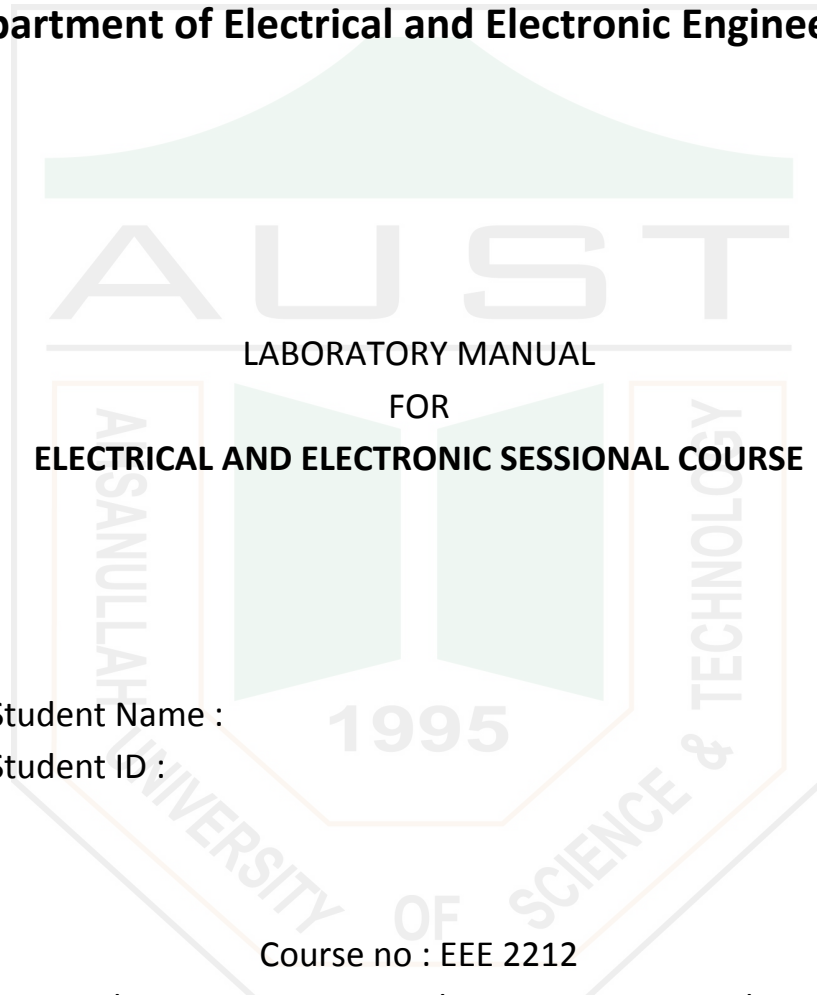# Ahsanullah University of Science and Technology

## Department of Electrical and Electronic Engineering

LABORATORY MANUAL

FOR

**ELECTRICAL AND ELECTRONIC SESSIONAL COURSE**

Student Name :

Student ID :

Course no : EEE 2212

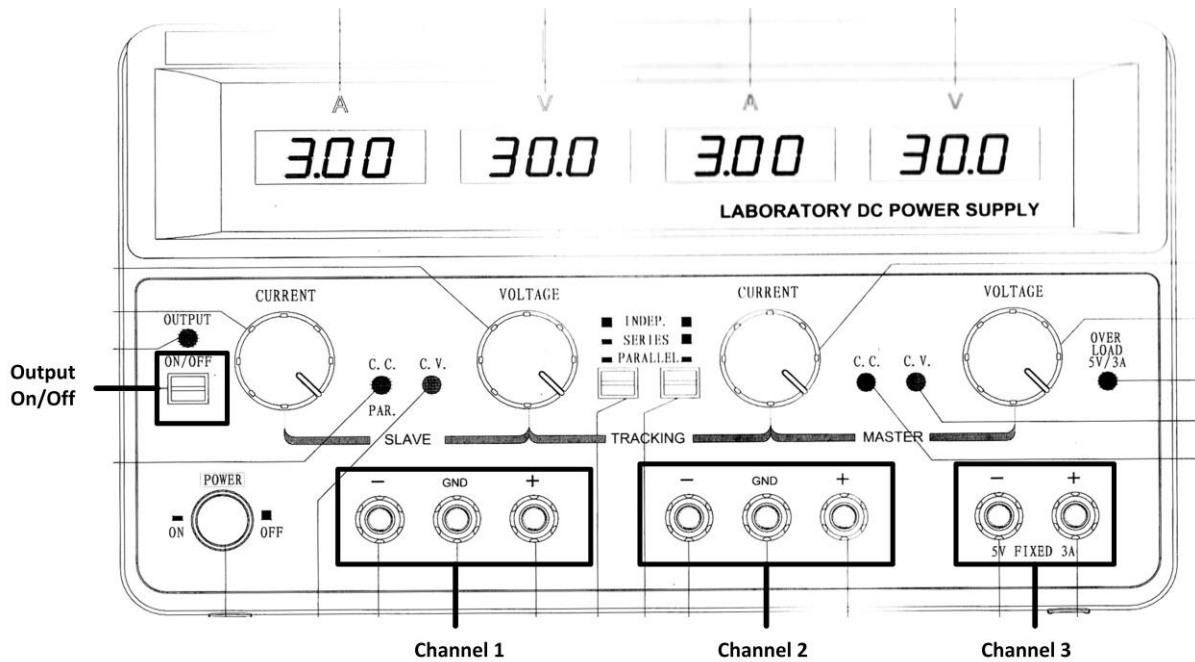Course Title : Measurement and Instrumentation Laboratory

For the students of

Department of Electrical and Electronic Engineering

2nd Year, 2nd Semester

| Experiment no. | 0 |
|---|---|
| **Name of the experiment:** | Introduction to Multi channel DC power supply |



The multi-channel DC supply have 3 different outputs:

| Voltage output at (reference to -) | Value | Type | Output tracking |
|---|---|---|---|
| Channel 1 | 0V to 30V | Variable | Dep. on the mode of operation |
| Channel 2 | 0V to 30V | Variable | |
| Channel 3 | +5 V | Always fixed value | Fixed |

The DC supply have 3 different modes of operation.

1. Independent
2. Series tracking operation
3. Parallel tracking operation

**1.Independent mode:**

When both the tracking push switch are pulled up, the supply is in independent mode. In this mode channel 1 and channel 2 voltages can be varied using the voltage knobs above the individual channels.

**2.Series tracking operation**

When the left TRACKING switch is pushed down and the right switch is pulled up the supply is in series tracking mode.

In this mode the positive terminal of the SLAVE (channel 1) supply output is internally connected to the negative terminal of the MASTER (channel 2) supply as shown in the fig 2 bellow:

In the series tracking mode, the output voltage of both MASTER and SLAVE supplies can be simultaneously varied with one control. The maximum SLAVE supply voltage is automatically set to the same value as the MASTER supply by using the MASTER VOLTAGE controls.
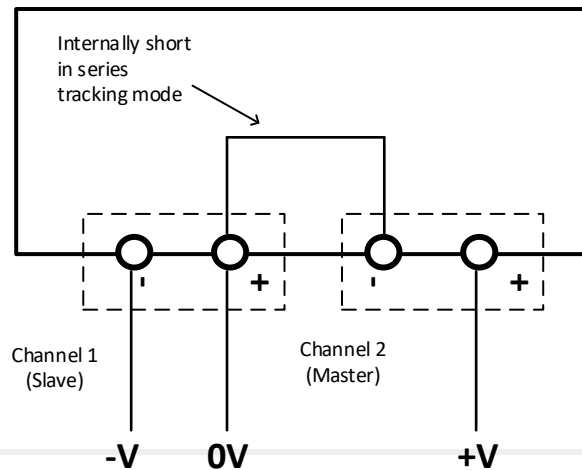
Fig 2. Series tracking mode and duel rail connection

**TIPS for connecting big circuit in breadboard:**

- Use horizontal bus bar in the breadboard for power delivery as shown in the figure.
- Avoid using unnecessary long wire for connections. Large wire cause circuits to be loosely connected, which can cause intermittent connection.
- While connecting a circuit in the breadboard, try to place the part such that the actual circuit closely resemble the schematic diagram. Later on it will be helpful if troubleshooting is required.
- To reduce cluttering if possible, try to place the component such as resistor, capacitor etc. using less wire/no wire (see the LDR and resistor connection in the figure)
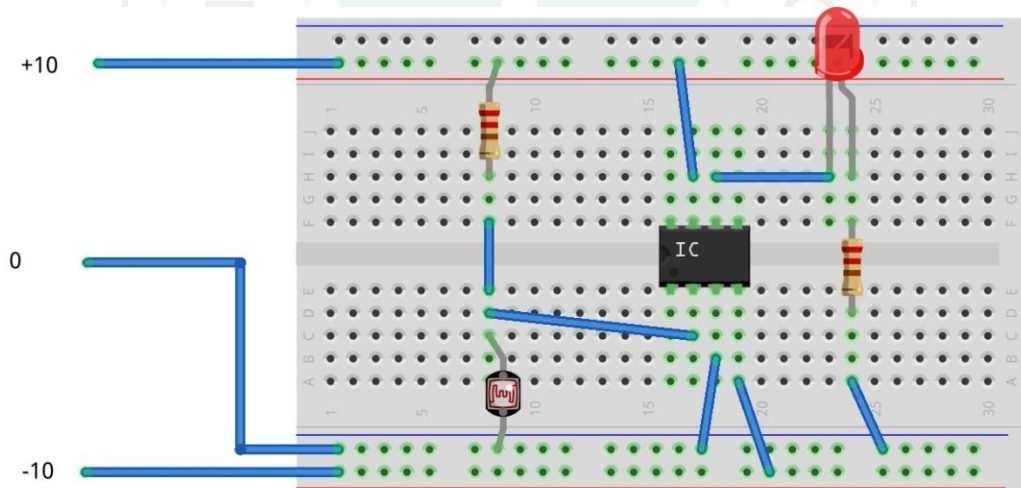


**Important Notes:**

1. To enable the output of the DC supply, push the output on/off button (see fig 1)
2. The GND terminal in the supply is 'earth reference'. Normally we don't need this.
3. Channel 3 is fixed 5V. The output is always enable for channel 3.

**Prepared by**

Md. Aminur Rahman
Dept. of EEE, AUST

| **Experiment no.** | 0 |
|---|---|
| **Name of the experiment:** | Experiment Name: Introduction to Digital Oscilloscope. |

Digital Storage Oscilloscope (DSO) can be controlled using various dial and soft button on the front panel. Front panel controls are divided into several sections. The major sections are a) Vertical b) Horizontal c) Trigger d) Entry etc.

**Vertical Section:**

The oscilloscope in the lab have 2 channels. Channels can be turned on or off by pushing '1' or '2' in the vertical section.  Each channel has its own voltage/div (big dial) and vertical position dial (small dial).

The big dial (voltage/div) is used for changing the voltage/div of ch-1 or ch-2. The voltage/div setting can be seen on top left side of the screen ①. By pushing the dial, the voltage/div goes into "Fine adjust" mode which can be used to change the voltage/div setting gradually.

The vertical position dial is used for changing the ground level of channel 1 or 2. The vertical position setting is visible when it is rotated (oscilloscope will display it as "offset voltage"). Trace position can also be viewed in the left side of the trace ②. By pushing the vertical position "dial" the trace (offset voltage) can be zeroed.

**Horizontal section:**

The big dial is used for changing the time/div setting of waveform/waveforms. The time/div information can be viewed on the top of the screen ③.

The small dial is used to "move" the waveform in horizontal direction (time delay). The horizontal delay information can be seen on top of screen ④. Also the horizontal triggering position can be seen as a notch on top of the screen ⑤. Waveform delay can be zeroed by pushing the dial.
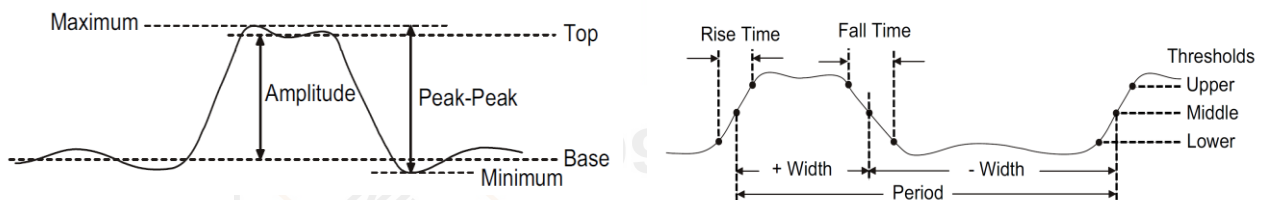
**Different Measurements:**



Fig. 1: Different measurement parameter

Peak-Peak: The peak-to-peak value is the difference between Maximum and Minimum values. Maximum: Maximum is the highest value in the waveform display. Minimum: Minimum is the lowest value in the waveform display.

**Adding a new measurement:**

1. Push *Meas* button in the Measure section of the DSO. Measurement Menu will appear on the right side of the screen (Fig. 2: Left).
2. From the *Source* submenu select which channel.
3. Select the measurement type from *Type* submenu (Fig. 2: right). Use the *entry dial* to highlight and then push the dial to add a measurement.
4. The measurement will appear on the bottom of the display.
5. Use *Clear meas* in measurement menu to delete one or more measurements from the bottom of the screen.
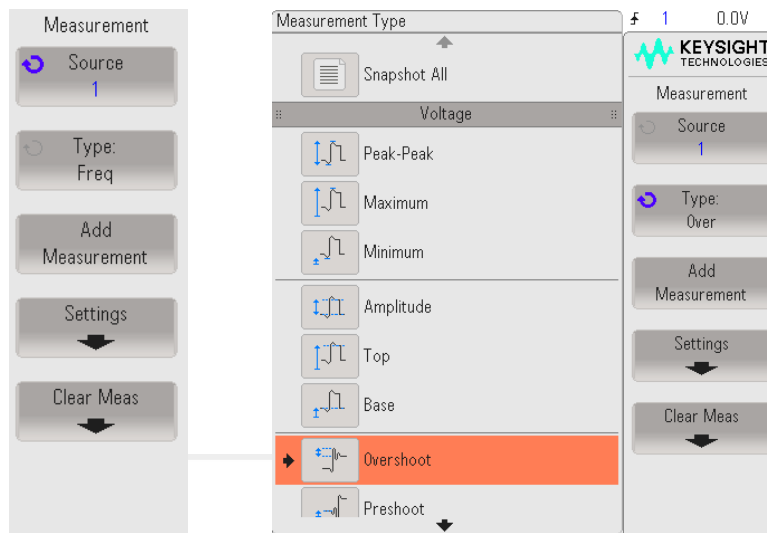
Fig. 2: (Left): Measurement menu (Right):Different measurement type

**Triggering a waveform:**

Most important part of the triggering section is *Level* adjustment dial and the *Trigger* button.

1. Push *Trigger* button. Trigger menu will appear left of the screen (Fig. 3: Left).
2. Select which wave-from the oscilloscope will use for triggering by selecting C*hannel 1* or C*hannel 2* in the *source* submenu Fig. 3: Right (if there is only one waveform and channel 1 is connected, *Channel 1* need to be selected). Oscilloscope will show which channel it is triggering on top of the display ⑥.
3. Use the *Level* dial such that the *trigger level* line intersects the wave from ⑦. Oscilloscope will show the trigger voltage level on top of the display ⑧.
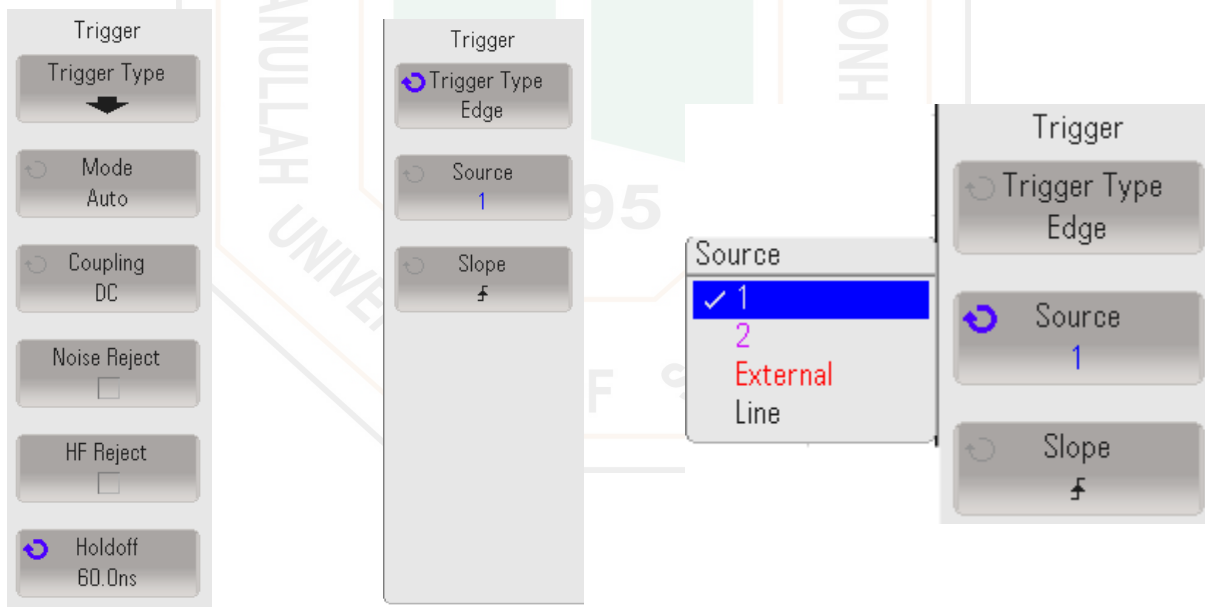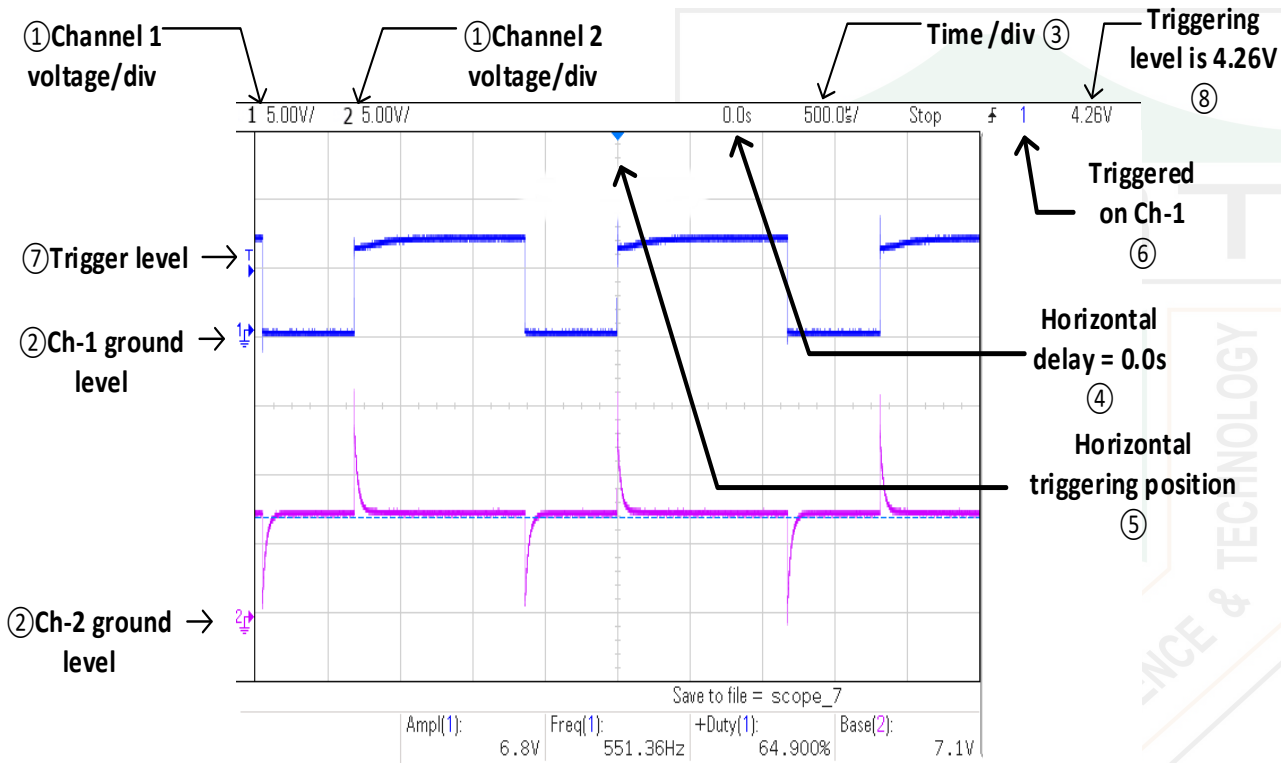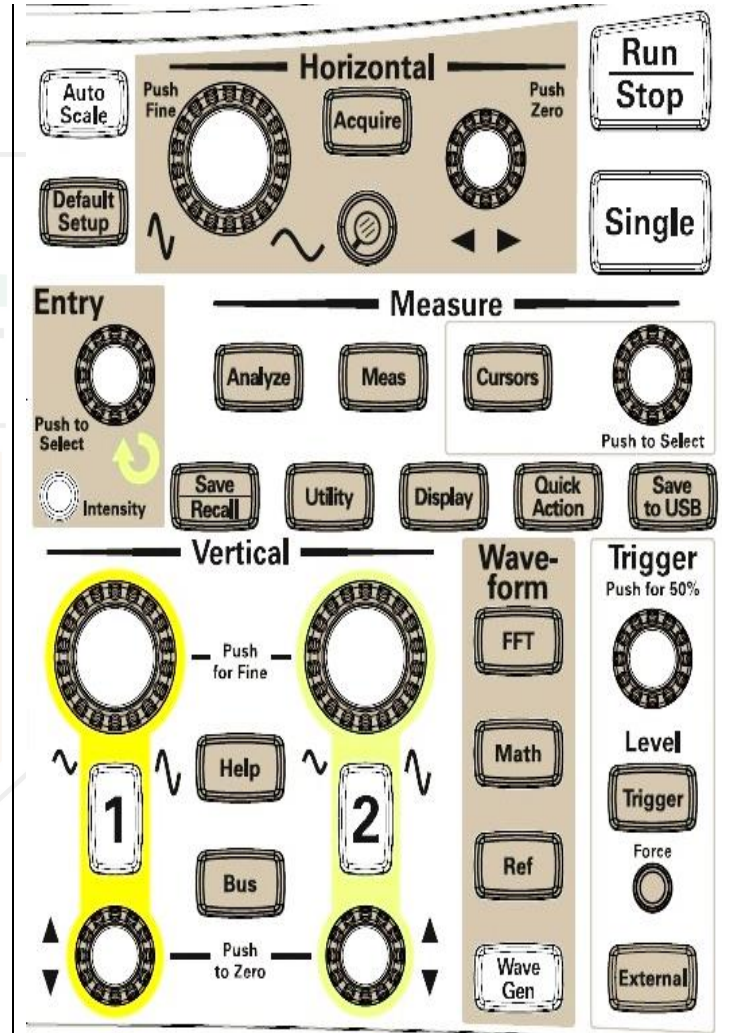


Fig. 3 (Left):*Trigger* Menu (Middle):*Trigger Type* submenu (Right):*Source* submenu

**Prepared by**

Md. Aminur Rahman
Dept of EEE, AUST

**Typical DSO screen**

①Channel 1 voltage/div

①Channel 2 voltage/div

Time /div ③

Triggering level is 4.26V ⑧

⑦Trigger level →

②Ch-1 ground level

②Ch-2 ground level

Triggered on Ch-1 ⑥

Horizontal delay = 0.0s ④

Horizontal triggering position ⑤

1 5.00V/   2 5.00V/        0.0s   500.0$\frac{\mu}{s}$/   Stop   ƒ 1   4.26V

Save to file = scope_7

Ampl(1):      Freq(1):      +Duty(1):      Base(2):
      6.8V        551.36Hz        64.900%        7.1V

**DSO Front panel**

**Experiment no.:** 03

**Name of the experiment:** Introduction to Thermistor, Light Dependent Resistor (LDR) and Photodiode.

**Objective:**
Study the characteristics and principle of operation of Thermistor, Light Dependent Resistor (LDR) and Photodiode.

**Theory:**

**Thermistor:**

Thermistor is a contraction of the term thermal resistors. It is generally composed of semiconductor materials. Most thermistors have a negative coefficient of temperature resistance i.e. their resistance decreases with increase of temperature. Thermistors are widely used in applications which involve measurements in the range of $-60^\circ$ to $15^\circ$C. The resistance of thermistors ranges from $0.5\Omega$ to $0.75M\Omega$. The thermistor exhibits a highly non-linear characteristic of resistance versus temperature. Figure1 shows typical resistivity vs. temperature characteristic for negative temperature coefficient thermistors.
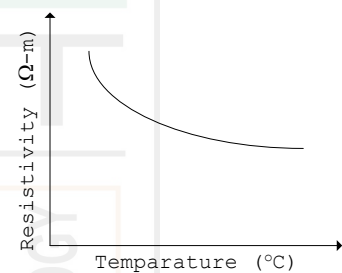


**Figure-1**

**Light Dependent Resistor:**

In this type of device, the electrical resistance of the material varies with the amount of light energy striking it. A particular photo cell illumination characteristic is shown in figure2.
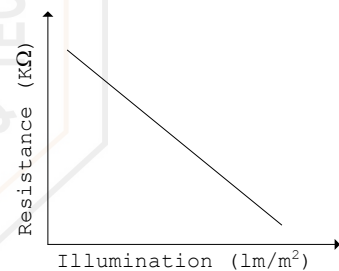


**Figure-2**

**Photodiode:**

A reverse biased photodiode passes only a small leakage current if the junction is not exposed to appropriate light. Under illumination, the current through the diode is almost directly proportional to the light intensity. In this experiment infrared photodiode is used. So the light source is an infrared LED.

**Apparatus:**

1. IC LM339
2. Resistor: 1K(9 pcs.), 220Ω, 100K
3. Potentiometer (100K)
4. Thermistor (NTC)
5. LDR
6. Photodiode
7. Infrared LED
8. Trainer board
9. Oscilloscope
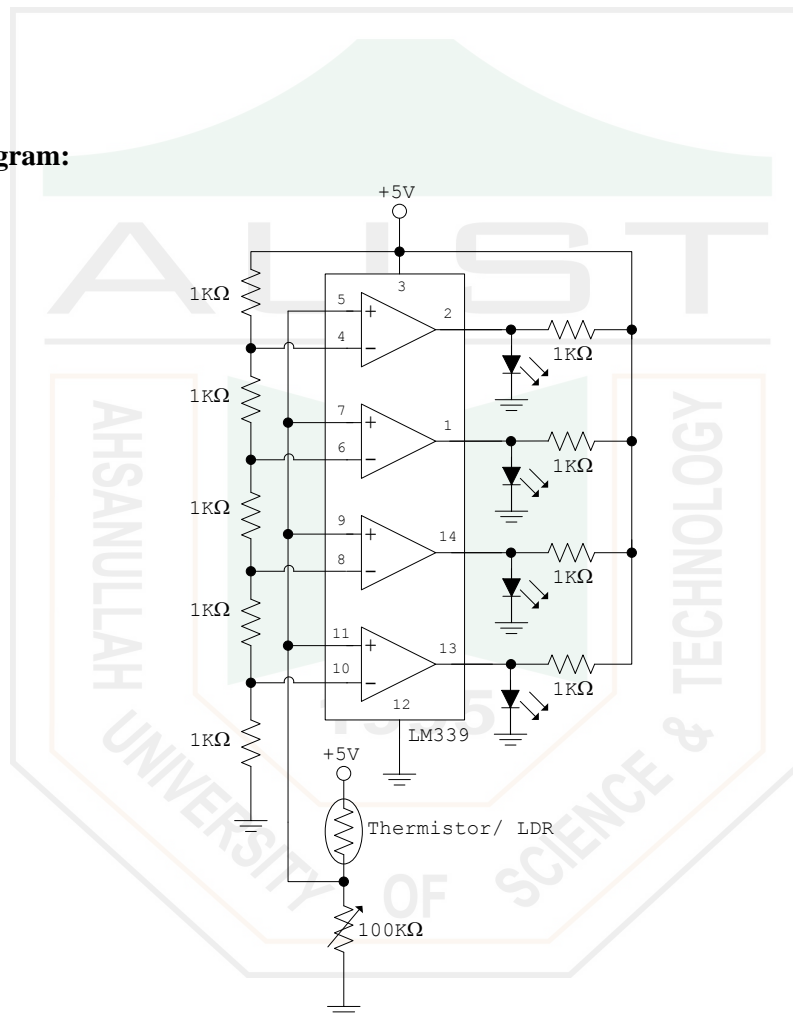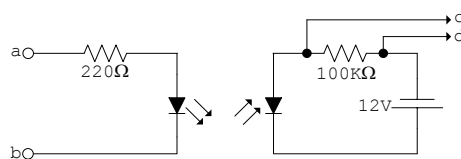10. DMM

**Circuit Diagram:**



**Figure3.**



**Figure4.**

**Procedure:**

1. Measure the resistance of the thermistor at room temperature and note it.
2. Measure the resistance of the LDR at dark. Also note the resistance of the same with maximum available light intensity.
3. Construct the circuit as shown in figure3 with the thermistor.
4. Adjust the potentiometer so that the voltage across the pot. is below 1V. Now Apply heat to the thermistor and observe the behavior of the LEDs connected to the circuit.
5. Stop heating when all the LEDs are ON. Observe the LEDs until the temperature of the thermistor reaches to room temperature.
6. Replace the thermistor with the LDR.
7. Keep the LDR at dark and adjust the pot. so that the voltage across the pot. is below 1V. Gradually increase the amount of light on the LDR and observe the behavior of the circuit output.
8. Construct the circuit as shown in figure4. Keep the infrared LED and photodiode face to face.
9. Apply TTL pulses of 1khz (approximately) between terminals 'a' and 'b'. Observe the waveshapes across the 100K resistor.
10. Increase frequency of the applied pulses and note the effects. Increase the distance between LED and photodiode. Observe the changes in waveshapes.

**Report:**

1. Explain the operation of the circuit shown in figure3.
2. List common applications of thermistor, LDR and photodiode.
3. Design a circuit using LDR to turn 'on' a lamp at the evening and turn it 'off 'at dawn.

**Reference:**

1. Robert F. Coughlin, Frederick F. Driscoll; *Operational Amplifiers and linear Integrated Circuits*; Prentice-Hall of India Private Limited, New Delhi, 4th edition, 1996.
2. A. K. Sawhney; *A course in- Electrical and Electronics measurement and Instrumentation*; Dhanpat Rai and Co. (Pvt.) Ltd., Delhi, India; Eleventh edition, 1998.

**Experiment number:** 04

**Name of the experiment:** Phase angle measurement with Polarized logic circuits.

**Pre-lab**

1. How to measure phase shift in oscilloscope and translate the value into phase angle.
2. How two DC power supply can be connected to output dual output (+15V, 0V and-15V)
3. Review the theory about average and Root Mean Square (R.M.S.) value.
4. Review how to view two different waveforms simultaneously in oscilloscope.
5. How to measure DC voltage from oscilloscope.

**Theory:**

Here, in the circuit under consideration, the only ac signal at the input is divided in two signals by using of a capacitor in one path. Therefore, these signals have a certain phase shift. The op-amps work as comparator and generate the polarized signals. These signals are added by another op-amp of gain 50% which practically XNORS the signals. The output is then converted to a dc signal by a bridge rectifier.

$$V_{DC} = \frac{1}{T} \int_0^{\theta+\phi} V_P \, dt$$

$$\Rightarrow V_{DC} = \frac{1}{180°} \left[ \int_0^{\phi} 0 \, dt + \int_{\phi}^{\theta+\phi} V_P dt \right]$$

$$\Rightarrow V_{DC} = \frac{1}{180°} [0 + V_P(\theta + \phi - \phi)]$$

$$\Rightarrow V_{DC} = \frac{V_P \, \theta}{180°}$$

$$\theta = \frac{V_{DC}}{V_P} \times 180°$$

Also,

$$T = \phi + \theta = 180°$$

Phase angle, $\phi = 180° - \theta$

Here,
$V_P$= Peak value
$V_{DC}$ = Average value
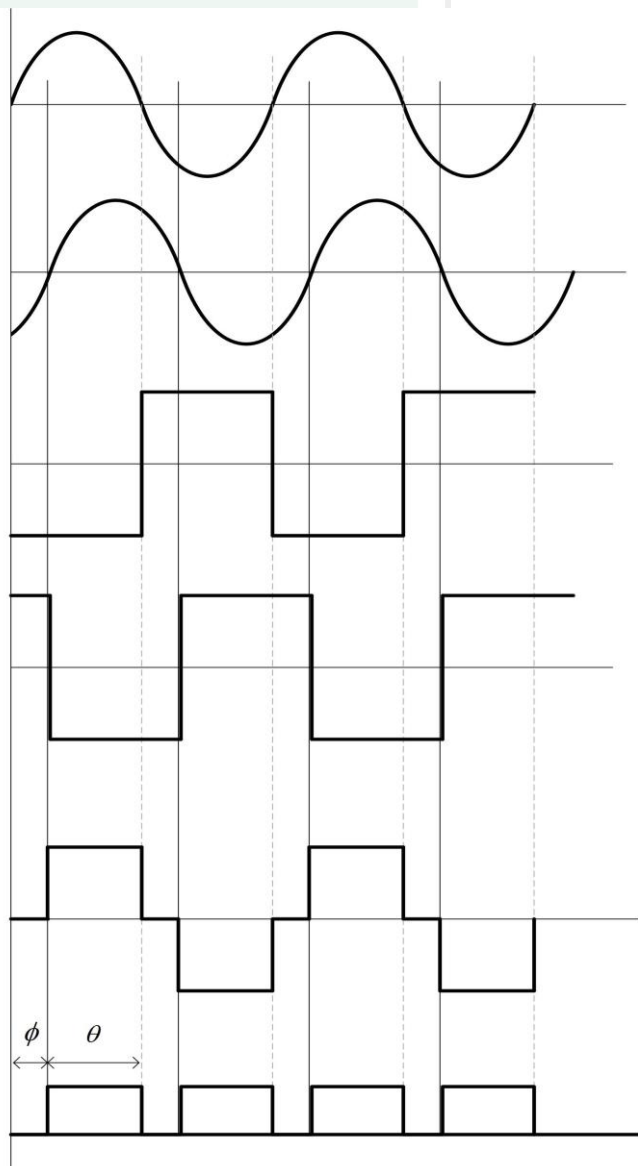T = Time period (180 deg)



**Figure 1: Waveforms at different point**

## Equipment:

1. LM741                3pcs
2. Resistors            100KΩ (2pcs), 50 KΩ (1pc), 10 KΩ (1pc), 8KΩ(1pc)
3. Capacitor            0.22µf(1pc)
4. Diodes               4pcs
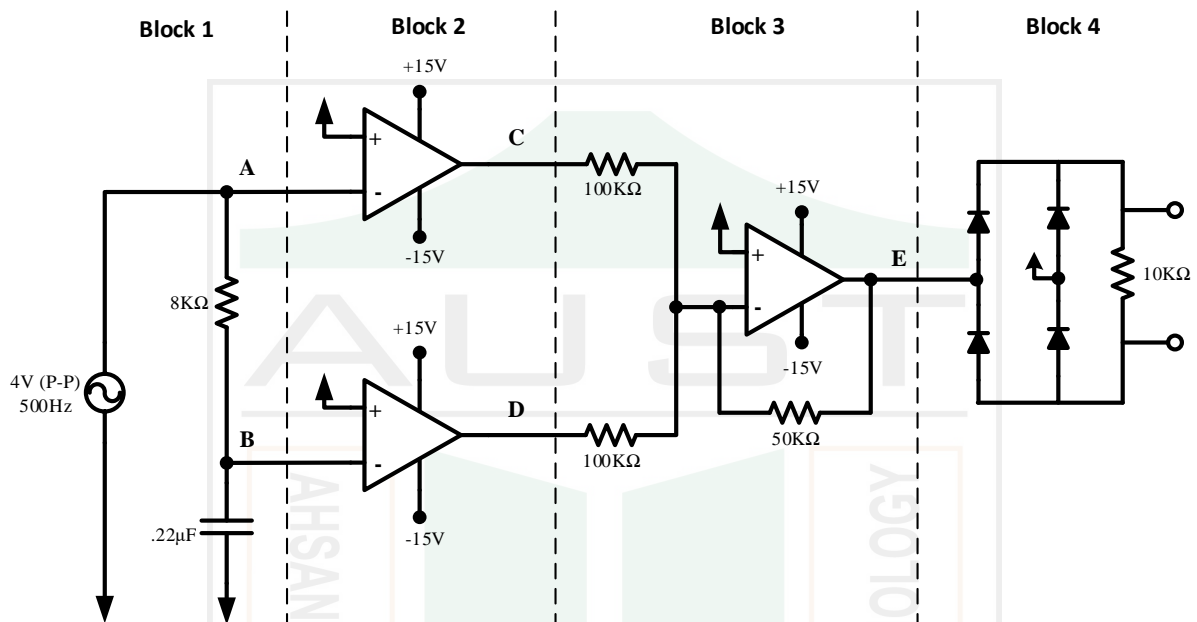5. Trainer board
6. Oscilloscope
7. Signal Generator



**Figure 2: Polarized logic circuits**

## Procedure:

1. Construct block 1 and observe the voltage waveform at A and B point using the two channels of the oscilloscope.
2. Measure the phase difference between point A and B
3. Construct block 2 and observe the voltage waveform at C and D point using the two channels of the oscilloscope. Does the wave shape resemble the waveform in figure 1?
4. Construct block 3 and observe the voltage waveform at E point. Again observe does the wave shape resemble the waveform in figure1 or not.
5. Construct block 4 and observe the voltage wave shape across the resistor.
6. Fill up the following table
7. Vary the plot, repeat the step 2 & 3 and tabulate the value of output voltage (dc) and phase angles for various pot settings.

## Report:

1. Draw the truth table for the experimental polarized logic circuit.
2. Draw the phase angle vs voltage output curve.
3. Explain the working principle of the experimental circuit.

**Experiment No: 05**
**Name of the Experiment: Study of Basic Differential Amplifier.**

**THEORY:**

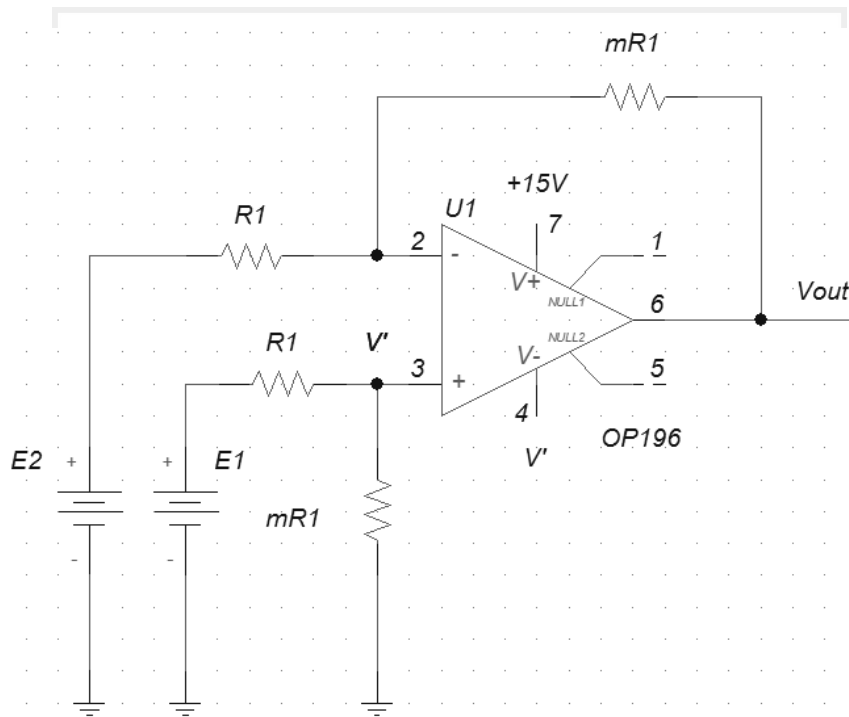Figure 01 shows the basic differential amplifier



**Figure 01:** basic differential amplifier

If $E_1$ is replaced by a short circuit, $E_2$ sees an inverting amplifier with a gain of $-m$. Therefore, the output voltage due to $E_2$ can be found by,

$V_0/mR_1 = -E_2/R_1$
$V_0 = -mE_2$

If $E_2$ is replaced by a short circuit, $E_2$ sees an non-= inverting amplifier with a gain of $-m$. Therefore, the output voltage due to $E_1$ can be found by,

$V'/R_1 = (V_0-V')/mR_1$
$V_0 = (1+m)V'$
$= (1+m) \times m/(1+m)E_1$
$= mE_1$

The output due to the two amplifier when working simultaneously,

$V_0 = m(E_1-E_2)$

## APPARATUS:

1. Op-amp(1 unit)
2. Resistors 1k(2 Nos.), 10k (2 Nos.) and 33k(1 No.)
3. Potentiometer (50k)
4. Multimeter (1 unit)
5. Trainer Board and
6. DC Power Supply

## CIRCUIT DIAGRAM:
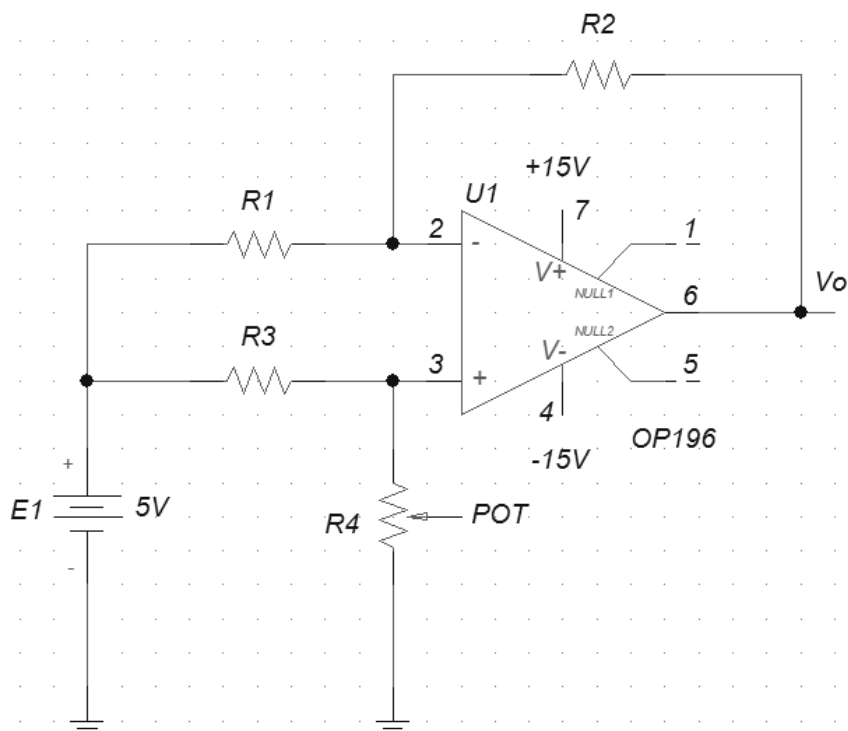


**Figure 02**: Circuit diagram for experiment

## PROCEDURE:

1. Measure the resistances and set up the circuit as shown in the figure 02. Use $R_1=R_3=10k$, $R_2=33k$ and 50k potentiometer as $R_4$.

2. Vary the potentiometer so that output voltage becomes zero. In this connection the input voltage is called the common-mode-input voltage, $E_{cm}$. Now V0will be zero if the resistor ratios are equal. This causes the common-mode-voltage gain, $V_0/E_{cm}$ to approach zero. It is the characteristic of a differential amplifier that allows a small signal voltage to be picked out of a larger noise voltage.

3. Set up the circuit as follows:
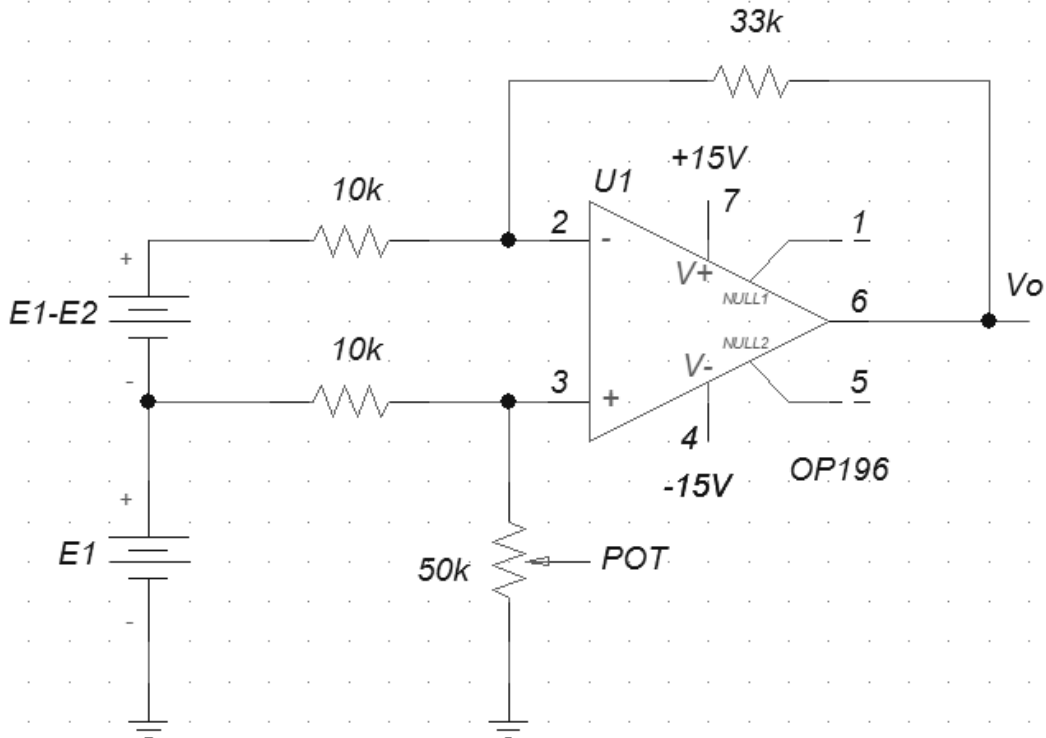


**Figure03:** Setup

4. Vary the $(E_1-E_2)$ from 0V to 6V with a step of 0.5 V and measure $V_0$. Record the data in the following table.

| Input Voltage $(E_1-E_2)$ | Output Voltage, $V_0$ |
| --- | --- |
| 0.5 | |
| 1.0 | |
| 1.5 | |
| 2.0 | |
| 2.5 | |
| 3.0 | |
| 3.5 | |
| 4.0 | |
| 4.5 | |
| 5.0 | |
| 5.5 | |
| 6.0 | |

5. Plot the $V_0$ versus $(E_1-E_2)$

6. Change $E_1$ to -5V and repeat the procedure with $R_1=R_2=1k$, $R_3=10k$ and $R_4$ with 50k potentiometer. Take reading in the following table:

13

| Input Voltage ($E_1$-$E_2$) | Output Voltage, $V_0$ |
|---|---|
| 0.5 | |
| 1.0 | |
| 1.5 | |
| 2.0 | |
| 2.5 | |
| 3.0 | |
| 3.5 | |
| 4.0 | |
| 4.5 | |
| 5.0 | |
| 5.5 | |
| 6.0 | |

**REPORT:**
1. Calculate the theoretical gain of the basic differential amplifier for both the cases.
2. Write down the disadvantages of a basic differential amplifier.
3. Calculate the value of the input resistance for $E_1$-$E_2$=3V for each case $E_1$=-5V and $E_2$=+5V.
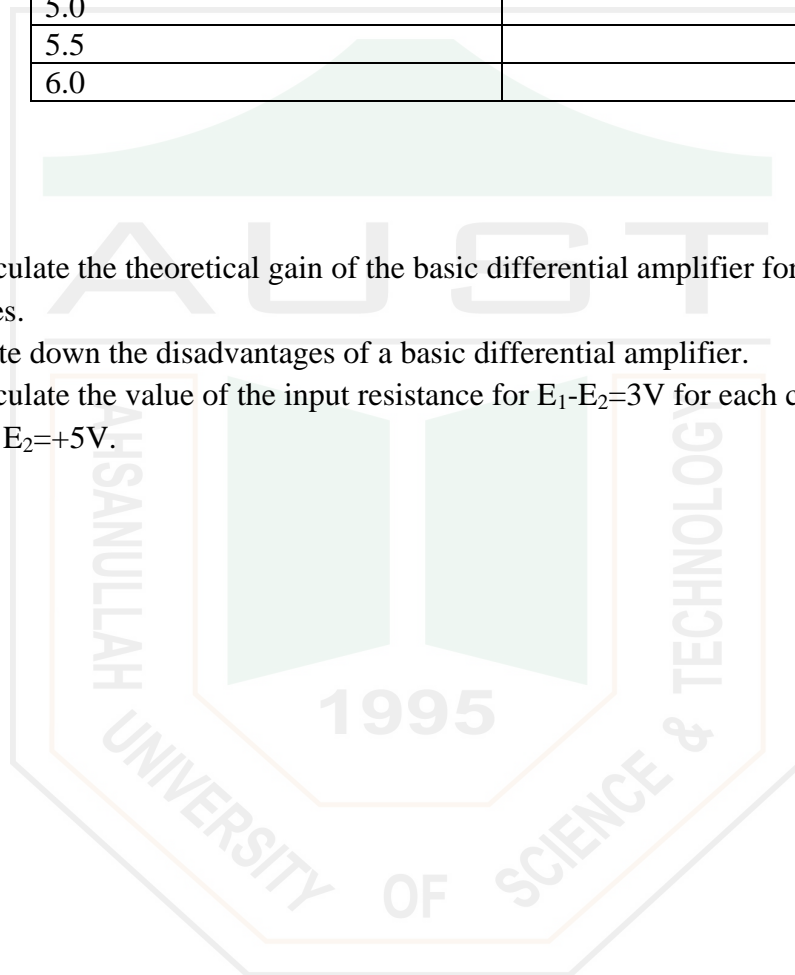
**Experiment number:**06

**Name of the Experiment:** Precision Rectifiers in Signal Processing.

**Objective:**
To study full-wave precision rectifiers for signal processing and measurement purposes.

**Theory:**
To measure an ac voltage precisely, a precision rectifier should be used to get full wave rectification of the input signal. After filtering out the higher harmonics using a low-pass filter, this rectified signal could be interlaced to a digital voltmeter chip for displaying the value of the input ac signal.

The circuit under consideration uses equal resistors and has an input resistance equal to 2.2k. For positive half cycle, the D1 diode conducts; input current divides in two paths and the second op-amp act as an inverter. The output voltage is therefore positive for both cycles giving the absolute value of the input signal in rectified form.

**Circuit Diagram:**



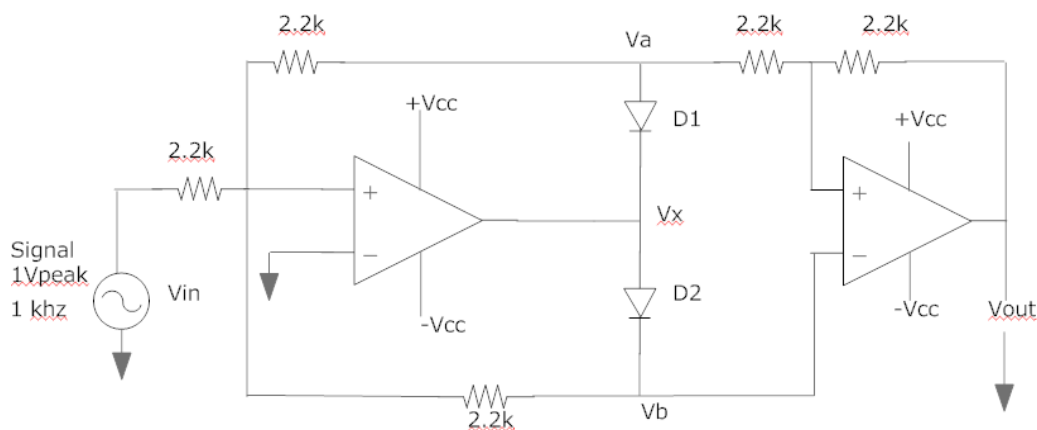**Fig:Precision full wave rectification**

+ve half cycle: Vx = -ve , Va = Vb = +ve , D1 diode conducts , current is divided at Va . So, second op-amp is in non-inverting mode.

-ve half cycle: Vx = +ve, Va = Vb = -ve, D2 diode conducts, current is added at Vb . So second op-amp is in non-inverting mode.

Feedback: $R_{in}= R_{out} = 2.2k$ ; gain feedback(like buffer), just compensates the input.

**Apparatus:**
1. IC741          2pcs
2. Resistors      2.2k-5 pcs
3. Diode          2pcs
4. Trainer board
5. Oscilloscope
6. Signal generator

**Procedure:**
1. Connect the circuit as shown in the figure. Take +ve Vcc and –Vcc , +20V and -20V respectively.
2. Adjust the signal generator.
3. Record the voltage wave forms at $V_{in}$ , $V_a$, $V_b$, $V_x$ and $V_{out}$

**Report:**
1. Draw the voltage wave form at $V_{in}$, $V_a$, $V_b$, $V_x$ and $V_{out}$
2. Explain the working principle of the experimental circuit.
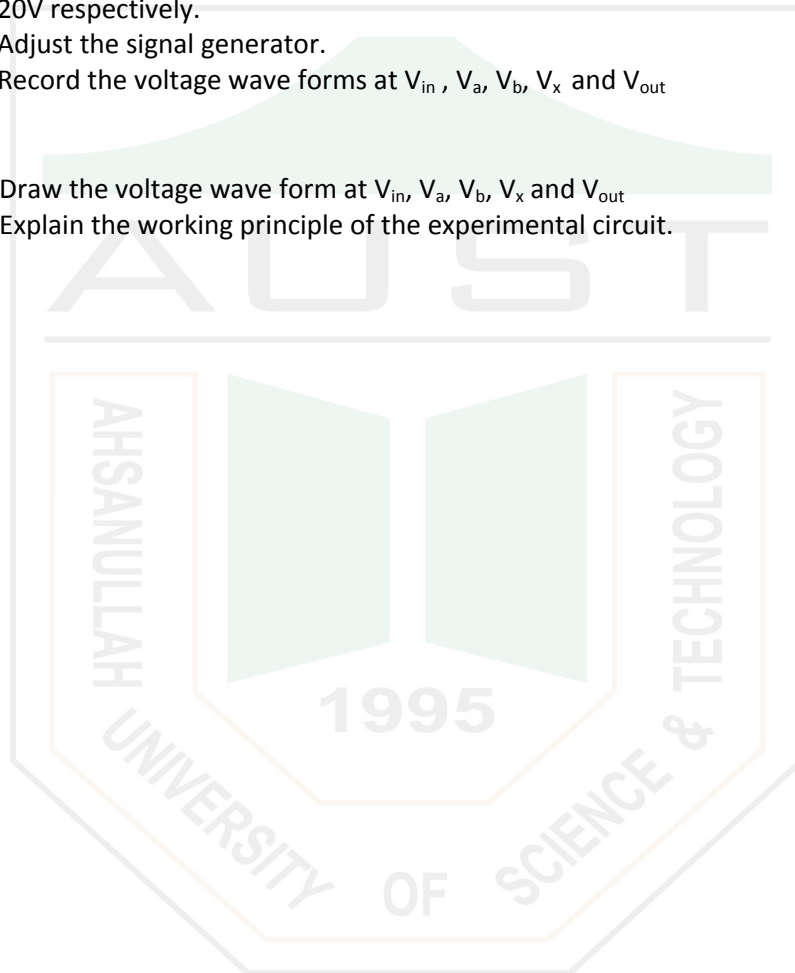
**Experiment number:** 07

**Name of the experiment:** Study of Instrumentation Amplifier

### Introduction:

The instrumentation is one of the most useful, precise, versatile amplifiers available today. It is made from three Op-Amps and seven resistors. It is actually made by connecting buffer amplifier to differential amplifier. Op-Amp A3 and its four equal resistors R from a differential amplifier with gain 1. Only the A3 resistor have to be matched. The primed resistor R` can be made variable to balance out any common mode voltage as shown in figure. The resistor aR is used to set the gain according to the following equation where a=aR/R.

$$V_0 / (E_1 - E_2) = 1 + (2/a)$$

### Circuit Diagram:
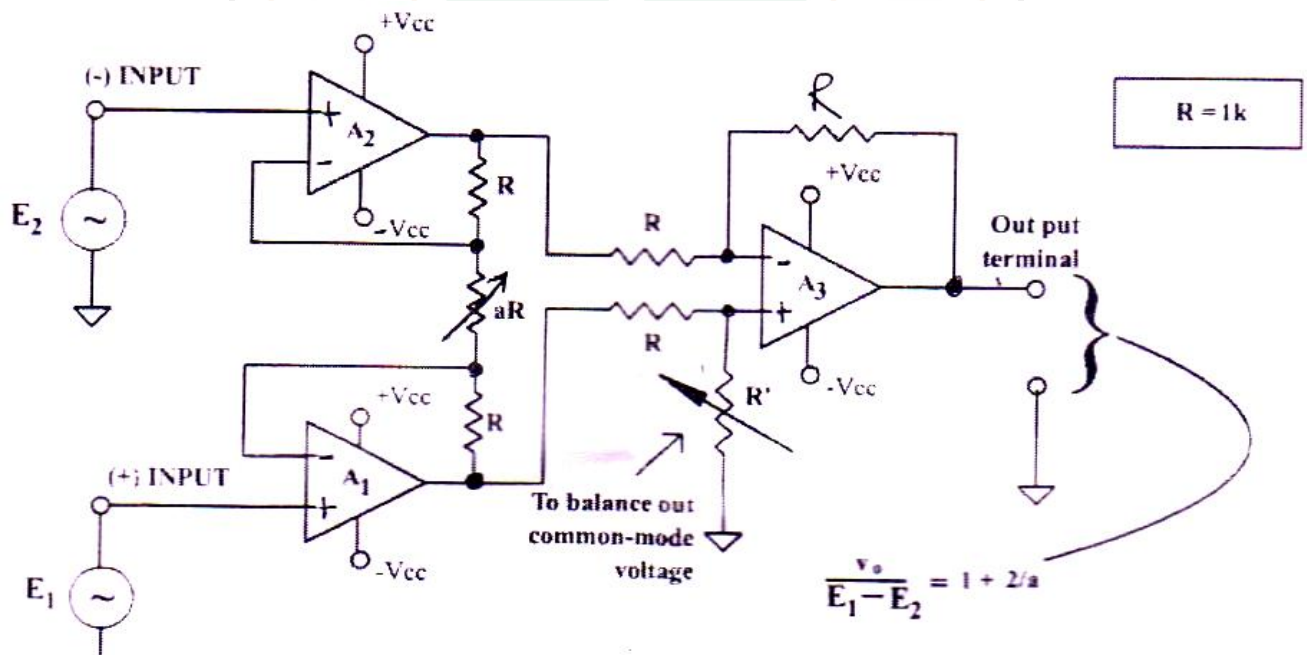


Fig: Instrumentation Amplifier

**Apparatus:**

1. IC 741          3 pcs
2. Resistors       1k-5 pcs
3. Pot             1k- 2 pcs
4. Trainer Board
5. Oscilloscope
6. Signal Generator

**Procedure:**

1. Connect the circuit as above. Use R=1K
2. Connect E2 to ground and provide a 5v P-P voltage at E$_1$.
3. Calculate *a* for a gain of 2.
4. Operate the *a*R pot again to get a gain of 2.
5. Draw the voltage wave shape at the output of the three Op-amps.
6. Compare the two values of *a*.

**Report:**

1. Describe the operation of the circuit and theoretically prove the equation of the instrumentation amplifier as given in the instruction.
2. Why it is a kind of a differential amplifier and how could we use it as an amplifier?
3. What are the advantage of instrumentation amplifier?

**(a)Prelab:**

1. Read theory of 555 timer operation and how the operational state diagram is generated.
2. How to measure using DSO

**(b)Objective:**

1. Understand the working principle of the 555 timer.
2. Understand the astable and monostable operation of the timer.
3. To measure an unknown capacitance using mono-stable timer.

**(c)Theory:**
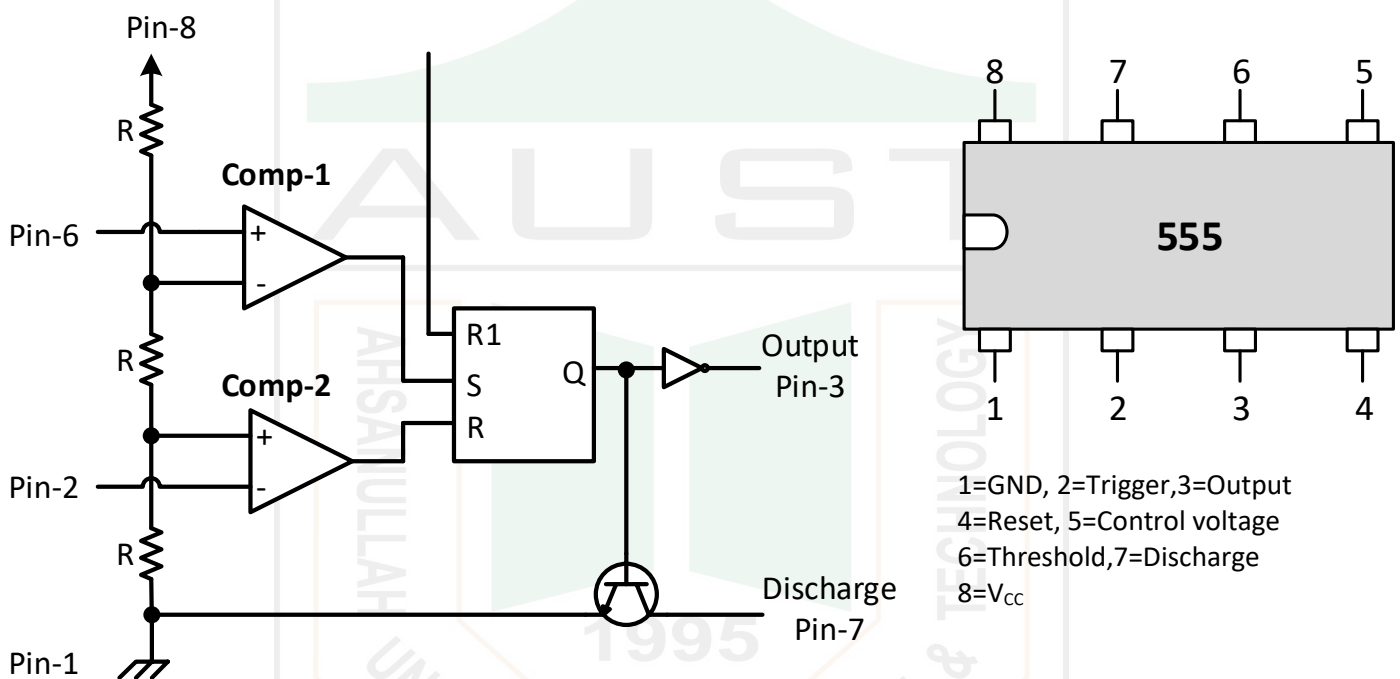
**555 Timer operation:**



**Fig 1: (Left)functional block diagram and (right) pin diagram of 555 timer IC**

A 555-timer's operation can be modeled using op-amp (used as comparators), SR flipflop, NPN transistor and three equal value resistances as shown in Fig 1.

From the left of the block diagram it can be seen that in-between pin 8 and pin 1 there are three resisters with equal value. Also as pin 8 is connected to Vcc each resister will drop 1/3 Vcc voltage. as a result, $Comp\ 1\ V_-$ will see 2/3 V$_{cc}$ and $Comp\ 2\ V_+$ will see 1/3 V$_{cc}$. Output of two comparators are connected to S and R pin of the SR flipflop. The output (Q) of the flipflop is connected to a inverter which will invert the output at pin 3 . Pin 7 of the timer is connected to a NPN transistor's collector and the emitter is connected to the ground. The base is connected to the Q pin of the flipflop and the transistor works as a switch in this setup.

As the op-amps are in comparators mode, we know if $V_+ > V_- = +V_{Sat} = ouput\ high$ or $V_+ < V_- = -V_{Sat} = ouput\ Low$. So comp-1 outputs comparing voltage of V$_6$ and 2/3V$_{cc}$ and similarly comp 2 outputs by comparing the voltage of V$_2$ and 1/3V$_{cc}$ . Based on the output of the comparators the flipflop outputs (Q) according to the truth table below:
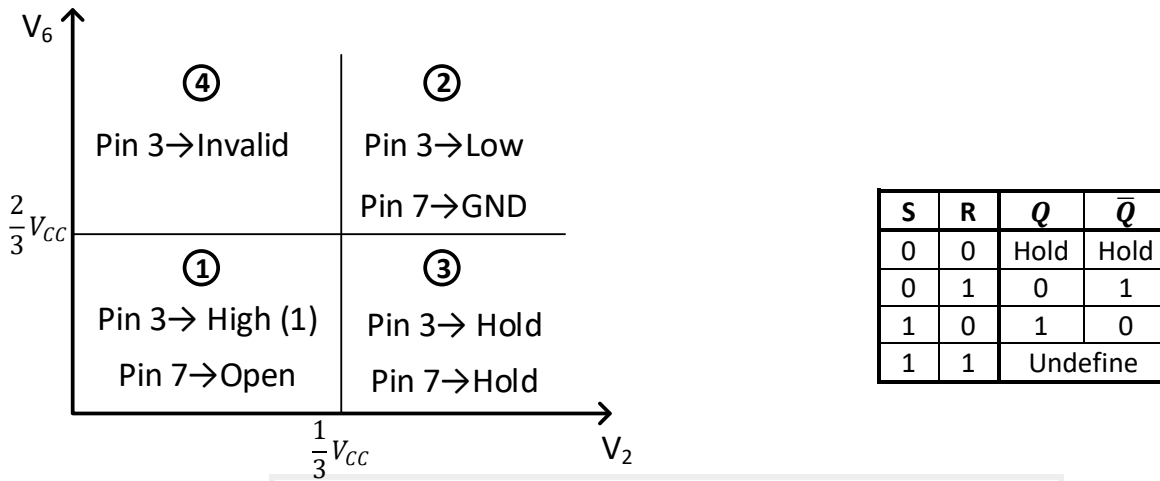
Fig 2: (Left) Operational state of the timer (Right) Truth table of SR flipflop

The pin 3 of the timer will output the inverted output of the flipflop. Based on the voltage at V2 and V6 pin the timer's operation can be shown using the following diagram.

**Astable Operation:**



Fig 3: Astable operation of the timer with waveform

| Time | $V_2 > \frac{1}{3}V_{CC}$ | $V_6 > \frac{2}{3}V_{CC}$ | Operational State | Output Pin 3 | Discharge Pin 7 |
|------|------|------|------|------|------|
| $0 - t_1$ | N | N | 1 | 1 | Open |
| $t_1 - t_2$ | Y | N | 3 | 1 (Hold) | Open (Hold) |
| $t_2{}^+$ | Y | Y | 2 | 0 | GND |
| $t_2 - t_3$ | Y | N | 3 | 0 (Hold) | GND (Hold) |
| $t_3{}^+$ | N | N | 1 | 1 | Open |
| $t_3 - t_4$ | Y | N | 3 | 1 (Hold) | Open (Hold) |
| $t_4{}^+$ | Y | Y | 2 | 0 | GND |
| $t_4 - t_5$ | Y | N | 3 | 0(Hold) | GND (Hold) |

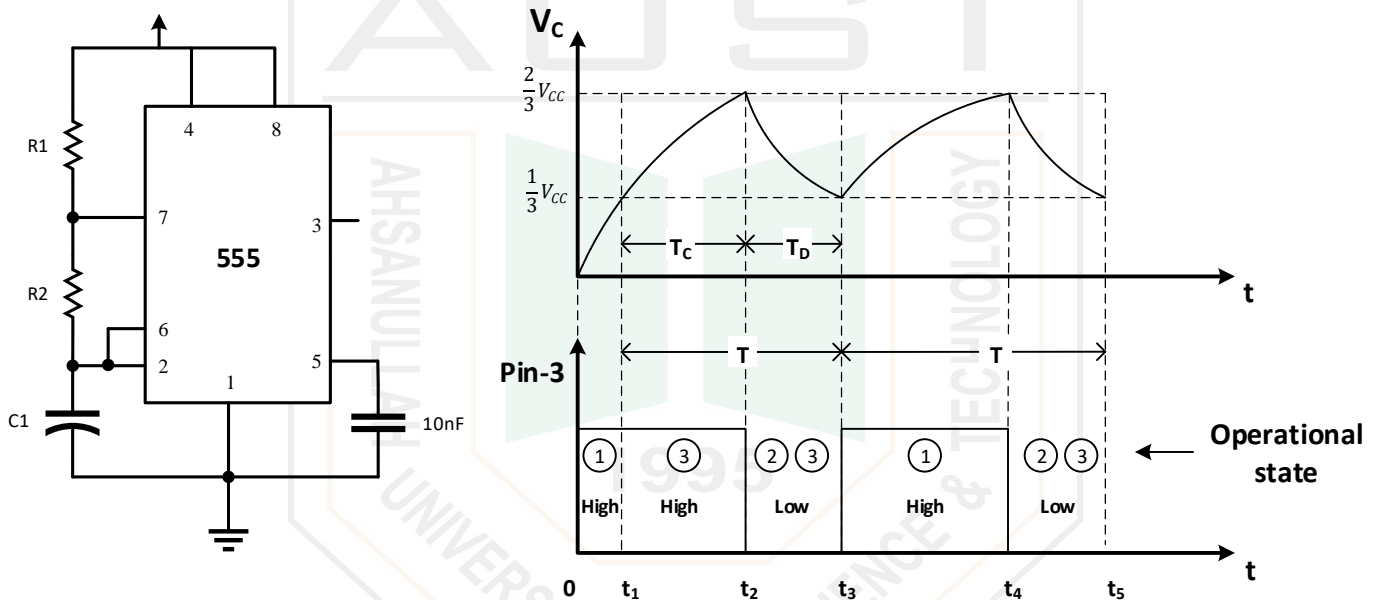**Monostable Operation:**
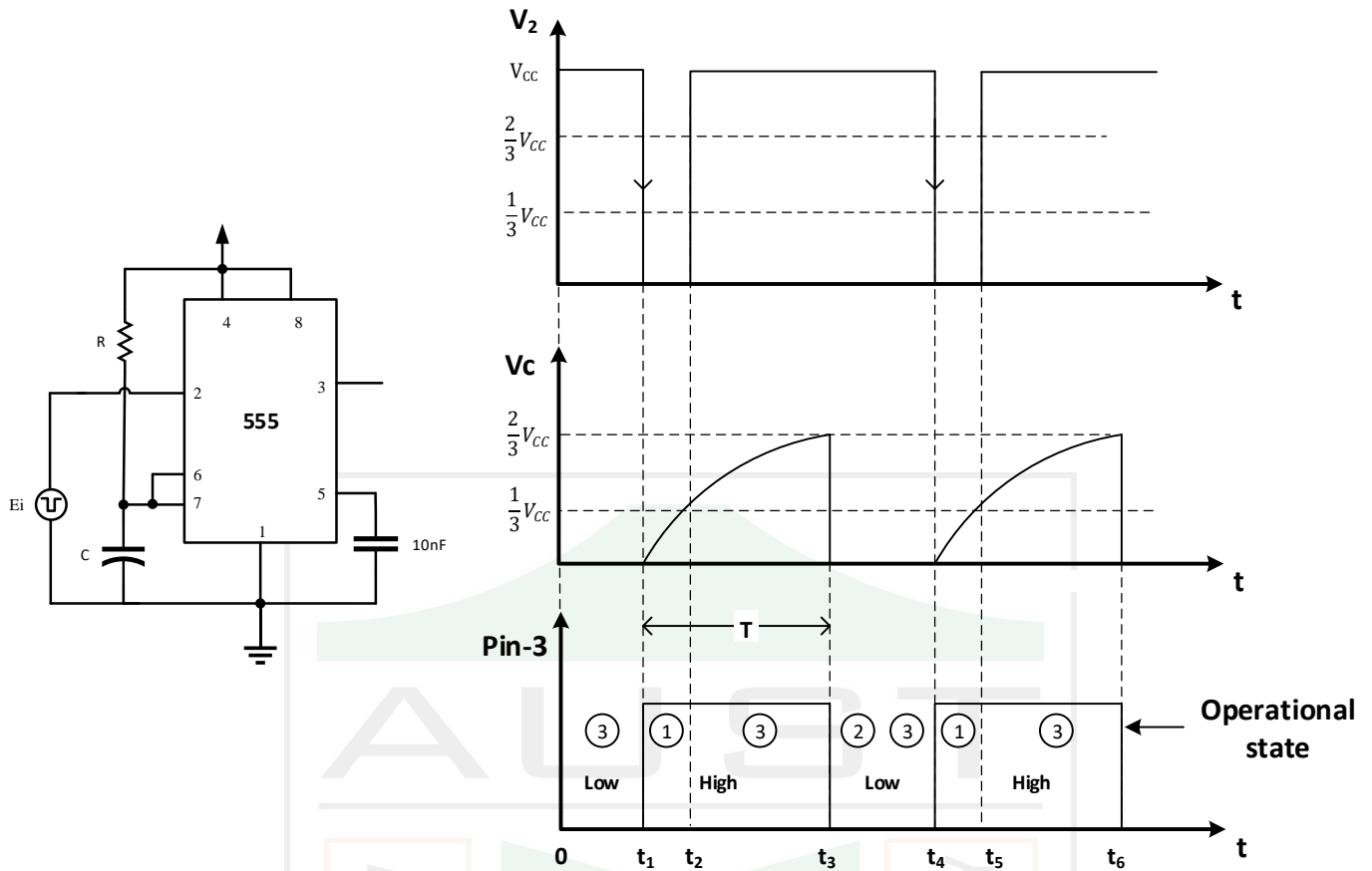


**Fig 4: Monostable operation of the timer with waveform**

| Time | $V_2 > \dfrac{1}{3}V_{CC}$ | $V_6 > \dfrac{2}{3}V_{CC}$ | Operational State | Output Pin 3 | Discharge Pin 7 | Comment |
|------|------|------|------|------|------|------|
| $0 - t_1$ | Y | N | 3 | 0 (Hold) | GND (Hold) | Initially Let output is Low |
| $t_1 - t_2$ | N | N | 1 | 1 | Open | |
| $t_2 - t_3$ | Y | N | 3 | 1 (Hold) | Open (Hold) | |
| $t_3{}^+$ | Y | Y | 2 | 0 | GND | |
| $t_3 - t_4$ | Y | N | 3 | 0 (Hold) | GND (Hold) | |
| $t_4{}^+ - t_5$ | N | N | 1 | 1 | Open | |
| $t_5 - t_6$ | Y | N | 3 | 1 (Hold) | Open (Hold) | |

**Capacitance measurement:**



**Fig 5: Circuit for capacitance measurement**

In this experiment two 555 timers are used. The 555 timer on the left side is operating in astable mode and the right one is in mono-stable mode as shown in Fig 5. For the 555 astable timer the output frequency is given by,

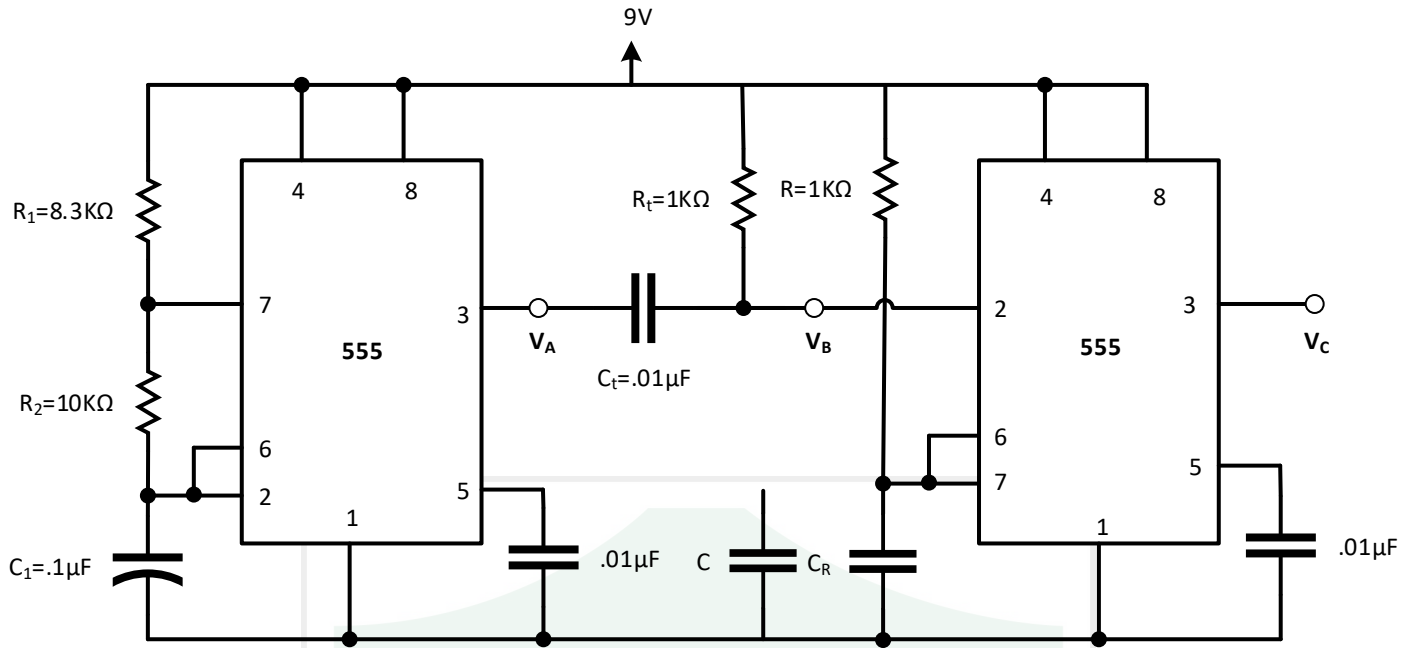$$f = \frac{1.44}{C_1(R_1 + 2R_2)} \quad \text{---------------(1)}$$

A mono-stable timer produces a pulse at the output whose width is dependent on the timing resistance and capacitance. For a 555-timer connected in mono-stable mode, the pulse width is given by

$$t_W = 1.1\, RC \quad \text{----------------(2)}$$

To measure capacitance, trigger the mono-stable timer at a constant frequency, f. The average DC voltage of the mono-stable timer will be,

$$V_{out} = t_W V_{CC}\, f$$
$$= 1.1\, V_{cc} f RC$$

So, for a reference capacitance, $C_R$ the equation becomes

$$V_1 = 1.1\, V_{CC} f R C_R \quad \text{----------------(3)}$$

If an unknown capacitor C is placed in parallel with the reference capacitor, equation will be:

$$V_2 = 1.1\, Vcc\, f R\, (C_R + C) \quad \text{----------------(4)}$$

Combining (1) and (2) we have,

$$\frac{V_1}{V_2} = \frac{C_R}{C_R + C}$$

$$\Rightarrow C = C_R \left(\frac{V_2}{V_1} - 1\right) \text{------------(5)}$$

**(c) Apparatus:**

| | | | |
|---|---|---|---|
| **1.** | Timer | NE 555 | **2** |
| **2.** | Resistor | 1KΩ | **2** |
| | | 10KΩ, 8.3KΩ | **1** |
| **3.** | Capacitor | .001 µF, .1µF, .22 µF, .47 µF | **1** |
| | | .01 µF | **3** |
| **4.** | Duel output power supply | | |
| **5.** | Digital Oscilloscope | | |
| **6.** | Digital Multimeter | | |
| **7.** | Connecting wire | | |

**(d) Procedure:**

1. Connect the circuit from left up to $V_A$ point as shown in fig 5.
2. Connect Ch-1 of the DSO to look the $V_{C1}$ also, connect Ch-2 across the $V_A$ to GND. Inspect the output of the astable timer along with the capacitor voltage and measure the frequency, duty cycle of the output.
3. Connect $R_t$ and $C_t$.
4. Connect Ch-1 to $V_A$ and Connect Ch-2 to $V_B$ point. Observe the wave shape.
5. Connect rest of the circuit. Use .001µF as $C_R$. Observe the wave shape at $V_B$ and $V_C$ point using the DSO.
6. Measure the voltage at $V_C$ using multimeter. This is $V_1$ voltage in eq. (3).
7. Connect .1 µF as C, parallel to the $C_R$.
8. Measure the voltage at $V_C$ using multimeter. This is $V_2$ voltage in eq. (3).
9. Calculate the value of C using eq-3.
10. Repeat step 7 -9 for C = 0.22µF, 0.47µF.

**(e) Report:**

1. Explain the operation of a 555 timer as monostable and astable multivibrator.
2. List other methods of measuring capacitance.
3. Is it possible to measure capacitance using only one timer? Explain how.
4. What is the purpose of $R_t$ and $C_t$?
5. What is the maximum range of Cs?

**(f) Prepared by**

- Md. Aminur Rahman

**Experiment no:** 08
**Name of the experiment:** Introduction to Analog to Digital (A/D) and Digital to Analog (D/A) conversions.

---

**Objective:**
To study the operation of *Analog to Digital* and *Digital to Analog* converters and be familiar with the parameters which serve to describe the quality of performance of the converters.

**Theory:**
**DIGITAL TO ANALOG CONVERSION (DAC)**
D/A conversion is the process of taking a value represented in digital code (such as straight binary or BCD) and converting it to a voltage or current that is proportional to the digital value.
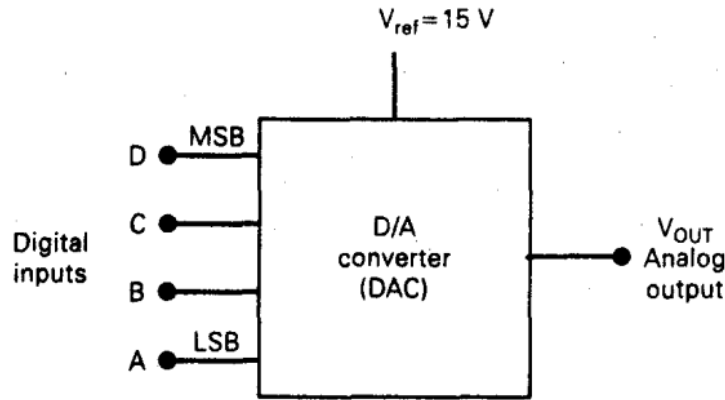


Figure-1

Notice that there is an input for a voltage reference, $V_{ref}$. This input is used to determine the full-scale output or maximum value that the D/A converter can produce. The digital inputs D, B, C and A are usually derived from the output register of a digital system. For each input number, the D/A converter output voltage is a unique value. In fact, for this case, the analog output voltage $V_{out}$ is equal in volts to the binary number. It could also have been twice the binary number or some other proportionality factor. The same idea would hold true if the D/A output were a current $I_{out}$.

In general, Analog output = K x digital input

Where, K is the proportionality factor and is a constant value for a given DAC connected to a fixed reference voltage.

**Resolution (Step Size) -** Resolution of a D/A converter is defined as the smallest change that can occur in the analog output as a result of a change in the digital input.

In this experiment R-2R Ladder circuit (figure-3) is used. Output equation of an R-2R Ladder circuit is given by-

$$V_o = -V_{ref} \cdot \frac{R_f}{R_i} \left[ \frac{B_1}{2^1} + \frac{B_2}{2^2} + \frac{B_3}{2^3} + \cdots \cdots \cdots + \frac{B_n}{2^n} \right]$$

Where, $V_{ref} = Reference\ voltage$
  $B_1 = Least\ Significant\ Bit\ (LSB)$
  $B_n = Most\ Significant\ Bit\ (MSB)$

**ANALOG TO DIGITAL CONVERSION (ADC)**
An analog to digital converter takes an analog input voltage and after a certain amount of time produces a digital output code that represents the analog input. The A/D conversion process is generally more complex

and time consuming than the D/A process, and many different methods have been developed and used. The basic ADC can be modeled as two processes: sampling and quantization.

Sampling is the first operation that takes place in ADC. Basically, the input analog signal is sampled or tested repeatedly over a period of time. This is done to determine the characteristic that contains the analog quantity, such as the signal's amplitude. A constantly varying input must be sampled at a much higher frequency than its own to ensure the accuracy of the conversion. For each sample taken, a voltage level is determined. By comparing the voltages detected by the sample pulses, the largest voltage would tend to indicate the peak and hence the amplitude of the input signal.

Quantization is the process of converting a continuous range of values into a finite range of discrete values. The difference between an input value and its quantized value is referred to as quantization error.

ADC0804 is a very commonly used 8-bit analog to digital convertor. It is a single channel IC. The digital outputs vary from 0 to a maximum of 255. The step size can be adjusted by setting the reference voltage at pin 9. When this pin is not connected, the default reference voltage is the operating voltage, $i.e.$, Vcc. The step size at 5V is 19.53mV (5V/255), $i.e.$, for every 19.53mV rise in the analog input, the output varies by 1 unit.



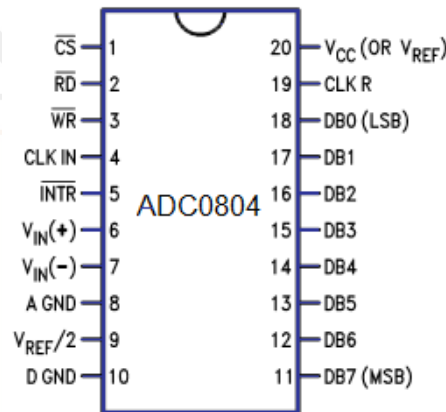Figure 2: Top view of ADC0804 IC

**Pin description of ADC0804:**

| Pin No | Function | Name |
|--------|----------|------|
| 1 | Activates ADC; Active low | Chip select |
| 2 | Input pin; High to low pulse brings the data from internal registers to the output pins after conversion | Read |
| 3 | Input pin; Low to high pulse is given to start the conversion | Write |
| 4 | Clock Input pin; to give external clock. | Clock IN |
| 5 | Output pin; Goes low when conversion is complete | Interrupt |
| 6 | Analog non-inverting input | Vin(+) |
| 7 | Analog inverting Input; normally ground | Vin(-) |
| 8 | Ground(0V) | Analog Ground |
| 9 | Input pin; sets the reference voltage for analog input | Vref/2 |
| 10 | Ground(0V) | Digital Ground |
| 11 | | D7 |
| 12 | 8 bit digital output pins | D6 |
| 13 | | D5 |

| 14 | | D4 |
|---|---|---|
| 15 | | D3 |
| 16 | | D2 |
| 17 | | D1 |
| 18 | | D0 |
| 19 | Used with Clock IN pin when internal clock source is used | Clock R |
| 20 | Supply voltage; 5V | Vcc |

**Apparatus:**
1. IC ADC0804
2. IC 741
3. Trainer board
4. Resistor: 10K (16 pcs.), 330Ω (8 pcs.)
5. Capacitor (150pF)
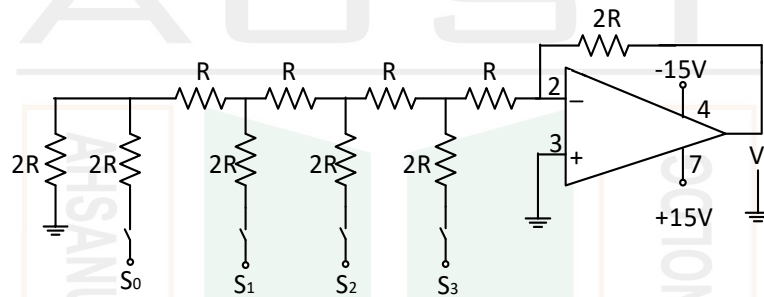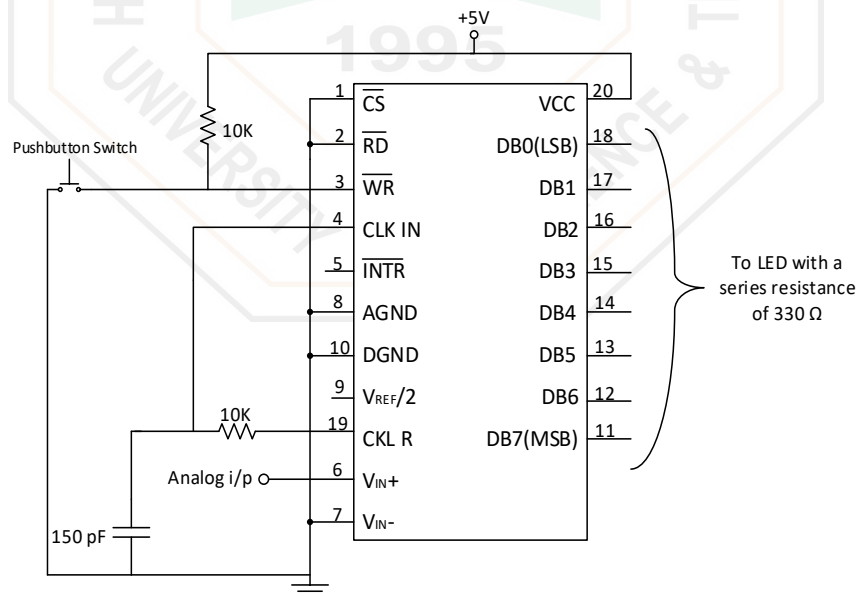6. LED (8 pcs.)
7. DMM

**Circuit Diagram:**



Figure-3



Figure-4

**Procedure:**

1.  Construct the circuit in figure-3. Connect $S_0$, $S_1$, $S_2$ and $S_3$ to the SPST switches of trainer board. Now fill up the following table:

| $S_3$ | $S_2$ | $S_1$ | $S_0$ | $V_0$ |
|-------|-------|-------|-------|-------|
| 0 | 0 | 0 | 0 | |
| 0 | 0 | 0 | 1 | |
| 0 | 0 | 1 | 0 | |
| 0 | 0 | 1 | 1 | |
| 0 | 1 | 0 | 0 | |
| 0 | 1 | 0 | 1 | |
| 0 | 1 | 1 | 0 | |
| 0 | 1 | 1 | 1 | |
| 1 | 0 | 0 | 0 | |
| 1 | 0 | 0 | 1 | |
| 1 | 0 | 1 | 0 | |
| 1 | 0 | 1 | 1 | |
| 1 | 1 | 0 | 0 | |
| 1 | 1 | 0 | 1 | |
| 1 | 1 | 1 | 0 | |
| 1 | 1 | 1 | 1 | |

2.  Construct the circuit as shown in figure-4.
3.  Vary the voltage of pin no. 6 from 0 to 5V and observe the output.

**Report:**

1.  Define quantization error.
2.  Calculate resolution for both ADC and DAC and verify the obtained readings.
3.  Discuss on different types of Analog to Digital converters.
4.  Design an 8-bit DAC.
5.  Change the connection in Figure-2, to allow an input signal that varies from -10V to +10V.
6.  What is the purpose of using Opamp in figure-3?

| Experiment No | : **11** |
| --- | --- |
| Name of the Experiment: | **Introduction to strain gauge, environmental measurements and ultrasonic transmitter/receiver**. |

## (a)Objective:

- Describe the construction and characteristics of a strain gauge.
- Describe the construction and characteristics of an ultrasonic receiver and transmitter.
- Describe the construction and characteristics of an air flow transducer.
- Describe the construction and characteristics of an air pressure transducer.
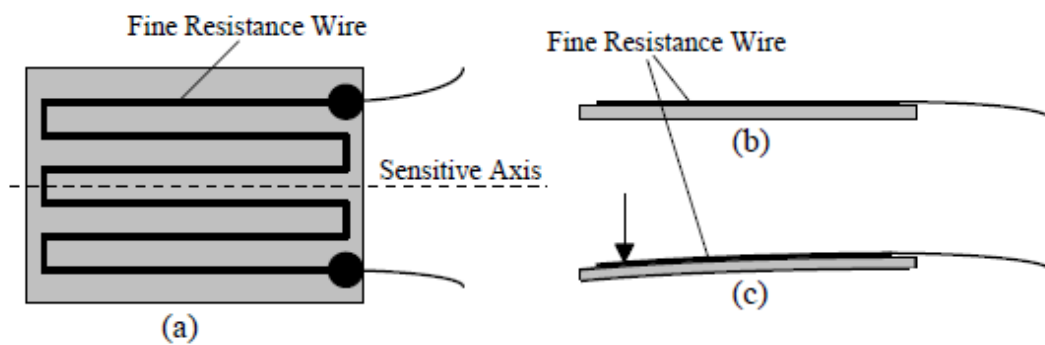
## (b)Theory:

### The Strain Gauge Transducer



**Fig 11.1**

Fig 11.1 shows the construction of a strain gauge, consisting of a grid of fine wire or semiconductor material bonded to a backing material. When in use, the unit is glued to the beam under test and is arranged so that the variation in length under loaded conditions is along the gauge sensitive axis (Fig 11.1(a)). Loading the beam increases the length of the gauge wire and also reduces its cross-sectional area (Fig 11.1(c)). Both of these effects will increase the resistance of the wire.
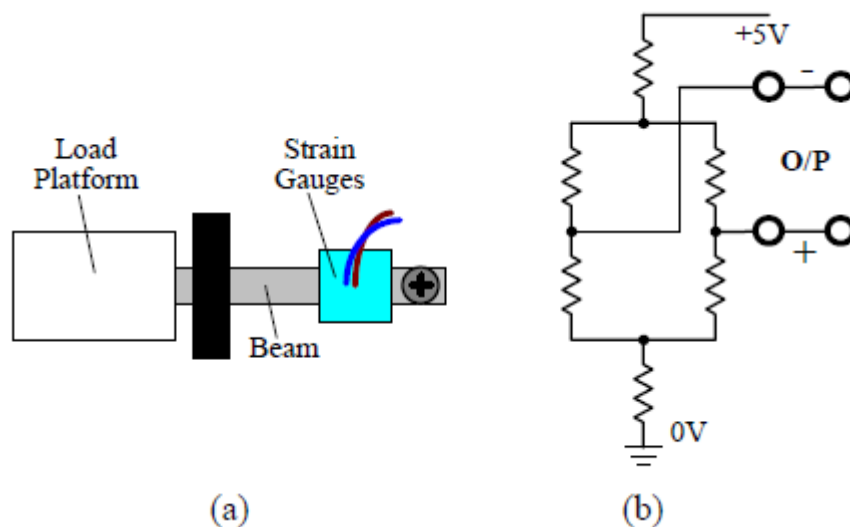


**Fig 11.2**

The layout and circuit arrangement for the DIGIAC 1750 unit is shown in Fig 11.2. Resistors are electro-deposited on a substrate on a contact block at the right-hand end of the assembly. The gauge is normally connected in a Wheatstone Bridge arrangement with the bridge balanced under no load conditions. Any change of resistance due to loading unbalances the bridge and this is indicated by the detector (Galvanometer).
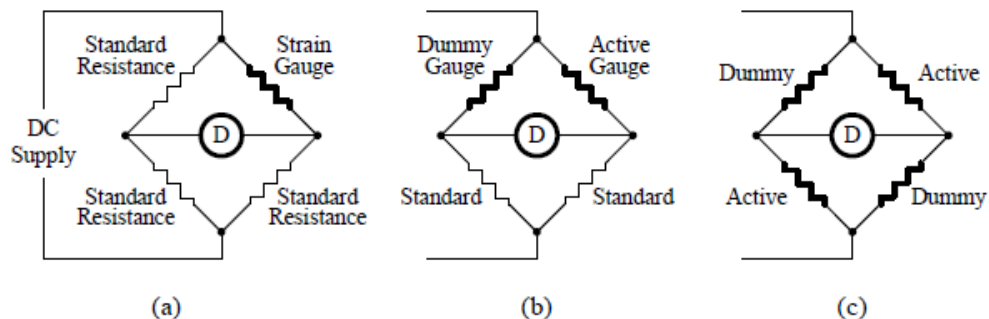


**Fig 11.3**

Fig 11.3(a) shows the basic Wheatstone Bridge arrangement with one strain gauge transducer. This circuit is liable to give inaccurate results due to thermal changes. A variation of temperature will also produce a change of resistance of the gauge and this will be interpreted as a change of loading. To correct for this an identical gauge is used and connected in circuit as shown in Fig 11.3(b). This gauge is placed near to the other gauge but is arranged so that it is not subjected to any loading. Any variation of temperature now affects both gauges equally and there will be no thermal effect on the bridge conditions. The gauge subjected to loading is referred to as the active gauge and the other is called the dummy gauge. The output from the circuit is small and to increase this, four gauges are normally used with two active gauges and two dummies as shown in Fig 11.3(c).

The DIGIAC 1750 uses two active gauges formed along the axis of the beam and two dummies formed at right angles to these.

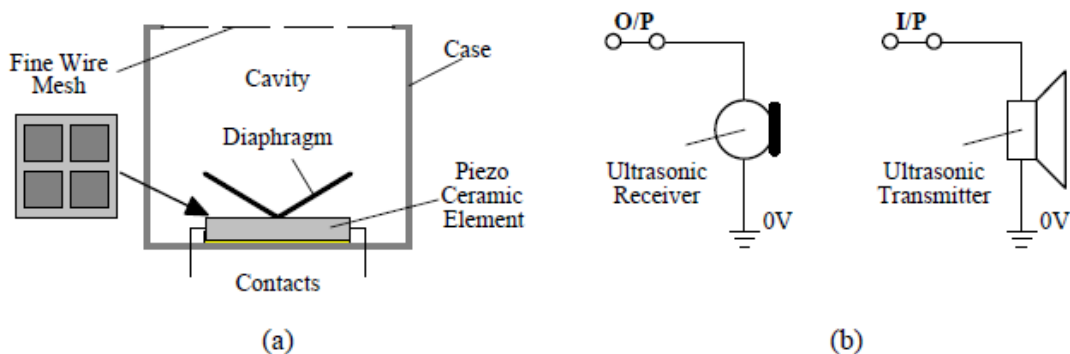### The Ultrasonic Transmitter/Receiver



**Fig 11.4**

The construction of both ultrasonic devices and their electrical circuit arrangements for the DIGIAC 1750 unit are shown in Fig 11.4. The receiver and transmitter are almost identical and consist of a slice of ceramic material with a small diaphragm fixed to it, inside the case of the unit. The operation of the receiver relies on the principle that certain ceramic materials produce a voltage when they are stressed. This is referred to as the piezo-electric principle. Vibration of the diaphragm stresses the ceramic material and produces an output voltage. The reciprocal applies to the transmitter. An applied alternating voltage produces stress which causes the ceramic

slice to vibrate. The dimensions of the components are arranged so that there is resonance (best response) at around 40 kHz. This is above the audible range (maximum 20kHz) and is therefore referred to as ultrasonic. The ceramic slice is arranged in four quarters which are connected in series for the receiver and in parallel for the transmitter.

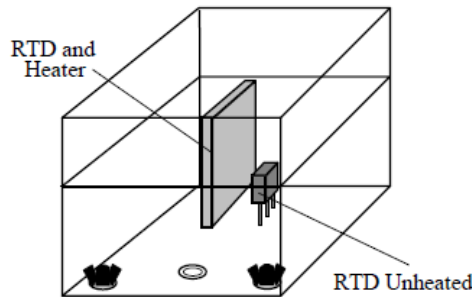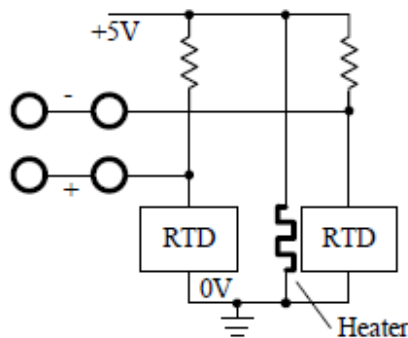## The Air Flow Transducer



**Fig 11.5**

Fig 11.5 shows the construction of an Air Flow Transducer, consisting of two RTD's (Resistance Temperature Dependent) mounted in a plastic case. One of the devices has an integral heating element incorporated with it and the other is unheated.

The operation of the device uses the principle that when air flows over the RTD's, the temperature of the heated unit will fall more than that of the unheated unit. The temperature difference will be related to the air flow rate which will in turn affect the resistance of the RTD's. With the DIGIAC 1750 Trainer, the transducers are enclosed in a clear plastic container and provision is made for air to be pumped over the device. Fig 11.6 shows the electrical circuit arrangement and main characteristics of the device in the DIGIAC 1750 Trainer



| Heater power | 1W |
|---|---|
| Output impedance | 1.7kΩ |
| Output voltage (-). Pump OFF | 2.1V |
| Output voltage (+). Pump OFF | 1.7V |
| Voltage change (airflow) | 0.06V |

**Fig 11.6**

## The Air Pressure Transducer

Fig 11.7 shows the external and internal construction of an air pressure transducer and also shows the electric circuit arrangement of the DIGIAC 1750 unit. The device consists of an outer plastic case which is open to the atmosphere via two ports. Within this case is an inner container from which the air has been evacuated and a strain gauge Wheatstone bridge circuit is fitted on the surface.
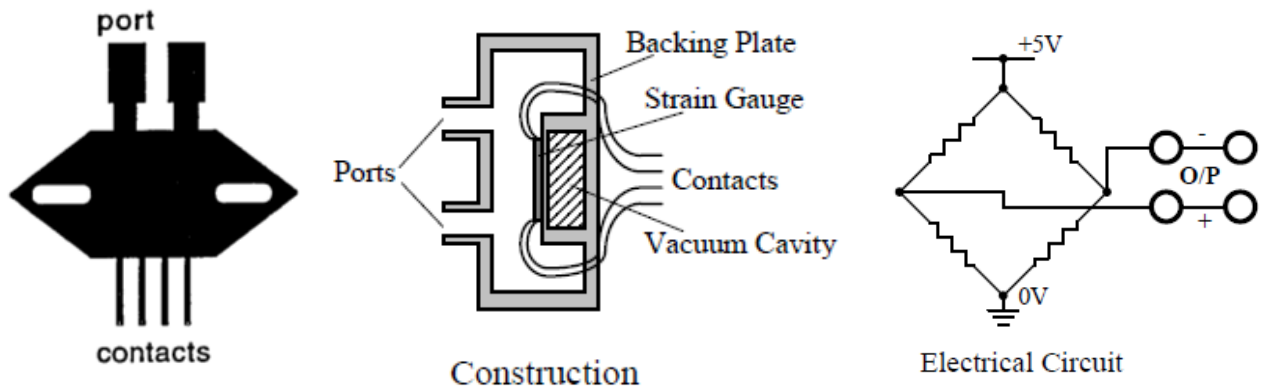
port
contacts
Ports
Backing Plate
Strain Gauge
Contacts
Vacuum Cavity
Construction

Electrical Circuit

**Fig 11.7**

The air pressure in the outer container will produce an output from the bridge and variation of the pressure will produce a variation of this output. The transducer output can be calibrated and may be called an absolute pressure transducer. Provision is made for air to be fed to the unit from the pump.

## (c)Procedure

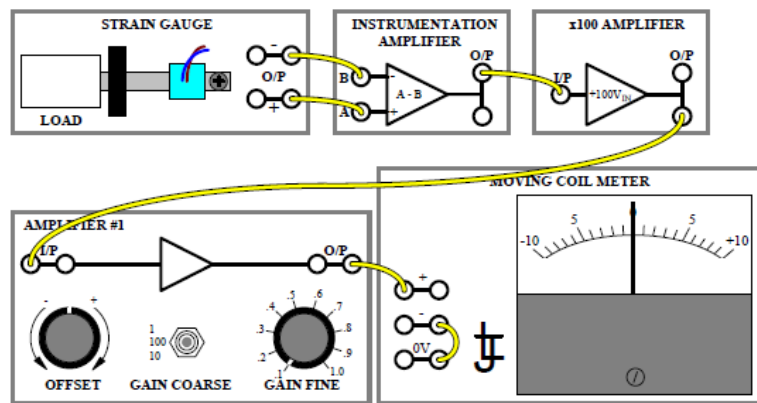### Characteristics of a Strain Gauge Transducer



**Fig 11.8**

You will need ten similar weights, such as ten equal value coins, to increase the loading in regular steps.

1.  Connect the circuit as shown in Fig 11.8 and set Amplifier #1 GAIN COARSE control to 100.
2.  Switch ON the power supply and with no load on the strain gauge platform, adjust the offset control of Amplifier #1 so that the output voltage is zero.
3.  Place all ten of your weights on the load platform and adjust the GAIN FINE control to give an output voltage of 7.0V as indicated on the moving coil meter. Note that this value of output voltage should cover all ranges of coins within the setting of the GAIN FINE control.
4.  Place one weight (coin) on the load platform and note the output voltage. Record the value in Table overleaf.
5.  Repeat the process, adding further weights one at a time, noting the output voltage at each step and recording the values in the table

| Number of weight | 0 | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|---|
| Output Voltage (V) | | | | | | | | | |

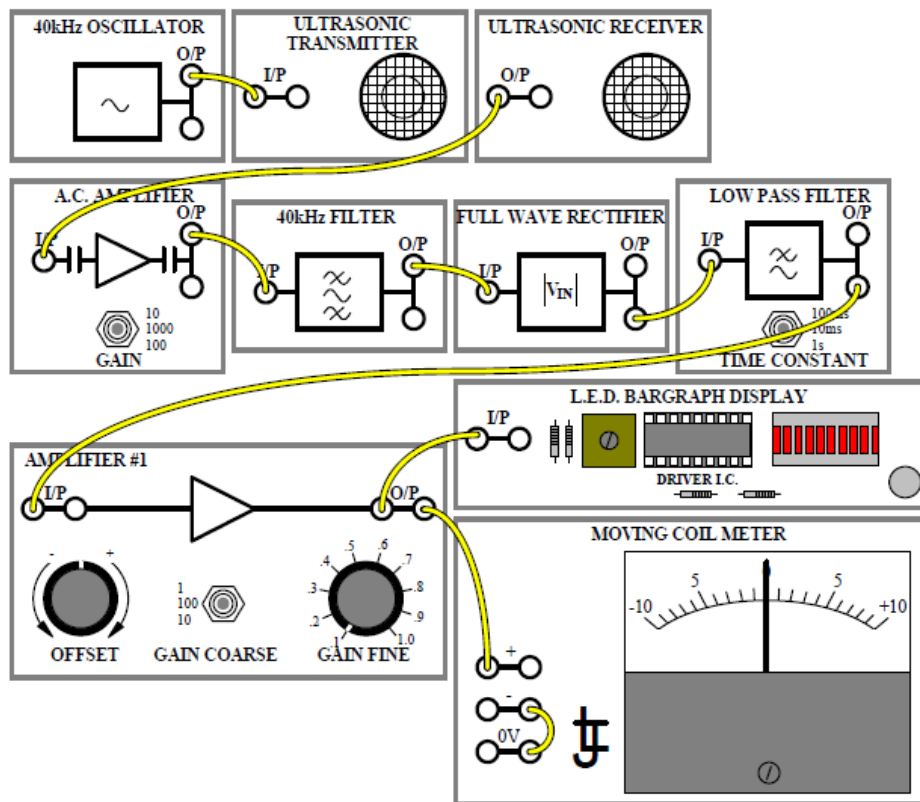## Characteristics of an Ultrasonic Transmitter/Receiver



**Fig 11.9**

1. Connect the circuit as shown in Fig 11.9. Set the AC Amplifier gain control to 1000 and Amplifier #1 GAIN COARSE control to 10 and GAIN FINE to 0.5. Switch the Low Pass Filter time constant to 100ms.
2. Switch ON the power supply and adjust Amplifier #1 OFFSET to give zero output on the Moving Coil Meter.
3. Note the bargraph display as you move your hand or any other object over the ultrasonic devices. The display should respond, indicating the receipt of a signal of frequency 40kHz by the ultrasonic receiver.
4. Place a small book (approximately 6 inches (15cm) × 4 inches (10cm) or other flat object 3 feet (90cm) above the Ultrasonic Transducers. Slowly move the object closer to the transducers, watching the output reading on the bargraph display, until the object is covering the transducers.

➢ **At which of the following positions is the maximum output obtained?**
   a. Object in contact with the Ultrasonic Tranducers.
   b. Object 4 inches (10cm) above the unit.
   c. Object 12 inches (30cm) above the unit.
   d. Object 3 feet (90cm) above the unit.

Remove any other equipment from the vicinity so that you have free access to the ultrasonic transmitter/receiver area.

5. Increase the Amplifier #1 GAIN FINE control to 1.0. Hold a thin object such as a pencil approximately 6 inches (15cm) above the Ultrasonic Transducers, move it horizontally and vertically and note the effect on the output response. This indicates how critical the direction angle is for the device.

> **Is the position of the reflector critical?    Yes   or   No**

6. Put a sheet of paper over the Ultrasonic Transducers to intercept the path and move your hand up and down above the transducers.

> **Does the beam pass through a piece of paper?    Yes   or   No**

In this exercise the received signal has been amplified, rectified, filtered (to remove all unwanted frequencies) and then amplified again to operate the display. Pulsed ultrasonic devices can be used for distance measurement to reflecting surfaces by measurement of the time between the transmission and return of the pulsed signal.

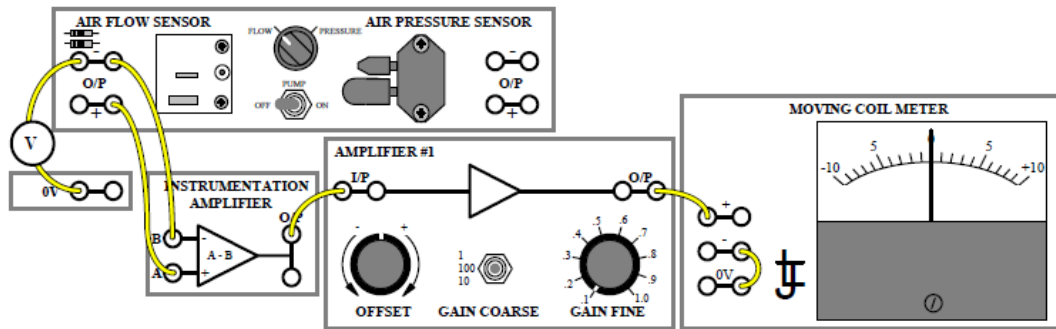## Characteristics of an Air Flow Transducer



**Fig 11.10**

Connect the circuit as shown in Fig 11.10 and set the GAIN COARSE control of Amplifier #1 to 10 and GAIN FINE control to 1.0. Check that the pump control is set to OFF.

1. Switch ON the power supply and allow the temperature to stabilize.
2. Adjust the OFFSET control of Amplifier #1 for zero output continuously during this time, setting the GAIN COARSE control to 100 when stabilized conditions are approached.
3. Set the Flow/Pressure control to FLOW.
4. Check that the OFFSET control is set for zero output voltage.
5. Use the digital multimeter to note the voltages at the - and + outputs from the transducer, then note the Amplifier #1 output voltage displayed on the Moving Coil Meter. Record the values in the table below. Switch the pump ON and note the voltages again when conditions have stabilized, recording the values in Table below.

|  | Pump OFF (Volt) | Pump ON (Volt) |
|---|---|---|
| **Transducer - Output Voltage** |  |  |
| **Transducer + Output Voltage** |  |  |
| **Amplifier #1 Output Voltage** |  |  |

6. Switch OFF the power supply and the pump.

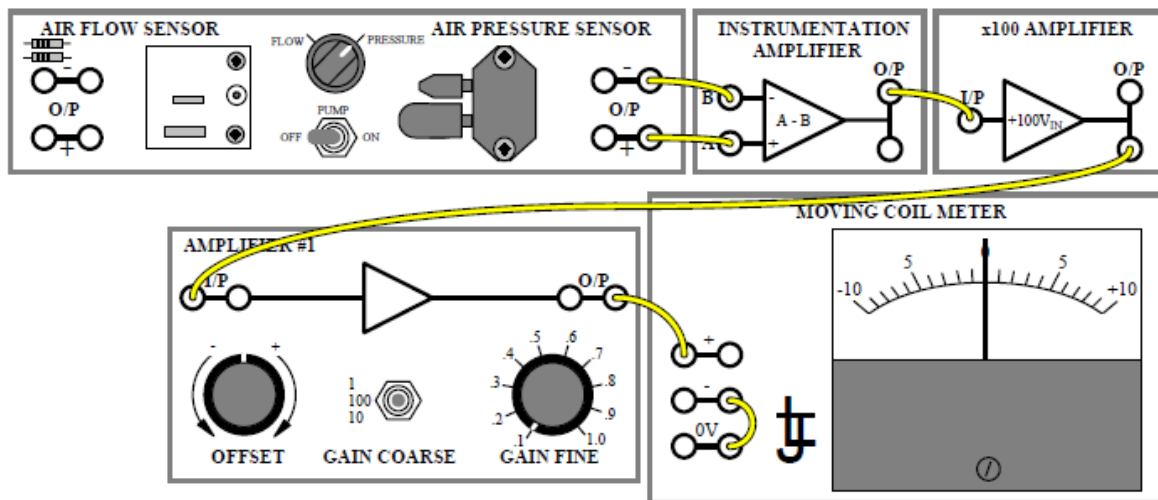## Characteristics of an Air Pressure Transducer



**Fig 11.11**

1. Connect the circuit as shown in Fig 11.11 and set the Amplifier #1 GAIN COARSE control to 10 and GAIN FINE control to 0.3. Ensure that the pump switch is set OFF.
2. Switch ON the power supply and adjust the OFFSET control of Amplifier #1 for zero output voltage. The unit is now calibrated zero for the current value of the atmospheric pressure.
3. Set the Flow/Pressure control to PRESSURE and then switch the pump ON. The output voltage from the Amplifier #1 will increase. Note the value of this voltage.

   Output voltage (Pump ON) = [        ] V

4. Switch OFF the power supply and the pump.

### (d) Report

1. Plot Weight vs voltage and comment on the result.
2. Design any measurement system which strain gauge and hence explain the operation.
3. Give practical example where the ultrasonic transmitter and receiver can be used. Design any measurement system which uses both the transducers and explain the operation.
4. Compare between air pressure and air flow transducers

### (e) Prepared by:

1. Hasib Md. Abid Bin Farid
2. Aminur Rahman

# Appendix

**Powers of ten**

| | | | |
|---|---|---|---|
| tera | T | 1000000000000 | $10^{12}$ |
| giga | G | 1000000000 | $10^{9}$ |
| mega | M | 1000000 | $10^{6}$ |
| kilo | k | 1000 | $10^{3}$ |
| hecto | h | 100 | $10^{2}$ |
| deca | da | 10 | $10^{1}$ |
| (none) | (none) | 1 | $10^{0}$ |
| deci | d | 0.1 | $10^{-1}$ |
| centi | c | 0.01 | $10^{-2}$ |
| milli | m | 0.001 | $10^{-3}$ |
| micro | µ | 0.000001 | $10^{-6}$ |
| nano | n | 0.000000001 | $10^{-9}$ |
| pico | p | 0.000000000001 | $10^{-12}$ |

**Capacitor Notations**
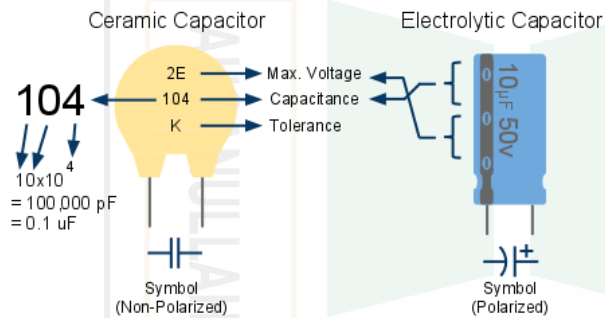


| Max. Operating Voltage | |
|---|---|
| **Code** | **Max. Voltage** |
| 1H | 50V |
| 2A | 100V |
| 2T | 150V |
| 2D | 200V |
| 2E | 250V |
| 2G | 400V |
| 2J | 630V |

| Capacitance Conversion Values | | |
|---|---|---|
| **Microfarads (µF)** | **Nanofarads (nF)** | **Picofarads (pF)** |
| 0.000001 µF | 0.001 nF | 1 pF |
| 0.00001 µF | 0.01 nF | 10 pF |
| 0.0001 µF | 0.1 nF | 100 pF |
| 0.001 µF | 1 nF | 1,000 pF |
| 0.01 µF | 10 nf | 10,000 pF |
| 0.1 µF | 100 nF | 100,000 pF |
| 1 µF | 1,000 nF | 1,000,000 pF |
| 10 µF | 10,000 nF | 10,000,000 pF |
| 100 µF | 100,000 nF | 100,000,000 pF |

| Tolerance | |
|---|---|
| **Code** | **Percentage** |
| B | ± 0.1 pF |
| C | ±0.25 pF |
| D | ±0.5 pF |
| F | ±1% |
| G | ±2% |
| H | ±3% |
| J | ±5% |
| K | ±10% |
| M | ±20% |
| Z | +80%, -20% |