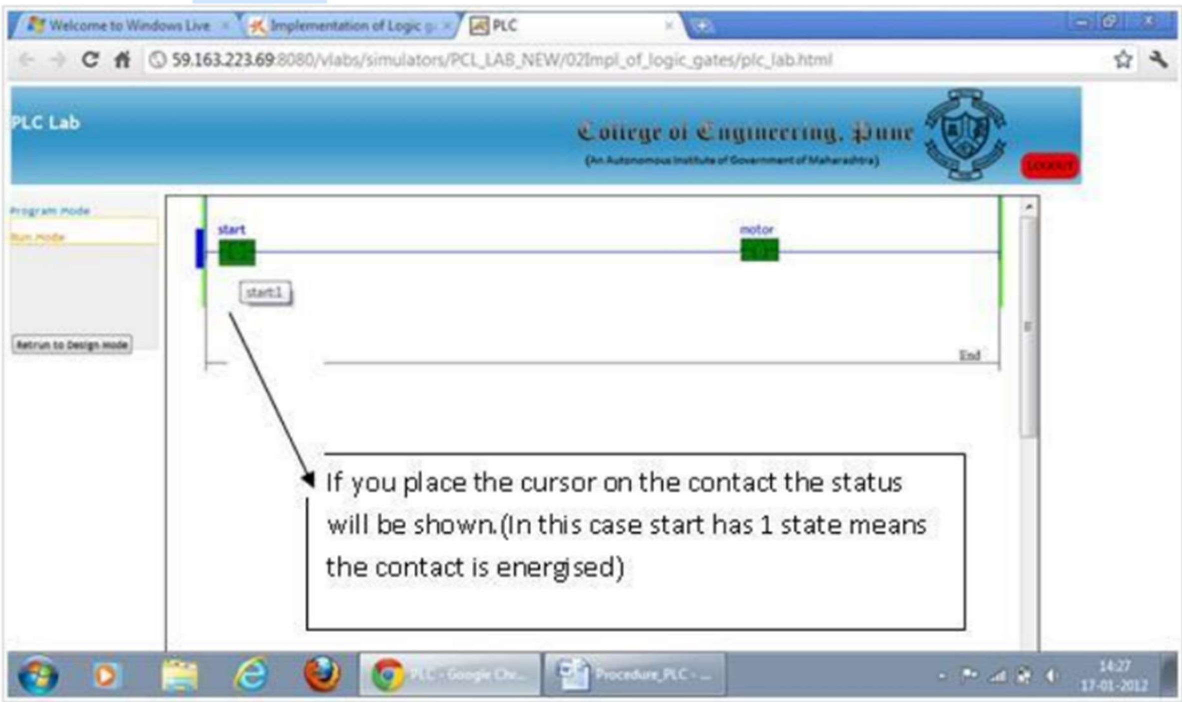
c. Double Click on the contact or coil and you can give tag name like "start", "stop" etc. Please ensure that the tag numbers are true replica of process connections. Similarly give tag name to coil like "motor", "Lamp" etc. The final ladder will look like this.



d. Please note that the tag names are case sensitive and if you are using them in circuit as bit make sure that the correct tag name appears.

e. Click the Run button available at the left side of the window so the ladder will be ready for running and you can test your logic. Both sides of the rung will become green and this is the indication of run mode. Please not that in run mode the contact is not available so if you want to make changes you will have to go in design/program mode.

f. Double click on the contact and the window for toggling the contact will appear. As you press toggle the state of contact will change. Hence the output contact status will change.



g. Please remember the ladder contacts or the state of the inputs and outputs are always in de-energised state. The de-energised is that state wherein the contacts are in non-active state.

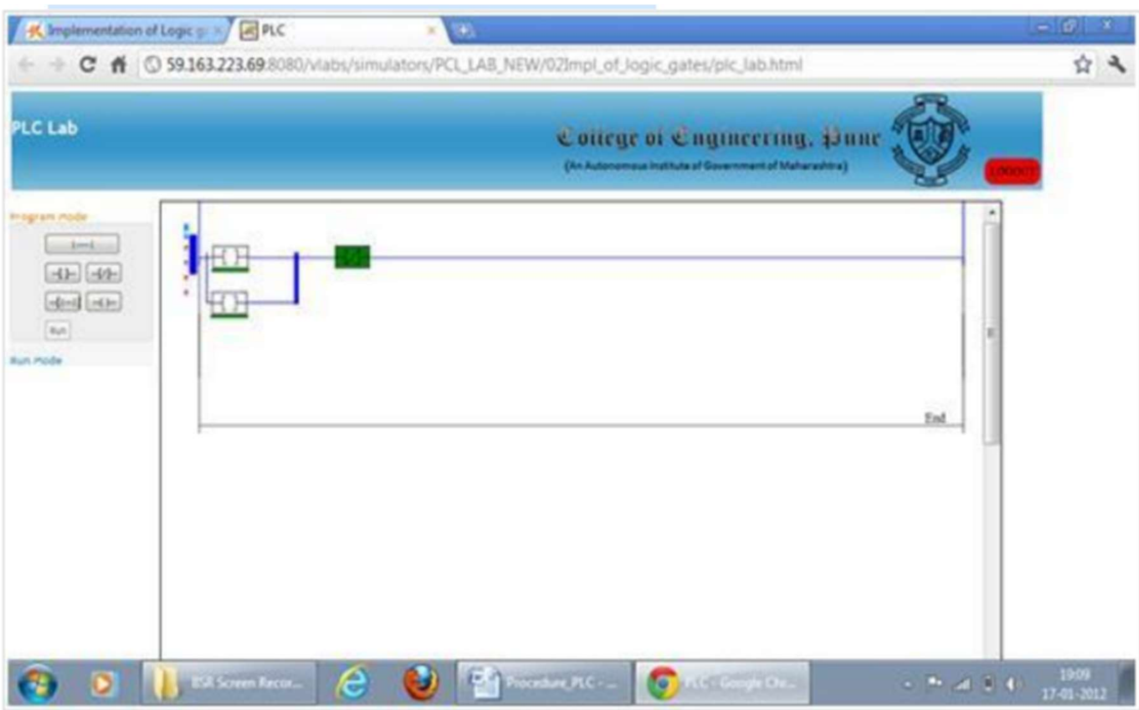
h. You can once again toggle the contact and the output state will change. To add any contact, you will have to go in programme mode. Click on the rung and add contacts.

i. To delete any contact or output click on the contact green line below the contact will appear press "delete" the contact will be deleted.

j. To reload the ladder, right click the mouse "Reload "button will appear and after pressing the button the page will be ready for new ladder development.

k. You can add seven elements in series and 5 elements in parallel.

l. To add element in parallel click on the contact where you wish to add parallel branch. Select the branch and drag it to location where you wish to terminate the branch. The screen will appear as shown below.

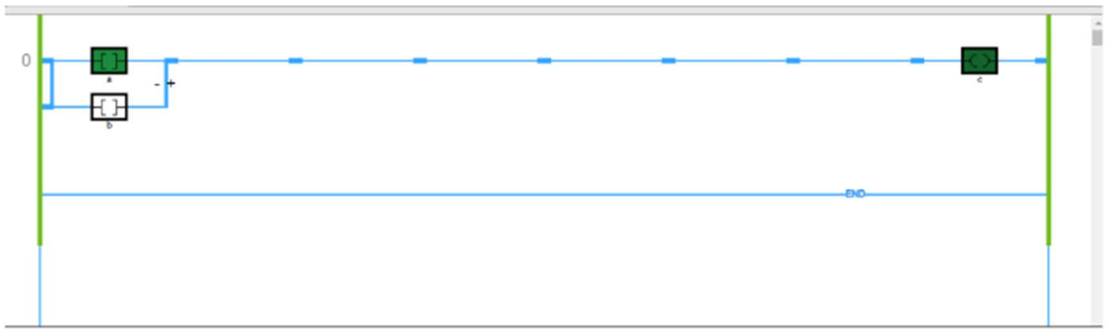


m. Repeat the procedure and check the correctness of the logic.

n. Similarly, you can check the logic for OR, NOR, and NAND gates. Validate the truth tables and confirm the results.

Observation:

OR gate



NOT gate

AND gate

Conclusion:

We have studied logic gates, Boolean values and constructed and tested different types of logic gates using ladder programming.

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:

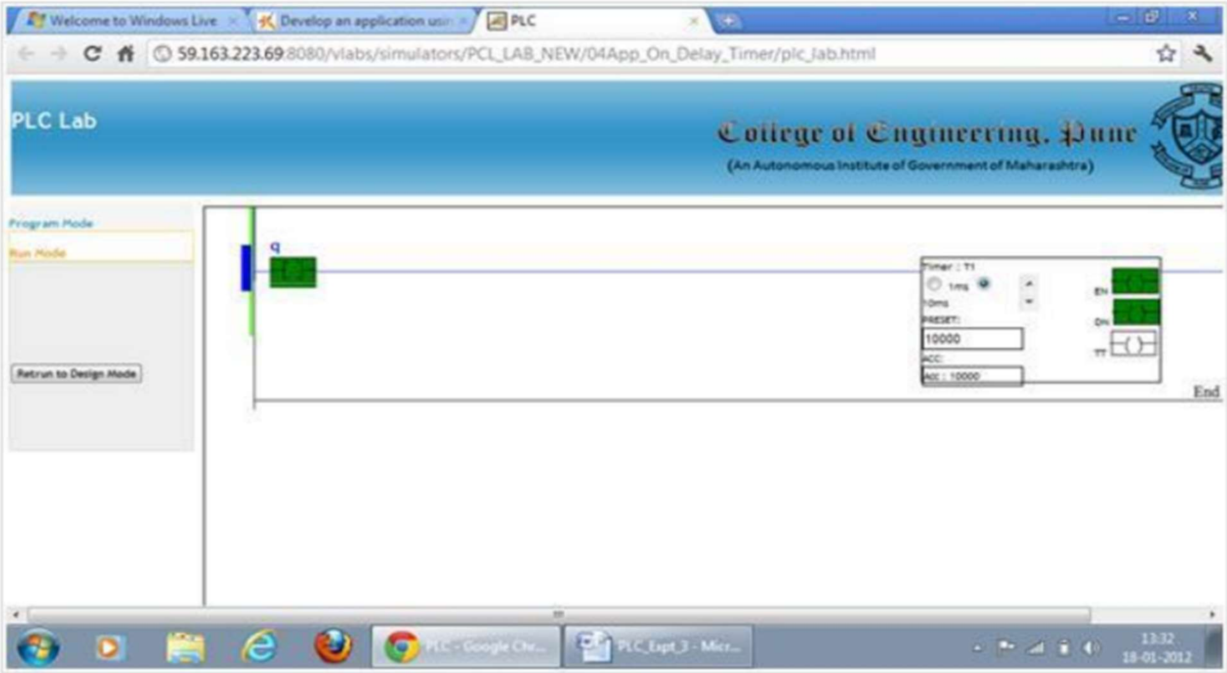
In this experiment the on-delay timer will be tested for its functionalities using Simulator. Following bits of the timer are to be observed.

- Initialising bit "q" in this case.
- Enable bit "EN"
- Done bit "DN"
- Timer timing bit "TT"

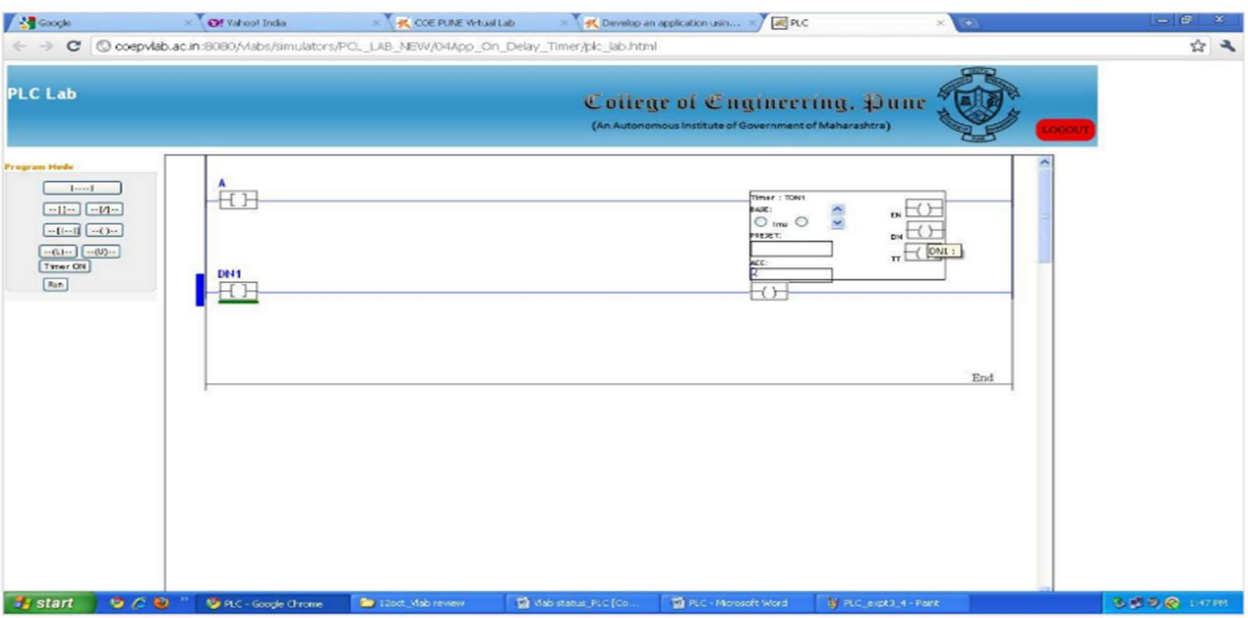
Preset value needs to be entered by the user.

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.

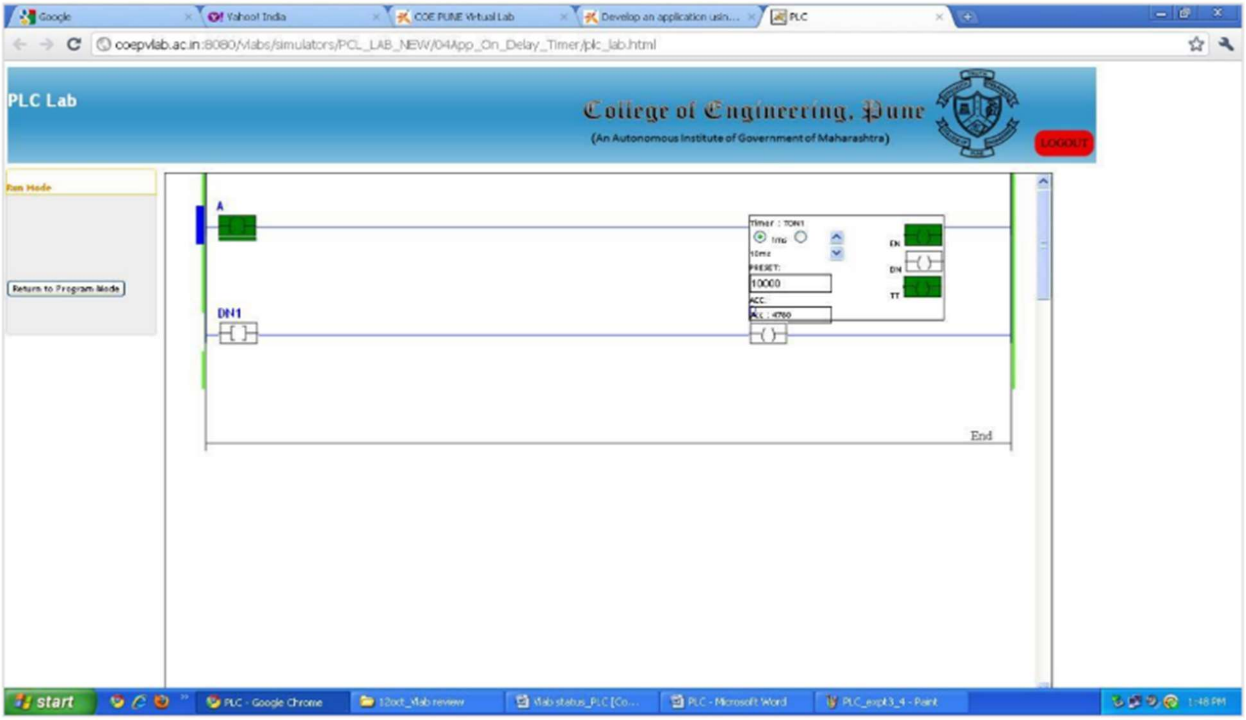
While configuring the timer the time base needs to be selected. There are two options viz. 1 mS and 10 mS are available for you to configure. Select appropriate as per the need of the application. The screen shot of the configured timer will appear like this.



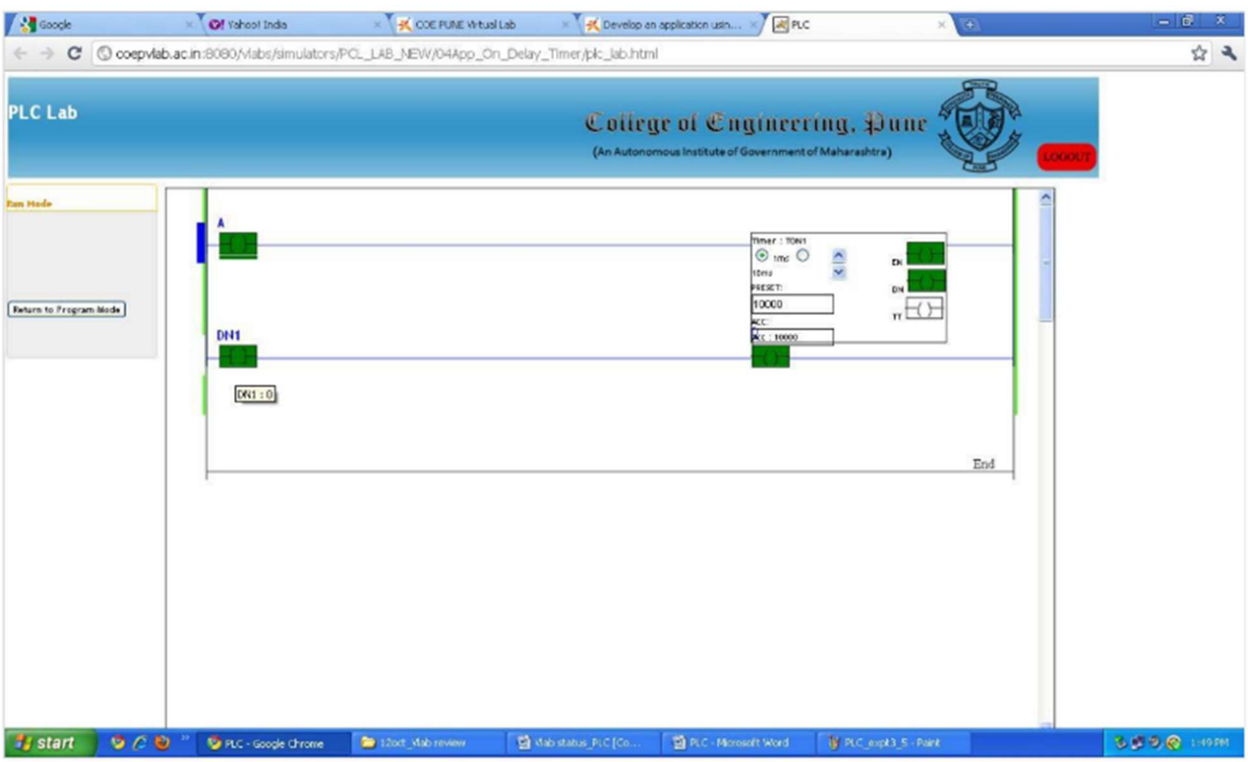
To test the EN, DN, and TT bits; double click on the bit and give tag name to the bit. The same tag name is to be used in the new rung to test the status or to energies the output. You can also test the cascading of the timer using these bits. For details see following screen shots. Observe the tag name for timer DN bit in Edit Mode and how it is used in the logic.



Observe the bit status in Run mode when input a is toggled.

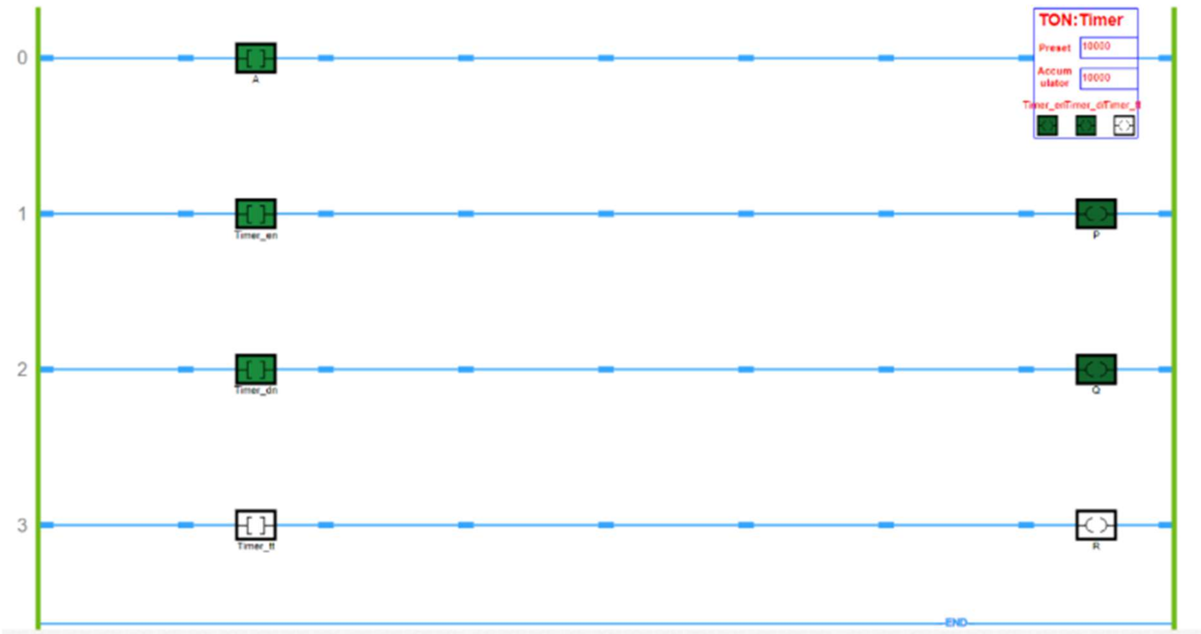


See following screen shot to observe the output bit status when delay is over.



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".

Observation:



Conclusion:

ON-delay timer circuits are not activated until after a certain amount of time (preset) has elapsed. They are used to control the timing of conveyor belts and other equipment.

Experiment 8: Develop an application using OFF Delay Timer

Aim: To Develop an application using OFF Delay Timer

Objectives:

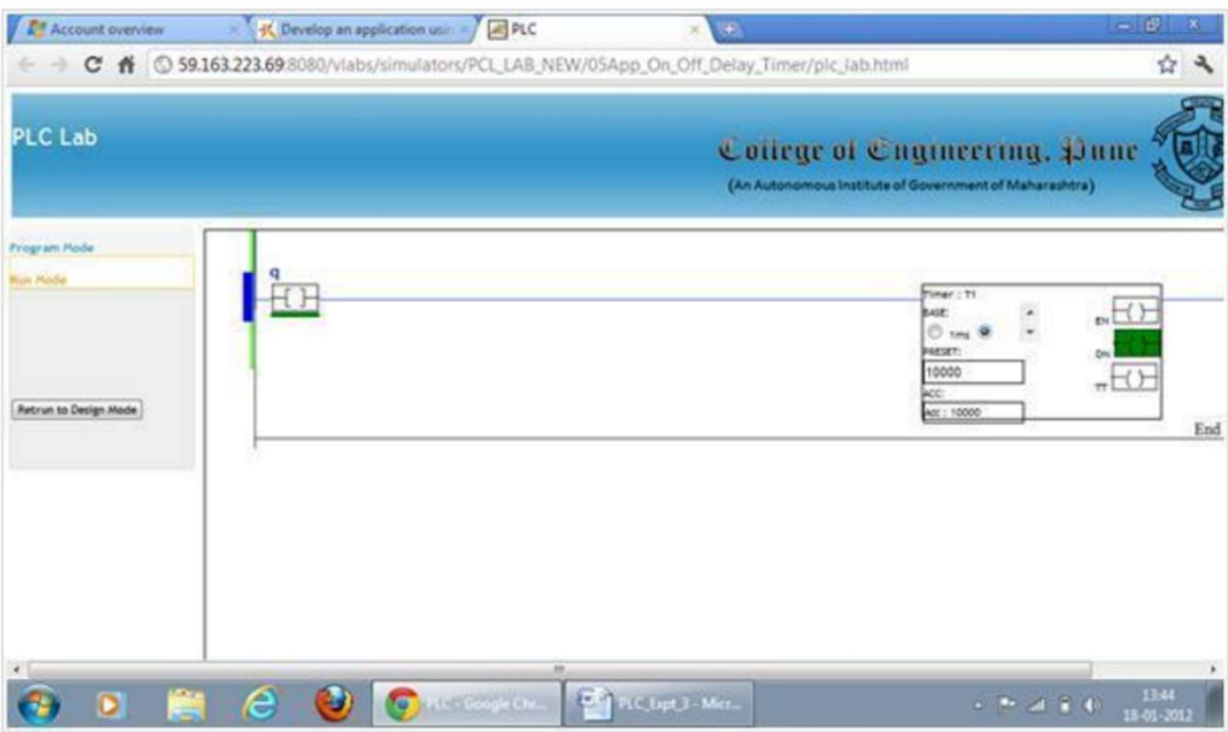
1. Study the timing diagram of OFF Delay Timer
2. Solve the assignment of Toff timer

Procedure:

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

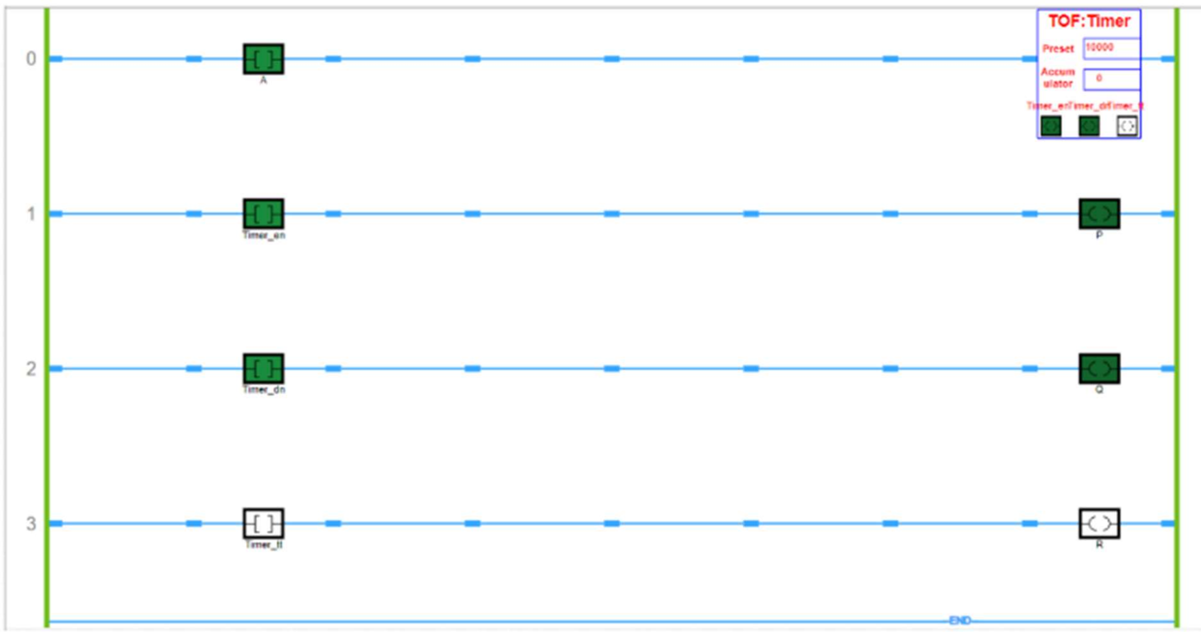
A typical difference can be observed in the operation (in Run mode).

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.

Observation:



Conclusion:

OFF-delay timers open or close the circuit as soon as power is removed. The contacts will not return to their normal position until the preset time delay has elapsed. They are used in air-conditioning systems, elevator doors, etc.

Experiment 9: Develop an application using UP/DOWN counter

Aim: To Develop an application using UP/DOWN counter

Objectives:

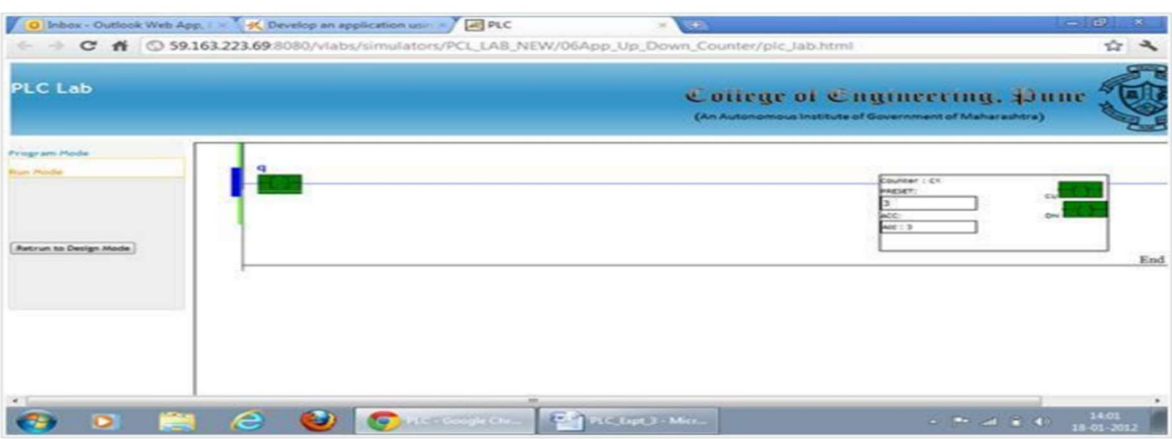
1. Study Counter timing diagram.
2. Develop an application specific ladder program using counters.

Procedure:

Implement the counter using Simulator

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 is 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".

Observation:

UP counter

DOWN counter

Conclusion:

Up counter counts from zero to the maximum number of counts. Down counter counts from the maximum value to zero value.