MADHA
Expertise | Empathy | Excellence
ENGINEERING COLLEGE

DEPARTMENT OF AERONAUTICAL ENGINEERING

LAB MANUAL

CE3362-FLUID MECHANICS AND MACHINERY LABORATORY

COURSE OBJECTIVES: Semester: III

- Upon Completion of this subject, the students can able to have hands on experience in flow measurements using different devices.
- Also perform calculation related to losses in pipes and also perform characteristic study of pumps, turbines etc.,

LIST OF EXPERIMENTS

A. FLOW MEASUREMENT

1. Verification of Bernoulli's theorem
2. Flow through Orifice/Venturi meter
3. Friction factor for flow through pipes
4. Impact of jet on fixed plate

B. METACENTRE

5. Determination of metacentric height

C. PUMPS

6. Characteristics of Centrifugal pump
7. Characteristics of Gear pump
8. Characteristics of Submersible pump
9. Characteristics of Reciprocating pump

D. TURBINES

10. Characteristics of Pelton wheel turbine
11. Characteristics of Francis turbine

COURSE OUTCOMES:

At the end of the course students will be able to:

CO1: Verify and apply Bernoulli equation for flow measurement like Orifice/Venturi meter.

CO2: Measure friction factor in pipes and compare with Moody diagram and verify momentum conservation law.

CO3: Determine the performance characteristics of Roto dynamic pumps.

CO4: Determine the performance characteristics of positive displacement pumps.

CO5: Determine the performance characteristics of turbines.

LABORATORY SAFETY PROCEDURE

Safety

Safety is our prime concern at all times and you will be asked to leave the lab if your conduct is deemed to compromise safety regulations. Do not perform unauthorized experiments by yourself. Never leave unattended an experiment that is in progress. The students are strictly advised to wear shoes when they come to the laboratory as a measure of safety.

Do's

1. Bring observation note books, lab manuals and other necessary things for the class.
2. Check the instruments for proper working conditions while taking and returning the same.
3. Thoroughly clean your laboratory work space at the end of the laboratory session.
4. Maintain silence and clean environment in the lab

Dont's

1. Do not operate the machines without the permission of the staff
2. Do not put hands or head while equipment is in running condition.
3. Do not fix or remove the test specimen while the main is switch on.

Exp No	Name of the Experiment	Date	Sign
1	CALIBRATION OF ROTAMETER		
2	DETERMINATION OF C0-EFFICIENT OF DISCHARGE OF VENTURIMETER		
3	DETERMINATION OF C0-EFFICIENT OF DISCHARGE OF ORIFICEMETER		
4	DETERMINATION OF CHARACTERISTIC OF CIRCULAR ORIFICE FLOWING UNDER THE CONSTANT HEAD		
5	DETERMINATION OF MEAN VELOCITY USING PITOT TUBE		
6	FLOW THROUGH NOTCHES (RECTANGLE & TRIANGLE)		
7	STUDY OF FRICTION LOSS IN PIPES		
8	STUDY OF MINOR LOSS IN PIPE		
9	PERFORMANCE CHARACTER OF CENTRIFUGAL PUMP		
10	PERFORMANCE CHARACTER OF RECIPROCATING PUMP		
11	PERFORMANCE STUDY OF GEAR OIL PUMP		
	TOPIC BEYOND SYLLABUS		
12	PERFORMANCE TEST ON A KAPLAN TURBINE		
13	CHARACTERISTIC CURVE OF FRANCIS TURBINE		
14	HYDRAULIC JUMP (DEMO)		
15	BERNOULLI’S EXPERIMENT		

EX.NO:…1……. **DATE**:………

CALIBRATION OF ROTAMETER

AIM:

To determine the Co-efficient of discharge of the Rotameter and to obtain the calibration error.

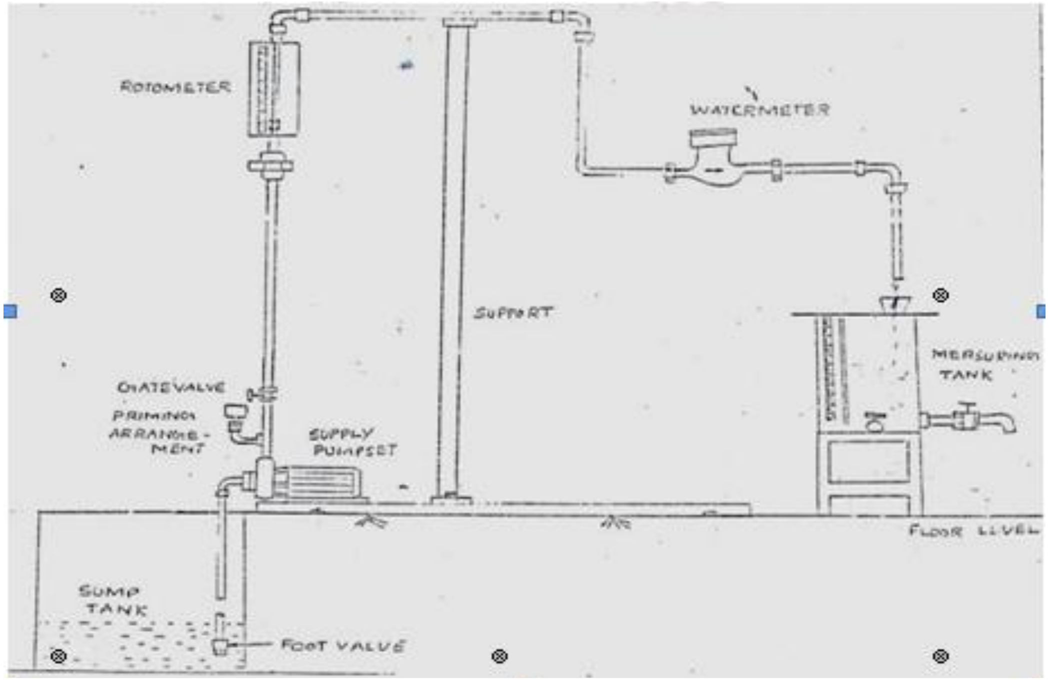
APPARATUS REQUIRED:

- Rotameter fitted with pipe line setup
- Stop watch
- Measuring scale & Tape

THEORY:

The Rotameter is the most popular flow meter. It consists essentially of a plummet or float which is free to move up or down in a vertical slightly tapered tube having its small end down. The fluid enters the lower end of the tube and causes the float to rise until the annular area between the tube and the float is such that the pressure drop across this construction is just sufficient to support the float. Typically, the tapered tube is of glass and caries etched upon it a nearly linear scale on which the position of the float may be usually noted as an indication of the flow.

Rotameter have proved satisfactory both for gasses and for liquids at high andlow pressures. Rotameter required straight runs of pipe before or after the point of installation. Pressure losses are substantially constant over the whole flow rang. In experimental work, for greatest precision, a Rotameter should be calibrated with the fluid which is to be entered. However, most modern Rota meters are precision-mode such that their performance closely corresponds to a mater calibration plate for the type in question.



Rotameter setup

FORMULA:

Co-efficient of discharge

$$C_d = \frac{\text{Actual discharge}}{\text{Theoretical discharge}} = \frac{Q_a}{Q_t}$$

Length of collecting tank (L) =……cm

Breadth of collecting tank (B)=…..cm

OBSERVATION:

Internal plan dimensions of the collecting tank

Sl.No	**Rotameter Reading** *Kg/cm²*	**Time For 5 CM raise of water (t)** *sec.*	**Actual Discharge (Q_{act})** *m³/sec.*	**Actual Discharge (Q_{theo})** *m³/sec*	**Co-efficient of Discharge (Cd)**	**Calibration error**

Actual discharge ,

$$Q_a = \frac{A*h}{t} \text{ m}^3/\text{s}$$

Where,

A - Area of the measuring tank m^2.

H - Rise of water level meters (say 5 cm)

t - Time in seconds for rise of water level (say 5 cm)

Theoretical discharge, Q_t = Rotameter reading × 1000 x60 m^3/s

Calibration error = $\frac{Qt-Qa}{Qt}$

MODEL CALCULATIONS:

1. Actual discharge

$$Q_a = \frac{A*h}{t} \text{ m}^3/\text{s}$$

= ....m^3/s

2. Theoretical discharge

Q_t = Rotameter reading × 1000 x60 m^3/s

= ....m^3/s

3. Co-efficient of discharge

$C_d = Q_a/Q_t$

= …………..

4. Calibration error

$$= \frac{Qt-Qa}{Qt}$$

= ………..

PROCEDURE:

- Note the cross section area of collecting tank.
- Slowly open the delivery valve of Rotameter.
- The actual discharge is measured with the help of Rotameter.
- Note down the value of Rotameter.
- The theoretical discharge is measured with the help of the Rotameter.

Graph:

- Actual Qa Vs Co-efficient of discharge.
- Actual Qa Vs calibration of an error

Result

- Average co-efficient of discharge (Cd) = ____________________
- Average calibration error =____________________

Thus the given Rotameter was calibration and connected at parameter.

EX. NO:2 **DATE: …………**

DETERMINATION OF COEFFICIENT OF DISCHARGE OF VENTURIMETER

OBJECTIVE:

To determine the coefficient of discharge for the venturimeter

APPARATUS REQUIRED:

- A venturimeter with known diameters at the mouth and throat fitted with stopcocks at the mouth and throat.
- A U-tube manometer containing mercury.
- Water measuring tank.
- A stopwatch.

Theoretical discharge

THEORY:

Venturimeter is a device, which works on the principle of Bernoulli's equation. It is used for measuring the rate of flow of fluid through a pipe. It consists of these parts.

- A short converging part.
- Throat
- Diverging part

A U-tube manometer is connected to the pipe and through which shows the head difference between them. There will not be any datum head H the water is horizontal. Hence, the pressure head is equal to the velocity head. The main principle involved is that the pressure at throat has maximum due to decreasing cross section, which is measured by using manometer. Thus, we measure discharge as well as co-efficient.

FORMULA:

Co-efficient of discharge, $(C_d) = \frac{Q_a}{Q_t}$

Theoretical Discharge, $Q_t = \frac{Cd * A1 * A2 \sqrt{2gH}}{\sqrt{(A1^2 - A2^2)}}$ m^3/s

A1 - cross sectional area of pipe, m^2
A2 - cross sectional area of throat, m^2
d1 - diameter of the pipe
d2 - throat/orifice diameter

g - Acceleration due to gravity = 9.81 m/s^2
H - Total head = h*12.6

Co-efficient of discharge of venturimeter

S.No	Manometer Reading in (cm)			Total Head (cm)	Time taken 10cm rise of water level (Seconds)	Discharge (Q) m^3/sec		Co-efficient of discharge (Cd)
	h1	h2	H			Actual discharge (Q_a)	Theoretical discharge (Q_t)	

MODEL CALCULATIONS:

1. Total head ,

$$H = (h_1 - h_2) * 12.6$$

$$= \dots\dots\dots \text{ cm}$$

2. Theoretical Discharge,

$$Q_t = \frac{Cd*A1*A2\sqrt{2gH}}{\sqrt{(A1^2 - A2^2)}}$$

$$= \dots\dots\dots\dots\dots m^3/s$$

3. Actual Discharge,

$$Q_a = \frac{A*R}{t*100}$$

$$= \dots\dots\dots\dots\dots m^3/s$$

4. Co-efficient of discharge,

$$(C_d) = \frac{Qa}{Qt}$$

$$= \dots\dots\dots\dots$$

Actual Discharge, $Q_a = \frac{A*R}{t*100}$ m^3/s

Where,A - Area of collecting tank (m^2)
R - Rise in water level of the collecting tank (cm)
t - Time for 'R' cm rise of water (sec)

PROCEDURE:

1. Check whether all the joints are leak proof and watertight.
2. Close all the pipes with cocks in the pressure feed pipes and manometer to prevent damage and over loading of the manometer.
3. Open the inlet value of the pipe. Switch on the pump and adjust and the control value to allow the meter to flow through Venturimeter.
4. Open the downstream and upstream cocks that connect the manometer to the Venturimeter for which the co-efficient of discharge is to be calculated.
5. Prime the manometer properly. Adjust the control value to maintain the flow and for the deserved rate of flow.
6. Measure the manometer head to find the venture discharge. Measure the time taken for 10mm rise in the collecting tank to find the actual discharge.
7. Calculate the co-efficient of discharge and repeat the procedure for the different flow rates.

GRAPH:

A graph is drawn between actual discharge Vs$\sqrt{h}$ by taking in X-axis and Q_a in Y-axis.

RESULT:

The co-efficient of discharge for the Venturimeter is found out and the necessary graph is plotted.

(i) From the table C_d =

(ii) From the graph C_d =

EX NO - 3 **DATE**

DETERMINATION OF COEFFICIENT OF DISCHARGE OF ORIFICE METER

AIM:

To determine the Coefficient of discharge (C_d) of the given Orificemeter.

APPARATUS REQUIRED:

1. Orificemeter with all accessories
2. Stop watch
3. Metre scale

THEORY:

Orificemeter is a device, used to measure the discharge of any liquid flowing through a pipeline. The pressure difference between the inlet and the orifice is recorded using a differential Manometer, and the time is recorded for a measured discharge.

$$\text{Theoretical discharge, } Q_t = \frac{a_1 a_2 \sqrt{2gh}}{\sqrt{a_1^2 - a_2^2}}$$

Where,
a_1 = Area of Inlet
a_2 = Area of Orifice
g = Acceleration due to gravity
h = Orifice head in terms of flowing liquid
= [$h_1 - h_2$]
h_1= Manometric head in one limb of the Manometer

h_2= Manometric head in other limb of the Manometer.

Actual Discharge, $Q_a = \dfrac{AH}{t}$

Where , A = Internal plan area of collecting tank

H = Rise of liquid

t = Time of collection

Coefficient of Orifice meter (C_d) is the ratio between the actual discharge (Q_a) and the theoretical discharge (Q_t)

i.e $C_d = \dfrac{Q_a}{Q_t}$

PROCEDURE:

1. The dimensions of the inlet and orifice are recorded and the internal plan dimensions of the collecting tank are measured.
2. Keeping the outlet valve closed, the inlet valve is opened fully.
3. The outlet valve is opened slightly and the manometric heads in both the limbs (h1 and h2) are noted.
4. The outlet valve of the collecting tank is closed tightly and the time t required for H rise of water in the collecting tank is observed using a stopwatch.
5. The above procedure is repeated by gradually increasing the flow and observing the required readings.
6. The observations are tabulated and the coefficient of Orificemeter (C_d) is computed.

OBSERVATION:

Diameter of the Inlet, d_1 = mm

Diameter of the orifice, d_2= mm

Internal plan dimensions of collecting tank, Length, L = mm

Breadth, B= mm

Acceleration due to gravity, g = 9810 mm/sec^2

TABULATION:

Sl. No.	Manometric Readings			$\sqrt{h}$	Time for H = 100mm rise of water(t)	Actual Discharge Q_a = AH/t	Theoretical Dsicharge $Q_t = a_1a_2\sqrt{(2gh)}/\sqrt{a_1^2 - a_2^2}$	Co efficient of Orificemeter $C_d = Q_a/Q_t$
	h_1	h_2	$h=h_1-h_2$					
	mm of water			mm$^{1/2}$	Sec	mm^3/s	mm^3/s	
1								
2								
3								
4								
5								
6								

MODEL CALCULATION:

1. Area of inlet, a_1 $= \pi d_1^2/4$

$=$ mm^2

2. Area of Orifice, a_2 $= \pi d_2^2/4$

$=$ mm^2

3. Internal plan area of collecting tank

$= L \times B$

$=$ mm^2

4. Actual Discharge Qa $= AH/t$

$=$ mm^3/s

5. Theoretical Discharge $Q_t = a_1 a_2 \sqrt{(2gh)} / \sqrt{a_1^2 - a_2^2}$

$=$ mm^3/s

6. Co efficient of discharge $= Q_a/Q_t$

$=$

GRAPH:

A graph Q_a vs. $\sqrt{h}$ is drawn taking $\sqrt{h}$ on X axis.

RESULT:

Coefficient of discharge of the Orificemeter (C_d)

1. Theoretically =
2. Graphically =

Ex no : 4 **Date:**

Flow through a small orifice

Aim:-

1. To investigate the discharge characteristics of circular orifices subjected to a constant head.
2. To investigate the trajectory of a horizontal jet issuing from an orifice and hence determine the coefficient of velocity for the orifice.

Apparatus Required:-

1. Constant head inlet tank (Figure 1).
2. Circular orifices with different diameters.
3. Hook gauge and scale.
4. Hydraulic bench.

Theory:-

Consider a small orifice in the side of a vessel with the head of water above the orifice kept constant.

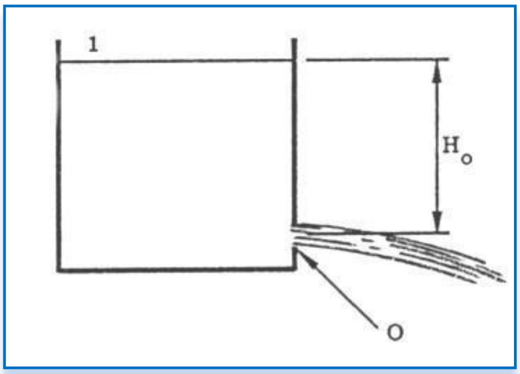


Figure 2: Discharge through an orifice

Applying Bernoulli's theorem between the surface of the water 1 and the orifice O yields

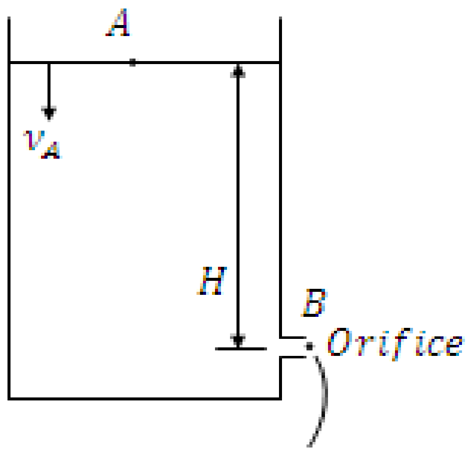
$$\text{H is constant} \Rightarrow v_A = 0$$

$$TH_A = TH_B + h_{L_{A-B}}$$

$$0 + 0 + H = \frac{v_B^2}{2g} + 0 + 0$$

$$\Rightarrow \boxed{v_B = \sqrt{2gH}}$$

$$\therefore Q_{th} = A\sqrt{2gH}$$



Two

Firs $Q_{act} = \frac{volume}{time} = C_d A\sqrt{2gH}$

between point A and B.

$$v_{actual} = c_v v_{theo} = c_v\sqrt{2gh}$$

C_v is the coefficient of velocity

Second: The stream line of the orifice contract reducing the area of flow. (Vena Contraction)

$$A_{actual} = C_c \cdot A$$

Where. C_c is the coefficient of contraction.

$$Q_{actual} = Cv.C_c A\sqrt{2gh}$$
$$Q_{actual} = Cd\, A\sqrt{2gh}$$

C_d in the range [0.6-0.65]

Consider the trajectory of a jet formed by the discharge of water through an orifice mounted in the side of a tank. The jet will be subjected to a downward acceleration of g due to gravity. Taking the origin of co-ordinates at the vena-contracta and applying the laws of motion in the horizontal and vertical planes then ignoring any effect of air resistance on the jet.

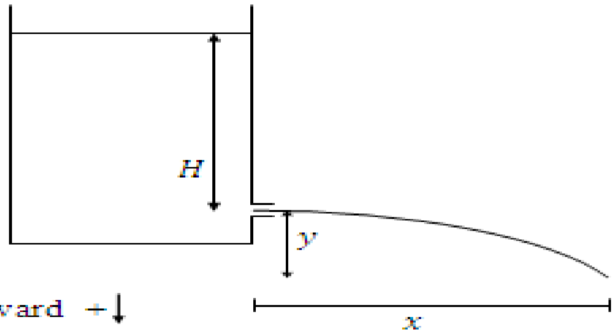
$$v = v_o + at$$
$$v^2 = v_o^2 + 2ax$$
$$x = v_o t + \frac{1}{2}at^2$$

In x-direction:

$$x = vt \Rightarrow t = \frac{x}{v}$$

In y-direction:

$$y = v_{oy}t + \frac{1}{2}gt^2, \text{ assuing positive is downward } +\downarrow$$
$$y = \frac{1}{2}gt^2$$
$$y = \frac{1}{2}g\left(\frac{x}{v}\right)^2 = \frac{1}{2}g\frac{x^2}{v^2}$$
$$v = \sqrt{\frac{1}{2}g\frac{x^2}{y}} \Rightarrow \boxed{v_{act} = \sqrt{\frac{gx^2}{2y}}} = \frac{x}{\sqrt{\frac{2y}{g}}}$$



Procedures:

1. Fit the 5mm diameter orifice into the side of the inlet head tank. Remove the overflow extension pipe. Start the pump and set up an inlet head of 25cm.
2. Measure the trajectory of the jet using the hook gauge. Record the horizontal and

vertical distances.

3. Replace the overflow extension tube and establish an inlet head of 500mm. Measure the trajectory of the jet
4. Repeat the experiment using the 8mm diameter orifice.

Model Calculation

1. $X = c_v\sqrt{y}2\sqrt{H}$

   =......................m

2. $c_c = \frac{cd}{c_v}$

   = ...

Results:

1. Record the results on a copy of the result sheet for discharge characteristics.
2. For each result calculate the flowrate
3. Plot a graph of square root of the head against the flow rate for each orifice diameter, the results should lie on a straight line passing through the origin to confirm that:

$$Q\alpha \sqrt{H}$$

Measure the slope of the line and hence calculate the coefficient of velocity from:

$$Q = \frac{slope}{A\sqrt{2g}}$$

Table 1 Collecting tank Value

D (mm)				
H (cm)				
$\sqrt{H}$(m				
V (L)				
T (sec)				
Q (m^3/s)				

Table 2: Trajectory of horizontal jet

D (mm)				
H (cm)				
x (cm)	**Vertical distance below orifice center line y (cm)**			
Slope of graph				

Ex No : 5 **Date :**

Determination of mean velocity by Pitot-Tube

AIM

To determine co-efficient of discharge of the Pitot tube.

APPARATUS

1. Pitot tube
2. Pipe
3. Manometer
4. Stop watch
5. Collecting tank fitted with a valve

Theory

The pitot tube can be used to measure the velocity of water in an open channel as well as in a closed pipe. For an open channel, a simple pitot tube will serve the purpose. However for a closed pipe in which the water is flowing under pressure, it is necessary to measure the static pressure also. Then the velocity head will be equal to the total Pitot-tube reading minus the static pressure. The static pressure is measured by inserting another L-shaped tube with its end pointing towards the flow downstream. The water will be drawn in this tube by means of suction. If, now, the tubes are connected by an inverted U-tube manometer, the difference of water height 'h' will give the velocity head. Such an arrangement is known as ''Pitot-meter". The static pressure can also be measured by inserting the other end of inverted U-tube to the pipes.

A Pitot tube is fixed inside a pipe connected to a supply water tank. The Pitot tube is connected to an inverted water manometer. The flow rate in the pipe is measured from given collecting tank. The flow rate is varied by adjusting the delivery valve.

The following formulae are employed to find the theoretical velocity and Pitot tube coefficient

Theoretical velocity, $V_a = \sqrt{2gh}$

Actual discharge, $Q_a = AH/T$

A = Internal plan area of collecting tank H

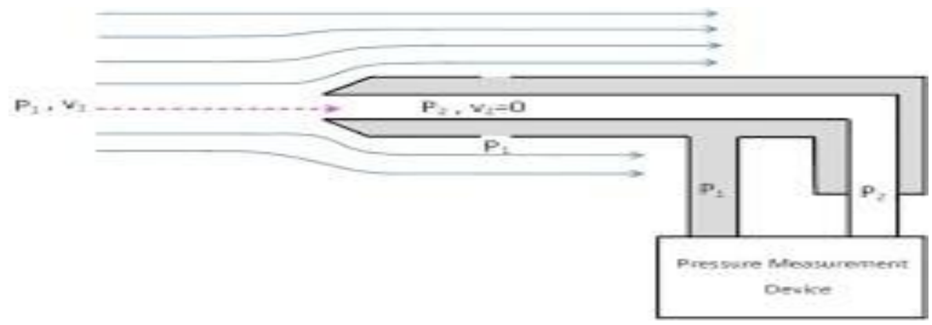
= Rise of liquid in collecting tank

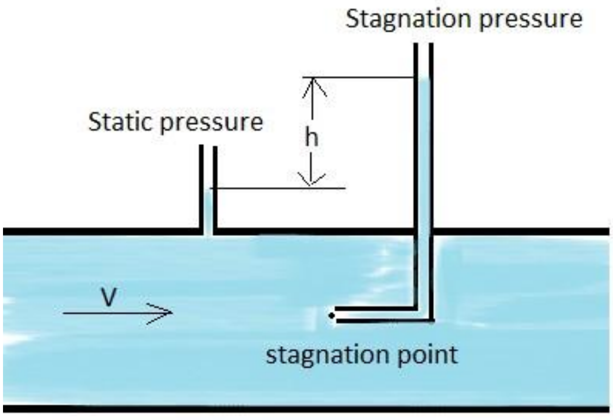
T = Time taken to collect liquid in the collecting tank

Actual velocity, $V_a = Q_a/a$

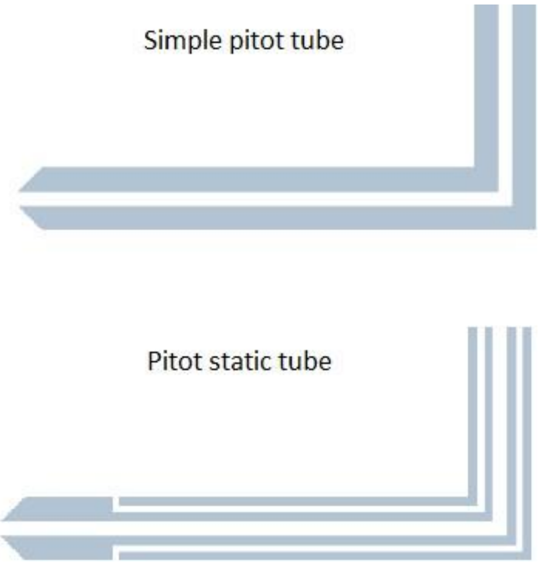
Error of the Pitot tube, $\Phi = V_a / V_{th}$

It is a device used for measuring the velocity of flow at any point in a pipe or a channel. It is based on the principle that if the velocity of flow at a point becomes zero, the pressure there is increased due to the conversion of the kinetic energy into pressure energy. In its simplest form, the Pitot-tube consists of a glass tube, bent at right angle as shown in figure below.





Simple pitot tube



The lower end, which is bent through 90 is directed in the upstream direction as shown in Figure. The liquid rises up in the tube to the conversion of kinetic energy in to pressure energy. The velocity is determined by measuring the raise of liquid in the tube.
Consider two points (1) and (2) at the same level in such a way that (2) is just at the inlet of the Pitot-tube and point (1) is far away from the tube.

Let
P1= intensity of pressure at point (1)
v1 = velocity of flow at (1)
P2= pressure at section (2)
V2 = velocity at point (1), which is zero
H= depth of tube in the liquid
h= rise of liquid in the tube above the free surface.

Applying Bernoulli’s equations at point (1) and (2), we get

$$\frac{P_1}{\rho g} + \frac{V_1^{\,2}}{2g} + Z_1 = \frac{P_2}{\rho g} + \frac{V_2^{\,2}}{2g} + Z_2$$

But $Z_1 = Z_2$ as points (1) and (2) are on the same line and V2 =0.

$\frac{P1}{g}$ = pressure head at (1) = H and $\frac{P2}{g}$ = pressure head at (2) = (h+H)

Substituting these values, we get

$$H + (V_1^2/2g) = (h+H)$$

This is theoretical velocity. Actual velocity is given by $(v1)act = Cv\sqrt{2gh}$,

Where Cv = Co-efficient of Pitot-tube

Velocity of flow in a pipe by Pitot-tube

For finding the velocity at any point in a pipe by Pitot-tube, the following arrangements are adopted:

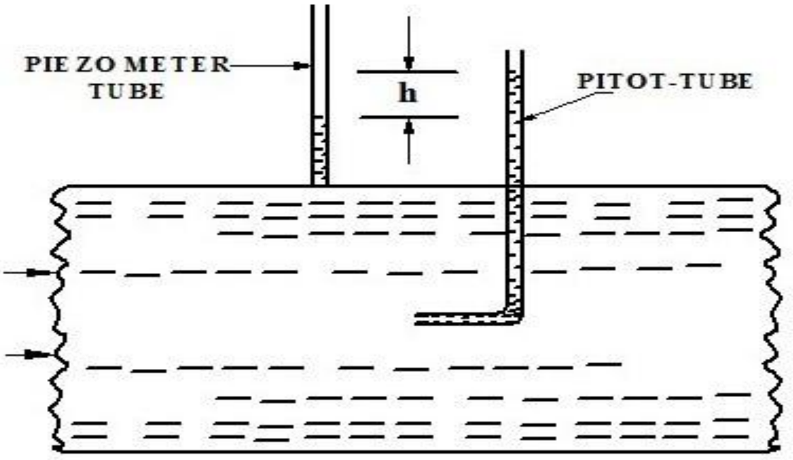


Fig. (b)

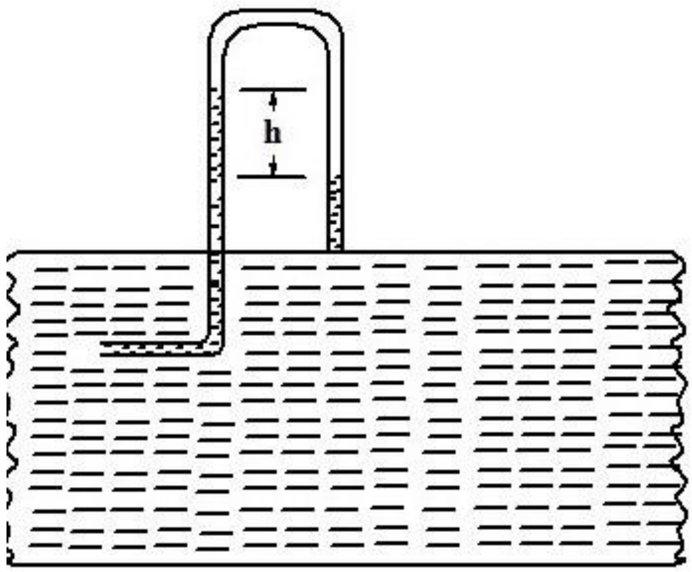


Fig. (c)

1. Pitot-tube along with a vertical piezometer tube as shown in Fig. (b).
2. Pitot-tube connected with piezometer tube as shown in Fig. (c).
3. Pitot-tube and vertical piezometer tube connected with a different U-tube manometer as shown in Fig. (d).

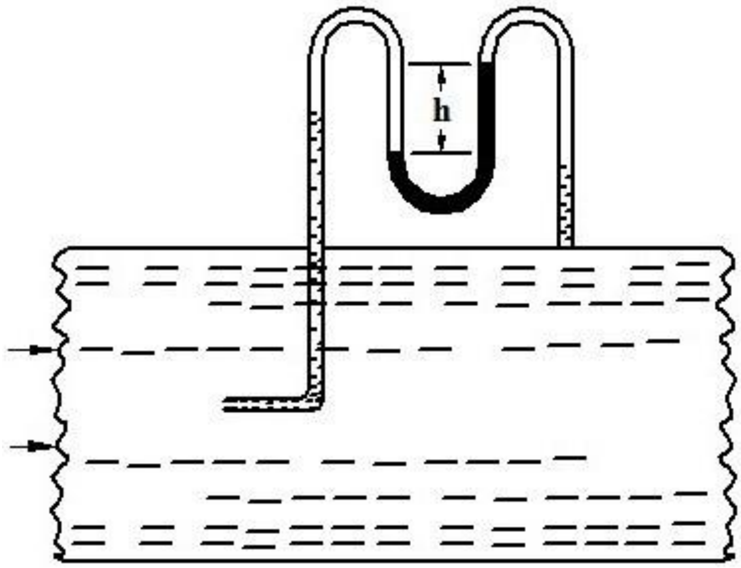


Fig. (d)

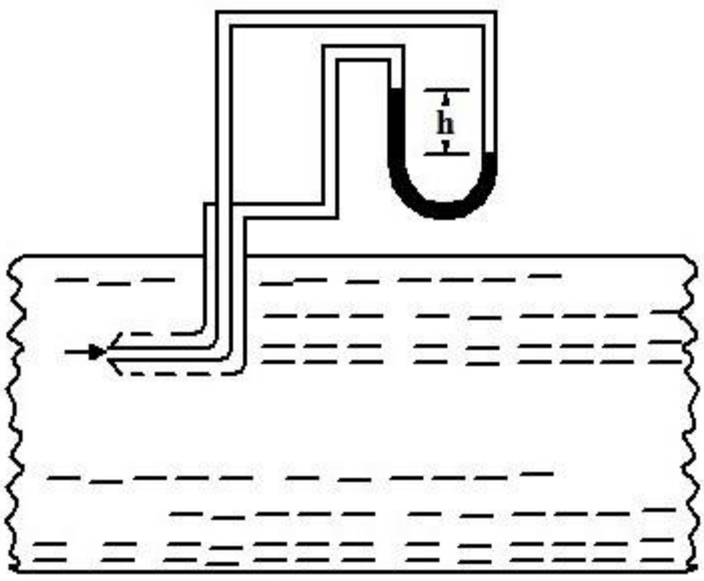


Fig. (e)

4. Pitot-static tube, which consists of two circular concentric tubes one the other with some annular space in between as shown in Fig. (e).
5. The outlet of these two tubes are connected to the different manometer where the difference of pressure head ‘h’ is measured by knowing the Difference of the levels of the manometer liquid say x. then

$h = x\left(\frac{Sg}{So} - 1\right)$

PROCEDURE

1. The diameter of the orifice and the internal plan dimensions of the collecting tank are measured.
2. The supply valve of the orifice tank is regulated and water is allowed to fill the orifice tank to a constant head (h)
3. The out let valve of the collecting tank is closed tightly and the time taken for “H ” rise of water in the collecting tank is noted.
4. The above procedure is repeated for different heads and the readings are tabulated.

MODEL CALCULATIONS

1. Head calculation

$h = x\left(\frac{Sg}{So} - 1\right)$

= ------------------------

$X = h_1 - h_2$ = -----------

2. Velocity calculation

$v = \sqrt{2gh}$ = ----------

3. Discharge calculation

   $Q_a = \frac{AH}{T}$ = - - - - - - - - - - -

4. Velocity calculation

   $V_a = \frac{Qa}{A}$ = - - - - - - - - - - -

5. Error in pitot tube

   $\Phi = \frac{Va}{V}$ = ...

GRAPHS

The following graph is drawn by taking Va on y – axis and $\sqrt{h}$
Va vs $\sqrt{h}$

RESULT

Error of the pitot tube, Φ = -------------------
(From experiment)
Error of the pitot tube, Φ = ---------------------
(From Va vs graph)

OBSERVATIONS AND CALCULATIONS

Table 1. Results of flow through Prandtl Pitot tube

S.No.	Manometer reading, cm			$V = \sqrt{2gh}$	Time for 10 cm rise of water, T sec	$Q_a = \frac{AH}{T}$	$V_a = \frac{Q_a}{a}$	$\phi = \frac{V_a}{V}$
	h_1	h_2	$h = (h_1 \sim h_2)\left(\frac{S_m}{S} - 1\right)$					
1								
2								
3								
4								
5								

EX.NO: **6**. **DATE**:………

FLOW THROUGH NOTCHES

OBJECTIVE:

To determine the coefficients of discharge of the rectangular, triangular and trapezoidal notches.

APPARATUS REQUIRED:

- Hydraulic bench
  Notches – Rectangular, triangular, trapezoidal shape.
- Hook and point gauge
- Calibrated collecting tank Stop watch

THEORY:

A notch is a sharp-edged device used for the measurement of discharge in free surface flows. A notch can be of different shapes – rectangular, triangular, trapezoidal etc. A triangular notch is particularly suited for measurement of small discharges. The discharge over a notch mainly depends on the head H, relative to the crest of the notch, measured upstream at a distance about 3 to 4 times H from the crest. General formula can be obtained for a symmetrical trapezoidal notch which is a combined shape of rectangular and triangular notches. By applying the Bernoulli Equation (conservation of energy equation) to a simplified flow model of a symmetric trapezoidal notch, theoretical discharge Q_{th} is obtained as:

. . . . (1)

Where 'H' is the water head measured above the crest, 'θ' is the angle between the side edges and 'B' is the bottom width of the notch.

When θ=0, this equation is reduced and applicable for rectangular notch or when B=0 (no bottom width) it is applicable for triangular notch. Hence the same equation (1) can be also used for both rectangular and triangular notches by substituting corresponding values (ie θ=0 or B=0).

If Q_{act} actual discharge is known then coefficient of discharge Cd of the notch can be expressed as

$Cd = Q_{act}/Q_{th}$.

DESCRIPTION

In open channel hydraulics, weirs are commonly used to either regulate or to measure the volumetric flow rate. They are of particular use in large scale situations such as irrigation schemes, canals and rivers. For small scale applications, weirs are often referred to as notches and invariablyare sharp edged and manufactured from thin plate material. Water enters the stilling baffles which calms the flow. Then, the flow passes into the channel and flows over a sharp-edged notch set at the other end of the channel. Water comes of the channel in the form of a nappe is then directed into the calibrated collection tank. The volumetric flow rate is measured by recording the time taken to collect a known volume of water in the tank.

A vertical hook and point gauge, mounted over the channel is used to measure the head of the flow above the crest of the notch as shown in Fig. 2.1. Hook gauge can be moved vertically to measure vertical movements.

FORMULA:

Coefficient of discharge , $C_d = \frac{Q_{act}}{Q_{th}}$

a) RECTANGULAR NOTCH

$$Q_{th} = \frac{2}{3}\sqrt{2g}\, B\, H^{3/2}$$

b) TRIANGULAR NOTCH

$$Q_{act} = \frac{\text{Volume Collected}}{\text{Time Taken}}$$

$$Q_{th} = \frac{8}{15}\sqrt{2g}\, H^{5/2} \tan\frac{\theta}{2}$$

c) TRAPEZOIDAL NOTCH

$$Q_{th} = \frac{8}{15}\sqrt{2g}\; H^{5/2} \tan\frac{\theta}{2} + \frac{2}{3}\sqrt{2g}\; B\, H^{3/2}$$

$$Q_{act} = \frac{\text{Volume Collected}}{\text{Time Taken}}$$

OBSERVATION AND COMPUTATIONS – I

A) For Rectangular notchNotch width ‘B’

= m

Initial reading of hook and point gauge h_0=

No.	*Theoretical Discharge Measurement*			*Actual Discharge Measurement*				*Cd*
	h1 (m)	H (m)	Theoretical Discharge, $Q_{th} = \frac{2}{3}\sqrt{2g}\,B\,H^{3/2}$	Water Rise in Collecting Tank R (m)	Time Taken ‘T’ (sec)	Volume of water collected (m3)ct	Discharge, **Q**act	**Qact Qth**
1								
2								
3								
4								
5								
6								

Area of collecting Tank A_{ct} = m^2

Tabulation 2.1 – Determination of Cd of rectangular notch.

Rectangular notch : Average Value of Cd = ……………...….

OBSERVATION AND COMPUTATIONS - II

A) For Triangular notch Notch angle ‘θ’ = 90º or60º

Initial reading of hook and point gauge h_0=

Area of collecting Tank A_{ct} =……

Tabulation 2.2 – Determination of Cd of triangular notch.

No.	*Theoretical Discharge Measurement*			*Actual Discharge Measurement*				*Cd*
	h1 (m)	H (m)	Theoretical Discharge, $Q_{th} = \frac{8}{15}\sqrt{2g}\ H^{5/2} \tan\frac{\theta}{2}$	Water Rise in Collecting Tank R (m)	Time Taken ‘T’ (sec)	Volume of water collected $(m^3)ct$	Discharge, **Q**act	**Qact / Qth**
1								
2								
3								
4								
5								
6								

Triangular notch: Average Value of Cd = ……………...….

OBSERVATION AND COMPUTATIONS III

For Trapezoidal notch

Notch Bottom Width 'B' = m   Notch angle 'θ' = °

Initial reading of hook and point gauge h_0=   Area of collecting Tank A_{ct}= m^2

S.No	*Theoretical Discharge Measurement*			*Actual Discharge Measurement*				*Cd*
	h1 (m)	H (m)	Theoretical Discharge, $Q_{th} = \frac{8}{15}\sqrt{2g}\ H^{5/2}\tan\frac{\theta}{2} + \frac{2}{3}\sqrt{2g}\ ]$	Water Rise in Collecting Tank R (m)	Time Taken 'T' (sec)	Volume of water collected $(m^3)_{ct}$	Discharge, **Q**act	**Qact Qth**
1								
2								
3								
4								
5								
6								

Tabulation – Determination of Cd of trapezoidal notch.
Trapezoidal notch: Average Value of Cd = …………….

MODEL CALCULATIONS:

Actual discharge,

$$Q_{act} = \frac{\text{Volume Collected}}{\text{Time Taken}}$$

= .... m^3/s

Theoretical discharge,

$$Q_{th} = \frac{2}{3}\sqrt{2g}\ B\,H^{3/2}$$

= .... m^3/s - For rectangular notch

$$Q_{th} = \frac{8}{15}\sqrt{2g}\ H^{5/2}\tan\frac{\theta}{2}$$

= .... m^3/s - For triangular notch

Coefficient of discharge , $C_d = \frac{Qact}{Q_{th}}$

=………….

PROCEDURE

Preparation for experiment:

1. Insert the given notch into the hydraulic bench and fit tightly by using bolts in order to prevent leakage.
2. Open the water supply and allow water till over flows over the notch. Stop water supply, let excess water drain through notch and note the initial reading of the water level '*h_0*'using the hook and point gauge. Let water drain from collecting tank and shut the valve of collecting tank after emptying the collecting tank.

Experiment steps:

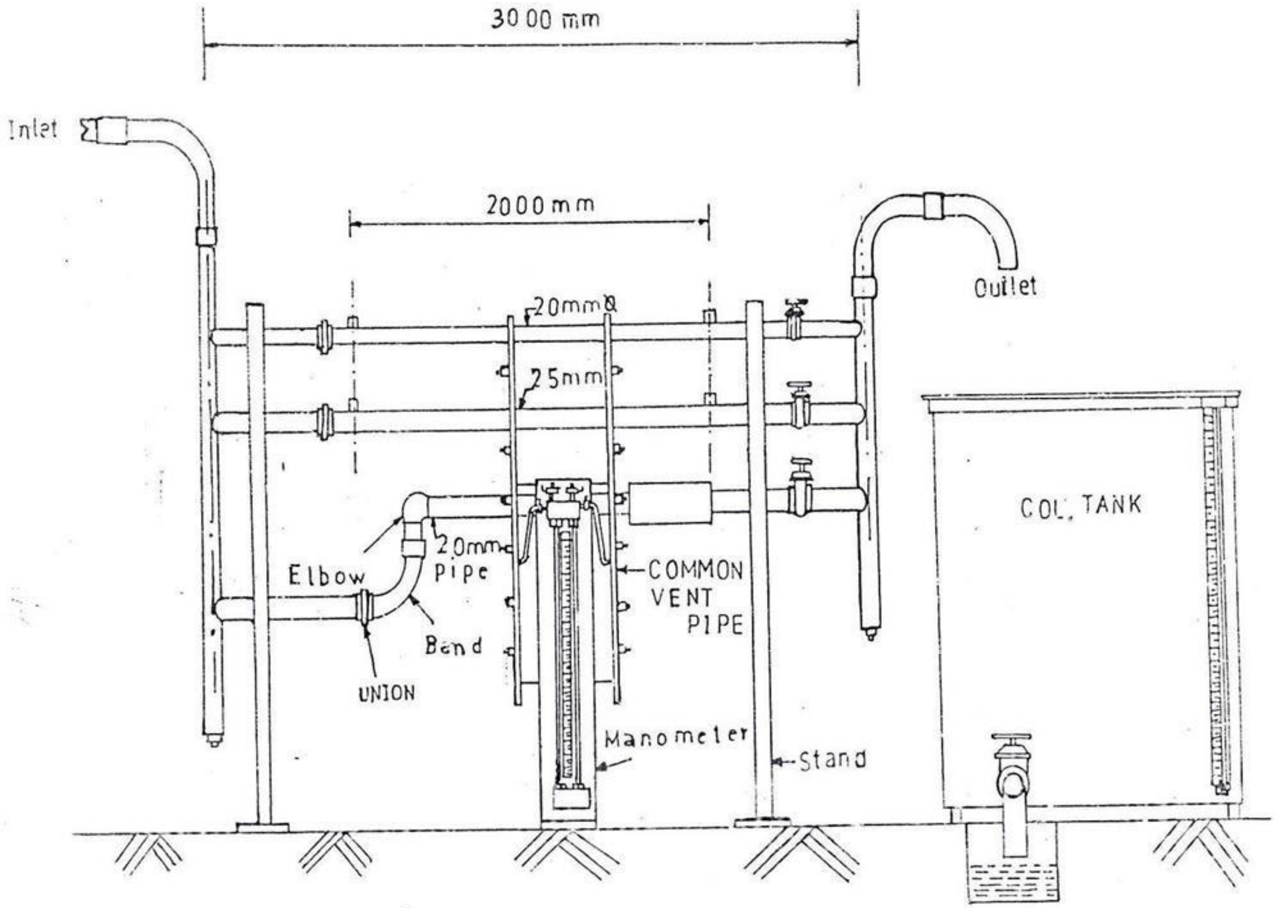
3. After initial preparation, open regulating valve to increase the flow and maintain water level over notch. Wait until flow is steady.
4. Move hook and point gauge vertically and measure the current water level 'h_1' to find the water head 'H' above the crest of the notch.
5. Note the piezometric reading 'z_0' in the collecting tank while switch on the stopwatch.
6. Record the time taken 'T' and the piezometric reading 'z_1' in the collecting tank after allowing sufficient water quantity of water in the collecting tank.

7. Repeat step 3 to step 6 by using different flow rate of water, which can be done by adjusting the water supply. Measure and record the H, the time and piezometric reading in the collecting tank until 5 sets of data have been taken. If collecting tank is full, just empty it before the step no 3.
8. To determine the coefficient of discharge for the other notch, repeat from step

After entering the readings in the Tabulation 2.1 and Tabulation 2.2, compute the necessary values

RESULTS

1. The coefficient of discharge of rectangular notch = …………
2. The coefficient of discharge of triangular notch = …………
3. The coefficient of discharge of trapezoidal notch = ………….



LOSSES IN PIPE LINES (MINOR & MAJOR)

Ex. no: 7 **Date:**

STUDY OF FRICTION LOSSES IN PIPES

AIM:

To determine the coefficient of friction (f) of the given pipe material.

APPARATUS REQUIRED:

1. A pipe provided with inlet and outlet valves
2. U tube Manometer
3. Collecting tank
4. Stop watch
5. Metre scale

THEORY:

When liquid flows through a pipe line, it is subjected to frictional resistance. The frictional resistance depends upon the roughness of the inner surface of the pipe. The loss of head between a selected length of pipe is observed for a measured discharge. The coefficient of friction (f) is calculated by using the expression

$$h_f = 4fLv^2/ 2gd$$

Where,

h_f = loss of head due to friction = $h_1 - h_2$ mm

h_1 = Manometric head in one limb of the manometer mm

h_2 = Manometric head in other limb of the manometer mm

L = Length of pipe between pressure tapping cocks.	mm
V = Velocity of flow in the pipe = Q_a/a	mm/s
Q_a = Actual Discharge = AH/t	mm^3/s
A = Internal Plan area of the collecting tank	mm^2
H = Height of collection in the collecting tank	mm
T = time of collection	sec
a= cross sectional area of the pipe = $\pi d^2/4$	mm^2
d = diameter of pipe	mm
g = acceleration due to gravity = 9810	mm/s^2

PROCEDURE:

1. The diameter of the pipe, the internal plan dimensions of the collecting tank and the length of the pipe line between the pressure tapping cocks are measured,
2. Keeping the outlet valve fully closed, the inlet valve is opened completely.
3. The outlet valve of the collecting tank is closed tightly and the time t required for H rise of water in the collecting tank is observed using a stop watch.
4. The above procedure is repeated by gradually increasing the flow and observing the required readings.
5. The observations are tabulated and the coefficient of friction is computed.

OBSERVATIONS:

Diameter of the pipe d = mm

Length of the pipe L = mm

Internal plan dimensions of collecting tank, Length, L = mm

Breadth, B= mm

Acceleration due to gravity, g = 9810 mm/sec^2

TABULATION:

Sl. No.	Manometric Readings			Time for H = 100mm rise of water(t)	Actual Discharge Q_a = AH/t	Velocity = Q_a/ a	V^2	Co-efficient of friction $f = 2gdh_f/4LV^2$
	h_1	h_2	$h_f=h_1-h_2$					
	mm of water			Sec	mm^3/s	mm/s	$(mm/s)^2$	
1								
2								
3								
4								
5								
6								

Mean Value of f =

MODEL CALCULATION:

1. Area of pipe, a = $\pi d^2 / 4$

= mm^2

2. Internal plan area of collecting tank

= L x B

= mm^2

3. Actual Discharge

$Q_a = AH/t$

= mm^3/ s

4. Velocity

$V = Q_a / a$

= mm/s

5. Coefficient of friction

$f = 2gdh_f / 4Lv^2$

=

GRAPH:

A graph h_f vs. v^2 is drawn taking v^2 on X axis.

RESULT:

The Coefficient of friction of the given pipe (f)

1. Theoretically =
2. Graphically =

Ex. no: 8 **Date:**

STUDY OF MINOR LOSSES IN PIPES

AIM:

To determine the co-efficient of minor loss of the given pipe.

APPARATUS REQUIRED:

1. Piping system
2. Measuring tank fitted with Piezometer
3. Differential U-tube manometer
4. Stop-watch

FORMULA:

Where there is any type of bend in pipe the velocity of flow changer, due to which the separation of the flow from the boundary and also formation of eddies takes place. Thus the energy is lost. The losses of head due to fittings in pipe

i.	Minor loss in bend	h_b	=	$KV_1^2 / 2g$
ii.	Minor loss in contraction	h_c	=	$KV_2^2 / 2g$
iii.	Minor loss in enlargement	h_e	=	$K(V_1-V_2)^2 / 2g$
iv.	Minor loss in elbow	h_d	=	$KV^2 / 2g$

Where, h_b = head loss due to bend
h_c = head loss due do contraction
he = head loss due to enlargement
h_d = head loss due to elbow
K = loss coefficient
V = velocity of fluid
V_1 = velocity of fluid in pipe of small diameter
V_2 = velocity of fluid in pipe of large diameter

2. Area of the collecting tank, L x B = cm^2

L - Length of the collecting tank (cm)

B - Breadth of the collecting tank (cm)

3. Discharge, Q = (A * y) / t cm^3 / s
A = Area of the collecting tank (cm^2)

y = Rise of water in the collecting tank (cm)

t = Time taken for 10 cm rise in the collecting tank (sec)

$a_1 = (\pi d_1^2) / 4$ (d_1 – diameter of the pipe in cm)

$a_2 = (\pi d_2^2) / 4$ (d_2 – diameter of the expanded pipe in cm)

4. Velocity, $V_1 = Q / a_1$

$V_2 = Q / a_2$

5. $h_e = (V_1 - V_2)^2 / 2g$

h_e - head lost due to sudden enlargement

g - Acceleration due to gravity (981 cm / s^2)

PROCEDURE:

1. Close all the valves provided
2. Fill the sump tank 75% with clean water and ensure that there are no foreign particles
3. Fill the manometer with measurement of mercury up to half of its level by opening the pipe from the fitting connected to the bottom most point of the manometer and connect the pipe back to its position
4. Open bypass valve
5. Ensure that ON/OFF switch given on the panel is at OFF position
6. Switch on the main power supply and switch on the pump
7. Open floe channel control valve of pipe for bend, sudden enlargement, sudden contraction and elbow fitting on ball valve and gate valve fitting
8. Open the pressure taps of manometer of related test section, very slowly to avoid the blow of water on manometer fluids
9. Now open the air release valve provided on the manometer, slowly to release the air in manometer
10. When there is no air in the manometer close the air release valves
11. Adjust water flow rate in desired section with the help of control valve for bypass valve
12. Record the manometer reading
13. Measure the flow of water, discharged through desired test section with using stop watch and measuring tank
14. Repeat the experiment for other fittings of selected pipe
15. When experiment is over for selected pipe open the control valve of other pipe
16. Repeat the experiment same procedure for different flow rate of water operating control valve and bypass valve

GRAPH :

A graph ,h_f Vs V_2^2 is drawn taking V_2^2 on X-axis

RESULT

1. The co-efficient of expansion of the given pipe C_e=

    i. Theoretically =

    ii. Graphically =

2. The co-efficient of contraction of the given pipe C_c =

    i. Theoretically =

    ii. Graphically = 3.The co-efficient of bend

of the given pipeC_b =

    i. Theoretically =

    ii. Graphically =

Ex. no: 9 **Date:**

PERFORMANCE CHARACTERISTICS OF A CENTRIFUGAL PUMP

AIM:

To determine the best driving condition of the given centrifugal pump at constant speed and to drive the characteristic curves.

APPARATUS REQUIERED:

1. Centrifugal pump
2. Meter scale
3. Stop watch THEORY:

Centrifugal pumps are used to induce flow or raise a liquid from a low level to a high level. These pumps work on a very simple mechanism. A centrifugal pump converts rotational energy, often from a motor, to energy in a moving fluid.

The two main parts that are responsible for the conversion of energy are the impeller and the casing. The impeller is the rotating part of the pump and the casing is the airtight passage which surrounds the impeller. In a centrifugal pump, fluid enters into the casing, falls on the impeller blades at the eye of the impeller, and is whirled tangentially and radially outward until it leaves the impeller into the diffuser part of the casing. While passing through the impeller, the fluid is gaining both velocity and pressure.

Note:

Since the centrifugal pump is not self-priming, the pump must be filled with water before starting, for this reason water is not allowed to drain and a Foot valve is provided

OBSERVATIONS:

Size of the tank	L=	m
	B=	m
Area of the Tank	A=	m^2
Correction head	X=	m
Energy meter constant	N_e=	rev/Kwh
Specific weight of water	w=	N/ m^2

TABULATION:

		Pressure gauge reading	Pressure head Gx	Vacuum gauge reading	Vacuum head Vx 0.0136	Total Head H = (G x 10) + (V x 0.0136)	Time for 10cmrise of rise of water in the	Actual Discharge	Time for N_r = 10 revolutions in the energy meter "T"	Input Power P_i = 3600 x N / N x T	Output Power Po =	Efficiency = (output power / input power)
i	Kg 2	/cm	'm' of water	'mm' of Hg	'm' of water	'm' of water	Seconds	m^3/sec	Seconds	KW	KW	%

1. Actual discharge $Q_a = AH/t$ (cumec)Where:

A= area of the collecting tank in "m^2"

H= rise of water in the collecting tank in "m"

t = time taken for 10 cm rise in the collecting tank in "sec"

2. Total Head h= GX10 + V x 0.0136 + X (in m of water)

Where:

G= pressure gauge reading

V= vacuum gauge reading

X= correction head in m of water

3. Input power $P_i = 3600 \times N_r / N_e \times T$ (KW)

Where:

Nr= No of revolutions counted in energy meter disc.

Ne= Energy meter constant

T = Time taken for N_r revolutions in energy meter.

4. Output power po = WQH (KW)

Where:

w= specific weight of water (9.81 KN/m^3)

5. Efficiency = output power/ input power x 100 %

PROCEDURE:

1. The internal plan dimensions of the collecting tank are measured.
2. Correction head (The difference in level between the center of the pressure gauge and vacuum gauge) is measured.
3. The energy meter constant is noted down.
4. The motor is started and the following readings are noted down
   a) Pressure gauge and vacuum gauge reading
   b) Time 't' for 10 cm rise of water in the collecting tank
   c) Time 'T' for 10 revolutions in the energy meter
5. The above procedure is repeated for various pressure gauge reading.

MODEL CALCULATIONS:

1. Total Head(h)	=	Gx10 + Vx0.0136 + X	(in 'm' of water)
	=		
2. Actual discharge	=	Qa = Ah/t	(cumec)
	=		
3. Input power (P_i)	=	3600 x N_r / N_e x T	(KW)
	=		
4. Output power po	=	WQH	(KW)
	=		
5. Efficiency	=	output power/ input power x 100 %	
	=		

GRAPHS:

The following graphs are drawn taking discharge on X-axis

1. Discharge Vs Total Head
2. Discharge Vs Input power
3. Discharge Vs Efficiency

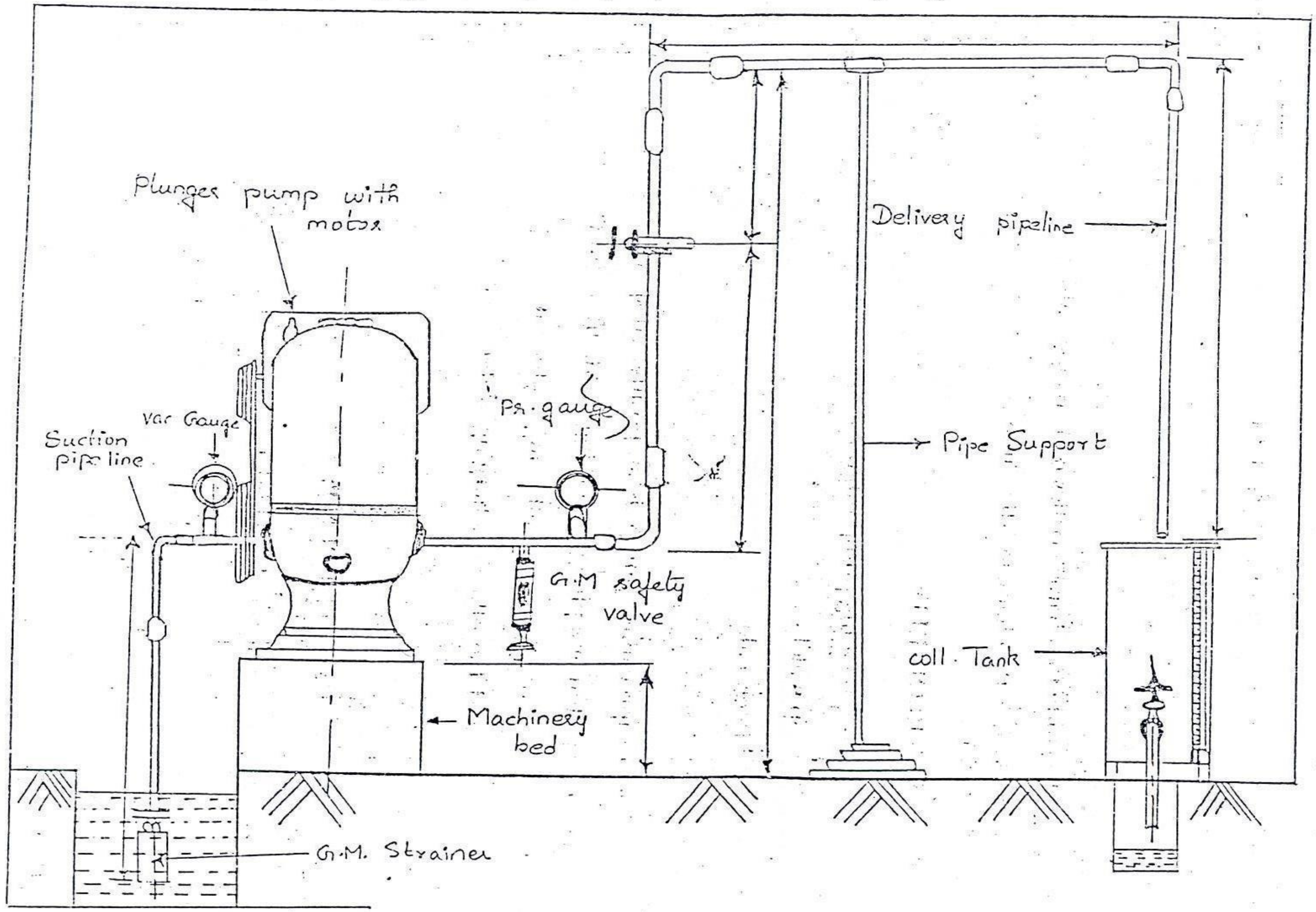
At the point of maximum efficiency of the graph, the corresponding value of discharge, head & Input power arrived from the graph

RESULT:

The characteristic curves are drawn from the point of maximum efficiency. The best driving conditions are found out and these conditions are obtained when,

1. The maximum efficiency =
2. Total head =
3. Discharge =
4. Input power =

EXPERIMENTAL SETUP OF RECIPROCATING PUMP.



Ex. no: 10 **Date:**

PERFORMANCE CHARACTERISTICS OF A CENTRIFUGAL PUMP

AIM:

To determine the best driving condition of the given reciprocating pump at constant speed and to drive the characteristic curves.

APPARATUS REQUIERED:

1. Reciprocating pump
2. Meter scale
3. Stop watch THEORY

A **reciprocating pump** is a class of positive-displacement pumps that includes the piston pump, plunger pump, and diaphragm pump. Well maintained, reciprocating pumps can last for decades. Unmaintained, however, they can succumb to wear and tear.[1] It is often used

where a relatively small quantity of liquid is to be handled and where delivery pressure is quite large. In reciprocating pumps, the chamber that traps the liquid is a stationary cylinder that contains a piston or plunger. During the motion of piston from left to right(refer fig.) a partial vacuum created inside the cylinder. Because of this low pressure water will rise from well through suction tube and fill the cylinder by forcing to open the suction valve. This operation is known as suction stroke.(motion of piston from left to right). In this stroke crank rotates θ=0˚ to θ=180˚. Also delivery valve will be closed and suction valve will be open during this stroke. When the crank rotates from θ=180˚ to θ=360˚ piston moves inwardly from position right to left. Now piston exerts pressure on the liquid and due to which suction valve closes and delivery valve opens.the liquid is then foced up through delivery pipe. This stroke is known as delivery stroke. Now the pump has completed one cycle. The same cycle repeated as the crank rotates.

OBSERVATIONS:

Size of the tank	L=	m
	B=	m
Area of the Tank	A=	m^2
Correction head	X=	m
Energy meter constant	N_e=	rev/Kwh
Length of the stroke	l=	m
Speed of the pump	N=	rpm
Diameter of the cylinder	d=	m
Specific weight of water	w=	N/ m^2

TABULATION:

Sl.No	Pressure gauge reading G	Pressure head G x 10	Vacuum gauge reading "V"	Vacuum head V x 0.0136	Total Head H = (G x 10) + (V x 0.0136)	Time for 10cm rise of rise of water in the tank "t"	Actual Discharge Q= Ah/t	Time for Nr = 10 revolutions in the energy meter "T"	Input Power Pi = 3600 x Nr / Ne x T	Output Power Po = wQH	Efficiency = (output power / input power) * 100
Unit	Kg/cm2	'm' of water	'mm' of Hg	'm' of water	'm' of water	Seconds	m3/sec	Seconds	KW	KW	%
1											
2											
3											
4											
5											

1. Actual discharge $Q_a = AH/t$ (cumec)

Where:

A= area of the collecting tank in "m^2"

H= rise of water in the collecting tank in "m"

t = time taken for 10 cm rise in the collecting tank in "sec"

1. Total Head h= GX10 + V x 0.0136 + X (in m of water)

Where:

G= pressure gauge reading

V= vacuum gauge reading

X= correction head in m of water

2. Input power $P_i = 3600 \times N_r / N_e \times T$ (KW)
Where:

Nr= No of revolutions counted in energy meter disc.

Ne= Energy meter constant

T = Time taken for N_r revolutions in energy meter.

3. Output power po = WQH (KW)
Where:

w= specific weight of water (9.81 KN/m^3)

4. Theoretical discharge $Q_t = 2ALN/60$ Where:
A= Area of the cylinder L= Length of the stroke N= Speed of the Pump

5. % slip= $(Q_t - Q_a)/ Q_t * 100$

6. Efficiency = output power/ input power x 100 %

PROCEDURE:

1. The internal plan dimensions of the collecting tank are measured.
2. Correction head (The difference in level between the center of the pressure gauge and vacuum gauge) is measurement

The energy meter constant is noted down.

3. The motor is started and the following readings are noted down
   a) Pressure gauge and vacuum gauge reading
   b) Time 't' for 10 cm rise of water in the collecting tank
   c) Time 'T' for N_r revolutions in the energy meter
4. The above procedure is repeated for various pressure gauge reading.

MODEL CALCULATIONS:

1. Total Head(h) = Gx10 + Vx0.0136 + X (in 'm' of water)

   =

2. Actual discharge = Qa = Ah/t (cumec)

   =

3. Input power (P_i) = 3600 x N_r / N_ex T (KW)

   =

4. Output power po = WQH (KW)

   =

5. Theoretical discharge Q_t = 2ALN/60
6. % slip= $(Q_t – Q_a)/ Q_t *100$

7. Efficiency = output power/ input power x 100 %

GRAPHS:

The following graphs are drawn taking discharge on X-axis

1. Discharge Vs Total Head
2. Discharge Vs Input power
3. Discharge Vs Efficiency

At the point of maximum efficiency of the graph, the corresponding value of discharge, head & Input power arrived from the graph

RESULT:

The characteristic curves of reciprocating pump are drawn from the point of maximum efficiency. The best driving conditions are found out and these conditions are obtained when,

1. The maximum efficiency =
2. Total head =
3. Discharge =
4. Input power =

Ex. no: 11 **Date:**

PERFORMANCE STUDY ON GEAR OIL PUMP

AIM:

To determine the best driving conditions of the giver gear oil pump at constant speed and to draw the characteristic curves.

APPARATUS REQUIRED:

1. Gear oil pump fitted with all accessories
2. Meter scale
3. Stop watch

FORMULA:

1. Actual Discharge (Q):

Q = Ah/t cumec

2. Total head (H) m of water:

H = Pressure head+ Vaccum head + Correction head (m)

3. Input power:

$I/P = (3600n) / Ect_2$

Where,

N – no.of revolutions

Fc – Energy meter constant (rev/kwh)

T_2 – time for n revolutions(s)

4. Output power:

O/P = VWQH Watts

OBSERVATIONS:

Size of the tank	L=	m
	B=	m
Area of the Tank	A=	m^2
Correction head	X=	m
Energy meter constant	N_e=	rev/Kwh
Specific weight of water	w=	N/ m^2

TABULATION:

Sl.No	Pressure gauge reading G	Pressure head G x 10	Vacuum gauge reading "V"	Vacuum head V x 0.0136	Total Head H = (G x 10) + (V x 0.0136)	Time for 10cm rise of rise of water in the tank "t"	Actual Discharge Q= Ah/t	Time for Nr = 10 revolutions in the energy meter "T"	Input Power Pi = 3600 x Nr / Ne x T	Output Power Po = wQH	Efficiency = (output power / input power) * 100
Unit	Kg/cm2	'm' of water	'mm' of Hg	'm' of water	'm' of water	Seconds	m3/sec	Seconds	KW	KW	%
1											
2											
3											
4											
5											

5. Efficiency:

$$\eta = \{(O/P) / (I/P)\} \text{ X } 100$$

PROCEDURE:

1. The internal dimensions of the collecting tank and the correction head are measured. The energy meter constant is noted.
2. The supply tank is filled with oil to the required height wsay ¾ of the tank.

3. The gate valve is opened in the delivery tube fully.
4. The motor is started and the following readings are noted.
   a. Pressure gauge & vacuum gauge readings
   b. Time for "h" m rise of oil in the tank.
   c. Time for "N_r" revolutions in the energy meter disc
   d. Take 4 or 5 sets of readings varying the delivery pressure.

GRAPH:

The following graph are drawn taking total "h" on X-axis.

1. Head VsDiscaharge
2. Head Vs Input power
3. Head Vs Efficiency

RESULT:

1. Maximum efficiency obtained =
2. Corresponding discharge =
3. Input power =
4. Corresponding total head of water =

EX.NO:12 DATE

PERFORMANCE TEST ON KAPLAN TURBINE

AIM:

To conduct load test on the Kaplan Turbine by keeping the speed as constant and to draw its characteristic curves.

APPARATUS REQUIRED:

- Kaplan turbine set up
- Sump tank
- Notch tank
- Centrifugal pump
- Collecting Tank

THEORY AND DESCRIPTION OF SET UP:

A Kaplan turbine is a type of propeller turbine which was developed by the Austrian engineer V. Kaplan (1876-1934). It is an axial flow turbine, which is suitable for relatively low heads, and hence requires a large quantity of water to develop large amount of power. It is also a reaction type of turbine and hence it operates in an entirely closed conduitfrom the headrace to the tailrace. The main components of Kaplan turbine are scroll casing, stay ring, arrangement of guide vanes, and the draft tube. Between the guide vanes and the runner the water in a Kaplan turbine turns through a right angle into the axial direction and then passes through the runner. The runner of a Kaplan turbine has four or six blades and it closely resembles a ship's propeller. The blades attached to a hub so shaped that water flows axially through the runner. Ordinarily the runner blades of a propeller turbine are fixed, but the Kaplan turbine runner blades can be turned about their own axis, so that their angle of inclination may be adjusted while the turbine is in motion. This adjustment of the runner blades is usually carried out automatically by means of a servomotor operating inside the hollow coupling of turbine and generator shaft.

The whole arrangement is attached to a rectangular notch provided. The whole arrangement is attached to a pump. The loading on the turbine is achieved with an electrical alternator connected to a lamp bank. Control panel on the turbine has digital units to display the turbine speed, head on turbine and electrical en

FORMULA:

1. **DISCHARGE** $Q = C_d A B^2 \sqrt{2gh / (1-B^4)}$

   A = $\pi d^2/4$ where $d_1 = 150mm$

   B= 0.6

   G= Acceleration due to Gravity

   h = (P_1-P_2) X 10 m of water

2. **INPUT POWER (P_i)**$= \frac{\rho g \times Qact \times H}{1000}$

3. **OUTPUT POWER(P_o** $= \pi NDT/ 60000$ **(KW)**

   T = Resultant Load = $((T_2-T_1) + T_o)$ Kg

4. **EFFICIENCY (η)** $\eta = \frac{Output\ power}{Input\ power}$ **X 100 %**

OBSERVATION:

Diameter of the brake drum (D) = ____________

Diameter of the rope (d) =_____________

Weight of lad hanger and rope =_____________

Sl. No	Inlet Pressure (P) *Kg/cm²*	Speed (N) *RPM*	Pressure gauge reading *Kg/cm²*			Discharge (Q) m^3/s	Load *Kg*		Net weight (T) (T2-T1 +To) *Kg*	Input power *(Kw)*	Output power *(kW)*	Efficiency (%)
			P_1	P_2	$h=P_1-P_2$		T_1	T_2				
1												
2												
3												
4												
5												

RESULT:

The best driving conditions of the Kaplan turbine for maximum efficiency condition are:

a. Maximum efficiency =________________*%*

b. Maximum output power =________________***Kw***

c. Maximum Speed =_________________r

EX.NO:13 **DATE: ………..**

CHARACTERISTICS CURVES OF FRANCIS TURBINE

AIM:

To conduct load test on Francis turbine and to study the characteristics of Francis turbine.

APPARATUS REQUIRED :

1. Francis wheel unit
2. Supply pump
3. Venturimeter
4. Brake drum
5. Dead Weight
6. Pressure gauge

THEORY AND DESCRIPTION OF THE SETUP:

A Francis turbine is an inward flow reaction turbine with mixed flow runner, in which water enters at high pressure. Around the runner, a set of stationary guide vanes direct the water into the moving vanes. The guide vanes also serve as gates. The gate openings can be adjusted by a handle. The guide vanes are surrounded by a chamber called ‘spiral chamber’. On the discharge side, the water passes to the tailrace by a tube ‘Draft tube’. The draft tube enables the turbine to be set at a higher level without sacrifice in head. Moreover, it entails regaining of pressure energy, thus increasing the efficiency of the turbines.

The input power supplied to the turbine is calculated from the net supply head on the turbine and the discharge through the turbine. The output power from the turbine is calculated from the readings taken on the rope brake drum and the speed of the shaft. A tachometer is used to measure the speed of the shaft. The efficiency of the turbine is computed from the output and the input.

FORMULA:

1. **DISCHARGE** $\mathbf{Q = C_d A B^2 \sqrt{2gh / (1-B^4)}}$

   A = $\pi d^2/4$ where $d_1 = 150mm$

   B= 0.6

   G= Acceleration due to Gravity

   h = (P_1-P_2) X 10 m of water

2. **INPUT POWER (P_i)=** $\frac{\rho g \times Qact \times H}{1000}$

3. **OUTPUT POWER(P_o = πNDT/ 60000 (KW)**

   T = Resultant Load = $((T_2-T_1) + T_o)$ Kg

4. **EFFICIENCY (η)** $\eta = \frac{Output\,power}{Input\,power}$ **X 100 %**

PROCEDURE:

1. The Francis turbine is started
2. All the weights in the hanger are removed
3. The pressure gauge reading is noted down and this is to be maintained constant for different loads
4. Pressure gauge reading is ascended down
5. The pressure gauge reading and speed of turbine are noted down
6. The experiment is repeated for different loads and the reading are tabulated

GRAPH;

The graph is drawn between speed along x-axis and output power and efficiency along y-axis. At the point of maximum efficiency output power and speed are noted from the graph and the specific speed is computed.

GRAPH TABLE;

Sl. No.	X-axis	Y- axis	
	Speed *Rpm*	**Output power** *kW*	**Efficiency** %
1			
2			
3			
4			
5			

OBSERVATION:

Diameter of the brake drum (D) = ______________

Diameter of the rope (d) =_______________

Sl. No	Inlet Pressure (P) *Kg/cm²*	Speed (N) *RPM*	Pressure gauge reading *Kg/cm²*			Discharge (Q) m^3/s	Load *Kg*		Net weight (T) (T2-T1 +T_o) *Kg*	Input power *(Kw)*	Output power *(kW)*	Efficiency (%)
			P_1	P_2	$h=P_1-P_2$		T_1	T_2				
1												
2												
3												
4												
5												

RESULT:

The best driving conditions of the Francis turbine for maximum efficiency condition are:

1. Maximum efficiency =___________________***%***

2. Maximum output power =___________________***Watts***

3. Maximum Speed =___________________***rpm***

EX.NO: 14 **DATE: ………..**

HYDRAULIC JUMP (DEMONSTRATION)

THEORY

The title type hydraulic flume may be used for uniform flow through open channels(i.e A phenomenon of flow through open channels, in which the rate of flow, depth of flow, area of flow and slope of bed remains constant). The change in any of the above condition causes the flow to be non- uniform.

The specific energy may be defined as energy per unit weight with respect to the datum passing through the bottom of the channel $E= h+ v^2 /\sqrt{2}g$ where h= depth of liquid flow and v= velocity of the liquid.

Critical depth: The depth of water in a channel corresponding to the minimum specific energy is known as critical depth = hc = $q^2/g)^{1/3}$. the velocity of water at critical depth is known as critical velocity $v_c= q/h_c$

A hydraulic jump occurs in an open channel when the flow changes from the supercritical to subcritical. The water level abruptly rises at the hydraulic jump. A large number of roller of turbulent eddies are formed at hydraulic jump, which cause dissipation of energy.

The hydraulic jump is analyzed by applying the impulse momentum equation to a control volume.

EXPERIMENT SET-UP

The set-up consists of a rectangular tilting flume having at the inlet and exit. The water is supplied to the flume by a centrifugal pump. A sluice gate is provided in the middle portion of the flume to create supercritical flow condition on its down stream so that a hydraulic jump can form.

A pointer gauge trolley can move on the rails at the top of the walls for the measurement of the water depths.

An venturimeter is provided on the supply pipe for the measurement of actual discharge.

THEORY

The depth y1 before hydraulic jump and the depth y2

after the hydraulic jump The loss energy in the

hydraulic jump is given by

$$dE= E_1-E_2 = (Y_2-Y_1)^3/ 4Y_1Y_2$$

PROCEDURE

- Measure the width of the flume. Take the pointer gauge at the bed of flume at suitable section upstream and downstream of the hydraulic of the hydraulic jump. H_0
- Open the supply value and adjust the inlet and exit gates so that the flow becomes uniform and steady.
- Gradually lower the sluice gate and adjust the exit gate so that a stable hydraulic jump is formed on the downstream of the sluice gate.
- Note the deflection h of manometer liquid in the u-tube manometer attached to the venturimeter on the supply line for computation of actual discharge

OBSERVATION AND CALCULATION

Width of flume L = 250 mm

Venturimeter dimension

D1 = 40mm ,

D2= 24 mm

Venturimeter coefficient

Cd = 0.95

s.no	VENTURIMETER		WATER DEPTH READING(cm)			Y1= H1''- H0''	Y2= H2''- H0''
	h1(cm)	h2(cm)	H1''	H2''	H0''		

RESULT

The loss energy in the hydraulic jump=

EXP NO :15 **DATE :**

BERNOULLI'S EXPERIMENT

AIM :

To verify the Bernoulli"s theorem.

APPARATUS USED:

1) A supply tank of water a tapered inclined pipe fitted with no.of piezometer tubes point,

2)measuring tank,

3)scale,

4) stop watch.

THEORY

Bernoulli"s theorem states that when there is a continues connection between the particle of flowing mass liquid, the total energy of any sector of flow will remain same provided there is no reduction or
addition at any point.

FORMULA USED

$H1= Z1+ P1/w+V1^2/2g$ $H2= Z2+ P2/w + V2^2/2g$

PROCEDURE

1. Open the inlet valve slowly and allow the water to flow from the supply tank.

2.Now adjust the flow to get a constant head in the supply tank to make flow in and out flow equal.

3.Under this condition the pressure head will become constant in the piezometer tubes. 4.Note down the quantity of water collected in the measuring tank for a given interval of time.

5.Compute the area of cross-section under the piezometer tube. 6.Compute the area of cross-section under the tube.

7.Change the inlet and outletsupply and note the reading.

TABULATIONS:

Location	1	2	3	4	5	6	7	8	9	10	11
Discharge Of piezometer Tube from **inlet(cumec)**											
Area of Crosssection Under foot Of each point (m^2)											
Velocity Of water Under foot Of each point(m/sec)											
$V^2/2g$											
p/ ρ											
p/ ρ+ **$V^2/2g$**											

RESULTS:

Hence, Bernoulli"s theorem verified.

DEPARTMENT OF AERONAUTICAL ENGINEERING

LABORATORY MANUAL

AS3361-THERMODYNAMICS AND STRENGTH OF MATERIALS LABORATORY

LIST OF EXPERIMENTS

Strength of Materials laboratory

1. Tension test on a mild steel rod
2. Double shear test on Mild steel and Aluminum rods
3. Torsion test on mild steel rod
4. Impact test on metal specimen
5. Hardness test on metals - Brinnell and Rockwell Hardness Number
6. Deflection test on beams
7. Compression test on helical springs
8. Strain Measurement using Rosette strain gauge
9. Effect of hardening- Improvement in hardness and impact resistance of steels.
10. Tempering - Improvement Mechanical properties Comparison

    (i) Unhardened specimen

    (ii) Quenched Specimen and

    (iii) Quenched and tempered specimen.
11. Microscopic Examination of

    (i) Hardened samples and

    (ii) Hardened and tempered samples.

STRENGTH OF MATERIALS LABORATORY

GENERAL INSTRUCTION

The following instructions should be strictly followed by the students in the strength of Materials Laboratory.

1. All the students are expected to come to the lab, with shoe, uniform etc., whenever they come for the laboratory class.

2. For each lab class, all the students are expected to come with observation note book, record note book, pencil, eraser, sharpener, scale, divider, graph sheets, French curve etc.

3. While coming to each laboratory class, students are expected to come observation note book prepared for the class.

4. All the students are expected to complete their laboratory work including calculations and get it corrected in the laboratory class itself.

5. While coming to the next lab classes are expected to submit the record note for correction.

6. All the equipments, tools accessories and expensive. Therefore, students are expected to handle the instruments with utmost care during the experiment.

7. The tools and accessories required for conducting the experiments can be obtained from the technician and the same should be returned as soon as the experiment over.

8. Breakage amount will be collected from the student(S) for causing damage to the instruments / equipments due to wrong operation or carelessness.

TABLE OF CONTENTS

TENSION TEST ON MILD STEEL ROD

Ex. No.: **Date:**

Aim:

To conduct a tension test on given mild steel specimen for finding the following:

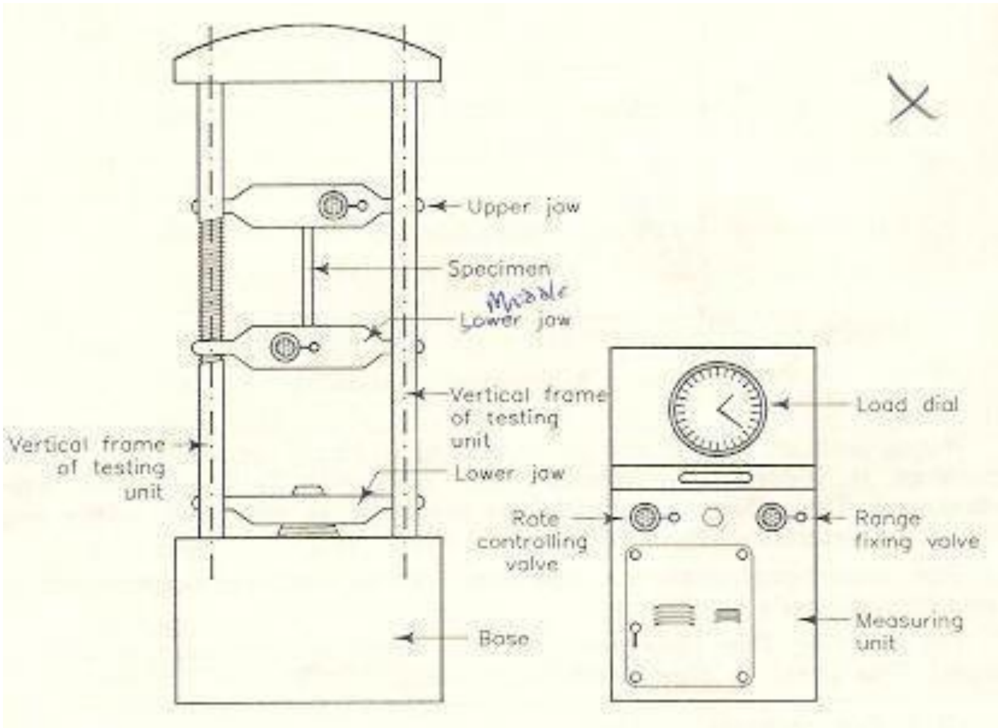
1. Yield stress
2. Ultimate stress
3. Nominal breaking stress
4. Actual breaking stress
5. Percentage Elongation in length
6. Percentage Reduction in area.

Apparatus Required:

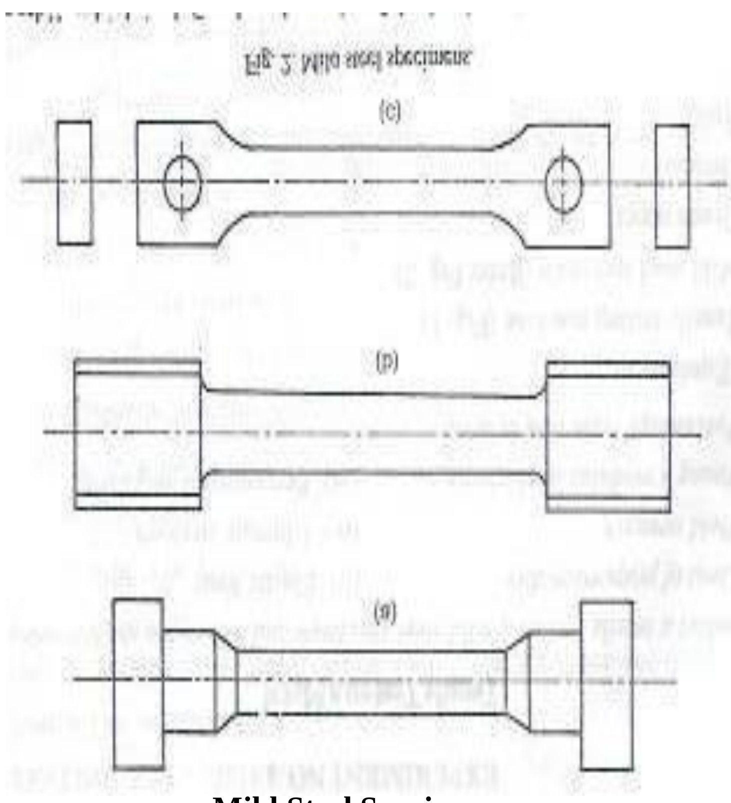
1. Universal Testing machine (UTM)
2. Mild steel specimen
3. Scale
4. Vernier caliper
5. Dot Punch
6. Hammer

Procedure:

1. Measure the length (L) and diameter (d) of the given specimen.
2. Mark the centre of the specimen using dot punch.
3. Mark two points P and Q at a distance of 150mm on either side of the centre mark so that the distance between P and Q will be equal to 300mm.
4. Mark two point A and B at a distance of 2.5 times the rod distance on the either side of the centre mark so that the distance between A & B will be equal to 5 times the rod diameter and is known as initial gauge length of rod. (li).
5. Insert the specimen in the middle cross head and top cross head grip of the machince so that the two points A and B coincide with grips.
6. Apply the load gradually and continue the applications of load. After sometime, there will be slightly pause in the increase of load. The load at this point is noted as yield point (Py).
7. Apply load continuously till the specimen fails and note down the ultimate load (Pa) and breaking load (P_b) from the digital indicator.
8. Remove the specimen from the machine and join the two pieces of the specimens.
9. Measure the distance between the two points A and B. This distance is known as final gauge length (li) of the specimen.
10. Measure the diameter of the rod at neck (dn).
11. Determine the yield stress, ultimate stress, nominal breaking stress, actual breaking stress, percentage elongation in length and percentage reduction in area using the following formula.



Universal testing machine



Mild Steel Specimen

Observation:

1. Material of the specimen =
2. Length of the specimen, L = mm
3. Diameter of the specimen, d = mm
4. Initial gauge length of the specimen I_i = mm
5. Final gauge length of the specimen I_i = mm
6. Diameter at neck d_n = mm
7. Yield load. P_y = KN
8. Ultimate load,P_u = KN
9. Breaking load, P_b = KN

Calculations:

1) Yield stress σ_y $= \dfrac{\text{Yield load } (P_y)}{\text{Initial Area } (A_i)}$

2) Ultimate stress σ_u $= \dfrac{\text{Ultimate load } (P_u)}{\text{Initial Area (Ai)}}$

3. Nominal breaking stress, $\sigma_{bn} = \dfrac{\text{Breaking load } (P_b)}{\text{Initial Area } (A_i)}$

4. Actual breaking stress, σ_{bn} $= \dfrac{\text{Breaking load (Pb)}}{\text{Neck Area } (A_n)}$

5. % Elongation in length $= \left(\dfrac{\text{Final gauge length } (I_i) - \text{Initial gauge length (Ii)}}{\text{Initial gauge length (Ii)}}\right) \text{x } 100$

6% Reduction in area = $\left(\dfrac{\text{Initial area (Ai)} - \text{Neck area } (A_n)}{\text{Initial gauge length (Ii)}}\right) \text{x } 100$

Where Ai = Initial Area = $\pi d^2 /4$

A_n = Area at neck = $\pi d_n^2 / 4$.

Result:

Tension test for the given specimen was conducted and the results are as follows:

1. Yield stress, σy = N/mm^2
2. Ultimate stress, σ_u = N/mm2
3. Nominal breaking stress, σ_{bn} = N/mm^2
4. Actual breaking stress, σ_{bn} = N/mm^2
5. Percentage Elongation in length =
6. Percentage Reduction in area =

TEST FOR TORSION ON MILD STEEL ROUND ROD

Experiment No: **Date:**

AIM:

To conduct torsion test on mild steel round rod and to determine the value of modulus of rigidity and maximum shear stress.

APPARATUS REQUIRED:

a. Torsion testing machine
b. Vernier caliper
c. Steel rule
d. Specimen

FORMULAE USED:

$$\text{Modulus of Rigidity, (C)} = \frac{T \times L}{J \times \theta} \ \text{N/mm}^2$$

$$\text{Maximum Shear Stress, } (\tau) = \frac{T \times R}{J} \ \text{N/mm}^2$$

Where,

T → Torque, N-mm
J → Polar Moment of Inertia, mm^4
L → Gauge Length, mm
θ → Angle of Twist, Radians
R → Mean radius of shaft, mm

PROCEDURE:

1. Before testing, adjust the measuring range according to the capacity of the test piece.
2. Hold the specimen in driving chuck and driven chuck with the help of handles.
3. Adjust the angle measuring dial at zero position, black pointer at the starting position and pen in its required position.
4. Bring the red dummy pointer in line with the black pointer.
5. Start the machine and now the specimen will be subjected to torsion.
6. Take down the value of torque from the indicating dial for particular value of angle of twist (for every 5° of rotation).
7. Repeat the experiment until the specimen breaks into two pieces. Note the value of torque at this breaking point.
8. Tabulate the readings and draw graph between angle of twist and torque.
9. Find the value of T/θ from the graph and find the value of modulus of rigidity.
10. Find the maximum shear stress.

OBSERVATION & TABULATION:

i. Gauge Length (L) = __________ mm

ii. Mean Diameter of Specimen(d):

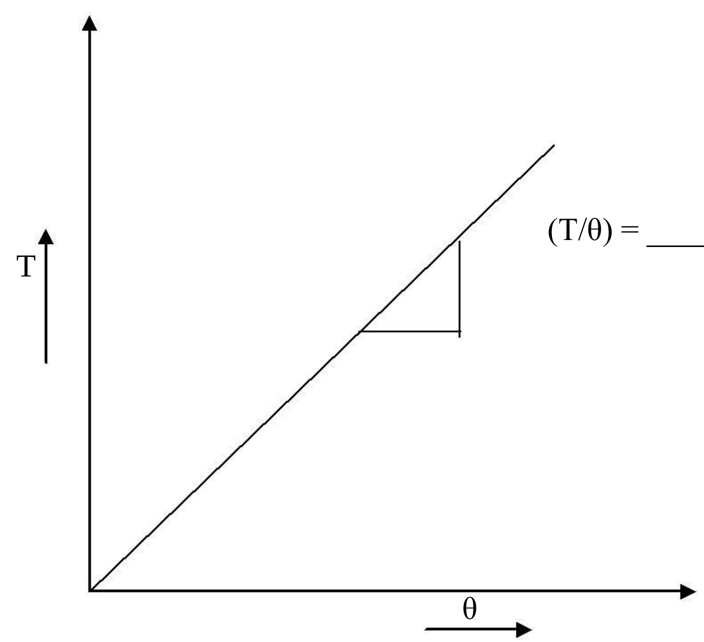
Vernier Caliper Reading: L.C. =__________ mm

Sl. No.	M.S.R. (mm)	V.S.R. (Div)	Observed Reading = M.S.R. + (V.S.R. x L.C.)	Correct Reading = Observed Reading ± Z.C.
			Mean	

Mean Diameter of the specimen (d) = __________ mm.

Sl. No.	Angle of Twist (θ) (Radians)	Torque (T) (N – mm)

GRAPH:



CALCULATIONS:

i. Polar Moment of Inertia (J) = $(\pi/32) \times d^4$

ii. Modulus of Rigidity, (C) $= \frac{T \times L}{J \times \theta}$ N/mm^2

iii. Maximum Shear Stress, (τ) $= \frac{T \times R}{J}$ N/mm^2

RESULT

For the given mild steel round rod specimen

Modulus of Rigidity, (C) = ____________ N/mm^2

Maximum Shear Stress, (τ) = ____________ N/mm^2

CHARPY IMPACT TEST

Ex. No.: **Date:**

Aim:

To determine the impact strength of the given specimen by conducting charpy impact test.

Apparatus and specimen required:

1. Impact testing machine with attachment for charpy test.
2. Charpy specimen
3. Vernier caliper
4. Scale.

Procedure:

1. Measure the length (l), breadth (b), & depth (d) of the given specimen.
2. Measure the position of notch (i.e. groove) from one end (l_g), depth of groove (d_g) and top width of the groove (w_g) in the given specimen.
3. Lift the pendulum and keep it in the position meant for charpy test.
4. Adjust the pointer to coincide with initial position (i.e. maximum value) in charpy scale.
5. Release the pendulum using the lever and note down the initial reading in the charpy scale.
6. Repeat the step 3 and 4.
7. Place the specimen centrally over the supports such that the groove in opposite to the striking face.
8. Release the pendulum again using the lever and note down the final reading in the charpy scale.
9. Find the impact strength of the given specimen by using the following relation: Impact strength = (Final charpy scale reading – Initial charpy scale reading)

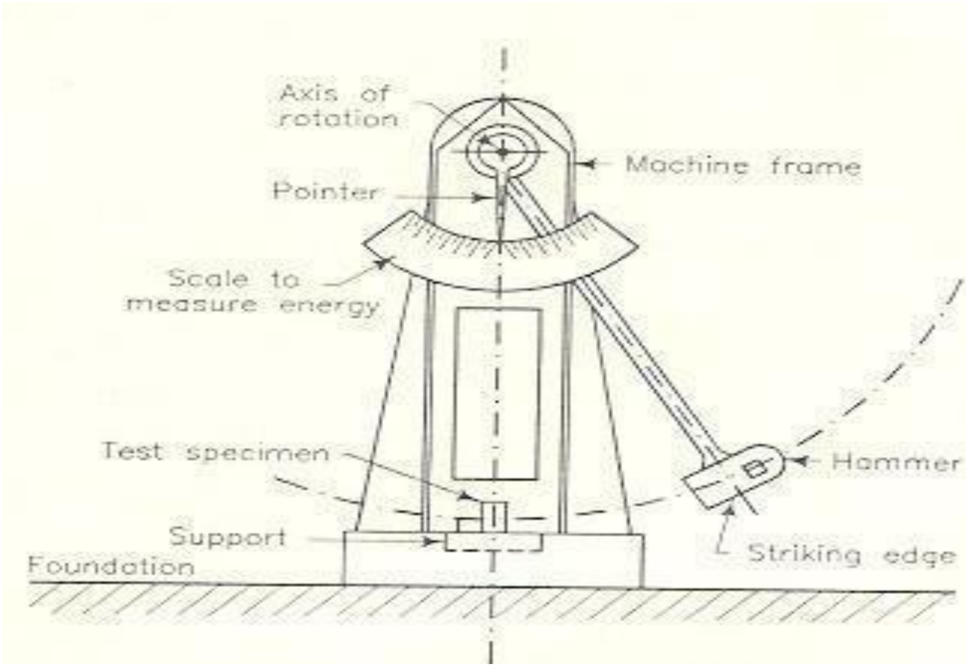
Observation:

1. Material of the given specimen	=	
2. Type of notch (i.e. groove)	=	
3. Length of the specimen, l	=	mm
4. Breadth of the specimen, b	=	mm
5. Depth of the specimen, d	=	mm
6. Position of groove from one end, (l_g)	=	mm
7. Depth of groove (d_g)	=	mm
8. Width of groove (w_g)	=	mm
9. Initial charpy scale reading	=	kg.m
10. Final charpy scale reading	=	kg.m

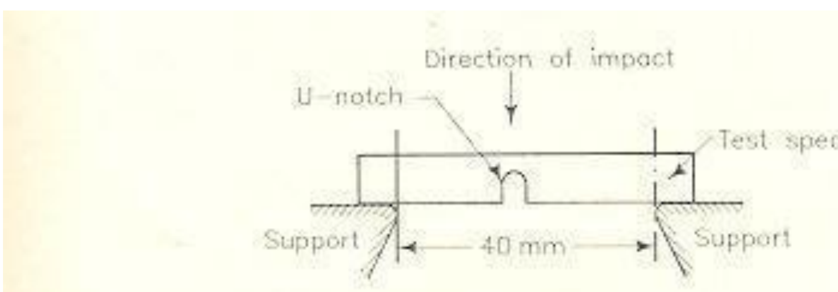
Result:

The impact strength of the given specimen is --------- Kg.m

CHARPY IMPACT TESTING MACHINE



SPECIMEN - CHARPY TEST



Charpy impact test.

FIG. 7-5

IZOD IMPACT TEST

Ex. No.: **Date:**

Aim:
To determine the impact strength of the given specimen by conducting Izod impact test.

Apparatus and specimen required:

1. Impact testing machine with attachment for Izod test.
2. Given specimen
3. Vernier caliper
4. Scale.

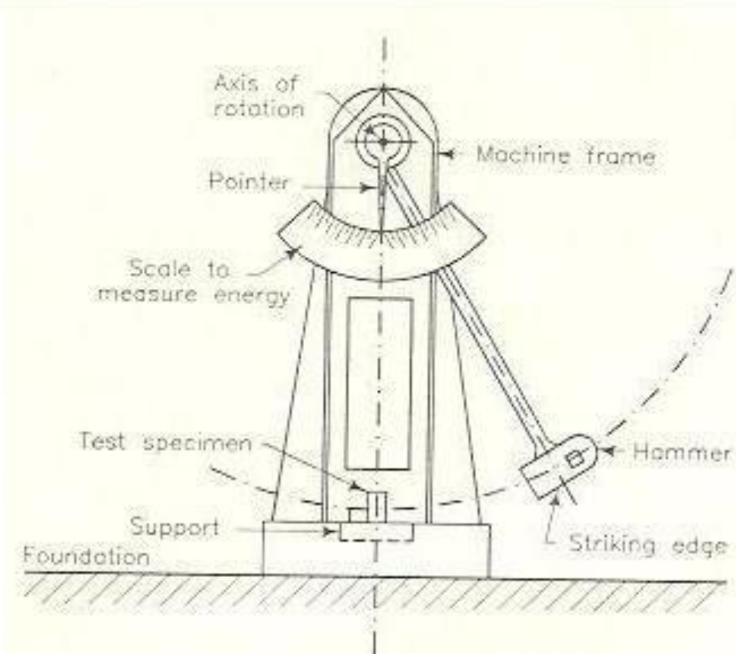
Procedure:
1. Measure the length (l), breadth (b), & depth (d) of the given specimen.
2. Measure the position of notch (i.e. groove) from one end (l_g), depth of groove (d_g) and top width of the groove (w_g) in the given specimen.
3. Lift the pendulum and keep it in the position meant for charpy Izod test.
4. Adjust the pointer to coincide with initial position (i.e. maximum value) in the izod scale.
5. Release the pendulum using the lever and note down the initial reading in the izod scale.
6. Repeat the step 3 and 4.
7. Place the specimen vertically upwards such that the shorter distance between one end of the specimen and groove will be protruding length and also the groove in the specimen should face the striking end of the hammer.
8. Release the pendulum again using the lever and note down the final reading in the izod scale.
9. Find the impact strength of the given specimen by using the following relation: Impact strength = (Final izod scale reading – Initial izod scale reading)

Observation:

1. Material of the given specimen	=	
2. Type of notch (i.e. groove)	=	
3. Length of the specimen, l	=	mm
4. Breadth of the specimen, b	=	mm
5. Depth of the specimen, d	=	mm
6. Position of groove from one end, (l_g)	=	mm
7. Depth of groove (d_g)	=	mm
8. Width of groove (w_g)	=	mm
9. Initial charpy scale reading	=	kg.m
10. Final charpy scale reading	=	kg.m

Result:
The impact strength of the given specimen is --------- Kg.m

IZODE IMPACT TESTING MACHINE



SPECIMEN - IZODE TEST

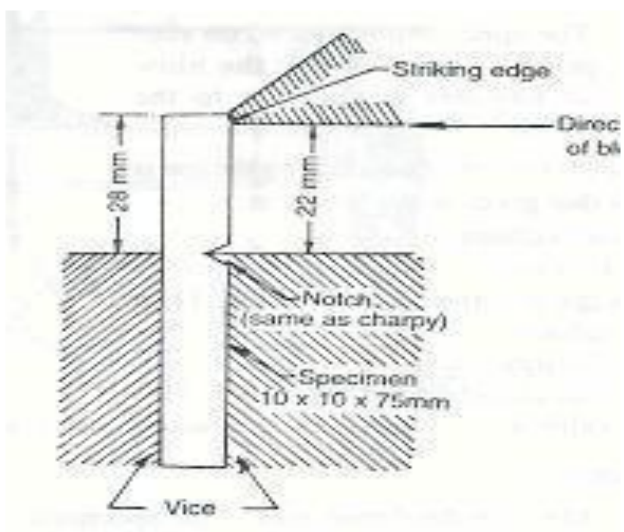


Fig. 10. Izod test.

ROCKWELL HARDNESS TEST

Ex. No.: **Date:**

Aim:

To determine the Rockwell hardness number for the given specimen.

Apparatus Required:

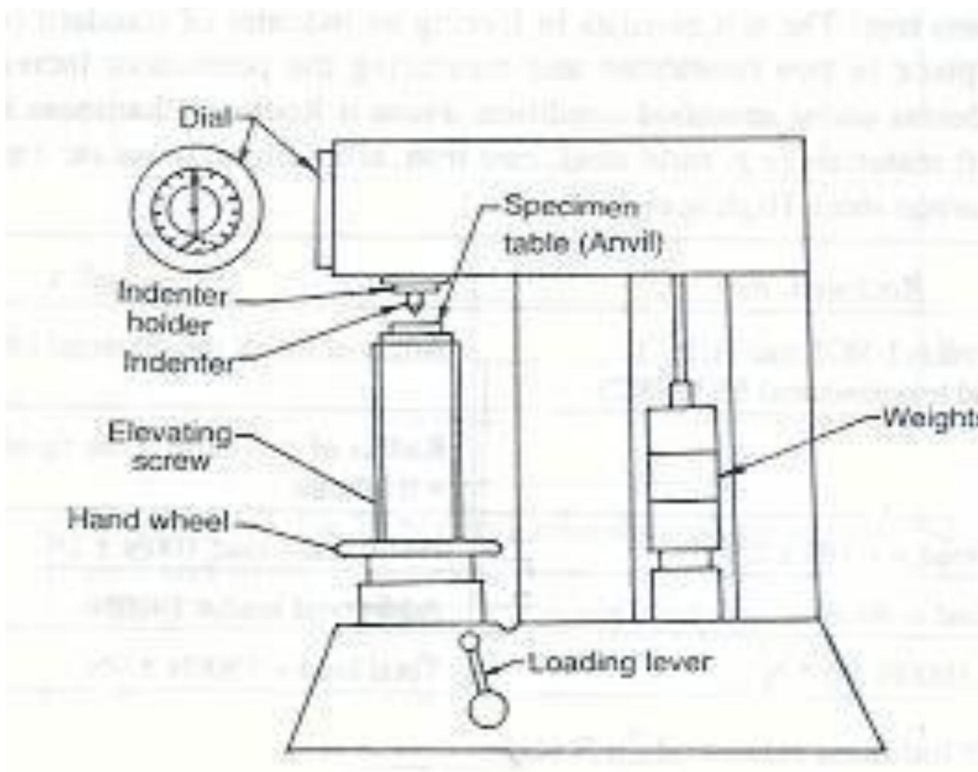
1. Rockwell hardness testing machine
2. Indentor
3. Test specimen
4. Stop watch

Procedure:

1. Identify the material of the given specimen
2. Know the major load, type of indenter and scale to be used for the given test specimen from the following table.

Sl.No.	Material type	Major load	Indenter	Scale
1	Hardened steel	150kg	Diamond cone 120º	C
2	Mild steel	100kg	1.58mm dia, steel ball	B
3	Aluminum	100kg	1.58mm dia. Steel ball	B
4	Brass	100kg	1.58mm dia. Steel ball	B
5	Copper	100kg	1.58mm dia. Steel ball	B

3. Fix the indentor and place the given specimen on the anvil of the machine.
4. Select the major load from the knob available on the right of the machine.
5. Raise the anvil using the rotating wheel till the specimen touches the indentor and then slowly turns the wheel till the small pointer on the dial reaches the red mark position. Now the specimen is subjected to a minor load of 10kg.
6. Push the loading handle in the forward direction to apply the major load to the specimen and allow the load to act on the specimen for 15 seconds.
7. Release the major load by pushing the loading handle in the backward direction and keep the minor 10kg load still on the specimen.
8. Read the Rockwell hardness number either from ‘C’ or ‘B’ scale, as the case may be, directly on the dial and record it.
9. Release the minor load of 10kg by rotating the hand wheel and lowering the screw bar.
10. Repeat the experiment to obtain at least 3 different sets of observations for the given specimen by giving a gap of at least 3mm between any two adjacent indentations and 1.5mm from the edge.
11. Find the average value, which will be the rckwell hardness number for the given specimen.



Rockwell hardness test equipment

Observation:

Sl.No.	Material	Major load	Indentor	Scale	Rockwell hardness number (RHC…… or RHB)
				Average	

Result:
The Rockwell hardness number for the given specimen = RHC ----------- (or) RHB -------

BRINELL HARDNESS TEST

Ex. No.: **Date:**

Aim:

To determine the Brinell hardness number for the given specimen.

Apparatus Required:

1. Brinell hardness testing machine
2. Microscope
3. Indenter
4. Test specimen
5. Stop watch

Procedure:

1. Identify the material of the given specimen
2. Know the value of P/D^2 and diameter of the indenter (D) type to be used for the given test specimen from the following table.

Sl.No.	Material type	P/D^2 value in kg/mm^2	Diameter of steel ball (D) indenter in mm
1	Steel and cast iron	30	2.5
2	Copperand Aluminum Alloys	10	2.5
3	Copper and Aluminum	5	2.5
4	Lead, Tin and Alloys	1	2.5

Where, P = Major load in kg.

3. Calculate the major load to be applied for the given test specimen by knowing the value of PD^2 and D.
4. Select the major load from the knob available on the right of the machine.
5. Fix the indentor and place the given specimen on the anvil of the machine.
6. Raise the anvil using the rotating wheel till the specimen touches the indentor and then slowly turns the wheel till the small pointer on the dial reaches the red mark position. Now the specimen is subjected to a minor load of 10kg.
7. Apply the major load to the specimen by pushing the loading – handle in the forward direction and allow the load to act on the specimen for 15 seconds.
8. Release the major load by pushing the loading handle in the backward direction.
9. Release the minor load of 10kg by rotating the hand wheel and lowering the screw bar.
10. Measure the diameter of indentation (d) using the microscope.

11. Calculate the Brinell hardness number for the given specimen using the following formula:

$$\text{Brinell hardness number} = \frac{\text{Load in kg}}{\text{Spherical area of Indentation of mm}^2}$$

$$= \frac{P}{\pi D/2\,[d - \sqrt{D^2 - d^2}]} \quad \text{kg/mm}^2$$

Where, P = Major load in kg.

D = Diameter of indenter in mm. d

= diameter of indentation in mm.

12. Repeat the experiment to obtain at least 3 different sets of observations for the given specimen by giving a gap of at least 3mm between any two adjacent indentations and 1.5mm from the edge.

11. Find the average value, which will be the Brinell hardness number for the given specimen.

BRINELL HARDNESS TEST

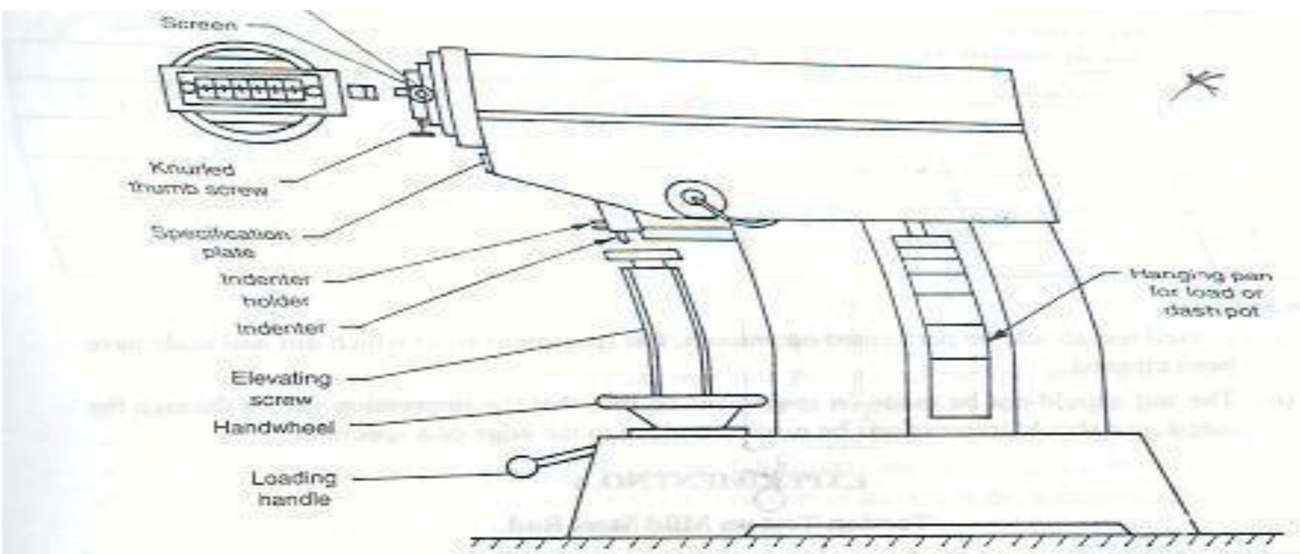


Fig. 7. Brinell hardness tester.

$$\text{Brinell hardness number, } BHN = \frac{2P}{\pi D\,(D - \sqrt{D^2 - d^2})}.$$

Test Requirements/Procedure:

Observation:

Sl.No.	Material	P/D^{2}value in kg/mm^2	Major load (P) in kg	Diameter of steel ball indentor (D) in mm.	Dia of indentation (d) in mm	Brinell hradmess number (BHN) in kg/mm^2
Average						

Result:

The Brinell hardness number for the given specimen = ----------- kg/mm^2

DEFLECTION TEST ON BEAMS

Ex. No.: **Date:**

Aim:

To determine the Young's modulus of the given specimen by conducting bending test.

Apparatus and Specimen required:

1. Bending Test Attachment
2. Specimen for bending test
3. Dial gauge
4. Scale
5. Pencil / Chalk

Procedure:

1. Measure the length (L) of the given specimen
2. Mark the centre of the specimen using pencil / chalk
3. Mark two points A & B at a distance of 350mm on either side of the centre mark. The distance between A & B is known as span of the specimen (l)
4. Fix the attachment for the bending test in the machine properly.
5. Place the specimen over the two supports of the bending table attachment such that the points A &B coincide with centre of the supports. While placing, ensure that the tangential surface nearer to heart will be the top surface and receives the load.
6. Measure the breadth (b) and depth (d) of the specimen using scale.
7. Place the dial gauge under this specimen at the centre and adjust the dial gauge reading to zero position.
8. Place the load cell at top of the specimen at the centre and adjust the load indicator in the digital box to zero position.
9. Select a strain rate of 2.5mm / minute using the gear box in the machine.
10. Apply the load continuously at a constant rate of 2.5mm/minute and note down the deflection for every increase of 0.25 tonne load up to a maximum of 6 sets of readings.
11. Calculate the Young's modulus of the given specimen for each load using the following formula:

Young's modulus, $E = \frac{Pl^3}{48I\delta}$

Where, P = Load in N

L = Span of the specimen in mm

I = Moment of Inertia in mm^4 ($bd^3/12$)

b = Breadth of the beam in mm.

d = Depth of the beam in mm

δ = Actual deflection in mm.

12. Find the average value of young's modulus that will be the Young's modulus of the given specimen.

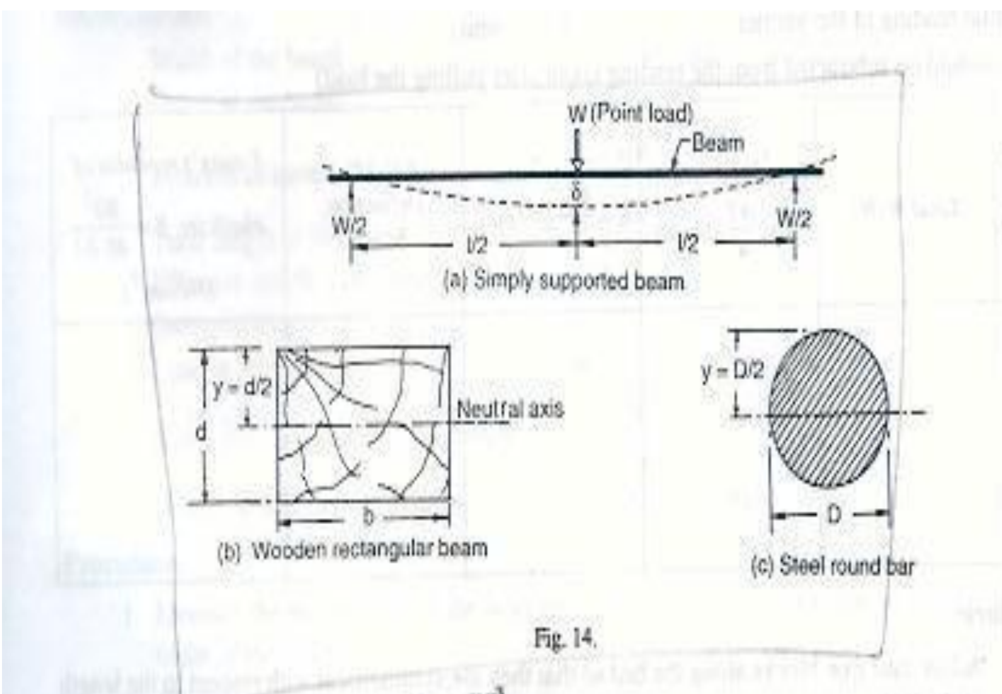
Observation:

1. Material of the specimen =
2. Length of the specimen, L = mm
3. Breadth of the specimen, b = mm
4. Depth of the specimen, d = mm
5. Span of the specimen, l = mm
6. Least count of the dial gauge, LC = mm

Sl.No.	Load in		Deflection in mm		Young's Modulus in N/mm^2
	T	N	observed	Actual	
Average					

Result:

The young's modulus of the given wooden specimen = --------------------- N/mm^2



$$\delta = \frac{Wl^3}{48\,EI}$$

DEFLECTION TEST – SPECIMEN SETUP

TEST ON COMPRESSION SPRING

Ex. No.: **Date:**

Aim:

To determine the modulus of rigidity and stiffness of the given compression spring specimen.

Apparatus and specimen required:

1. Spring test machine
2. Compression spring specimen
3. Vernier caliper

Procedure:

1. Measure the outer diameter (D) and diameter of the spring coil (D) for the given compression spring.
2. Count the number of turns i.e. coils (n) in the given compression specimen.
3. Place the compression spring at the centre of the bottom beam of the spring testing machine.
4. Rise the bottom beam by rotating right side wheel till the spring top rouches the middle cross beam.
5. Note down the initial reading from the scale in the machine.
6. Apply a load of 25kg and note down the scale reading. Increase the load at the rate of 25kg upto a maximum of 100kg and note down the corresponding scale readings.
7. Find the actual deflection of the spring for each load by deducting the initial scale reading from the corresponding scale reading.
8. Calculate the modulus of rigidity for each load applied by using the following formula:

Modulus of rigidity, $N = \underline{64PR^3n}\ d^4\delta$

Where, P = Load in N

R = Mean radius of the spring in mm (D –d /2)
d = Diameter of the spring coil in mm
δ = Deflection of the spring in mm
D = Outer diameter of the spring in mm.

9. Determine the stiffness for each load applied by using the following formula:
   Stiffness, K = P/δ
10. Find the values of modulus of rigidity and spring constant of the given spring by taking average values.

Observation:

1. Material of the spring specimen =
2. Outer diameter of the spring. D = mm
3. Diameter of the spring coil, d = mm
4. Number of coils / turns, n = Nos.
5. Initial scale reading = cm = mm

Sl.No.	Applied Load in		Scale reading in		Actual deflection in mm	Modulus of rigidity in N/mm^2	Stiffness in N/mm
	kg	N	cm	mm			
						Average	

Result:

The modulus of rigidity of the given spring = ------------------- N/mm^2

The stiffness of the given spring = ------------------- N/mm^2

DOUBLE SHEAR TEST ON STEEL BAR

Ex. No.: **Date:**

Aim:

To determine the maximum shear strength of the given bar by conducting double-shear test.

Apparatus and specimen required:

1. Universal Testing machine (UTM)
2. Mild steel specimen
3. Device for double shear test
4. Veriner caliper / screw gauge

Description:

In actual practice when a beam is loaded the shear force at a section always comes to play along with bending moment. It has been observed that the effect of shearing stresses compared to bending stress is quite negligible. But sometimes, the shearing stress at a section assumes much importance in design calculations. Universal testing machine is used for performing shear, compression and tension. There are two types of UTM.1. Screw type2. Hydraulic type. Hydraulic machines are easier to operate. They have a testing unit and control unit connected to each other with hydraulic pipes. It has a reservoir of oil, which is pumped into a cylinder, which has a piston. By this arrangement, the piston is made to move up. Same oil is taken in a tube to measure the pressure. This causes movement of the pointer, which gives reading for the load applied.

Procedure:

1. Measure the diameter (d) of the given specimen.
2. The inner diameter of the hole in the shear stress attachment is slightly greater than that of the specimen.
3. Fit the specimen in the double shear device and place whole assembly in the UTM.
4. Apply the load till the specimen fails by double – shear.
5. Note down the load at which the specimen fails (P).
6. Calculate the maximum shear strength of the given specimen by using the following formula:

$$\text{Maximum shear strength (c/s area in double shear)} = \frac{\text{Load at failure (P) in N}}{\text{2 x cross – sectional area of the bar in mm}^2}$$

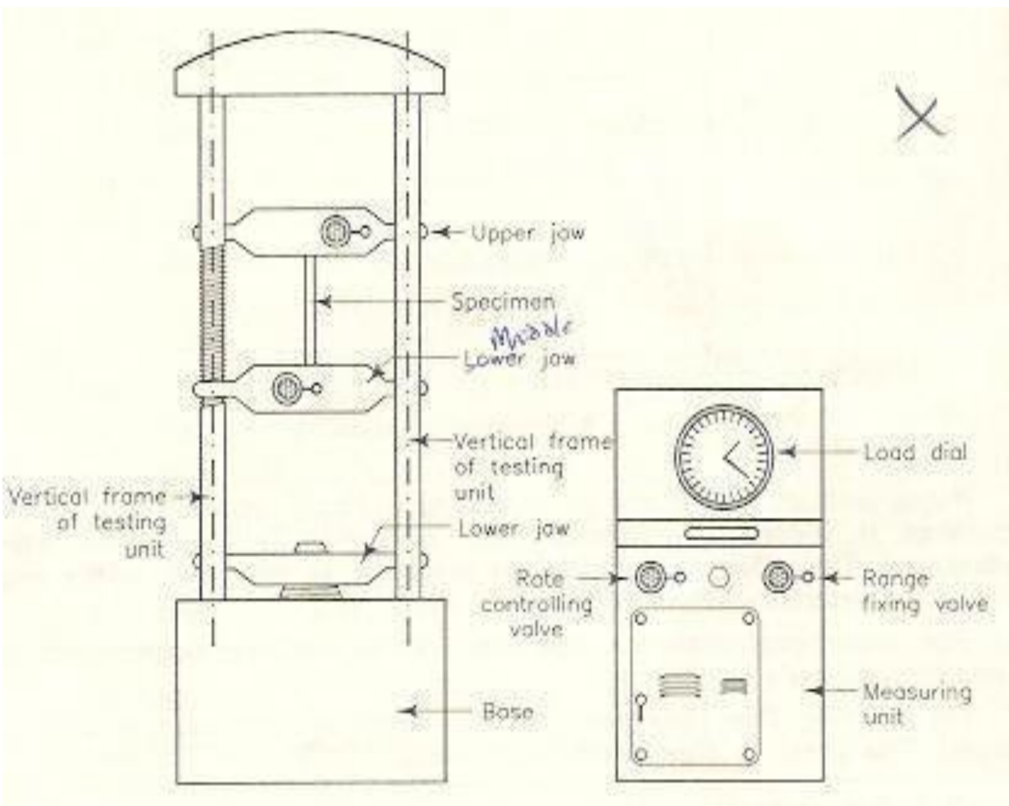
Observation:

1. Material of the specimen =
2. Diameter of the specimen, d = mm
3. Cross sectional area in double shear, (A) = $2 \times \pi d^2/4$ mm^2
4. Shear Load taken by specimen at the time of failure (P) = ------ KN

Result:

The maximum shear strength of the given specimen = ---------------------- N/mm^2

UNIVERSAL TESTING MACHINE



STRAIN MEASUREMENT ON CANTILEVER BEAM

Ex. No.: **Date:**

Aim:

To determine the Strain of the cantilever beam subjected to Point load at the free end and to plot the characteristic curves.

Apparatus required

Cantilever Beam Strainguage Trainer Kit
Weights and Multimeter

Formula used

Strain, $S = 6PL / BT^2E$

Where,
P=Load applied in Kg.
L = Effective length of the beam in cm.
B = Width of the beam in cm.
T = thickness of the beam in cm.
E = young's modulus = $2x10^9 Kg/cm^2$.
S = Micro strain.

Theory:

When the material is subjected to any external load, there will be small change in the Mechanical properties like thickness of the material or change in the length depending upon the nature of load applied to the material. The change in mechanical properties will remain till the load is released. The change in the property is called Strain (or) material gets strained.

Strain $S = \partial L/L$

Since the change in length is very small, it is difficult to measure ∂L, so the strain is measured in micro strain. Since it is difficult to measure the length, Resistance strain gauge are used to measure strain in the material directly. Strain gauges are bonded directly on the material using special adhesive s. As the material get strained due to load applied the resistance of the strain gauge changes proportional to the load applied. This change in resistance is used to convert mechanical property into electrical signal which can be easily measured and stored for analysis.

The change in the resistance of the strain gauge depends on the sensitivity of the strain gauge which is expressed in terms of a gauge factor, S_g

$$S_g = \Delta R /R$$

The output $\Delta R/R$ of a strain gauge is usually converted into voltage signal with a Wheatstone bridge. If a single gauge is used in one arm of Wheatstone bridge and equal but fixed resistors is used in the other arm, the output voltage is $E_o = E_i / 4(\Delta R_g /R_g)$

$$\mathbf{E_o = 1/4(EiS_g \Delta)}$$

The input voltage is controlled by the gauge size and the initial resistance of the gauge. As a result, the output voltage E_O usually ranges between 1 to 10 ΔV / micro units of strain.

Procedure:

1. The instrument is switched on (i.e.,). The display glows to indicate the instrument is ON.
2. The Instrument is allowed to be in ON position for 10 minutes for initial worm-up.
3. From the selector switch, FULL or HALF bridge configuration is selected.
4. The potentiometer is adjusted for ZERO till the displays reads ' 000’
5. 1 Kg load is applied on the pan of the cantilever the CAL Potentiometer is adjusted till the display reads 377 micro strains. When the weights are removed the display should come to ZERO, in case of any variation, ZERO Potentiometer is adjusted again and the procedure is repeated again. Now the instrument is calibrated to read micro strains.
6. Then the loads are applied on the pan in steps of 100 gm up to 1kg. When the cantilever is strained, instrument displays exact micro strain.
7. The readings are noted down in the tabular column . Percentages error in readings, hysteresis and accuracy of the instrument can be calculated by comparing with the theoretical results.

Observation:

Sl.No.	Weight (gms)	Actual readings (using formula) Micro strains	Display readings		Error %
			While loading micro strains	While unloading micro strains	
1	100				
2	200				
3	300				
4	400				
5	500				
6	600				
7	700				
8	800				
9	900				
10	1000				

$$\% \text{ ERROR} = \frac{(\text{Actual reading} - \text{Display reading}) \times 100}{\text{Max Weight (gms)}}$$

Result:

Thus the strain of the cantilever beam subjected to free end loading, is obtained in micro strains and the characteristics curves – Load Vs Strain, Output Voltage Vs Strain and Actual Vs Display readings are plotted.

TEMPERING- IMPROVEMENT MECHANICAL PROPERTIES COMPARISON

Ex. No.: **Date:**

Aim:

To perform the heat treatment tempering on the given material C-40 steel.

Apparatus required:

1. Muffle furnace: tongs
2. Given material: C-40 steel
3. Quenching medium: water
4. Rockwell test setup

Procedure:

Quenching:
It is an operation of rapid cooling by immersing a hot piece into a quenching bath.

Tempering:
It is defined as the process of reheating the hardened specimen to some temperature before the critical range followed by any rate of cooling such are heating permit the trapped temperature to transform and relieve the internal stresses.

1. The given specimen is subjected to Rockwell hardness test and Rockwell hardness number is measured before hardening that the specimen is subjected to rough grinding.
2. The specimen is placed inside the combustion chamber of muffle furnace and is noted up to 830˚C
3. Then the specimen is soaked for 10 minutes at the same temperature 830˚C.
4. After soaking it is taken out from the furnace and it is quenched in the water.
5. The specimen is cooled, now the tempering is completed.
6. Again the specimen is subjected to Rockwell hardness test and Rockwell hardness number is measured.

Tabulation:

S.NO	Specimen Material	Load(Kgf)	Penetration	Scale	RHN

Result:

The heat treatment tempering on the given material C-40 steel and its Rockwell hardness number is measured

1. Rockwell hardness number before tempering =
2. Rockwell hardness number after tempering =

MECHANICAL PROPERTIES FOR UNHARDENED OR HARDENED SPECIMEN

Ex. No.: **Date:**

Aim

To find hardness number and impact strength for unhardened, hardened specimen or Quenched and tempered specimen and compare mechanical properties.

Material and equipment:

Unhardened specimen, Hardened or Quenched and tempered specimen, muffle furnace, Rockwell testing machine, impact testing machine.

Procedure:

HARDENING:

It is defined as a heat treatment process in which the steel is heated to a temperature within or above its critical range, and held at this temperature for considerable time to ensure thorough penetration of the temperature inside the component and allowed to cool by quenching in water, oil or brine solution.

Case (I) - Unhardened specimen

1. Choose the indenter and load for given material.
2. Hold the indenter in indenter holder rigidly
3. Place the specimen on the anvil and raise the elevating screw by rotating the hand wheel up to the initial load.
4. Apply the major load gradually by pushing the lever and then release it as before.
5. Note down the readings in the dial for corresponding scale.
6. Take min 5 readings for each material.

Case (II) - For Hardened specimen

1. Keep the specimen in muffle furnace at temperature of 700˚ to 850˚ for 2 hours
2. The specimen is taken from muffle furnace and quenched in water or oil.3. Then above procedure is followed to test hardness

Case (III) - For Tempered specimen

1. Keep the specimen in muffle furnace at temperature of 650˚ for 2 hours
2. Allow the specimen for air cooling after taking from muffle furnace
3. Then same procedure is followed foe the specimen

Observation:

Rockwell hardness test:

Cases for hardness =

Cross sectional area=

S.No	Material	temperature	Load (Kgf)	Indenter detail	scale	RHN			
						Trial 1	trail 2	Trail 3	Mean
1	Deep casehardened steel								
2	Deep casehardened steel								
3	Mild steel								
4	Mild steel								

CHARPY TEST

S.No	Material and Condition	Energy absorbed(Joules)	Cross-sectional area below the notch(mm)	Impact strength(J/mm)
1	Mild steel-unhardened			
2	Quenched			

Result:

Thus the hardening – heat treatment process is carried out.

MICROSCOPIC EXAMINATION OF (i) HARDENED SAMPLES AND (ii) HARDENED AND TEMPERED SAMPLES.

Ex. No.: **Date:**

Aim:
To prepare a specimen for microscopic examination.

Tools required:
Linisher – polisher grades of emery sheets (rough and Fine), disc polisher, metallurgical microscopes.

Procedure

The specimen preparation consists of following stages:

i) Rough grinding
ii) Intermediate Polishing
Iii) Fine Polishing
iv) Etching

(i) Rough grinding:

It is first necessary for specimen to obtain a reasonable flat surface. This is achieved by using a motor driven energy belt called Linisher-Polisher. The specimen should be kept over the moving belt which will abrade the specimen and make the surface flat. In all grinding and polishing operations, the specimen should be moved perpendicular the existing scratches, so that the deeper scratches will be replaced to a shallower one. This operation is done until the specimen is smooth, free from rust, burs, troughs and deep scratches.

(ii) Intermediate Polishing:

It is carried out using energy paper of cogressively fine grades. The emery paper should be of good quality. The different grades of emery paper used are 120,240,320,400 and 1/0,2/0,3/0,4/0 (Grain size from coarse to fine). The emery paper should be kept against the specimen and moved gently until a fine matrix of uniformly spaced scratches appears on the object. Final grade is then chosen and the specimen is turned perpendicular to the previous direction. This operation is usually done dry.

(iii) Fine Polishing:

An approximate flat scratch free surface is obtained by the use of wet rotary wheel covered with abrasive of alumina powder of 0.05 microns. In this operation, water is used as lubricant and carrier of the abrasive fine scratches and very thin layer produced due to previous operations.

(iv) Etching:

The polished surface is washed with water and etching is done by rubbing the polished surface gently with cotton wetted with etching reagent. After etching the specimen is again washed and then dried, it is then placed under the metallurgical microscope to view the microstructure of it. Thus the specimen is identified.

Result:

Thus the specimen was prepared for microscope observation for its identification.

MADHA
Expertise | Empathy | Excellence
ENGINEERING COLLEGE

DEPARTMENT OF AERONAUTICAL ENGINEERING

LABORATORY MANUAL

AE3411- AERODYNAMICS LABORATORY

AERODYNAMICS LABORATORY

TABLE OF CONTENTS

NOMENCLATURE

SL.NO	SYMBOL	DESCRIPTION
1	C_p	*Pressure coefficient*
2	V	*Free stream velocity of the fluid*
3	p_{atm}	*Atmospheric pressure*
4	p_0	*Total pressure*
5	p	*Static pressure*
6	q	*Dynamic pressure*
7	p_{air}	*Free stream fluid density (air at sea level and 15 °c) is 1.225 kg / m^3*
8	p_w	*Water density is 1000 kg / m^3*
9	g	*Acceleration due to gravity is 9.81 m / s^2*
10	C_l	*Coefficient of lift force*
11	C_d	*Coefficient of drag force*
12	C_s	*Coefficient of side force*
13	F_L	*Lift force*
14	F_D	*Drag force*
15	F_S	*Side force*
16	α	*Angle of attack*
17	S	*Surface area (s) = span (b) × chord (c)*
18	ΔH	*Change in manometer head*
19	μ_∞	*Dynamic Viscosity of air 1.789 x 10^{-5} kg/m s*
20	Re/l	*Reynolds Number per unit length*

INTRODUCTION

Aerodynamics

Aerodynamics is the branch of fluid mechanics dealing with air motion and reaction of a body moving within that of air.

Aerodynamic centre

The point in the chord line about which pitching moment is constant. It will not vary with angle of attack.

Airfoil

The cross section of any surface which can produce aerodynamic lift from the atmosphere.

Angle of attack

Angle between the free stream wind direction and chord line.

Angle of incidence

Angle between the chord line and longitudinal axis of the airplane.

Centre of pressure

The point at which the total resultant pressure force acts.

Chord

It is the straight line joining the leading and trailing edge of an airfoil section.

Drag

It is an aerodynamic force opposing the direction of motion. Drag is inevitable to minimize completely but its effect can be reduced to some extents. It can be due to surface viscosity (friction drag), pressure differences due to shape of an object (form drag), lift acting on a finite wing (induced drag) and other energy loss mechanism in the flow such as wave drag to shockwaves and in efficiencies
in engines.

Drag coefficient (C_D)

It is defined as drag divided by dynamic pressure multiplied by reference area.

C_D = Drag/(Dynamic pressure x Reference area)

Dynamic pressure

It is product of density and square of velocity divided by two.

Lift

It is force acting perpendicular to the direction of flight. Force generated by an airfoil section acting at right angles to airstream flowing past it. In level flights lift should be equal to the weight of aircraft.

Lift coefficient

C_L = Lift/Dynamic pressure x Reference Area

NACA Airfoil

These airfoils are wing cross sections designs invented by NACA organization.

Pressure Coefficient

It is a non dimensional form of pressure.

Stall

Normally the lift increases with the increase in angle of attack. When angle of attack is reached to a certain value, the flow over the upper surface of the wings separates from the body and the lift starts decreasing even with increase in the angle of attack and this condition (decrease in lift with increase in angle of attack) is called stall.

Streamline

The imaginary line along which the tangent at every point will give velocity at that direction.

Wing loading

Wing loading = The total weight of aircraft/Span area of wing.

Wing span

It is the total length of the wing (measured from wing tip to wing tip)

Aspect ratio

Ratio of span to chord of an airfoil. Hence a high aspect ratio wing has great span and narrow chord and vice – versa.

Boundary layer

The thin layer adjacent to the body where velocity varies from zero on the body and 0.99 V_α on the edge of the boundary layer.

EXP NO : 01 DATE :	CALIBRATION OF SUBSONIC WIND TUNNEL

Aim

To calibrate subsonic test section velocity for various rotational speed of propeller and to plot the variation of velocity and static pressure inside the test section with the propeller rotational speed.

ApparatusRequired

- Subsonic Wind Tunnel
- Pitot Static probe
- U – Tube Manometer

Theory

Wind tunnel is referred to a facility which provides a controllable flow field fortesting aerodynamic models and studying flow phenomena. Low speed is referred tothe air flow speed lower than 100 m/s, for which the incompressible flow condition is satisfied.

Types of wind tunnels

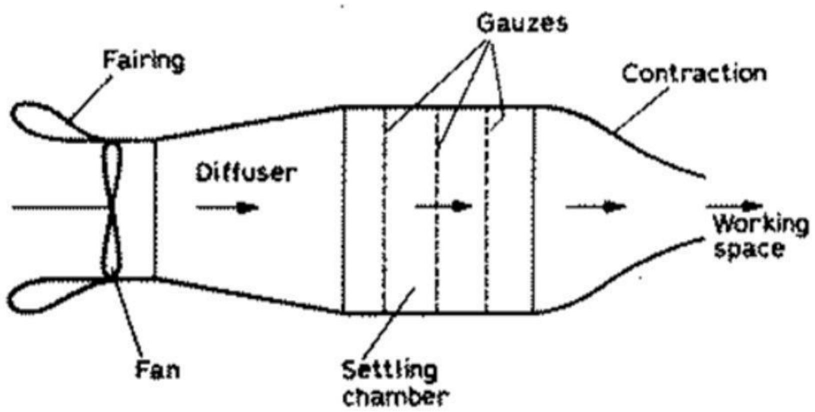
Based on set up models

- Closed-type wind tunnel
- Open-type wind tunnel
- Aerodynamic wind tunnel
- Environmental wind tunnel
- Low turbulence level wind tunnel

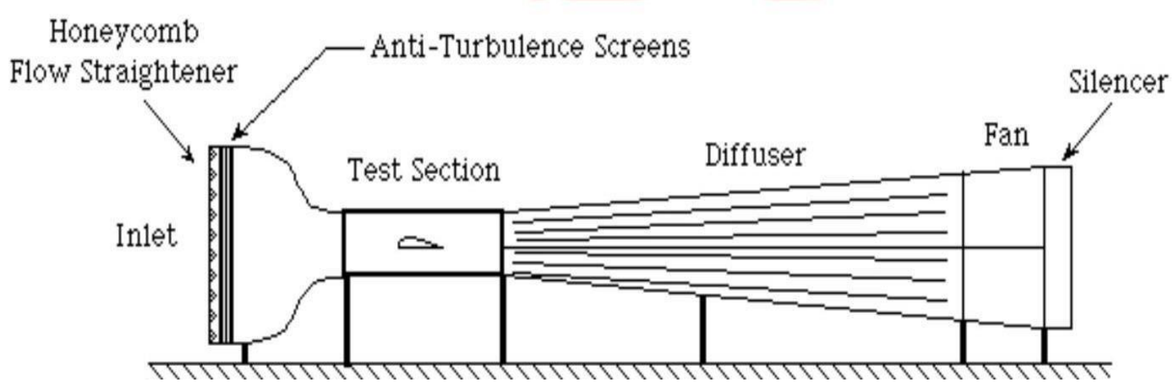
Based on cross section of the test section

- Rectangular (general purpose)
- Circular (axi-symmetric model)
- Elliptical (aircraft model)

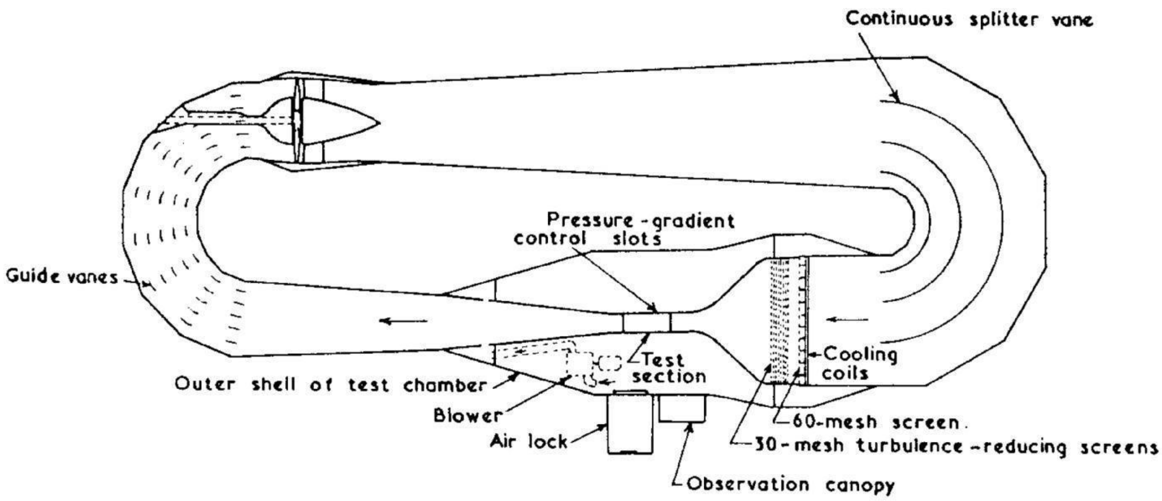
Open-type (blower) wind tunnel



Open type (Suction) wind tunnel



Closed circuit



Components of wind tunnel

Fan drive: provide a pressure increase of flow, to overcome the pressure loss in the tunnel circuit.

Test section: provide desirable flow condition and space for model testing or experiment, where **the** instrumentation is situated. (Reynolds number is of the major concern to manage the issue of dynamic similarity.)

Diffuser: a device to lower the air flow speed, consequently reduce the pressure loss due to Friction.

Guide vanes: to guide the flow through the turning duct, and reduce the extent of secondary flows.

Transition duct: the device to connect the upstream and downstream components of Different cross-sectional shapes.

Settling Chamber: A large space in front of the Nozzle to lower the air flow speed, and to maintain the flow in uniform distribution and lower turbulence intensity.

Nozzle (Convergent section): to accelerate the flow speed to reach the desirable level in the test section, meanwhile reduce the turbulence intensity.

FormulaUsed

Dynamic Pressure, $q = \frac{\rho_\infty V_\infty^2}{2} = \rho_W \, g \, \Delta H$

Static Pressure, $p_\infty = p_{tot} - q$

Velocity of the test section, $V_\infty = \left(\frac{2q}{\rho_a}\right)^{\frac{1}{2}}$

Reynolds Number per unit length, $\frac{Re}{m} = \frac{\rho_\infty V_\infty}{\mu_\infty}$

Where,

$\Delta H = H_1 - H_2$ Difference in head column of U- tube Manometer (m)

q - Dynamic pressure (N/m^2)

$\rho_\infty = \rho_a$ - Density of free stream air 1.225 kg/m^3

g – Acceleration due to gravity m/s^2

V_∞ - Velocity of air inside the test section (m/s)

ρ_w - Density of water 1000 kg/m^3

p_∞ - Static pressure(N/m^2)

p_{tot}- Total Pressure or atmospheric pressure (N/m^2)

Re/l - Reynolds Number per unit length

μ_∞ - Dynamic Viscosity of air 1.789×10^{-5} kg/(m.s)

Procedure

- Make sure there is no model in the test section of wind tunnel.
- Switch on the motor and allow the air to flow through the test section.

- Now increase the propeller speed to 150 rpm and wait for 30 sec to establish uniform flowinside the test section.
- Note down the reading in right and left limb i.e h_1& h_2 of U – tube Manometer
- Increase the propeller speed in step of 50 RPM from 150 to 750 RPM and note down thereading.
- Once reading is noted down decrease the speed of propeller to 100 rpm and switch off themotor.
- Calculate the static pressure and velocity in test section using the above formula.

Tabulation

Calculation of Test Section Velocity

P_{atm}=_________________ on _______________

T_{atm}= _________________ on _______________

ρ_{atm} = _________________ on _______________

S.No	Speed of Propeller RPM	Manometer Reading *(x 10 $^{-2}$m)*			Dynamic Pressure	V_∞	Static Pressure	Reynolds Number per unit length
		H_1	H_2	H_1-H_2				

Calculation

Graph

Speed of propeller vs Velocity of test section.

Speed of propeller vs Static pressure.

Result

Thus, the plot for RPM vs Test section Velocity has been drawn.

EXP NO : 02 DATE :	DETERMINATION OF LIFT FOR THE GIVEN AIRFOIL SECTION

Aim

To determine the Lift for the Given Airfoil Section**.**

Apparatus Required

1) Low speed wind tunnel
2) Symmetric aerofoil model
3) Cambered aerofoil

Formula Used:

Lift Coefficient $$C_L = \frac{L}{\frac{1}{2}\rho V^2 S}$$

Drag Coefficient $$C_D = \frac{D}{\frac{1}{2}\rho V^2 S}$$

L = Lift in N
D = Drag in N
ρ = Density of air in kg/m^3
V = Velocity in m^3/s
S = Wetted area in m^2

Procedure:

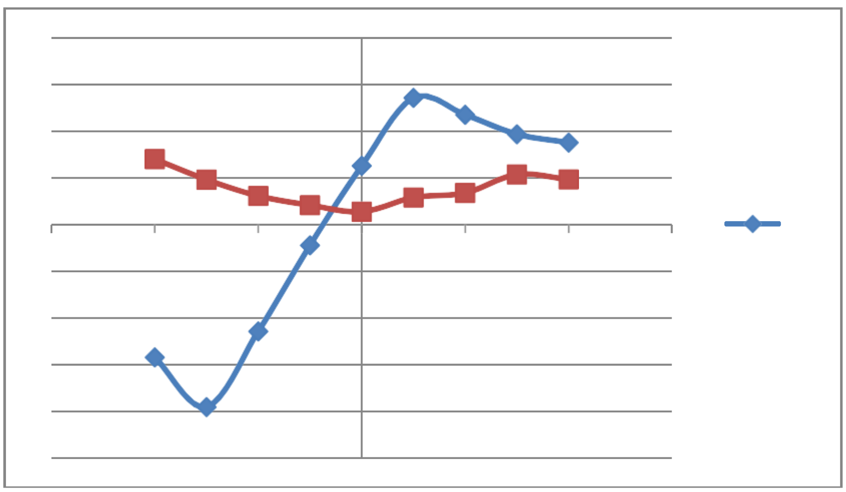
a) Prepare a wind tunnel and calibrate it with the lift drag balance and ensure it is fully serviceable.
b) The operating instructions are to be meticulously followed.
c) Fix the model on the vertical string and lock it.
d) Close the test section and ensure that no items are left inside the test section before closing.
e) Blank all the points.
f) Set the lift force indicator to zero.
g) Fix the required air velocity using the velocity indicator
h) Now by changing the angle of attack the corresponding lift force is noted down.
i) The same is repeated for different angle of attacks.

TABULAR COLUMN

Sl.No.	Angle of attack (α)	L	D	C_L	C_D

Graph: aerofoil (symmetric and cambered)

1) Coefficients of lift Vs angle of attack.
2) Coefficient of drag Vs angle of attack.
3) C_D Vs C_L.



Result:

Thus, the Lift for the Given Airfoil Section were determined and the graphs are plotted between C_L and α.

EXP NO : 03 DATE :	PRESSURE DISTRIBUTION OVER SMOOTH CIRCULAR CYLINDER

Aim

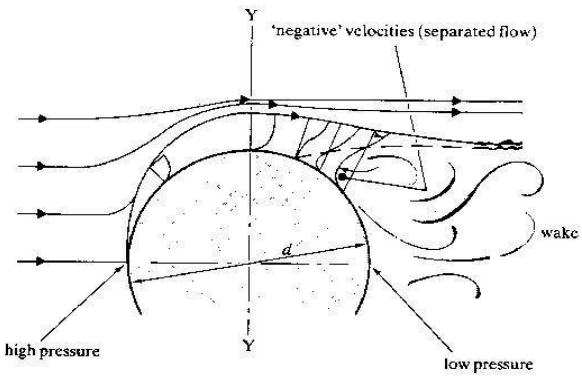
To determine the surface pressure coefficient over smooth circular cylinder for different Reynolds number and plot the variation of pressure coefficient along surface and for different Reynolds numbers.

Apparatus Required

- Subsonic Wind tunnel
- Smooth circular cylinder
- Multi - tube manometer
- Pitot - Static Tube

Theory

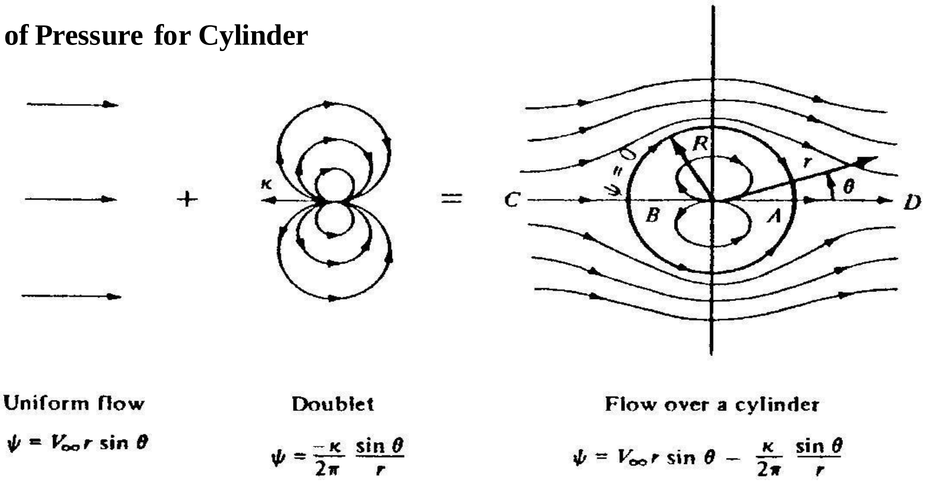
A body moving through a fluid experiences a drag force, which is usually divided intotwo components: frictional drag and pressure drag. Frictional drag comes from friction between the fluid and the surfaces over which it is flowing. This friction is associated with the development of boundary layers, and it scales with Reynolds number as we have seen above. Pressure drag comes from the eddying motions that are set up in the fluid by the passage of the body. This drag is associated with the formation of a wake, which can be readily seen behind a passing boat, and it is usually less sensitive to Reynolds number than the frictional drag. Formally, both types of drag are due to viscosity (if the body was moving through an inviscid fluid there would be no drag at all), but the distinction is useful because the two types of drag are due to different flow phenomena. Frictional drag is important for attached flows (that is, there is no separation), and it is related to the surface area exposed to the flow. Pressure drag is important for separated flows, and it is related to the cross-sectional area of the body.



We can see the role played by friction drag (sometimes called viscous drag) and pressure drag (sometimes called form drag or profile drag) by considering an airfoil at different angles of attack. At small angles of attack, the boundary layers on the top and bottom surface experience only mild pressure gradients, and they remain attached along almost the entire chord length. The wake is very small, and the drag is dominated by the viscous friction inside the boundary layers.

However, as the angle of attack increases, the pressure gradients on the airfoil increase in magnitude. In particular, the adverse pressure gradient on the top rear portion of the airfoil may become sufficiently strong to produce a separated flow. This separation will increase the size of the wake, and the pressure losses in the wake due to eddy formation Therefore the pressure drag increases. At a higher angle of attack, a large fraction of the flow over the top surface of the airfoil may be separated, and the airfoil is said to be stalled. At this stage, the pressure drag is much greater than the viscous drag.

Co-efficient of Pressure for Cylinder



Superposition of Uniform flow over Doublet flow

Radial velocity

$$V_r = \frac{1}{r}\frac{\partial\psi}{\partial\theta} = \frac{1}{r}(V_\infty r\, cos\theta)(1 - \frac{R^2}{r^2})$$

$$V_r = (1 - \frac{R^2}{r^2})(V_\infty cos\theta)$$

Tangential velocity

$$V_\theta = -\frac{\partial\psi}{\partial r} = -[(V_\infty r\, sin\theta)\frac{2R^2}{r^3} + \left(1 - \frac{R^2}{r^2}\right)(V_\infty sin\theta)]$$

$$V_\theta = -\left(1 + \frac{R^2}{r^2}\right)(V_\infty sin\theta)$$

$$when\, r = R$$

$$V_r = 0$$

$$V_\theta = -2(V_\infty sin\theta)$$

$$C_p = 1 - \left(\frac{V}{V_\infty}\right)^2$$

Formula Used

$$C_p = 1 - 4\, sin^2\theta$$

Velocity of test section $\boldsymbol{V} = \sqrt{\frac{2(p0-p)}{\rho}}$

Coefficient of pressure $C_p = \frac{p - p\infty}{0.5\rho V^2{}_\infty}$

Reynolds Number per unit length, $\frac{R_e}{m} = \frac{\rho_\infty V_\infty}{\mu_\infty}$

p_0–Total Pressure=$p_{atm}+\rho_w g\Delta H$

p –Static Pressure=$p_{atm}+\rho_w g\Delta H$

Where,

ΔH –Change in manometer head

ρ_w–Density of Water(Liquid used in multi-tube manometer)

ρ–Density of Air (in laboratory)

p – Pressure at a tapping

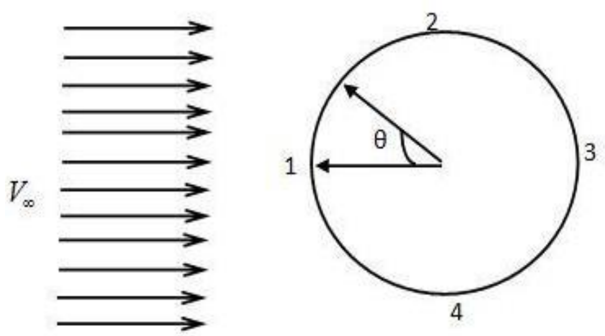
p_∞–Freestream static pressure

V_∞–Freestream velocity

g – Acceleration due to gravity m/s^2

Re/l - Reynolds Number per unit length

μ_∞ - Dynamic Viscosity of air 1.789 x 10^{-5} kg/ (m s)



Procedure

- Mount the circular cylinder model in the test section of wind tunnel.
- Connect the pressure port of the circular cylinder to the multi-tube manometer.
- Set the manometer levels without air bubbles by turning the knobs on and off.
- Note down the initial readings from the manometer before starting the experiment.
- Switch on the motor,
- Increase the RPM of the propeller to 250 rpm using the controls.
- Note down the reading in manometer for connected ports (0,90,180,270)
- Now change the orientation of port 1 to 30^0 using rotating knob (angle changing mechanism) attached in test section to flow direction note down head reading using multi-tube manometer for angular position 30,120,210,300.
- Now change the orientation of port 1 to 60^0 using rotating knob attached in test section to flow direction note down head reading using multi-tube manometer for angular position 60,150,240,330.
- Now bring the port to initial position that is to 0^0 orientation to flow direction.
- Repeat the step 6 to 9 for 350,450,550 RMP of the propeller.
- After the experiment reduce the speed of propeller to 100 RPM and switch off the driver and Main remove the cylinder from the tunnel.
- Calculate the co-efficient of pressure from manometer readings and compare with theoretical formulae.

Tabulation

Calculation of Test Section Velocity

P_{atm}=on

T_{atm}=on

ρ_{atm} =on

Sl. No.	**RPM**	**Pitot Reading, p_0**				**Static Reading, p**				$V = \sqrt{\frac{2(p_0 - p)}{\rho}}$
		H_1	H_2	$H_1 – H_2$	$p_{atm}+\rho_w g\Delta H$	H_1	H_2	$H_1 – H_2$	$p_{atm}+\rho_w g\Delta H$	

Calculation of *Cp*

RPM						
Velocity						
Port No	**Angular position**	**Static pressure at Tap, *p***			$p_{atm} + \rho_w g \Delta H$	$C_p = \frac{p - p_\infty}{0.5\,\rho V^2_\infty}$
		H_1	H_2	$H_1 - H_2$		

Calculation

Precautions

- Check the manometer level properly without any errors.
- Take care while increasing the speed.
- Do not stand behind the wind tunnel while operating it.

Graph

Pressure coefficient vs. Angular position.

Result

Thus, the pressure Coefficient over the smooth circular cylinder has been found and required plot has been drawn.

EXP NO : 04 DATE :	PRESSURE DISTRIBUTION OVER ROUGH CIRCULAR CYLINDER

Aim

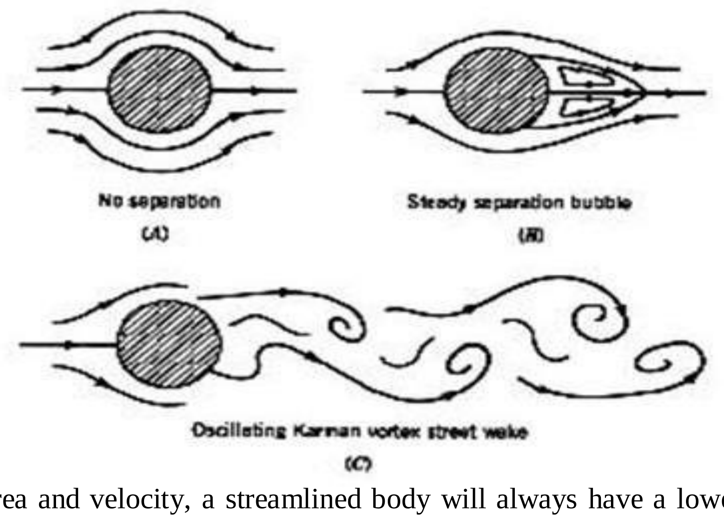
To determine the surface pressure coefficient over rough circular cylinder for different Reynolds number and plot the variation of pressure coefficient along surface and for different Reynolds numbers.

Apparatus Required

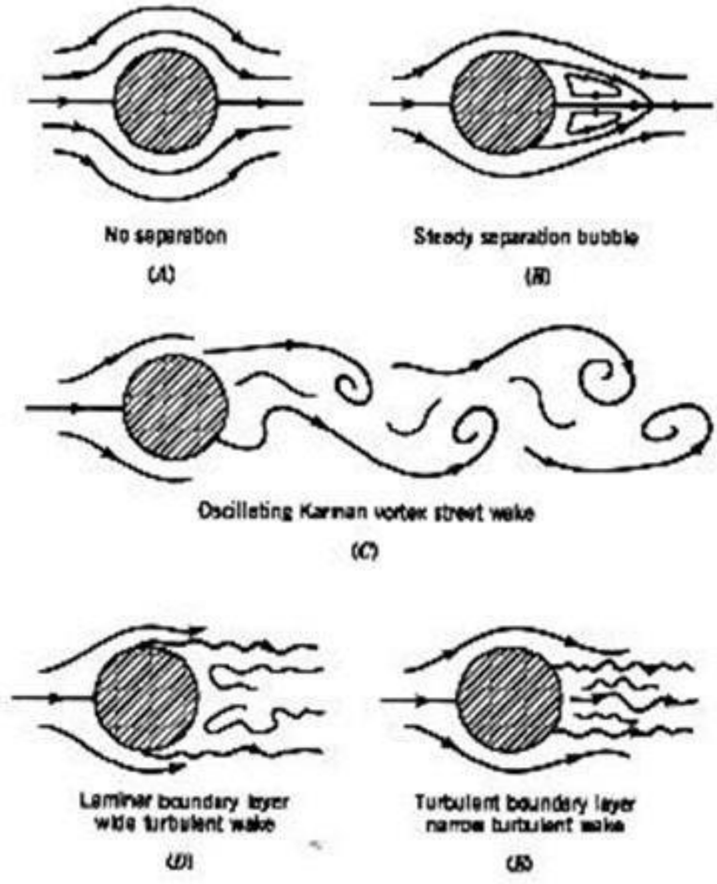
- Subsonic Wind tunnel
- Rough circular cylinder
- Multi - tube manometer
- Pitot – Static Tube

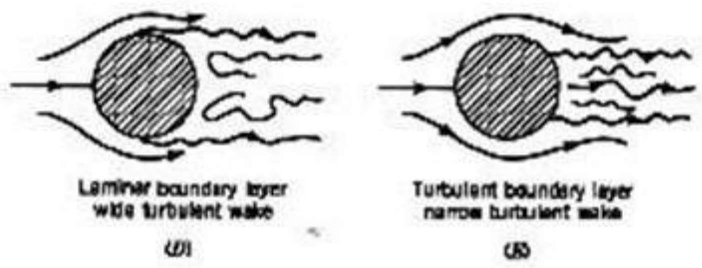
Theory

Sphere is a three dimensional body where cylinder is two dimensional body. When thedrag is dominated by viscous drag, we say the body is streamlined, and when it is dominated by pressure drag, we say the body is bluff. Whether the flow is viscous-drag dominated or pressure-drag dominated depends entirely on the shape of the body. A streamlined body looks like a fish, or an airfoil at small angles of attack, whereas a bluff body looks like a brick, a cylinder, or airfoil at large angles of attack. For streamlined bodies, frictional drag is the dominant source of air resistance. For a bluff body, the dominant source of drag is pressure drag.



For a given frontal area and velocity, a streamlined body will always have a lower resistance than a bluff body. For example, the drag of a cylinder of diameter can be ten times larger than a streamlined shape with the same thickness.





Formula Used

Velocity of test section $\boldsymbol{V} = \sqrt{\frac{2(\boldsymbol{p0}-\boldsymbol{p})}{\boldsymbol{\rho}}}$

Coefficient of pressure $\boldsymbol{C_p} = \frac{\boldsymbol{p}-\boldsymbol{p^{\infty}}}{\boldsymbol{0.5\rho V^2}_{\infty}}$

Reynolds Number per unit length, $\frac{R_e}{m} = \frac{\rho_{\infty} V_{\infty}}{\mu_{\infty}}$

p_0 –Total Pressure$=p_{atm}+\rho_w g \Delta H$

p –Static Pressure$=p_{atm}+\rho_w g \Delta H$

Where,

ΔH –Change in manometer head

ρ_w–Density of Water(Liquid used in multi-tube manometer)

ρ–Density of Air (in laboratory)

p – Pressure at a tapping

p_∞–Freestream static pressure

V_∞–Freestream velocity

g – Acceleration due to gravity m/s^2

Re/l - Reynolds Number per unit length

μ_∞ - Dynamic Viscosity of air 1.789 x 10^{-5} kg/ (m s)

Procedure

- Mount the rough circular cylinder model in the test section of wind tunnel.
- Connect the pressure port of the circular cylinder to the multi-tube manometer.
- Set the manometer levels without air bubbles by turning the knobs on and off.
- Note down the initial readings from the manometer before starting the experiment.
- Switch on the motor.
- Increase the RPM of the propeller to 250 rpm using the controls.
- Note down the reading in manometer for connected ports (0,90,180,270)
- Now change the orientation of port 1 to 30^0 using rotating knob (angle changing mechanism) attached in test section to flow direction note down head reading using multi-tube manometer for angular position 30,120,210,300.
- Now change the orientation of port 1 to 60^0 using rotating knob attached in test section to flow direction note down head reading using multi-tube manometer for angular position 60,150,240,330.
- Now bring the port to initial position that is to 0^0 orientation to flow direction.
- Repeat the step 6 to 9 for 350,450,550 RMP of the propeller.
- After the experiment reduce the speed of propeller to 100 RPM and switch off the driver and Main remove the cylinder from the tunnel.
- Calculate the co-efficient of pressure from manometer readings and compare with theoretical formulae.

Tabulation

Calculation of Test SectionVeloc ity

P_{atm}=on

T_{atm}=on

ρ_{atm}=on

Sl. No.	RPM	Pitot Reading, *p0*				Static Reading, *p*				$V = \sqrt{\frac{2(p_0 - p)}{\rho}}$
		H_1	H_2	$H_1–H_2$	$p_{atm} + \rho_w g \Delta H$	H_1	H_2	$H_1–H_2$	$p_{atm} + \rho_w g \Delta H$	

Calculation of Cp

RPM						
Velocity						
Port No	**Angular position**	**Static pressure at Tap, p**			$p_{atm} + \rho_w g \Delta H$	$C_p = \frac{p - p_\infty}{0.5\, \rho V^2_\infty}$
		H_1	H_2	$H_1 - H_2$		

Calculation

Precautions

- Check the manometer level properly without any errors.
- Take care while increasing the speed.
- Do not stand behind the wind tunnel while operating it.

Graph

Pressure coefficient vs Angular position.

Result

Thus the pressure Coefficient over the Rough circular cylinder has been found and required plot has been drawn.

EXP NO : 05 DATE :	**PRESSURE DISTRIBUTION OVER A SYMMETRICAL AIRFOIL**

Aim

To determine the surface pressure coefficient over symmetrical airfoil at various angle of attack and compare the pressure distribution over the airfoil for different Reynolds number

Apparatus Required

- Subsonic Wind tunnel
- Airfoil model
- Multi-tube manometer

Theory

A body immersed in a flowing fluid is exposed to both pressure and viscous forces. The sum of the forces that acts normal to the free-stream direction is the lift, and the sum that acts parallel to the free-stream direction is the drag. The geometric and dynamic characteristics of airfoils are shown in figure. This experiment is concerned with computation of the pressure distribution on a stationary airfoil mounted in the test section of a wind tunnel. We will consider only two-dimensional airfoils where tip and root effects are neglected.

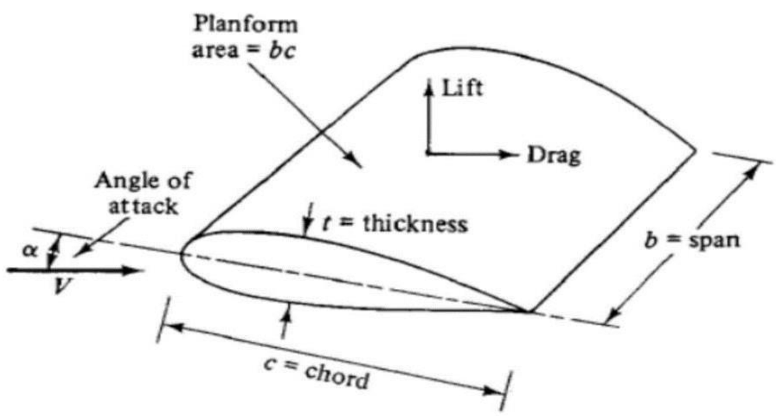


Figure. Geometry and Dynamic parameters of Airfoil

Because the velocity of the flow over the top of the airfoil is greater than the free-stream velocity, the pressure over the top is negative. This follows directly from the application of Bernoulli's equation. Similarly the velocity along the underside of the airfoil is less than the free-stream velocity and the pressure there is positive.

Hence, both the negative pressure over the top and the positive pressure along the bottom contribute to the lift. There are a variety of ways to measure lift. In this experiment, the lift force, L, on the airfoil will be determined by integration of the measured pressure distribution over the airfoil's surface. Typical pressure distribution on an airfoil and its

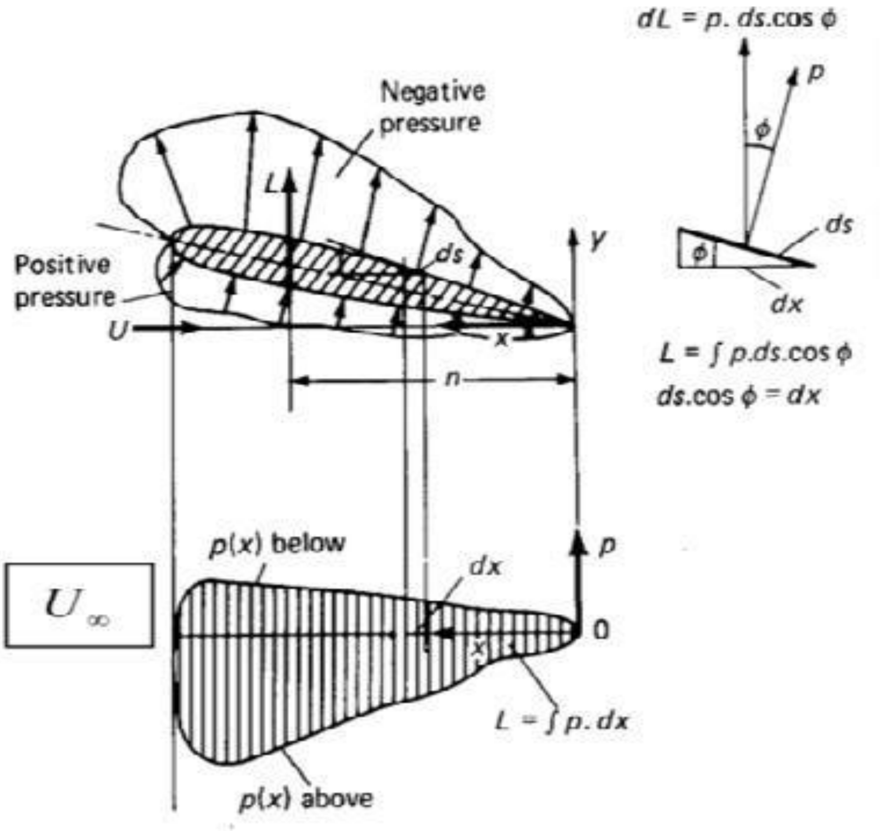


Figure. Pressure distribution over an airfoil

Formula Used

Velocity of test section $V = \sqrt{\frac{2(p0-p)}{\rho}}$

Coefficient of pressure $C_p = \frac{p-p\infty}{0.5\rho V^2{}_{\infty}}$

p_0–Total Pressure=p_{atm}+$\rho_w g\Delta H$

p –Static Pressure=p_{atm}+$\rho_w g\Delta H$

ΔH–Change in manometer head

ρ_w–Density of Water (Liquid used in multi-tube manometer)

ρ–Density of Air (in laboratory)

p – Pressure at a tapping

p_∞–Freestream static pressure

V_∞–Freestream velocity

Procedure

- Fix symmetrical airfoil inside the test section of the wind tunnel at zero degree angle of attack.
- Connect the pressure port of the airfoil to the multi-tube manometer.
- Switch on the driver of the tunnel.
- Increase the RPM of the propeller to 250 rpm using the controls.
- Note down the reading in manometer for connected ports(1 to 21)
- Now increase the angle of attack to 5 degree and now down the surface head.
- Similarly increase the angle of attack to 10,15 note down the head reading.
- Repeat the step 5 to 7 for different Reynolds number (i..e, for different speed of the propeller).
- After the experiment reduce the speed of propeller to 100 RPM and switch off the driver andMain remove the cylinder from the tunnel.
- Find the pressure coefficient on the surface of the airfoil at different angle of attack using theformula.

Tabulation

Calculation of Test Section Velocity

P_{atm}=on

T_{atm}=on

ρ_{atm} =on

Sl. No.	**RPM**	**Pitot Reading, p_0**				**Static Reading, p**				$V = \sqrt{\frac{2(p_0 - p)}{\rho}}$
		H_1	H_2	$H_1–H_2$	$patm+ \rho wg\Delta H$	H_1	H_2	$H_1–H_2$	$patm+\rho wg\Delta H$	

Calculation of C_p

RPM							
Velocity							
Angle of attack (α)							
Port No	Tap location,x	x/C	Static Pressure at Tap, *p*				$C_p = \frac{p - p_\infty}{0.5\rho V^2_\infty}$
			H_1	H_2	$H_1 – H_2$	$p_{atm} + \rho w g \Delta H$	

RPM							
Velocity							
Angle of attack (α)							
Port No	Tap location,x	x/C	Static Pressure at Tap, *p*				$C_p = \frac{p - p_\infty}{0.5\rho V^2_\infty}$
			H_1	H_2	$H_1 – H_2$	$p_{atm} + \rho w g \Delta H$	

Calculation

Graph

Pressure coefficient vs chord length for different angle of attack.

Result

Thus the pressure Coefficient over the symmetrical airfoilhas been found and required plot has been drawn.

EXP NO : 05 DATE :	PRESSURE DISTRIBUTION OVER CAMBERED AIRFOIL

Aim

To find the C_p for a given airfoil at various angle of attacks for constant speed.

Apparatus Required

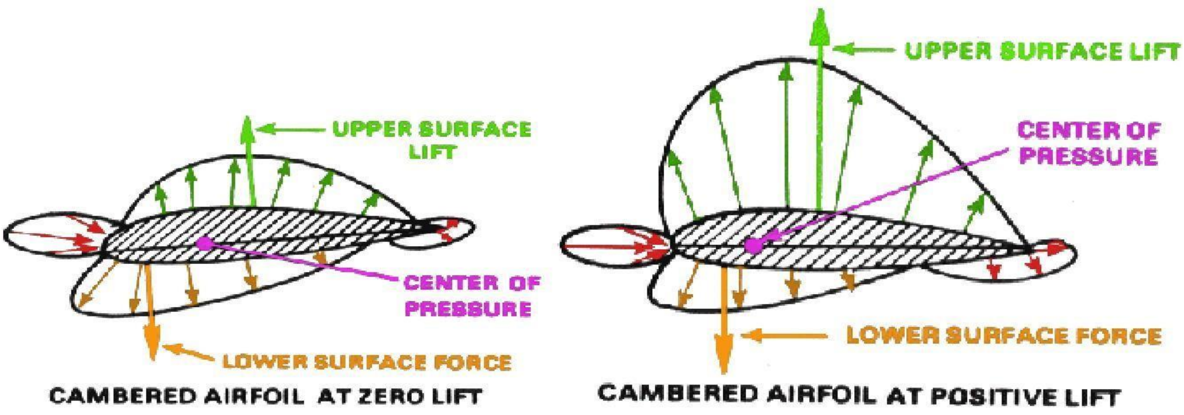
- Wind tunnel
- Aerofoil model
- Multi-tube Manometer

Theory

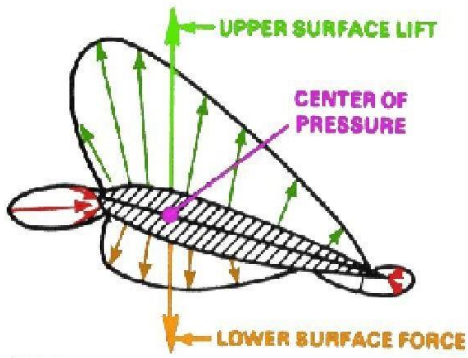
Distribution of pressure over an airfoil section may be a source of an aerodynamic twisting force as well as lift. A typical example is illustrated by the pressure distribution pattern developed by this cambered (nonsymmetrical) airfoil

The upper surface has pressures distributed which produce the upper surface lift. The lower surface has pressures distributed which produce the lower surface force. Net lift produced by the airfoil is the difference between lift on the upper surface and the force on the lower surface. Net lift is effectively concentrated at a point on the chord called the Center Of Pressure.

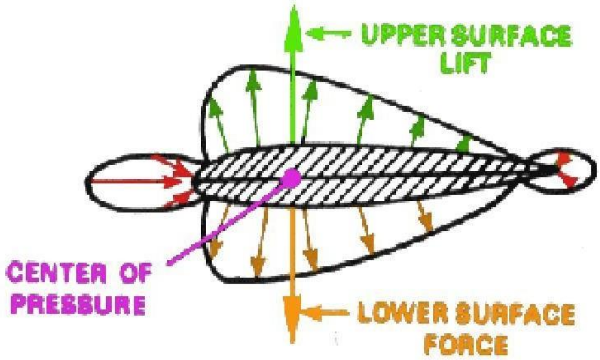
When the angle of attack is increased: Upper surface lift increases relative to the lower surface force. Since the two vectors are not located at the same point along the chord line, a twisting force is exerted about the center of pressure. Center of pressure also moves along the chord line when angle of attack changes, because the two vectors are separated. This characteristic of nonsymmetrical airfoils results in



undesirable control forces that must be compensated for if the airfoil is used in rotary wing applications. When the angle of attack is increased to develop positive lift, the vectors remain essentially opposite each other and the twisting force is not exerted. Center of pressure remains relatively constant even when angle of attack is changed. This is a desirable characteristic for a rotor blade, because it changes angle of attack constantly during each revolution.



SYMMETRICAL AIRFOIL AT POSITIVE LIFT



SYMMETRICAL AIRFOIL AT ZERO LIFT

Formulae Used

Velocity of test section $\boldsymbol{V} = \sqrt{\frac{2(p0-p)}{\rho}}$

Coefficient of pressure $C_p = \frac{p-p\infty}{0.5\rho V^2_\infty}$

$p0$–Total Pressure=$patm+\rho wg\Delta H$

p –Static Pressure=$patm+\rho wg\Delta H$

ΔH –Change in manometer head

ρw–Density of Water(Liquid used in multi-tube manometer)

ρ–Density of Air (in laboratory)

p – Pressure at a tapping

p_∞–Freestream static pressure

V_∞–Freestream velocity

Procedure

- Mount the aerofoil model on the stand provided in the test section of wind tunnel. The trailingedge should be faced towards fan.
- Connect the pressure port of the airfoil to the multi-tube manometer.
- Switch on the driver of the tunnel.
- Increase the RPM of the propeller to 250 rpm using the controls.
- Note down the reading in manometer for connected ports(1 to 21)
- Now increase the angle of attack to 5 degree and now down the surface head.
- Similarly increase the angle of attack to 10,15 note down the head reading.
- Repeat the step 5 to 7 for different Reynolds number(i..e, for different speed of the propeller)
- After the experiment reduce the speed of propeller to 100 RPM and switch off the driver andMain remove the cylinder from the tunnel.
- Find the pressure coefficient on the surface of the airfoil at different angle of attack using theformula.

Tabulation

Calculation of Test Section Velocity

P_{atm}=on

T_{atm}=on

ρ_{atm} =on

Sl. No.	**RPM**	**Pitot Reading, *p0***				**Static Reading, *p***				$V = \sqrt{\frac{2(p_0 - p)}{\rho}}$
		H_1	H_2	$H_1 – H_2$	$p_{atm} + \rho_w g \Delta H$	H_1	H_2	$H_1 – H_2$	$p_{atm} + \rho_w g \Delta H$	

Calculation of C_p

RPM							
Angle of attack (α)							
Velocity							
Chord length							
Port No	**Tap location,x**	**x/C**	**Static pressure at Tap, *p***				$C_p = \frac{p - p_\infty}{0.5\, \rho V^2_\infty}$
			H_1	H_2	$H_1 - H_2$	$p_{atm} + \rho_w g \Delta H$	

Calculation

Precautions

- Check the manometer level properly without any errors.
- Take care while increasing the speed.
- Do not stand behind the wind tunnel while operating it.

Graph

Pressure coefficient vs chord length for different angle of attack.

Result

Thus, the pressure Coefficient over the cambered airfoil has been found and required plot has been drawn.

EXP NO : 07 DATE :	FORCE MEASUREMENT USING WIND TUNNEL BALANCING SET UP

Aim

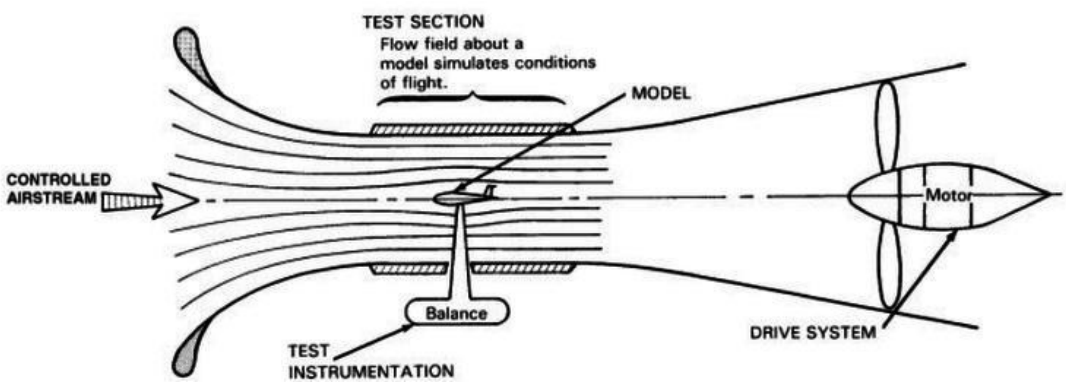
To measure the lift force, drag force and pitching moment on a missile model for different angles of attack.

Apparatus Required

- Subsonic Wind Tunnel
- 3 Axis Wind Tunnel Balance
- Pitot - Static Probe
- Multi-tube Manometer

Theory

The wind tunnels main function is to provide flow simulation on a model introduced in a fluid flow. Global forces and momentums on the model are mainly obtained by using different wind tunnel balances; although in special tests, local balances or pressure distribution measurement can be used as well. Range, accuracy and response time of the measurements are the main parameters that define such systems. The wind tunnel balances are extensively used and are an accurate method for measurements acquisition, with a wide range of measures and a fast response to loads changes. This system requires an important initial calibration effort but once the measurements are probed to be correct, the system can be used to test several low cost models with a reduced effort. Other option for aerodynamic load measurements is the pressure measurement in several model points by means of a pressure scanner or scanivalve system. This system requires a very complex and expensive test model. The measurement points are built in the model surface by making holes and connecting them w ith the scanivalve by means of a tube that transfer the pressure. These holes introduce also modifications in the flow around the model thus modifying the real behavior of the model.



Procedure

- Mount the missile model in the test section of the wind tunnel and make necessary connections for the wind tunnel balance.
- Switch on the AC mains.
- Turn on the inverter to the wind tunnel motor and set the desired RPM.
- Measure the corresponding pitot - static readings and calculate velocity in the test section using the above formula.
- Switch on the wind tunnel balance and note down the readings of lift, drag and pitching moment.
- The experiment is repeated for different angles of attack and speeds.
- Care should be taken that the wind tunnel has to be brought to rest before changing the angle of attack.
- The readings are tabulated and the lift, drag and pitching moment are plotted against angle of attack.

Formula Used

To measure velocity in wind tunnel, $\boldsymbol{V} = \sqrt{\frac{\boldsymbol{2(p0-p)}}{\boldsymbol{\rho}}}$

p_0 – Total Pressure = $p_{atm} + \rho_w g \Delta H$

p – Static Pressure = $p_{atm} + \rho_w g \Delta H$

ΔH – Change in manometer head

H_1 – Height of liquid column where the static port is connected (m) (Max height)

H_2 – Height of liquid column where the pitot port is connected (m) (min height)

ρ_w – Density of Water (Liquid used in multi-tube manometer)

ρ – Density of Air (in laboratory)

Theoretical calculation

Lift Coefficient $c_l = \frac{L}{{}^1/_2\, \rho_\infty V_\infty{}^2 S}$

Drag Coefficient $c_d = \frac{D}{{}^1/_2\, \rho_\infty V_\infty{}^2 S}$

Moment Coefficient $c_m = \frac{M}{{}^1/_2\, \rho_\infty V_\infty{}^2 S\, l}$

Reynolds Number Re= $\rho_{air} V_\infty d/\mu_\infty$

Velocity of Test section $V_\infty = \sqrt{\frac{2\rho_W g \Delta H}{\rho_{air}}}$

Where, S – Reference area of missile $(\frac{\pi d^2}{4})$ (m^2)

d – Diameter of the missile (m)

ρ_a - Density of free stream air 1.225 (kg/m^3)

g – Acceleration due to gravity (9.81 m/s^2)

V_∞ - Velocity of air inside the test section (m/s)

ρ_w - Density of water 1000 (kg/m^3)

Re - Reynolds Number per unit length

μ_∞ - Dynamic Viscosity of air 1.789 x 10^{-5} (kg/m s)

Tabulation

Calculation of Test Section Velocity

P_{atm}=_________________ on _______________

T_{atm}= _________________ on _______________

ρ_{atm} = _________________ on _______________

Sl. No.	RPM	Pitot Reading, p_0				Static Reading, p				$V = \sqrt{\frac{2(p_0 - p)}{\rho}}$
		H_1	H_2	$H_1 – H_2$	$p_{atm} + \rho_w g \Delta H$	H_1	H_2	$H_1 – H_2$	$p_{atm} + \rho_w g \Delta H$	

RPM :			Velocity :	
Sl. No.	Angle of Attack (deg)	Lift(N)	Drag(N)	Pitching Moment(N-m)

Calculation

Graph

Plot for Coefficient of forces vs Angle of Attack.

Result

Thus, the lift, drag and moment were measured for the missile model for different angles of attack and speeds using the 3 axis wind tunnel balance.

EXP NO : 08 DATE :	FLOW OVER A FLAT PLATE AT DIFFERENT ANGLES OF INCIDENCE.

Aim

To study the pressure distribution over the missile model due to air flow and to visualize the flow separation and Determine the base drag of a missile model.

Apparatus Required

- Subsonic Wind Tunnel
- Smoke Generator
- Missile Model

Procedure

- Switch "ON" the Main which is connected to the 440 V, 32 A, 3 ph, AC power supply with neutral and earth connection.
- Check all the switches of the controller are in "OFF" position before starting.
- Switch "ON" the MCB (*Miniature Circuit Breaker*) of Console Board.
- Put-on the mains and observe the main indicator lights are glowing at the bottom of the control panel.
- Fix the smoke distributor at the starting portion of the test section.
- Fix a model in the test section, at required orientation.
- Switch "ON" the Heater which is connected to the 2 ph, AC power supply.
- Wait 5 minutes, till the heater gains some heat energy.
- Let the blended fuel (25 % of Kerosene and 75 % of Diesel) to flow over the heater by open the fuel flow control valve slightly.
- Switch "ON" the blower and light in the console board.
- Ensure the out-coming of smoke and connect the hose to the smoke distributor. Now observe the smoke being forced out of the smoke distributor at the entry of the test section.
- Control the main flow of air in the test section by controlling the AC motor speed, so that the smoke flow pattern to persist across the model. Keep the speed at very low; higher velocities will defuse the smoke.
- Observe the flow pattern over the model.

- Never switch "ON" the heater for long time with or without the fuel being supplied to the unit.

Figure. Missile model

- After the experiment, close the fuel flow control valve, switch “OFF” the heater and light, and run the blower for some time. This is just to exhaust out any smoke left in the smoke generator unit.
- Then switch “OFF” all accessories.

Precautions

- Do not switch on the heater without the liquid paraffin in the reservoir.
- Do not generate the smoke when it is not required.
- Long breathing of the smoke is very disturbing
- Overheating of coils may not produce smoke.
- A balanced of correct heating controlling the blower speed will generate the correct amount of smoke in the test section.

Result

Thus, the formation of layers over the models due to air flow and the flow separation has been visualized.

EXP NO : 09 **DATE :**	**FLOW VISUALIZATION ON BLUFF BODIES USING WATER FLOW CHANNEL**

Aim

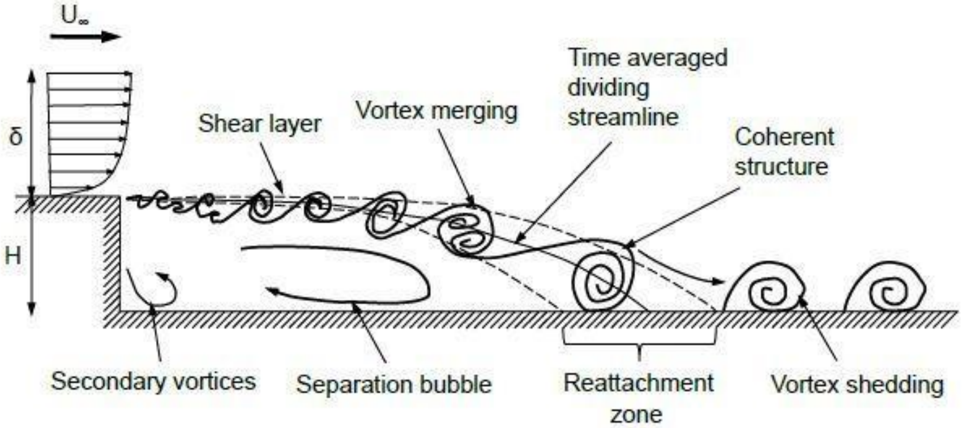
To determine the pressure distribution over backward facing step. Plot the variation of pressure coefficient over model for various Reynolds number.

Apparatus Required

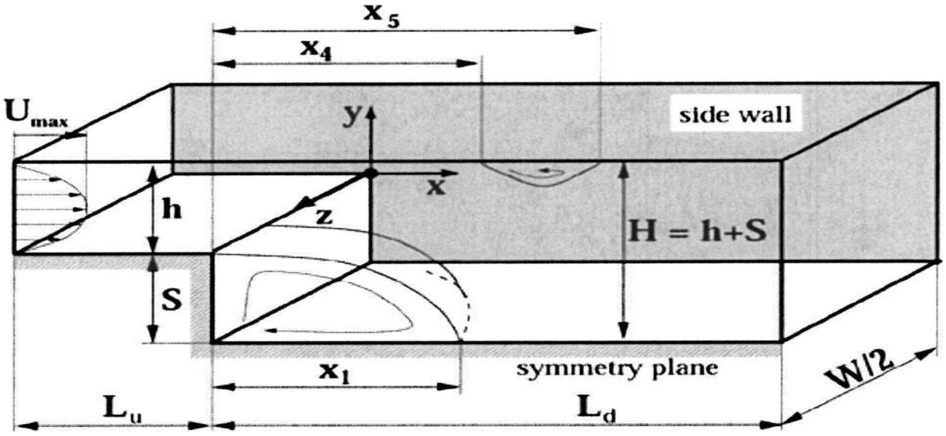
- Subsonic Wind Tunnel
- Pitot - Static Probe
- Multi-tube Monometer
- Backward Facing Step Model

Theory

Wind tunnel studies of separation over rearward facing steps and bluff geometries provide some insight into the physical aspects of flow separation and recirculating flow formation in the cavity zone. When a two-dimensional turbulent flow separates from the sharp corner of a back step, a shear layer with high vorticity and low static pressure is formed which spreads linearly downstream (for steady, two-dimensional laminar flow the spreading is parabolic)G Momentum diffuses from this turbulent mixing layer into the cavity zone which sets the wake fluid into motion, and thus the sharp velocity discontinuity at the wake boundary is smoothed out. In the 'reattachment zone the pressure increase arising out of the compression creates a steep pressure gradient near the surface such that part of the flow near the surface returns upstream to feed the recirculation zone. Due to this entrainment process the velocity and pressure variations are large, and the wake boundary is not well defined.



For, theore … hosen as the streaml … ırt of the new shear … ıid recirculates as a large eddy. The corner ,eddy is formed as a result of a shear-layer separation which is induced by the reverse flow now approaching the step as a forward-facing step.



Sketch of the flow configuration and definition of length scales

Procedure

- Mount the backward facing step in the test section of the wind tunnel and connect all the pressure tappings to the Multi-tube manometer in the proper order.
- Switch on the AC mains.
- Note down the initial readings of the multi-tube manometer.
- Turn on the inverter to the wind tunnel motor and set the desired RPM.
- Measure the corresponding pitot – static readings and calculate velocity in the test section using the above formula.
- Note down the final readings from the manometer.
- The experiment is repeated for different speeds.
- The readings are tabulated and the coefficient of pressure for the various tappings is calculated from the given formula.
- The Cp values are plotted against x/h.

Formula used

To measure velocity in wind tunnel,

Velocity of test section $\boldsymbol{V} = \sqrt{\frac{2(p0-p)}{\rho}}$

Coefficient of pressure $C_p = \frac{p-p\infty}{0.5\,\rho V^2{}_\infty}$

p_0–Total Pressure=p_{atm}+$\rho_w g\Delta H$

p –Static Pressure=p_{atm}+$\rho_w g\Delta H$

ΔH –Change in manometer head

ρ_w–Density of Water(Liquid used in multi-tube manometer)

ρ–Density of Air (in laboratory)

p – Pressure at a tapping

p_∞–Freestream static pressure

V_∞–Freestream velocity

h - Channel height upstream of the step

S- Step height

Tabulation

Calculation of Test SectionVeloc ity

P_{atm}=on

$T_{a\ tm}$=on

ρ_{atm} =on

Sl. No.	RPM	Pitot Reading, p_0				Static Reading, p				$V = \sqrt{\frac{2(p_0 - p}{\rho}}$
		H_1	H_2	H_1–H_2	$patm$+ $\rho wg\Delta H$	H_1	H_2	H_1–H_2	$patm$+ $\rho wg\Delta H$	

Calculation of C_p

RPM							
Free stream static pressure							
Velocity							
Height of the step							
Port. No	**Tap location, x**	**x/h**	**Static pressure at Tap,** p				$C_p = \frac{p - p_\infty}{0.5\ \rho V^2_\infty}$
			H_1	H_2	$H_1 - H_2$	$p_{atm} + \rho_w g \Delta H$	

RPM							
Free stream static pressure							
Velocity							
Height of the step							
Port. No	**Tap location, x**	**x/h**	**Static pressure at Tap,** p				$C_p = \frac{p - p_\infty}{0.5\ \rho V^2_\infty}$
			H_1	H_2	$H_1 - H_2$	$p_{atm} + \rho_w g \Delta H$	

Calculation

Graph

x/h vs C_p

Result

Thus, the pressure distribution on the backward facing step for different speeds was estimated and plotted.

EXP NO : 10 DATE :	FLOW VISUALIZATION USING HELE-SHAW APPARATUS

Aim

To find the power generated by a wind turbine for different speeds.

Apparatus Required

- Subsonic Wind Tunnel
- Pitot – Static Probe
- Multitube Manometer
- Wind turbine mount
- Digital multimeter
- Breadboard, LED, Resistor, Connecting Wires

Formulae Used

Power generated $P_g = V\,I$

Power Available $P_a = 0.5\,\rho_{air}\,V_\infty{}^3 A$

Power Coefficient $c_p = \frac{Power\ generated}{Power\ avaliable}$

Velocity of test section $V = \sqrt{\frac{2(p0-p)}{\rho}}$

p_0 – Total Pressure = $p_{atm} + \rho_w g \Delta H$

p – Static Pressure = $p_{atm} + \rho_w g \Delta H$

ΔH – Change in manometer head

ρ_w – Density of Water (Liquid used in multi-tube manometer)

ρ – Density of Air (in laboratory)

V – Voltage (Volts)

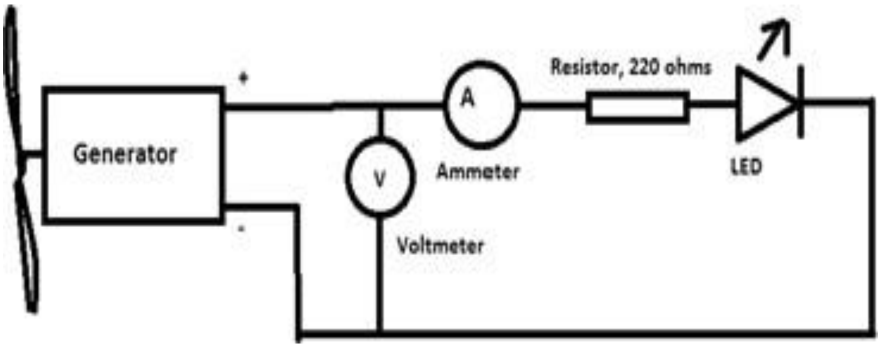
I – Current (Amperes)

V_∞ - Velocity of air inside the test section (m/s)

d – Diameter of the disk (m)

A – Area of disk (m^2)

Circuit



Procedure

- Mount the wind turbine in the test section of the wind tunnel.
- Make all the connections to measure the current and voltage with the help of the circuit diagram.
- Switch on the AC mains.
- Note down the initial readings of the multi-tube manometer.
- Turn on the inverter to the wind tunnel motor and set the desired RPM.
- Measure the corresponding pitot – static readings and calculate velocity in the test section using the above formula.
- Note down the current and voltage readings.
- The experiment is repeated for different speeds.
- The readings are tabulated and the power generated can be calculated from the current and voltage values.
- The power generated is plotted against RPM.

Tabulation

Calculation of Test Section Velocity

P_{atm}= ________________on

T_{atm}= ________________on

ρ_{atm} = ________________on

Sl. No.	RPM	Pitot Reading, p_0				Static Reading, p				$V = \sqrt{\frac{2(p_0 - p)}{\rho}}$
		H_1	H_2	$H_1 – H_2$	$patm + \rho wg \Delta H$	H_1	H_2	$H_1 – H_2$	$patm + \rho wg \Delta H$	

Calculation of Power Generated

S.No	Speed of Propeller RPM	Volts (V)	Current (A)	Power Generated p_g (W)	Power Available p_a (W)	Power Coefficient $C_p = \frac{p_g}{p_a}$

Graph

Wind tunnel speed vs Power available.

Wind tunnel speed vs power generated.

Wind tunnel speed vs power coefficient.

Result

Thus, the power generated by the wind turbine for different speeds was estimated and plotted.

EXP NO : 12 DATE :	**STUDY OF FANNO FLOW**

Aim

To study about Fanno flow which refers to adiabatic flow through a constant area duct where the effect of friction is considered.

Theory

Fanno flow refers to adiabatic flow through a constant area duct where the effect of friction is considered. Compressibility effects often come into consideration, although the Fanno flow model certainly also applies to incompressible flow. For this model, the duct area remains constant, the flow is assumed to be steady and one-dimensional, and no mass is added within the duct. The Fanno flow model is considered an irreversible process due to viscous effects. The viscous friction causes the flow properties to change along the duct. The frictional effect is modeled as a shear stress at the wall acting on the fluid with uniform properties over any cross section of the duct.

The Fanno flow model begins with a differential equation that relates the change in Mach number with respect to the length of the duct, dM/dx. Other terms in the differential equation are the heat capacity ratio, γ, the Fanning friction factor, f, and the hydraulic diameter, D_h

$$\frac{dM^2}{M^2} = \frac{\gamma M^2}{1-M^2}\left(1+\frac{\gamma-1}{2}M^2\right)\frac{4f}{D_h}dx$$

Assuming the Fanning friction factor is a constant along the duct wall, the differential equation can be solved easily. One must keep in mind, however, that the value of the Fanning friction factor can be difficult to determine for supersonic and especially hypersonic flow velocities. The resulting relation is shown below where L* is the required duct length to choke the flow assuming the upstream Mach number is supersonic. The left-hand side is often called the Fanno parameter.

$$\frac{4fL^*}{D_h} = \left(\frac{1-M^2}{\gamma M^2}\right) + \left(\frac{\gamma+1}{2\gamma}\right)\ln\left[\frac{M^2}{\left(\frac{2}{\gamma+1}\right)\left(1+\frac{\gamma-1}{2}M^2\right)}\right]$$

Equally imp ... ge in entropy over the heat capacity at constant pressure, c_p.

$$\Delta S = \frac{\Delta s}{c_p} = \ln\left[M^{\frac{\gamma-1}{\gamma}}\left(\left[\frac{2}{\gamma+1}\right]\left[1+\frac{\gamma-1}{2}M^2\right]\right)^{\frac{-(\gamma+1)}{2\gamma}}\right]$$

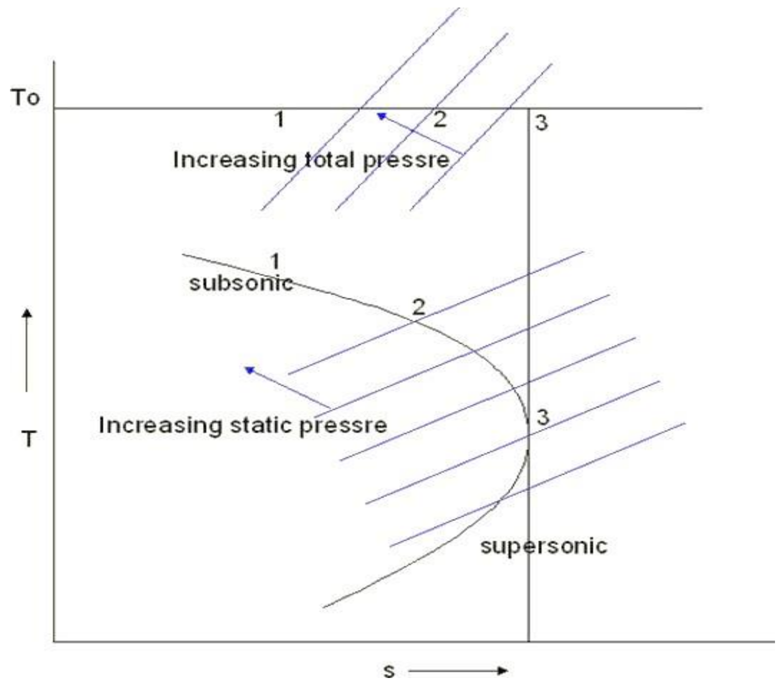


Fig. h-s diagram for Fanno flow

The above equation can be rewritten in terms of a static to stagnation temperature ratio, which, for a calorically perfect gas, is equal to the dimensionless enthalpy ratio, H

$$H = \frac{h}{h_0} = \frac{c_p T}{c_p T_0} = \frac{T}{T_0}$$

$$\Delta S = \frac{\Delta s}{c_p} = \ln\left[\left(\frac{1}{H} - 1\right)^{\frac{\gamma-1}{2\gamma}} \left(\frac{2}{\gamma - 1}\right)^{\frac{\gamma-1}{2\gamma}} \left(\frac{\gamma+1}{2}\right)^{\frac{\gamma+1}{2\gamma}} (H)^{\frac{\gamma+1}{2\gamma}}\right]$$

The equation above can be used to plot the Fanno line, which represents a locus of states for given Fanno flow conditions on an H-ΔS diagram. In the diagram, the Fanno line reaches maximum entropy at H = 0.833 and the flow is choked. According to the Second law of thermodynamics, entropy must always increase for Fanno flow. This means that a subsonic flow entering a duct with friction will have an increase in its Mach number until the flow is choked. Conversely, the Mach number of a supersonic flow will decrease until the flow is choked. Each point on the Fanno line corresponds with a different Mach number, and the movement to choked flow is shown in the diagram.

The Fanno line defines the possible states for a gas when the mass flow rate and total enthalpy are held constant, but the momentum varies. Each point on the Fanno line will have a different momentum value, and the change in momentum is attributable to the effects of friction.

Additional Fanno flow relations

The area and mass flow rate in the duct are held constant for Fanno flow. Additionally, the stagnation temperature remains constant. These relations are shown below with the * symbol representing the throat location where choking can occur. A stagnation property contains a 0 subscript.

$$A = A^* = \text{constant}$$
$$T_0 = T_0^* = \text{constant}$$
$$\dot{m} = \dot{m}^* = \text{constant}$$

Differential equations can also be developed and solved to describe Fanno flow property ratios with respect to the values at the choking location. The ratios for the pressure, density, temperature, velocity and stagnation pressure are shown below, respectively. They are represented graphically along with the Fanno parameter.

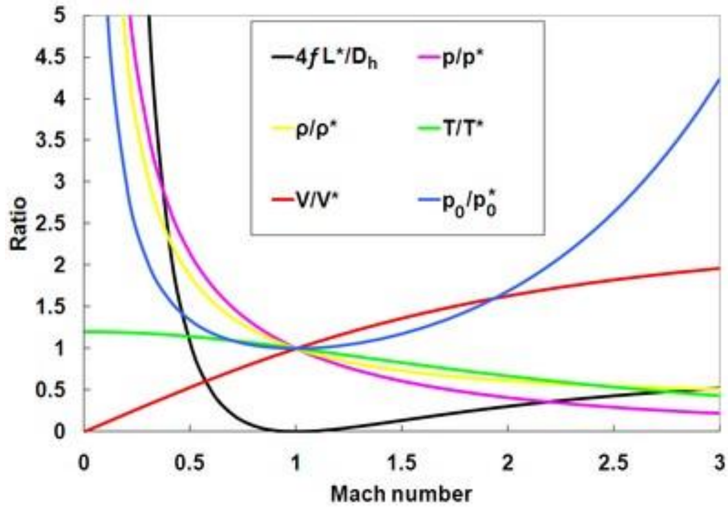


Figure. Common thermodynamic property ratios plotted as a function of Mach number using the Fanno flow model.

$$\frac{p}{p^*} = \frac{1}{M}\frac{1}{\sqrt{\left(\frac{2}{\gamma+1}\right)\left(1+\frac{\gamma-1}{2}M^2\right)}}$$

$$\frac{\rho}{\rho^*} = \frac{1}{M}\sqrt{\left(\frac{2}{\gamma+1}\right)\left(1+\frac{\gamma-1}{2}M^2\right)}$$

$$\frac{T}{T^*} = \frac{1}{\left(\frac{2}{\gamma+1}\right)\left(1+\frac{\gamma-1}{2}M^2\right)}$$

$$\frac{V}{V^*} = M\frac{1}{\sqrt{\left(\frac{2}{\gamma+1}\right)\left(1+\frac{\gamma-1}{2}M^2\right)}}$$

$$\frac{p_0}{p_0^*} = \frac{1}{M}\left[\left(\frac{2}{\gamma+1}\right)\left(1+\frac{\gamma-1}{2}M^2\right)\right]^{\frac{\gamma+1}{2(\gamma-1)}}$$

Applications

The Fanno flow model is often used in the design and analysis of nozzles. In a nozzle, the converging or diverging area is modeled with isentropic flow, while the constant area section afterwards is modeled with Fanno flow. For given upstream conditions at point 1 as shown in Figures 3 and 4, calculations can be made to determine the nozzle exit Mach number and the location of a normal shock in the constant area duct. Point 2 labels the nozzle throat, where M = 1 if the flow is choked. Point 3 labels the end of the nozzle where the flow transitions from isentropic to Fanno. With a high enough initial pressure, supersonic flow can be maintained through the constant area duct, similar to the desired performance of a blow down-type supersonic wind tunnel. However, these figures show the shock wave before it has moved entirely through the duct. If a shock wave is present, the flow transitions from the supersonic portion of the Fanno line to the subsonic portion before continuing towards M = 1. The movement in Fig is always from the left to the right in order to satisfy the second law of thermodynamics.

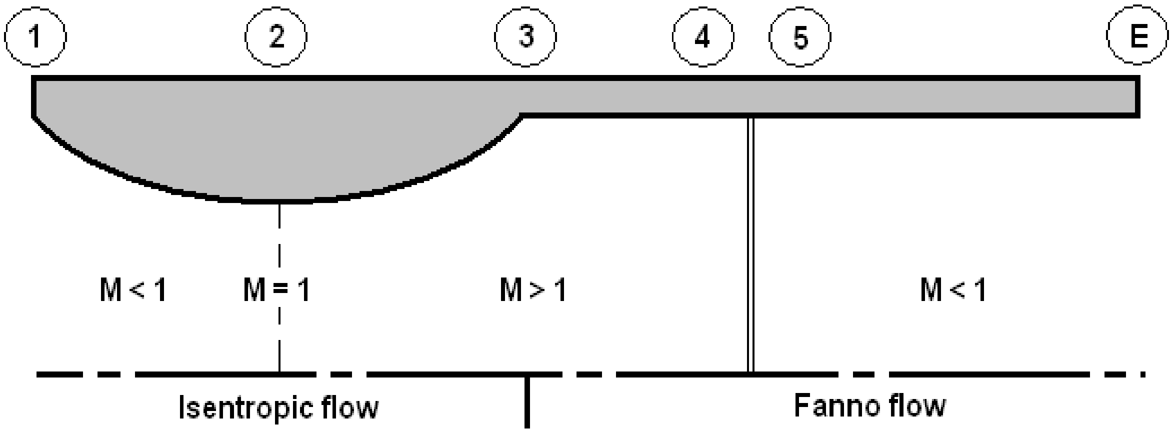


Figure. A supersonic nozzle leading into a constant area duct is depicted. The initial conditions exist at point 1. Point 2 exists at the nozzle throat, where M = 1. Point 3 labels the transition from isentropic to Fanno flow. Points 4 and 5 give the pre- and post-shock wave conditions, and point E is the exit from the duct.

The Fanno flow model is also used extensively with the Rayleigh flow model. These two models intersect at points on the enthalpy-entropy and Mach number-entropy diagrams, which is meaningful for many applications. However, the entropy values for each model are not equal at the sonic state. The change in entropy is 0 at M = 1 for each model, but the previous statement means the change in entropy from the same arbitrary point to the sonic point is different for the Fanno and Rayleigh flow models. If initial values of si and Mi are defined, a new equation for dimensionless entropy versus Mach number can be defined for each model. These equations are shown below for Fanno and Rayleigh flow, respectively.

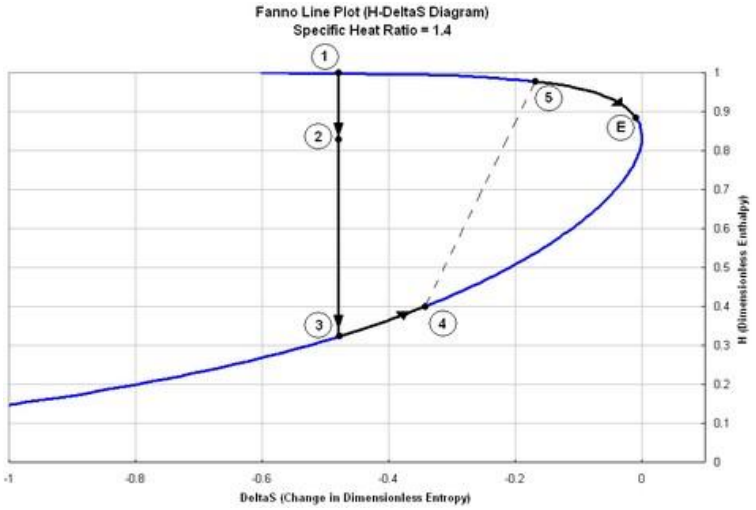
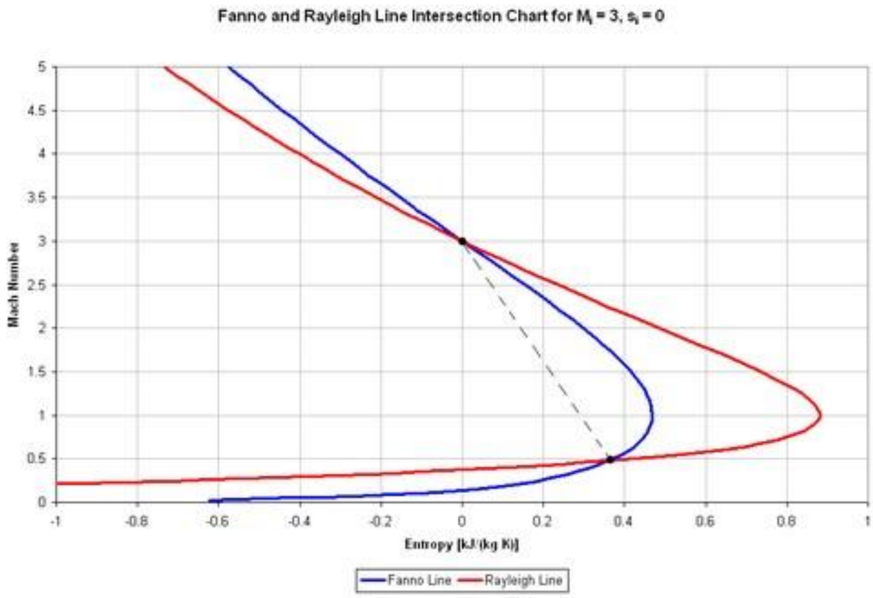


Figure. The H-S diagram is depicted for the conditions of Figure 3. Entropy is constant for isentropic flow, so the conditions at point 1 move down vertically to point 3. Next, the flow follows the Fanno line until a shock changes the flow from supersonic to subsonic. The flow then follows the Fanno line again, almost reaching a choked condition before exiting the duct.



Figrue. Fanno and Rayleigh Line Intersection Chart

$$\Delta S_F = \frac{s - s_i}{c_p} = \ln\left[\left(\frac{M}{M_i}\right)^{\frac{\gamma-1}{\gamma}}\left(\frac{1+\frac{\gamma-1}{2}M_i^2}{1+\frac{\gamma-1}{2}M^2}\right)^{\frac{\gamma+1}{2\gamma}}\right]$$

$$\Delta S_R = \frac{s - s_i}{c_p} = \ln\left[\left(\frac{M}{M_i}\right)^{2}\left(\frac{1+\gamma M_i^2}{1+\gamma M^2}\right)^{\frac{\gamma+1}{\gamma}}\right]$$

Fig. shows the Fanno and Rayleigh lines intersecting with each other for initial conditions of si = 0 and Mi = 3. The intersection points are calculated by equating the new dimensionless entropy equations with each other, resulting in the relation below.

$$\left(1+\frac{\gamma-1}{2}M_i^2\right)\left[\frac{M_i^2}{(1+\gamma M_i^2)^2}\right] = \left(1+\frac{\gamma-1}{2}M^2\right)\left[\frac{M^2}{(1+\gamma M^2)^2}\right]$$

Interestingly, the intersection points occur at the given initial Mach number and its post-normal shock value. For Fig, these values are M = 3 and 0.4752, which can be found the normal shock tables listed in most compressible flow textbooks. A given flow with a constant duct area can switch between the Fanno and Rayleigh models at these points.

Result

Thus, the analysis of Fanno flow has been studied.

EXP NO : 13 DATE :	STUDY OF PROFILE DRAG OF BODIES BY WAKE SURVEY METHOD

Aim

To find the drag of an object in a moving fluid is determined by wake survey method.

Apparatus Required

- Subsonic Wind Tunnel
- Model

Theory

Wake surveys are a common method for measuring profile drag. The wake survey measures static pressures and the decrease in total pressure within the wake and compares those values to the free stream total pressure. This pressure deficit, or essentially the momentum loss of the flow, can then be directly related to the profile drag. An excellent and more complete discussion of how to extract drag from wake pressure deficits can be found. Nevertheless, a brief review of the general approach is listed in this section. A pictorial representation of the velocities before, station 0, and aft of the airfoil, stations 2 and 1. The shaded area represents the total moment loss and integrating this area gives the profile drag. A wake survey measures drag by collecting pressure readings in the wake and in the free stream. The difference in these pressures translates to the loss of momentum in the flow. The losses are greatest in the center of the wake and decrease moving outward until free stream momentum is achieved as can be seen in Fig. These losses occur because of the boundary layer interaction with the airfoil. While it is difficult to measure the velocities in the wake, Bernoulli's equation provides a relationship between velocity, static pressure, and dynamic pressure.

$$p_t = p + \frac{1}{2}\rho V^2 = p + q$$

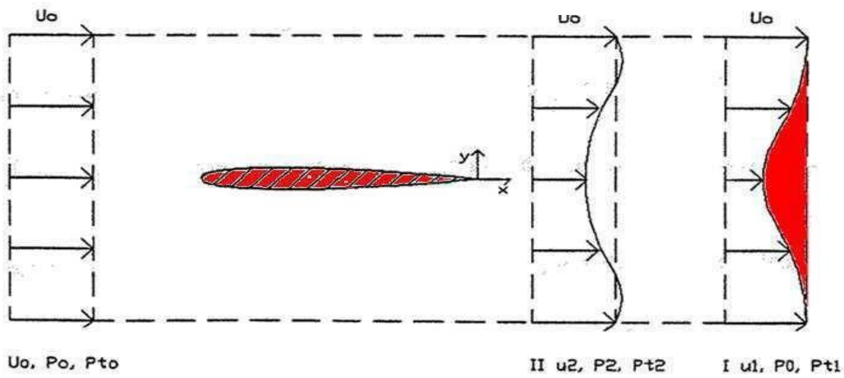


Figure. Momentum loss of flow over an Airfoil

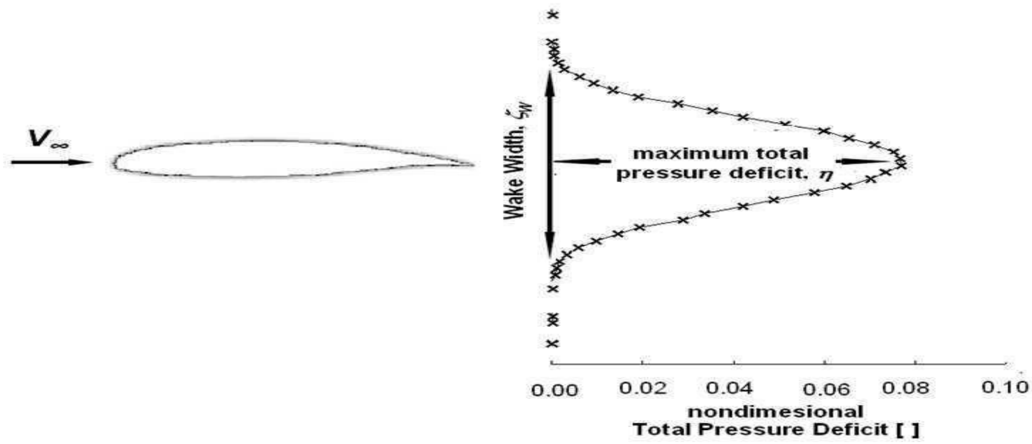


Figure. Typical traversing Pitot-static probe and measured pressure deficit in wake.

Using this relationship, the deficit in total pressure, pt, within the wake can be related to profile drag of the wing section.

$$c_d \cong \frac{1}{cq_0} \int_{\text{wake}} (p_{t_0} - p_t)\, dy$$

The wake pressure deficit has to be measured sufficiently far enough from the trailing edge to allow the static pressure to recover to free stream conditions. Because of the laws of continuity in the free stream flow, static and total pressures are easily measured and may be taken anywhere in the free stream. Further corrections can be applied to adjust for the static pressure in the wake not having fully recovered at the location of the wake survey. The individual samples of total pressures in the wake, p_t , can then be integrated across the width of the wake and related to profile drag.

Procedure

- Switch “ON” the Main which is connected to the 440 V, 32 A, 3ph, AC power supply with neutral and earth connection.
- Check all the switches of the controller are in “OFF” position before starting.
- Switch “ON” the MCB (Miniature Circuit Breaker) of Console Board.
- Put-on the mains and observe the main indicator lights are glowing at the bottom of the control panel.
-
- Fix automotive model in the test section, at required orientation.
- Switch “ON” the blower and light in the console board.

- Control the main flow of air in the test section by controlling the AC motor speed, so that the smoke flow pattern to persist across the model. Keep the speed at very low; higher velocities will defuse the smoke.
- Observe the flow pattern over the model.
- Never switch "ON" the heater for long time with or without the fuel being supplied to the unit.
- After the experiment, close the fuel flow control valve, switch "OFF" the heater and light, and run the blower for some time. This is just to exhaust out any smoke left in the smoke generator unit.
- Then switch "OFF" all accessories.
- Tabulate the values of different drag coefficient with respect to various air velocities.
- Plot the graph between various air velocities and drag coefficient.

Result

Thus, the investigation of profile drag of objects by wake survey method was done.

EXP NO : 14	FLOW VISUALIZATION AT SUBSONIC VELOCITY
DATE :	A) USING TUFT B) OIL FLOW VISUALIZATION

a) Tuft technique

Aim

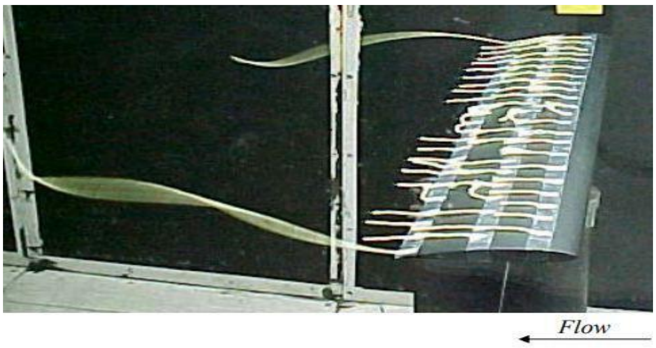
To visualize the flow separation over a Aerofoil using tuft technique.

Apparatus

- Wind tunnel
- Aerofoil model and oil

Description

The simplest and most frequently used method for surface flow visualization is to attach tufts to the surface of interest. The tufts must be of light, flexible material that will align itself with the local surface flow as a result of direct aerodynamic force. The most commonly used material is light yam with weights and lengths chosen according to model size and test speeds. Very small monofilament has also been used. There are also polyester and cotton sewing threads, such as Clark's O.N.T. mercerized cotton No. 60, which can be treated with a fluorescent material. The thread is a multiple-strand material and tends to unravel with time. Tufts do affect the aerodynamic forces to some extent as we will show, but there are many situations in which the method is so easy and economical that it is the first choice. Two basic methods of attaching tufts to a surface are by scotch tape or by glue. When tape is used, the tufts are usually made on a "tuft board." The tuft material is strung back and forth around pins, then the tape is applied to the tufts and the tuft material is cut at the edge of the tape. This gives a length of tape with tuft attached that is applied to the model. The model surface is cleaned with naphtha or other solvents to remove oil so that the tape will hold under the adverse conditions of high-speed flow.



When tufts are glued to the model, nitrocellulose cement such as Duco is used, thinned 50% with acetone or methyl ethyl ketone. Often 10% pigmented lacquer is added both to obscure the portion of the tuft under the glue and to make the glue dots visible by using a contrasting color. The glue dots are kept as small as possible. Tufts readily show where flow is steady and where it is unsteady. Regions of complete separation and buffeting flow are readily identified. The resolution of the determination is of the order of the spacing of the tufts. The possibility of significant influence of the tufts themselves on the flow is very high and must always be kept in mind. This can be investigated by removing tufts upstream of indicated flow separation.

Precautions

- Do not stand behind the wind tunnel while operating it.

Result

Thus, the flow separation over a Aerofoil using tuft technique was studied.

b) Oil Flow Visualization

Aim

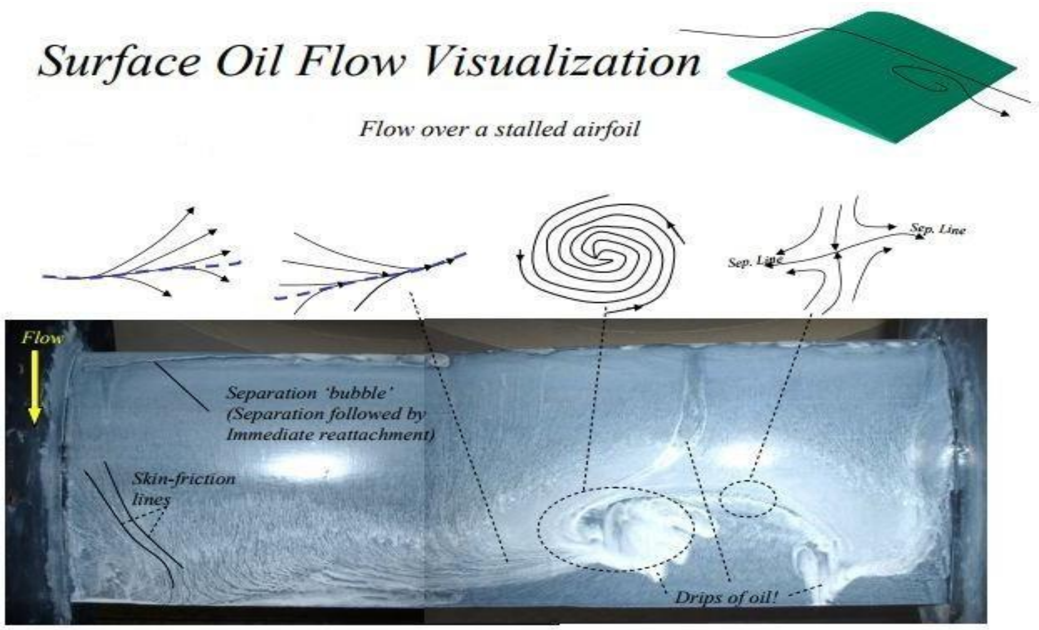
To visualize the flow separation over a Aerofoil by Oil Flow Visualization.

Apparatus

- Wind tunnel
- Aerofoil model and oil

Description

Flow visualization is the study of methods to display dynamic behavior in liquidsand gases. The field dates back at least to the mid-1400, where sketched images of fine particles of sand and wood shavings which had been dropped into flowing liquids. Since then, laboratory flow visualization has become more and more exact, with careful control of the particulate size and distribution. An advance in photography has also helped extend our understanding of how fluids flow under various circumstances. More recently, computational fluid dynamics (CFD) has extended the abilities of scientists to study flow by creating simulations of dynamic behavior of fluids under a wide range of conditions. The result of this analysis is usually a 2-D or 3-D grid of velocity vectors, which may be uniformly or non-uniformly spaced. The goal is then to analyze this vector field to identify features such as turbulence, vortices, and other forms of structure.



Procedure

- Mount the aerofoil model at 20^0 angle on the stand provided in the test section of wind tunnel. The trailing edge should be faced towards fan.
- On the Smoke Generator and wait for few minutes to generate smoke.

- Visualize the flow over Aerofoil and the streamlines separating from aerofoil surface.
- Take the pictures of Separation.

Precautions

- Do not stand behind the wind tunnel while operating it.

Result

Thus, the flow separation over a Aerofoil by Oil Flow Visualization was studied.

EXP NO : 15 DATE :	FLOW VISUALIZATION STUDIES IN SUPERSONIC FLOWS BY SCHLIEREN SYSTEM

Aim

To study the use of Schlieren system to visualize shock.

Apparatus Required

- Supersonic Wind Tunnel
- Schlieren system

Introduction

Schlieren method visualizes the distribution of fluid density within a fluid, as fluid density controls the index of refraction. Regions of density gradient deflect light beams, shifting their position on the image plane. The relative change in light intensity can be used to infer the original density and flow field.

Optical Principle

There are several methods commonly used to visualize refractive index or density changes in liquids, gases, liquids and solids. Generically these include shadowgraphs, schlieren and interferometric techniques. These systems are used to visualize temperature gradients, shock waves in wind tunnels, non-homogeneous areas in sheet glass, convection patterns in liquids, etc.

Shadowgraph systems are often used where the density gradients are large. This technique also can accommodate large subjects, is relatively simple in terms of materials required and in terms of cost is probably the least expensive technique to set-up and operate.

Interferometric techniques are usually highly sensitive, complex in set-up, can provide quantitative information but are expensive systems and can only deal with relatively small subjects. Schlieren systems are intermediate in terms of sensitivity, system complexity and cost. Schlieren systems can be configured to suit many different applications and sensitivity requirements. Typically, however, they are still considered too complex and expensive to implement in many cases where a simple density visualization system is needed.

While discussed and introduced in the early '50's, the focusing schlieren method is one that uses no unusual or expensive optical elements and yet achieves high sensitivity at relatively low cost. Plus it relies on low-tech system "ingredients". As you can see by the attached illustration showing two students setting up a demo system, it is perfect for use in teaching situations where cost tends to be a primary concern.

Unlike other systems, the focusing schlieren technique uses a broad light source instead of a small or point light source. This source is usually made up of a regular array of black and white elements. To make one requires simply a large translucent, opalescent, sheet of plastic onto which opaque stripes are fastened creating a black line/bright line effect. This screen is illuminated from behind with a single flood lamp or a flash, or a bank of them. If this screen is located closer to the "imaging" lens than infinity, then the image of this screen is reproduced at a distance somewhat larger than 1 focal length of the lens. If one makes a high contrast negative photograph of the image of the screen, then the bright areas of the subject will reproduce as dark, opaque, lines at the location of the image of the screen and the black lines will reproduce as clear lines.

Then, if a subject of some kind is placed between the source and the camera lens, the image of this subject will be reproduced sharply at a distance larger then that at which a sharp image of the screen is reproduced. This essentially means that image points of this closer subject are made up, at the focus of the image of this closer subject, by light rays that come to the camera lens from a large number of source locations. Before these light rays become part of the sharply focused image of the subject they are part of the sharply focused image of the source. Now if the negative reproduction of the light source or screen is placed in the position where it was when its photograph was made, then at any position farther away from the camera lens no light will be seen because all the light from the clear areas of the source would be stopped by the dark lines of the reproduction of the source.

If during the "steady state" condition of the systems the interfering physical reproductions of the gridlines of the source are displaced slightly from the pint where they cause a total extinction of light, then a grayish field of uniform brightness will be perceived. This field will become brighter until the physical lines completely cover the dark lines of the image of the source grid allowing all the light present in the clear, bright, areas of the source grid to pass.

If the system is adjusted so that the obstruction and the image of the source grid are slightly misaligned and a uniform, grey field is seen, then local variations in refractive index that cause the light beam to move will become visible as local areas in the image plane that are brighter or darker than the middle grey selected that indicates the "steady state" condition. Thus local variations in density can be visualized. They can be recorded with simple auxiliary photographic equipment.

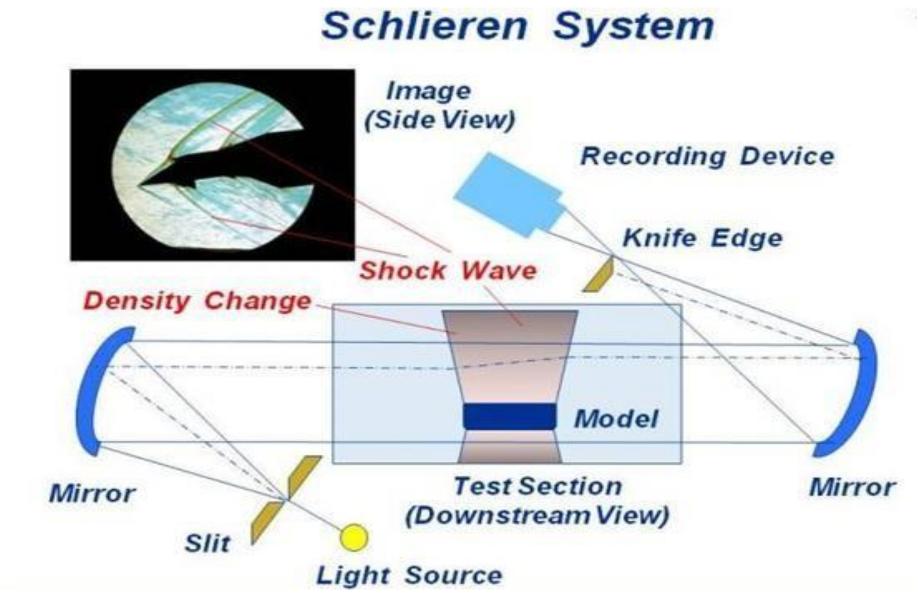


Figure. Operating principle of schlieren system

Light rays are bent whenever they encounter changes in density of a fluid. Schlieren systems are used to visualize the flow away from the surface of an object. The schlieren system shown in this fig ure uses two concave mirrors on either side of the test section of the wind tunnel. A mercury vapor lamp or a spark gap system is used as a bright source of light. The light is passed through a slit which is placed such that the reflected light from the mirror forms parallel rays that pass through the test section. On the other side of the tunnel, the parallel rays are collected by another mirror and focused to a point at the knife edge. The rays continue on to a recording device like a video camera.

Now if the parallel rays of light encounter a density gradient in the test section, the light is bent, or refracted. In our schematic, a shock wave has been generated by a model placed in the supersonic flow through the tunnel test section. Shock waves are thin regions of high gradients in pressure, temperature and density. A ray of light passing through the shock wave is bent as shown by the dashed line in the figure. This ray of light does not pass through the focal point, but is stopped by the knife edge. The resulting image recorded by the camera has darkened lines that occur where the density gradients are present. The model completely blocks the passing of the light rays, so we see a black image of the model. But more important, the shock waves generated by the model are now seen as darkened lines on the image. We have a way to visualize shock waves.

The earliest schlieren photographs of shock waves were black and white images. The image shown here is a color schlieren image produced by putting a prism near the slit and breaking the white light into different colors. Notice that the resulting image is two dimensional while, in reality, shock waves are three dimensional. So the schlieren photograph provides some valuable information about the location and strength of the shock waves, but it requires some experience to properly interpret the results of the process.

Procedure

- It is desirable to use a large aperture, relatively long focal length lens for the imaging lens. Some surplus copier lenses can be applied for this purpose. The source grid should be placed at a distance equal to four focal lengths of the lens. The subject field will be located at two focal lengths from the lens. Therefore the lens will be making a life sized reproduction of the subject.
- Due to this choice of distances, the lens will make an image of the source grid at a distance of 16 inches from the lens. This can be determined from the basic relationship that the reciprocal of the lens' focal length is equal to the reciprocal of the object distance plus the reciprocal of the image distance.
- To make a reproduction of the source grid a piece of high contrast film material (possibly stiffened by a piece of glass or held in or taped to a frame) is placed where the source grid comes to a sharp focus. It goes without saying that this must be done in such a manner that only light from the source grid falls on the photosensitive emulsion. This can be accomplished by baffling or building a light tight box around the lens to film area.
- It is important to note that the reproduction of the source grid will reflect any refractive index in homogeneities present due to the inclusion of imperfect windows in the system and this feature alone often results in major cost benefits in terms of system setup since

  relatively low quality optical components can be used between the one lens in the system and source.

- After the film is exposed and developed, it is replaced in its original position. This can be checked by visually examining the registration between the image of the source grid and its physical negative reproduction as well as tracking the appearance of the subject field on a ground glass placed, as mentioned above, at two focal lengths from the lens. It should be noted that slight miss registration between the two grids allows the operator to vary the brightness of the subject field on the ground glass.
- Once this relationship has been established, placing a density gradient two focal lengths from the camera lens will cause this gradient to be visible on the ground glass. If the source of the gradients is a hot soldering iron, for example, the warm air rising from the iron should be clearly visible as rising plumes of light and dark from the iron itself.

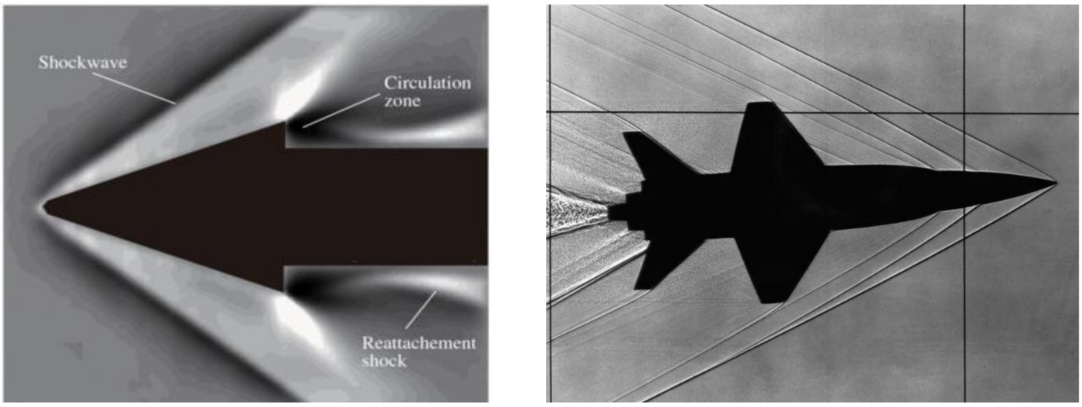


Fig. Shock captured by schlieren system

- A camera can be aimed at the ground glass and images of the disturbances created by the rising density gradients can then ultimately be photographed. Conversely, if the whole system (from the lens back to the ground glass) is built in a light tight container and the lens is equipped with shutter, then film can be the ultimate image and exposures are made by simply operating the shutter or firing a flash that illuminates the source grid for a brief instant. This is a focusing schlieren photograph made directly onto film showing warm air rising from hot soldering irons.

Result

Thus, the use of schlieren system to visualize the shock is studied.

DEPARTMENT OF AERONAUTICAL ENGINEERING

LABORATORY MANUAL

AE3412- PROPULSION LABORATORY

LIST OF EXPERIMENTS

1. Study of aircraft piston and its components.
2. Determine the velocity profiles of free jets.
3. Determine Velocity profiles of wall jets.
4. Wall pressure measurements of a subsonic diffusers and ramjet ducts.
5. Flame stabilization studies using conical and hemispherical flame holders.
6. Cascade testing of compressor blades.
7. Velocity and pressure measurements high speed jets.
8. Wall Pressure measurements of supersonic nozzle.
9. Wall pressure measurements on supersonic inlet
10. Flow visualization of supersonic flow.
11. Performance test of propeller
12. Study of gas turbine engines and its components

EXP NO: 1

STUDY OF PISTON ENGINES

INTRODUCTION

A Piston engine is a heat engine that uses one or more pistons to convert pressure into a rotating motion. The main types are the internal combustion engine used extensively in motor vehicles, the steam engine which was the mainstay of the industrial revolution and the niche application Stirling engine.

There may be one or more pistons. Each piston is inside a cylinder, into which a gas is introduced, either already hot and under pressure (steam engine), or heated inside the cylinder either by ignition of a fuel air mixture (internal combustion engine) or by contact with a hot heat exchanger in the cylinder (Stirling engine). The hot gases expand, pushing the piston to the bottom of the cylinder. The piston is returned to the cylinder top (Top Dead Centre) either by a flywheel or the power from other pistons connected to the same shaft. In most types the expanded or "exhausted" gases are removed from the cylinder by this stroke. The exception is the Stirling engine, which repeatedly heats and cools the same sealed quantity of gas.

In some designs the piston may be powered in both directions in the cylinder in which case it is said to be double acting.

COMPONENTS AND THEIR FUNCTIONS

The major components seen are connecting road, crank shaft(swash plate), crank case, piston rings, spark plug, cylinder, flywheel, crank pin and valves or ports.

In all types the linear movement of the piston is converted to a rotating movement via a connecting rod and a crankshaft or by a swash plate. A flywheel is often used to ensure smooth rotation. The more cylinders a reciprocating engine has, the more vibration-free (smoothly) it can run also the higher the combined piston displacement volume it has the more power it is capable of producing.

A seal needs to be made between the sliding piston and the walls of the cylinder so that the high pressure gas above the piston does not leak past it and reduce the efficiency of the engine. This seal is provided by one or more piston rings. These are rings made of a hard metal which are sprung into a circular grove in the piston head. The rings fit tightly in the groove and press against the cylinder wall to form a seal.

ENGINE TERMINOLOGY

Stroke: Either the up or down movement of the piston from the top to the bottom or bottom to top of the cylinder (So the piston going from the bottom of the cylinder to the top would be 1 stroke, from the top back to the bottom would be another stroke)

Induction: As the piston travels down the cylinder head, it 'sucks' the fuel/air mixture into the cylinder. This is known as 'Induction'.

Compression: As the piston travels up to the top of the cylinder head, it 'compresses' the fuel/air mixture from the carburetor in the top of the cylinder head, making the fuel/air mix ready for ignighting by the spark plug. This is known as 'Compression'.

Ignition: When the spark plug ignites the compressed fuel/air mixture, sometimes referred to as the power stroke.

Exhaust: As the piston returns back to the top of the cylinder head after the fuel/air mix has been ignited, the piston pushes the burnt 'exhaust' gases out of the cylinder & through the exhaust system.

The following is an additional parameter for a 2 stroke engine

Transfer Port: The port (or passageway) in a 2 stroke engine that transfers the fuel/air mixture from the bottom of the engine to the top of the cylinder

TYPES OF PISTON ENGINES

It is common for such engines to be classified by the number and alignment of cylinders and the total volume of displacement of gas by the pistons moving in the cylinders usually measured in cubic centimeters (cc).

In-line Engine

This type of engine has cylinders lined up in one row. It typically has an even number of cylinders, but there are instances of three- and five- cylinder engines. An in-line engine may be either air cooled or liquid cooled. It is better suited for streamlining. If the engine crankshaft is located above the cylinders, it is called an inverted engine. Advantages of mounting the crankshaft this way include shorter landing gear and better pilot visibility. An in-line engine has a higher weight-to-horsepower ratio than other aircraft engines. A disadvantage of this type of engine is that the larger it is, the harder it is to cool. Due to this, airplanes that use an inline engine use a low- to medium-horsepower engine, and are typically used by light aircraft.

Opposed Engine

An opposed-type engine has two banks of cylinders opposite each other. The crankshaft is located in the center and is being driven from both sides. The engine is either air cooled or liquid cooled, but air cooled versions are used mostly in aviation. It can be mounted either vertically or horizontally. The advantage of a horizontally-opposed engine is that it allows better visibility and eliminates fluid lock typically found on bottom cylinders. An opposed engine also has a relative advantage in being mostly free of vibration. This is due to the fact that the pistons are located left and right of the crankshaft and act as balance weights for each other.

V-Type Engine

Cylinders in this engine are arranged in two in-line banks, tilted 30-60 degrees apart from each other. The engine can be either air cooled or liquid cooled.

Radial Engine

This type of engine has a row of cylinders arranged in a circle around a crankcase located in the middle. The combination of cylinders must be an odd number in each row and may contain more than one row. The odd number of cylinders allows for every other cylinder to be on a power stroke, allowing for smooth operation. The power output is anywhere from 100 to 3,800 HP.

4 Stroke engine

Engines based on the four-stroke or Otto cycle have one power stroke for every four strokes (up-down-up-down) and are used in cars, larger boats, and many light aircraft. They are generally quieter, more efficient, and larger than their two-stroke counterparts. There are a number of variations of these cycles, most notably the Atkinson and Miller cycles. Most truck and automotive diesel engines use a four-stroke cycle, but with a compression heating ignition system. This variation is called the diesel cycle. The four strokes refer to intake, compression, combustion and exhaust strokes that occur during two crankshaft rotations per working cycle of Otto Cycle and Diesel engines. The four steps in this cycle are often informally referred to as "suck, squeeze (or squash), bang, blow."

2 Stroke engine

The two-stroke internal combustion engine differs from the more common four-stroke engine by completing the same four processes (intake, compression, combustion, exhaust) in only two strokes of the piston rather than four. This is accomplished by using the beginning of the compression stroke and the end of the combustion stroke to perform the intake and exhaust functions. This allows a power stroke for every revolution of the crank, instead of every second revolution as in a four-stroke engine. For this reason, two-stroke engines provide high specific power, so they are valued for use in portable, lightweight applications such as chainsaws as well as large-scale industrial applications like locomotives. Two-stroke engines are still commonly used in high-power, handheld applications where light weight is essential, primarily string trimmers and chainsaws. To a lesser extent, these engines may still be used for certain small, portable, or specialized machine applications. These include outboard motors, high-performance, small-capacity motorcycles, mopeds, under bones, scooters, snowmobiles, karts, ultra lights, model airplanes (and other model vehicles) and lawnmowers. In the past, two-stroke cycles were experimented with for use in diesel engines, most notably with opposed piston designs, low-speed units such as large marine engines, and V8 engines for trucks and heavy machinery

A Very Basic 2 Stroke Engine Cycle

Stroke	**Piston Direction**	**Actions Occurring during This Stroke**	**Explanation**
Stroke 1	Piston travels up the cylinder barrel	Induction & Compression	As the Piston travels up the barrel, fresh fuel/air mix is sucked into the crankcase (bottom of the engine) & the fuel/air mix in the cylinder (top of the engine) is compressed ready for ignition

Stroke 2	Piston travels down the cylinder barrel	Ignition & Exhaust	The spark plug ignites the fuel/air mix in the cylinder, the resulting explosion pushes the piston back down to the bottom of the cylinder, as the piston travels down, the transfer port openings are exposed & the fresh fuel/air mix is sucked from the crankcase into the cylinder. As the fresh fuel/air mix is drawn into the cylinder, it forces the spent exhaust gases out through the exhaust port.

A Very Basic 4 Stroke Engine Cycle

Stroke	**Piston Direction**	**Inlet & Exhaust Valve Positions**	**Actions Occurring During This Stroke**	**Explanation**
Stroke 1	Piston travels down the cylinder barrel	Inlet valve open/Exhaust valve colsed	Induction stroke	As the Piston travels down the cylinder barrel, the inlet valve opens & fresh fuel/air mixture is sucked into the cylinder
Stroke 2	Piston travels up the cylinder barrel	Inlet & exhaust valve closed	Compression stroke	As the piston travels back up the cylinder, the fresh fuel/air mix is compressed ready for ignition
Stroke 3	Piston travels down the cylinder barrel	Inlet & exhaust valve closed	Ignition (power) stroke	The spark plug ignites the compressed fuel/air mix, the resulting explosion pushes the piston back to the bottom of the cylinder
Stroke 4	Piston travels up the cylinder barrel	Inlet valve closed/Exhaust valve open	Exhaust stroke	As the piston travels back up the cylinder barrel, the spent exhaust gases are forced out of the exhaust valve

EXP NO :2

STUDY OF JET ENGINES

INTRODUCTION

A **jet engine** is a reaction engine that discharges a fast moving jet of fluid to generate thrust in accordance with Newton's third law of motion. This broad definition of jet engines includes turbojets, turbofans, rockets, ramjets, pulse jets and pump-jets, but in common usage, the term generally refers to a gas turbine Brayton cycle engine, an engine with a rotary compressor powered by a turbine, with the leftover power providing thrust. Jet engines are so familiar to the modern world that gas turbines are sometimes mistakenly referred to as a particular application of a jet engine, rather than the other way around. Most jet engines are internal combustion engines but non combusting forms exist also.

Jet engines are primarily used by jet aircraft for long distance travel. The early jet aircraft used turbojet engines which were inefficient. Modern jet aircraft usually use high-bypass turbofan engines which help give high speeds as well as, over long distances, better fuel efficiency than many other forms of transport. A large proportion of the worlds oil consumption (about 7.2% in 2004) is burnt in jet engines.

MAJOR COMPONENTS OF A JET ENGINE AND THEIR FUNCTIONS

The major components of a jet engine are similar across the major different types of engines, although not all engine types have all components.

Cold Section:

- **Air intake (Inlet)** — The standard reference frame for a jet engine is the aircraft itself. For subsonic aircraft, the air intake to a jet engine presents no special difficulties, and consists essentially of an opening which is designed to minimize drag, as with any other aircraft component. However, the air reaching the compressor of a normal jet engine must be traveling below the speed of sound, even for supersonic aircraft, to sustain the flow mechanics of the compressor and turbine blades. At supersonic flight speeds, shockwaves form in the intake system and reduce the recovered pressure at inlet to the compressor. So some supersonic intakes use devices, such as a cone or ramp, to increase pressure recovery, by making more efficient use of the shock wave system.
- **Compressor** or **Fan** — The compressor is made up of stages. Each stage consists of vanes which rotate, and stators which remain stationary. As air is drawn deeper through the compressor, its heat and pressure increases. Energy is derived from the **turbine** (see below), passed along the **shaft**.

Common:

- **Shaft** — The shaft connects the **turbine** to the **compressor**, and runs most of the length of the engine. There may be as many as three concentric shafts, rotating at independent speeds, with as many sets of turbines and compressors. Other services, like a bleed of cool air, may also run down the shaft.

Hot section:

- **Combustor** or **Can** or **Flame holders** or **Combustion Chamber** — This is a chamber where fuel is continuously burned in the compressed air.
- **Turbine** — The turbine is a series of bladed discs that act like a windmill, gaining energy from the hot gases leaving the **combustor**. Some of this energy is used to drive the **compressor**, and in some turbine engines (i.e. turboprop, turbo shaft or turbofan engines), energy is extracted by additional turbine discs and used to drive devices such as propellers, bypass fans or helicopter rotors. One type, a **free turbine**, is configured such that the turbine disc driving the compressor rotates independently of the discs that power the external components. Relatively cool air, bled from the compressor, may be used to cool the turbine blades and vanes, to prevent them from melting.
- **Afterburner** or **reheat** (chiefly UK) — (mainly military) Produces extra thrust by burning extra fuel, usually inefficiently, to significantly raise Nozzle Entry Temperature at the **exhaust**. Owing to a larger volume flow (i.e. lower density) at exit from the afterburner, an increased nozzle flow area is required, to maintain satisfactory engine matching, when the afterburner is alight.
- **Exhaust** or **Nozzle** — hot gases leaving the engine exhaust to atmospheric pressure via a nozzle, the objective being to produce a high velocity jet. In most cases, the nozzle is convergent and of fixed flow area.
- **Supersonic nozzle** — if the Nozzle Pressure Ratio (Nozzle Entry Pressure/Ambient Pressure) is very high, to maximize thrust it may be worthwhile, despite the additional weight, to fit a convergent-divergent (de Laval) nozzle. As the name suggests, initially this type of nozzle is convergent, but beyond the throat (smallest flow area), the flow area starts to increase to form the divergent portion. The expansion to atmospheric pressure and supersonic gas velocity continues downstream of the throat, whereas in a convergent nozzle the expansion beyond sonic velocity occurs externally, in the exhaust plume. The former process is more efficient than the latter.

The various components named above have constraints on how they are put together to generate the most efficiency or performance. The performance and efficiency of an engine can never be taken in isolation; for example fuel/distance efficiency of a supersonic jet engine maximizes at about mach 2, whereas the drag for the vehicle carrying it is increasing as a square law and has much extra drag in the transonic region. The highest fuel efficiency for the overall vehicle is thus typically at Mach ~0.85.

For the engine optimization for its intended use, important here is air intake design, overall size, number of compressor stages (sets of blades), fuel type, number of exhaust stages, metallurgy of components, amount of bypass air used, where the bypass air is introduced, and many other factors. For instance, let us consider design of the air intake.

TYPES, DESCRIPTION, ADVANTAGES AND DISADVANTAGES OF JET ENGINES

There are a large number of different types of jet engines, all of which achieve propulsion from a high speed exhaust jet.

Type	**Description**	**Advantages**	**Disadvantages**
Water jet	Squirts water out the back through a nozzle	Can run in shallow water, powerful, less harmful to wildlife, (indeed used by squid)	Can be less efficient than a propeller, more vulnerable to debris
Motor jet	Most primitive air breathing jet engine. Essentially a supercharged piston engine with a jet exhaust.	Higher exhaust velocity than a propeller, offering better thrust at high speed	Heavy, inefficient and underpowered
Turbojet	Generic term for simple turbine engine	Simplicity of design, efficient at supersonic speeds (~M2)	A basic design, misses many improvements in efficiency and power for subsonic flight, relatively noisy.
Turbofan	First stage compressor greatly enlarged to provide bypass airflow around engine core, and it provides significant amounts of thrust. Most common form of jet engine in use today- used in airliners like the Boeing 747 and military jets, where an afterburner is often added for supersonic flight.	Quieter due to greater mass flow and lower total exhaust speed, more efficient for a useful range of subsonic airspeeds for same reason, cooler exhaust temperature.	Greater complexity (additional ducting, usually multiple shafts), large diameter engine, need to contain heavy blades. More subject to FOD and ice damage. Top speed is limited due to the potential for shockwaves to damage engine.

Rocket	Carries all propellants and oxidants on-board, emits jet for propulsion	Very few moving parts, Mach 0 to Mach 25+, efficient at very high speed (> Mach 10.0 or so), thrust/weight ratio over 100, no complex air inlet, high compression ratio, very high speed (hypersonic) exhaust, good cost/thrust ratio, fairly easy to test, works in a vacuum-indeed works best exoatmospheric which is kinder on vehicle structure at high speed, fairly small surface area to keep cool, and no turbine in hot exhaust stream.	Needs lots of propellant- very low specific impulse — typically 100-450 seconds. Extreme thermal stresses of combustion chamber can make reuse harder. Typically requires carrying oxidizer on-board which increases risks. Extraordinarily noisy.
Ramjet	Intake air is compressed entirely by speed of oncoming air and duct shape (*divergent*)	Very few moving parts, Mach 0.8 to Mach 5+, efficient at high speed (> Mach 2.0 or so), lightest of all air-breathing jets (thrust/weight ratio up to 30 at optimum speed), cooling much easier than turbojets as no turbine blades to cool.	Must have a high initial speed to function, inefficient at slow speeds due to poor compression ratio, difficult to arrange shaft power for accessories, usually limited to a small range of speeds, intake flow must be slowed to subsonic speeds, noisy, fairly difficult to test, finicky to keep lit.
Turboprop (Turbo shaft similar)	Strictly not a jet at all — a gas turbine engine is used as power plant to drive propeller shaft (or rotor in the case of a helicopter)	High efficiency at lower subsonic airspeeds (300 knots plus), high shaft power to weight	Limited top speed (airplanes), somewhat noisy, complex transmission

Propfan/Unducted Fan	Turboprop engine drives one or more propellers. Similar to a turbofan without the fan cowling.	Higher fuel efficiency, potentially less noisy than turbofans, could lead to higher-speed commercial aircraft, popular in the 1980s during fuel shortages	Development of prop fan engines has been very limited, typically more noisy than turbofans, complexity
Pulsejet	Air is compressed and combusted intermittently instead of continuously. Some designs use valves.	Very simple design, commonly used on model aircraft	Noisy, inefficient (low compression ratio), works poorly on a large scale, valves on valved designs wear out quickly
Pulse detonation engine	Similar to a pulsejet, but combustion occurs as a detonation instead of a deflagration, may or may not need valves	Maximum theoretical engine efficiency	Extremely noisy, parts subject to extreme mechanical fatigue, hard to start detonation, not practical for current use
Air-augmented rocket	Essentially a ramjet where intake air is compressed and burnt with the exhaust from a rocket	Mach 0 to Mach 4.5+ (can also run exoatmospheric), good efficiency at Mach 2 to 4	Similar efficiency to rockets at low speed or exoatmospheric, inlet difficulties, a relatively undeveloped and unexplored type, cooling difficulties, very noisy, thrust/weight ratio is similar to ramjets.
Scramjet	Similar to a ramjet without a diffuser; airflow through the entire engine remains supersonic	Few mechanical parts, can operate at very high Mach numbers (Mach 8 to 15) with good efficiencies[5]	Still in development stages, must have a very high initial speed to function (Mach >6), cooling difficulties, very poor thrust/weight ratio (~2), extreme aerodynamic complexity, airframe difficulties, testing difficulties/expense

Turbo rocket	A turbojet where an additional oxidizer such as oxygen is added to the air stream to increase maximum altitude	Very close to existing designs, operates in very high altitude, wide range of altitude and airspeed	Airspeed limited to same range as turbojet engine, carrying oxidizer like LOX can be dangerous. Much heavier than simple rockets.

The motion impulse of the engine is equal to the air mass multiplied by the speed at which the engine emits this mass:

$$I = m\,c$$

where m is the air mass per second and c is the exhaust speed. In other words, the plane will fly faster if the engine emits the air mass with a higher speed or if it emits more air per second with the same speed. However, when the plane flies with certain velocity v, the air moves towards it, creating the opposing ram drag at the air intake:

$$m\,v$$

Most types of jet engine have an air intake, which provides the bulk of the gas exiting the exhaust. Conventional rocket motors, however, do not have an air intake, the oxidizer and fuel both being carried within the airframe. Therefore, rocket motors do not have ram drag; the gross thrust of the nozzle is the net thrust of the engine. Consequently, the thrust characteristics of a rocket motor are completely different from that of an air breathing jet engine.

The air breathing engine is only useful if the velocity of the gas from the engine, c, is greater than the airplane velocity, v. The net engine thrust is the same as if the gas were emitted with the velocity c-v. So the thrust is actually equal to

$$S = m\,(c-v)$$

Turboprops have a wide rotating fan that takes and accelerates the large mass of air but by a relatively small amount. This low speed limits the speed of any propeller driven airplane. When the plane speed exceeds this limit, propellers no longer provide any thrust ($c-v < 0$).

turbojets and other similar engines accelerate a much smaller mass of the air and burned fuel, but they emit it at the much higher speeds possible with a de Laval nozzle. This is why they are suitable for supersonic and higher speeds.

Low bypass turbofans have the mixed exhaust of the two air flows, running at different speeds (c1 and c2). The thrust of such engine is

$$S = m1 (c1 - v) + m2 (c2 - v)$$

where m1 and m2 are the air masses, being blown from the both exhausts. Such engines are effective at lower speeds, than the pure jets, but at higher speeds than the turbo shafts and propellers in general. For instance, at the 10 km attitude, turbo shafts are most effective at about 0.4 mach, low bypass turbofans become more effective at about 0.75 mach and true jets become more effective as mixed exhaust engines when the speed approaches 1 mach - the speed of sound.

Rocket engines are best suited for high speeds and altitudes. At any given throttle, the thrust and efficiency of a rocket motor improves slightly with increasing altitude (because the back-pressure falls thus increasing net thrust at the nozzle exit plane), whereas with a turbojet (or turbofan) the falling density of the air entering the intake (and the hot gases leaving the nozzle) causes the net thrust to decrease with increasing altitude. Rocket engines are more efficient than even scramjets above roughly Mach 15.

For all jet engines the propulsive efficiency (essentially energy efficiency) is highest when the engine emits an exhaust jet at a speed that is the same as the airplane velocity.

EXP NO :3

STUDY OF FREE CONVECTION APPARATUS

INTRODUCTION

Convection is a mode of heat transfer where by a moving liquid transfers heat from a surface. When the fluid movement is caused by density differences in the fluid, due to temperature variations, it is called FREE OR NATURAL CONVECTION. This provides students with a sound information about the features of free convection heat transfer from a heated vertical rod a vertical rod duct is fitted with a heated vertically placed cylinder air gets heated and dense around this cylinder, causing it to rise. This in turn gives rise to continuous flow of air upward in the duct. The instrumentation provides the heat input and temperature at different point on the heated cylinder.

AIM

To determine the theoretical and actual heat transfer co-efficient using natural convection apparatus.

APPARATUS REQUIRED

Natural convection apparatus

SPECIFICATIONS

Rod length $L = 500mm$

Rod diameter $d = 40mm$

FORMULA USED

Actual method:

Average temperature of heater $= (T_2+T_3+T_4+T_5) / 4$

Average temperature of air $= (T_1+T_6) / 2$

Power input of heater 'Q' $= VI = hA\Delta T$

Overall heat transfer co-efficient 'h' $= Q / A\Delta T$

Where ΔT – (avg temp of heater rod) – (avg temp of air)

Theoretical method:

$Nu = hl_c /K = 0.53(G_r P_r)^{1/4}$ for $G_r P_r < 10^5$

G_r – grashoff's number – $\beta g\Delta T L^3 / \gamma^2$

P_r – prandtl number

γ – kinematic viscosity

β – 1 / (mean temp of air + 273)

K – Thermal conductivity

Nu – Nusette's number = hl /K

$Nu = hl /K = 0.56(G_r P_r)^{1/4}$ for $10^5 < G_r P_r < 10^8$

$Nu = hl /K = 0.13(G_r P_r)^{1/3}$ for $10^8 < G_r P_r < 10^{12}$

PROCEDURE

Switch on the unit and adjust the regulator to provide suitable power input.
Allow some time for the unit to reach steady state condition
Note the specimen temperature (T_2,T_3,T_4,T_5) and note down the inlet temperature (T_1) and outlet temperature (T_6)
Note ammeter and voltmeter reading.
Repeat the procedure and take 3 sets of readings
Calculate the theoretical and actual heat transfer co-efficient using the given formula

RESULT

Thus the theoretical and actual heat transfer co-efficient using natural convection apparatus are calculated.

TABULATION

Voltmeter Reading in volts	Ammeter Reading in ampere	Q = VI watts	Thermocouple readings							Actual heat transfer co-efficient	Theoretical heat transfer coefficient
			T_1	T_2	T_3	T_4	T_5	T_6	ΔT		

EXP NO :4

STUDY OF FORCED CONVECTION APPARATUS

INTRODUCTION

The important relationship between Reynolds number, prandtl number and nusette number in heat exchanger design may be investigated in this self contained unit.

AIM

To determine the theoretical and actual heat transfer coefficient using forced convection apparatus.

APPARATUS REQUIRED

Forced convection apparatus.

EXPERIMENTAL SETUP

The experimental setup consist of a tube through which air is sent in by the blower the test section consists of a long electrical surface heater on the tube., which serves as a contact heat flux source on the flowing medium. The inlet and outlet temperatures of the flow are measured by the thermocouples and also the temperatures at several locations along the surface of heater from which average temperature can be measured. An orifice meter in the tube is used to measure the air flow rate with a U-tube water manometer.

An ammeter and a voltmeter are provided to measure the power input to the heater. A power regulator is provided to vary the power input to the heater. A multi-point digital temperature indicator is provided to measure the above thermocouple's input. A value is provided to regulate the flow of air.

FORMULA USED

Actual method:

Average temperature of heater = $(T_2+T_3+T_4+T_5) / 4$

Average temperature of air = $(T_1+T_6) / 2$

Power input of heater 'Q' = $VI = hA(LMTD) = mc_p\Delta T$

Where ΔT – (avg temp of heater) – (avg temp of air)

h – Heat transfer co-efficient

A – Heat transfer area = Л DL

D – Diameter of the tube

L – Length of the tube

LMTD – logarithmic mean temperature difference

$$= \frac{[(\text{avg temp of tube} - \text{outlet air temp})-(\text{avg temp of tube} - \text{inlet air temp})]}{\ln [(\text{avg temp of tube} - \text{outlet air temp}) / (\text{avg temp of tube} - \text{inlet air temp})]}$$

m – Mass flow rate of air = ρAV

V – velocity of air = Q / area of the tube

c_p – specific heat of air

Actual heat transfer co-efficient $h_{act} = Q / A(LMTD)m\ c_p\ \Delta T$

Where Q = volume of air flowing through the tube it can be calculated as follows

$$Q = C_d\ a_1 a_2 \sqrt{(2gh_0)} / \sqrt{(a_1^2 - a_2^2)}$$

Where C_d = 0.6

a_1 – area of the tube = $Л\ d_1^2/ 4$

a_2 – area of the orifice = $Л\ d_2^2/ 4$

d_1 – diameter of the pipe

d_2 – diameter of the orifice

h_0 – head of the air causing the flow

$= (h_1 – h_2)\ \rho_w / \rho_a$

ρ_w – density of water

ρ_a – density of air

Theoretical method:

Reynolds's number $R_e = VD / \gamma$

Nusette's number $Nu = hD / K$

Where D – Diameter of the tube

V – velocity of air

γ – kinematic viscosity

K – Thermal conductivity

Also Nu $=.023 \times R_e^{0.8} \times P_r^{\ 0.4}$

PROCEDURE

Switch on the mains.

Switch on the blowers.

Adjust the regulator to any desired power input the heater.

Adjust the position of the valve to any desired flow rate of air.

Wait till the steady state temperature is reached.

Note manometer readings h1 and h2.

Note the temperature along the tube.

Note the air inlet and outlet temperatures.

Note the voltmeter and ammeter readings.

Adjust the position of the valve and vary the flow rate of air and repeat the experiment.

For various valve openings and for various power inputs, the readings may be taken to repeat the experiment.

RESULT

The theoretical and actual heat transfer coefficients are determined using forced convection apparatus

EXP NO :5

BOMB CALORIMETER

INTRODUCTION

A bomb calorimeter will measure the amount of heat generated when matter is burnt in a sealed chamber (bomb) in an atmosphere of oxygen gas.

This isothermal bomb calorimeter provides a simple inexpensive yet accurate method for determination of heat of combustion (calorific value) of solid and liquid fuels. The out fit is complete for analysis as per method recommended by ISI (IS 1359-1954).

AIM

To determine the calorific value of the given solid or non-volatile liquid fuel using a bomb calorimeter

OPERATING PRINCIPLE

A known amount of sample is burnt in a sealed chamber (bomb) the air is replaced by pure oxygen. The sample is ignited electrically. As the sample burns heat is generated. The raise in temperature is noted since baring loss of heat the amount of heat generated by burning of the sample must be equal to the amount of heat absorbed by the calorimeter assembly. By knowing the energy equivalent of the calorimeter and the temperature raise, the calorific value can be found out.

PROCEDURE

Find the weight of the empty crucible using a physical balance.
A small quantity of liquid fuel (diesel) is taken in the crucible and is again weighed with fuel in it.
The crucible with fuel is placed over the support. A fuse wire is connected between the electrodes.
The bomb is closed air tight and is filled with oxygen at a pressure of about 25 bars.
The bomb is placed inside the calorimeter vessel filled with water. Noted the initial temperature of water using the digital thermometer.
The calorimeter water is stirred using a motor drive. The fuel is ignited electrically by passing a high voltage through the fuse wire which causes the fuse wire to burn.
Heat liberated by the fuel causes the temperature to rise.
After steady condition is reached the temperature raise is measured using the digital thermometer provided.

OBSERVATION

Weight of the crucible without fuel (m_1) = gm
Weight of the crucible with fuel (m_2) = gm
Initial reading of the digital thermometer (t_1) = ·c
Final reading of the digital thermometer (t_2) = ·c

CALCULATION

Mass of fuel burnt (m) = m_2 - m_1
Temperature rise (t) = t_2 - t_1

W = energy equivalent of the calorimeter assembly = 9735 J/·c
C_v = calorific value of fuel in J/gm or KJ/Kg

Then W * t = C_v * m
C_v = W * t / m

RESULT

Thus the calorific value of given solid or non volatile liquid fuel is found using bomb calorimeter.

EXP NO :6

STUDY OF PERFORMANCE OF PROPELLER

AIM

To study the performance of the propeller

BASIC PROPELLER PRINCIPLE

The aircraft propeller consists of two or more blades and a central hub to which the blades and are attached. Each blade is essentially of rotating wing. As a result of their construction, propeller blade produce forces/thrust to pull or push the aeroplane through air.

Power to rotate the propeller blades is furnished by the engines. Low powered engine propeller is mounted on the propeller shaft and that is geared to the engine crank shaft.

PROPELLER NOMENCLATURE

In order to explain the theory and construction of propellers it is necessary first to define the parts of various types of propellers and give the nomenclature associated with the propeller.

The cross section of a propeller blade is shown in the figure the leading edge of the blade trailing edge, the cambered side, or back and the flat side or face. The blade has an aerofoil shape similar to that of an aeroplane wing; it is through that it is a small wing; which has been reduced in length, width and thickness (small wing shape). When the blade start rotating, airflows around the blade fast as it flows around the wing of an aeroplane and blade is lifted forward

The nomenclature of an adjustable propeller is illustrated in the figure. This is metal propeller with two blades clamped into a steel hub assembly. The hub assembly is supporting unit for the blades, and it provides mounting structure in which propeller is attached to the engine propeller shaft. The propeller hub is split on a plane parallel to the plane of rotation of the propeller to allow for the installation of the blades. The sections of the hubs are held in place by means of clamping rings secured by means of bolts.

NOMENCLATURE FOR A CROUND ADJUSTABLLE PROPELLER

The figure shows two views of various cross sections of propeller blades. The blade shank is that portion of the blade near the butt of the blade it is usually made thick to give its strength, and it is cylindrical where it fits the hub barrel, but the cylindrical portion of the shank contributes little or no thrust. In order designs, the aero foil shape is carried to the hub by means of blade cuffs which are thin sheet metals and it function like cowling.

BLADE ELEMENT THEORY

The theory for the design of aircraft propeller was known as blade element theory. IT Is some time referred to as the DRYE WIECKI theory as the polish scientist name is DRYE WIECKI.

The theory assumes to the tip of the blade is divided into various mall, rudimentary aerofoil sections. For example if a propeller blade is 54 inch long and can be divided into 54 one-into aerofoil sections. Figure shows one of these aerofoil sections located at radius ‘r’, the chord ‘c’ will depend on the plan form or general shape of the blade.

According to the blade element theory, many aerofoil sections or elements being joined together side by side, unit to form an aerofoil (the blade) that can create thrust when revolving in a plane around central axis.

The thrust developed by a propeller is in accordance. With Newton’s third law of motion. In the case of propeller the first action is acceleration of a mass of air to rear of the aeroplane. This means that if propeller is exerting a force of 200 pounds in accelerating a given mass of air, it is the same time exerting at a force of 2000 pounds in pulling the aeroplane in the direction of opposite that in which the aeroplane is pulled forward. The quantitative realization slip among mass, acceleration, and force can be determined by the use of formula Newton’s second law.

$$F=m*a$$

True pitch propeller is one that makes use of the blade. Elemental theory. Each element of the blade travels at different rates of speed that is tip section travels faster than the section closer to the hub.

BLADE STATION

Blade stations are designated distances in inches measured along the blade from the centre of the hub the figure shows the location of a point on the blade at the 42 inches in each station this division of blade into station provides a convenient means of discussing the performance of the propeller blade locating blade marking and damage finding the proper point for measuring the blade angle and locating anti-glare areas

BLADE ANGLE

Blade angle is defined as the angle between the chord particular blade section and the plane of rotation

BLADE PITCH

Blade pitch is the distance advanced by the propeller in one revolution

GEOMETRIC PITCH

The propeller would have been advanced in one revolution

EXPERIMENTAL MEAN PITCH

The distance traveled by the propeller in one revolution without producing thrust

EFFECTIVE PITCH

Actual distance advanced by the propeller in one revolution

PITCH DISTRIBUTION

The angle gradually decreases towards the tip and towards the shank

ANGLE OF ATTACK

This is the angle formed between the chord of the blade and direction of relative air flow

PROPELLER SLIP

Slip is defined as difference between the geometric pitch and the effective pitch

FORCES ACTING ON A PROPELLER

- Thrust force
- Centrifugal force
- Torsion or twisting force
- Aerodynamic twisting force
- Aerodynamic twisting movement (ATM)
- Centrifugal twisting movement (CTM)

THRUST FORCE

Thrust force is a thrust load that tends to bend propeller blade forward as the aircraft is pulled through the air

CENTRIFUGAL FORCE

Centrifugal force is the physical force that tends to throw the rotating propeller blades away from the hub

TORSION OR TWISTING FORCE

Torsion force is the force of air resistance tends to bend the propeller blade in a direction that is opposite to the direction of rotation

AERODYNAMIC TWISTING FORCE

It is the force that tends to turn the blade to higher blade angle

AERODYNAMIC TWISTING MOMENT

It is the force that tends to turn the blade angle towards low blade angle

PROPELLER EFFICIENCY

Propeller efficiency has been achieved by use of this aerofoil section near the tips of the propeller blades and very sharp leading and trailing edge

.

Propeller efficiency id calculated = thrust horsepower / torque horse power

Its is the ratio of thrust horse power to the torque horse power. Thrust horse power is the actual amount of horse power that an engine propeller transforms x thrust

PROPELLER CHART

For a given pitch angle B, the efficiency of the propeller is a function of dimensionless quantity T, the advance ratio such as a plot for a family of pitch angle that is valuable in a propeller can be plotted. This is called the propeller chart.