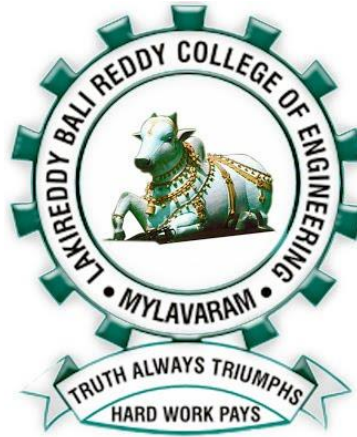


**LAKIREDDY BALI REDDY COLLEGE OF
ENGINEERING**

(AUTONOMOUS)

L B REDDY NAGAR, MYLAVARAM – 521 230, N.T.R DISTRICT, ANDHARA PRADESH



LABORATORY MANUAL

AERODYNAMICS LABORATORY

(R20 & R23 Regulations)

Name of the Student	
Roll No	
Program & Semester	
Academic Year	

LIST OF EXPERIMENTS

S.No	Name of the Experiment	Page No
1.	Determination of Lift and Drag for Symmetrical Airfoil	
2.	Determination of Lift and Drag for Unsymmetrical Airfoil	
3.	Pressure Distribution over smooth cylinder	
4.	Flow Visualization over objects in water flow channel	
5.	Potential flow visualization over objects using Hele-shaw Apparatus	
6.	Pressure distribution over symmetrical airfoil	
7.	Pressure distribution over unsymmetrical airfoil	
8.	Experimental studies on convergent nozzle	
9.	Experimental Studies on convergent-divergent nozzle (Part 1)	
10.	Experimental Studies on convergent-divergent nozzle (Part 2)	
11.	Visualization of shock waves by using Shadowgraph technique	
12.	Sample Viva-Voce Questions	

AWARD LIST

S.No	Name of the Experiment	Page No	Signature of the Instructor with Marks

No. of Exp. Conducted:

No. of Exp. Done:

Avg. Marks Awarded:

Signature of the Instructor

DETERMINATION OF LIFT AND DRAG OF A SYMMETRICAL AEROFOIL

Aim: To identify the series of a symmetrical airfoil and to draw the characteristic curves for the given airfoil at different free stream velocities.

Apparatus: Wind tunnel, airfoil, 3-component strain gauge balance.

Description: Wind tunnel is a device which produces the flow of air under controlled conditions.

Specifications:

Type	:	Low speed, open circuit and suction type
Test section	:	300mm X 300mm
Air speed	:	Up to 35m/sec
Contraction ratio	:	9:1
Drive	:	Axial flow fan driven by DC motor.
Overall size	:	1 m X 2 m X 5 m (approx)
Power requirement	:	AC 3PH, 440V, 15 Amp electrical supply
Material of construction	:	FRP/MS

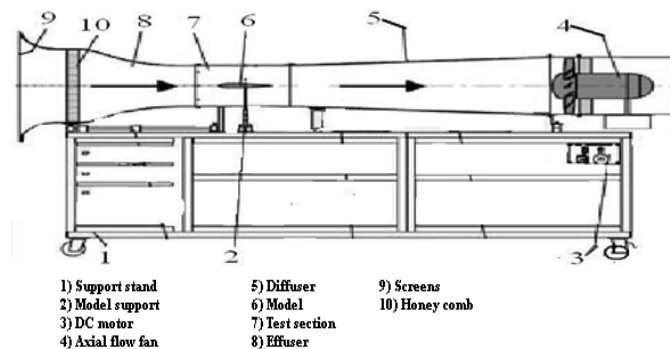


Fig 1: Low-speed wind tunnel

Symmetrical Airfoil: Airfoil is a device that produces relative force when in motion to the surrounding air; can lift or control a plain of flight. The airfoil given here is NACA 0015 symmetrical airfoil which has Chord (C) = 0.1m and Span(S) = 0.295m. The number 0015 mean to thickness is 15% of the chord. The airfoil is shown in figure 2.

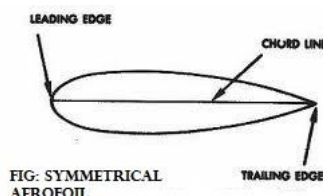


Fig 2: Symmetrical airfoil

Tabular column:

S.NO	Velocity (m/s)	Angle of attack(α) (degrees)	Lift Force(L) (N)	Lift coefficient (C_L)	Drag force(D) (N)	Drag coefficient (C_D)

S.NO	Velocity (m/s)	Angle of attack(α) (degrees)	Lift Force(L) (N)	Lift coefficient (C_L)	Drag force(D) (N)	Drag coefficient (C_D)

Calculations:

For the given aerofoil NACA 0015,

Chord (c) = 0.1m, Span (b) = 0.295m,

Then the cross-sectional area of airfoil (S) = 0.0295 m²

Let, V = Velocity in the test section, (m/s)

g= Acceleration due to gravity (m/sec²)

ρ = Density of air kg/m³,

q_∞ = Dynamic pressure = $\frac{1}{2} \rho_\infty V_\infty^2$

L = Life force (N),

D = Drag force (N),

The lift coefficient is $C_L = L / (q_\infty S)$.

The drag coefficient is $C_D = D / (q_\infty S)$

3-Component strain gauge balance: The 3-component strain gauge balance is capable of measuring lift force, drag force and pitching moment of a model mounted in the wind tunnel. The concept used here is the bending-beam principle. The forces acting on the spring element causes strain on its surface which is measured by strain gauges. The gauges are connected to the Wheatstone bridge where the small change in strain will result in the corresponding change in strain gauge resistance that producing a certain amount of voltage difference. The difference will be sent to the data acquisition system which gives the forces.

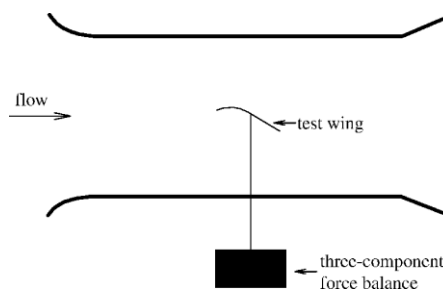


Fig 3: Three-component balance

Experimental procedure:

- 1) Open the test section side door and place the symmetrical airfoil in the test section and attach the same to the three-component balance.
- 2) Switch ON the console and observe the digital 3-component balance and velocity indicator is 'ON'.
- 3) Now increase the speed control knob slowly in the clockwise direction, so that the speed increases gradually. Set the required value of velocity by adjusting the knob.
- 4) The digital indicator of 3-component balance will show the lift and drag on the airfoil.
- 5) Tabulate the reading and find out the corresponding lift and drag coefficients.
- 6) Now repeat steps 4 and 5 for different angles of attack of the airfoil.
- 7) Repeat the same procedure for different velocities of the test section.
- 8) Plot the variation of lift and drag coefficients with angle of attack and variation of drag coefficient with lift coefficient.

Precautions:

- 1) Care should be taken while mounting the 3-component balance.
- 2) Take care while changing the angle of attack of the aerofoil.
- 3) Do not place any loose parts in the test section.

Space for Calculations:

Graphs: Draw the graphs between

- (1) Angle of attack (α) on X-axis and lift coefficient (C_L) on Y-axis.
- (2) Angle of attack (α) on X-axis and drag coefficient (C_D) on Y-axis.
- (3) Lift coefficient (C_L) on X-axis and drag coefficient (C_D) on Y-axis.
- (4) Angle of attack (α) on X-axis and ratio of lift and drag coefficients (L/D) on Y-axis.

Result:

DETERMINATION OF LIFT AND DRAG OF AN UNSYMMETRICAL AEROFOIL

Aim: To identify the series of an unsymmetrical airfoil and to draw the characteristic curves for the given airfoil at different free stream velocities.

Apparatus: Wind tunnel, airfoil, 3-component strain gauge balance.

Description: Wind tunnel is a device which produces the flow of air under controlled conditions.

Specifications:

Type	:	Low speed, open circuit and suction type
Test section	:	300mm X 300mm
Air speed	:	Up to 35m/sec
Contraction ratio	:	9:1
Drive	:	Axial flow fan driven by DC motor.
Overall size	:	1 m X 2 m X 5 m (approx)
Power requirement	:	AC 3PH, 440V, 15 Amp electrical supply
Material of construction	:	FRP/MS

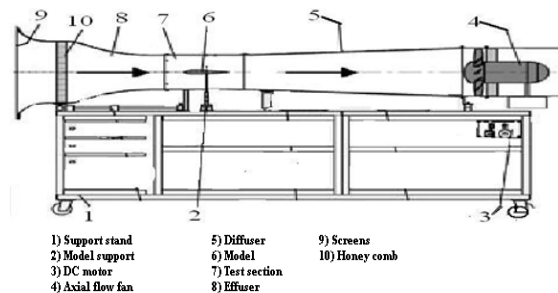


Fig 1: Low-speed wind tunnel

Unsymmetrical Airfoil: Airfoil is a device that produces relative force when in motion to the surrounding air; can lift or control a plain of flight. The unsymmetrical airfoils will also have different nomenclature as compared to symmetrical aero foil and these are also specified by a unique number as for NACA. For example, NACA 2412, the first digit indicates that the maximum camber is 0.02c (where C is chord), the second digit refers to the distance of maximum camber is about 0.4c and the last two digits refers to the maximum thickness that is about 0.12c. The unsymmetrical airfoil is shown in figure 2.

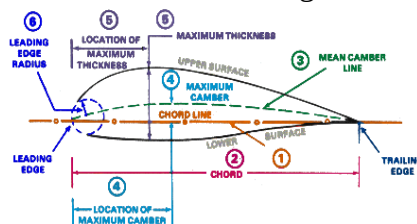


Fig 2: Unymmetrical airfoil

Tabular column:

S.NO	Velocity (m/s)	Angle of attack(α) (degrees)	Lift Force(L) (N)	Lift coefficient (C_L)	Drag force(D) (N)	Drag coefficient (C_D)

S.NO	Velocity (m/s)	Angle of attack(α) (degrees)	Lift Force(L) (N)	Lift coefficient (C_L)	Drag force(D) (N)	Drag coefficient (C_D)

Calculations:

For the given aerofoil,

Chord (c) = 0.1m, Span (b) = 0.295m,

Then the cross-sectional area of airfoil (S) = 0.0295 m²

Let, V = Velocity in the test section, (m/s)

g= Acceleration due to gravity (m/sec²)

ρ = Density of air kg/m³,

q_∞ = Dynamic pressure = $\frac{1}{2} \rho_\infty V_\infty^2$

L = Life force (N),

D = Drag force (N),

The lift coefficient is $C_L = L / (q_\infty S)$.

The drag coefficient is $C_D = D / (q_\infty S)$

3-Component strain gauge balance: The 3-component strain gauge balance is capable of measuring lift force, drag force and pitching moment of a model mounted in the wind tunnel. The concept used here is the bending-beam principle. The forces acting on the spring element causes strain on its surface which is measured by strain gauges. The gauges are connected to the Wheatstone bridge where the small change in strain will result in the corresponding change in strain gauge resistance that producing a certain amount of voltage difference. The difference will be sent to the data acquisition system which gives the forces.

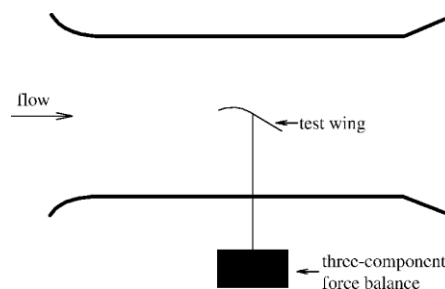


Fig 3: Three-component balance

Experimental procedure:

- 9) Open the test section side door and place the symmetrical airfoil in the test section and attach the same to the three-component balance.
- 10) Switch ON the console and observe the digital 3-component balance and velocity indicator is 'ON'.
- 11) Now increase the speed control knob slowly in the clockwise direction, so that the speed increases gradually. Set the required value of velocity by adjusting the knob.
- 12) The digital indicator of 3-component balance will show the lift and drag on the airfoil.
- 13) Tabulate the reading and find out the corresponding lift and drag coefficients.
- 14) Now repeat steps 4 and 5 for different angles of attack of the airfoil.
- 15) Repeat the same procedure for different velocities of the test section.
- 16) Plot the variation of lift and drag coefficients with angle of attack and variation of drag coefficient with lift coefficient.

Precautions:

- 4) Care should be taken while mounting the 3-component balance.
- 5) Take care while changing the angle of attack of the aerofoil.
- 6) Do not place any loose parts in the test section.

Space for Calculations:

Graphs: Draw the graphs between

- (1) Angle of attack (α) on X-axis and lift coefficient (C_L) on Y-axis.
- (2) Angle of attack (α) on X-axis and drag coefficient (C_D) on Y-axis.
- (3) Lift coefficient (C_L) on X-axis and drag coefficient (C_D) on Y-axis.
- (4) Angle of attack (α) on X-axis and ratio of lift and drag coefficients (L/D) on Y-axis.

Result:

PRESSURE DISTRIBUTION OVER A SMOOTH CYLINDER

Aim: To estimate and plot the static pressure variation over a smooth cylinder for different freestream velocities.

Apparatus: Low speed wind tunnel, smooth cylinder with pressure taps, multi tube manometer.

Note: Apparatus Description is same as given in the previous experiments.

Experimental Model (Smooth Cylinder): The construction of smooth cylinder with pressure taps is as shown in the figure below.

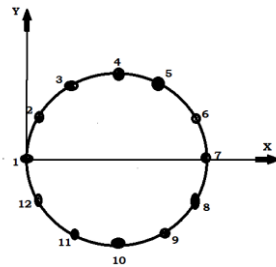


Fig 2: Smooth cylinder with pressure tap locations

Experimental procedure:

- 1) Switch ON the console.
- 2) Place the smooth cylinder in the test section along the direction of the flow.
- 3) Connect the outlets of pressure tapings to the multi tube manometer.
- 4) Set the required velocity in the test section by adjusting the knob.
- 5) Note down the manometer reading corresponding to tap number in the tabular column.
- 6) Calculate the dynamic pressure and test section static pressure by taking the head difference in the corresponding manometers.
- 7) Calculate the pressure coefficients at different locations of the cylinder and tabulate the readings.
- 8) Repeat steps 5, 6 & 7 for different positive and negative angles of attack.

Precautions:

- 1) Make sure that there are no loose parts in the test section before the start of the experiment.
- 2) Operate the fan speed control knob gently.
- 3) Make sure that pressure taps are connected to corresponding tube of multi tube manometer.
- 4) Do not place any devices behind the driving unit.

Tabular column:

S. No.	parameter	Static Pressure Port No												Static Pressure Port
		1	2	3	4	5	6	7	8	9	10	11	12	
	Initial Reading													
	Final Reading													
	Δh (m)													
	Measured pressure (p) (N/m ²)													
	Pressure Coefficient ($C_p = \frac{P - P_\infty}{\frac{1}{2} \rho_\infty V_\infty^2}$)													

Calculations:

The pressure coefficient of a symmetrical aerofoil can be calculated by using the following formula,

$$\text{Coefficient of pressure (C}_p\text{)} = \frac{P - P_\infty}{q_\infty} = \frac{P - P_\infty}{\frac{1}{2} \rho_\infty V_\infty^2}$$

Where, P = Measured pressure (N/m²).

P_∞ = Static pressure in the test section without model (N/m²).

ρ_∞ = Density of air (Kg/m³).

V_∞ = velocity in the test section (m/s)

Graphs: Draw the graph between pressure coefficient (C_p) and distance along the chord (X/C) for different angles of attack.

Space for Calculations:

Result:

FLOW VISUALIZATION OVER OBJECTS IN WATER FLOW CHANNEL

AIM: To visualize the flow filed over given objects at various Reynolds numbers.

APPARATUS: Water flow channel, models, Dye and stopwatch.

WATER FLOW CHANNEL:

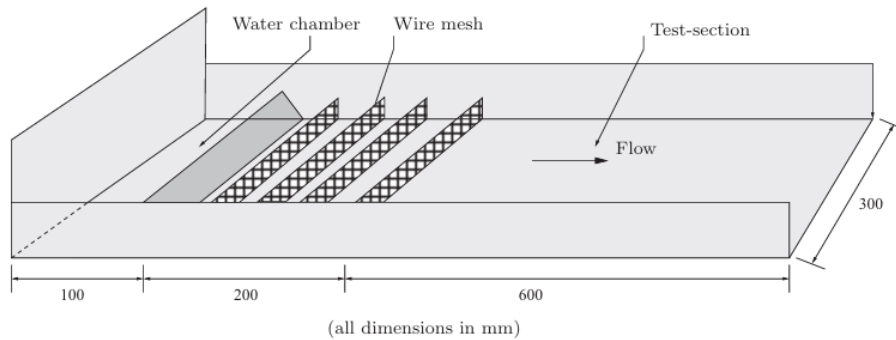


Fig 1. Schematic diagram of water flow channel

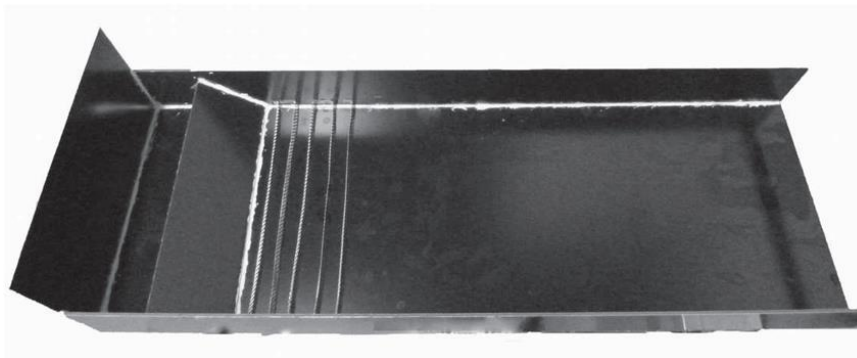


Fig 2. Pictorial view of water flow channel

Water flow channel is a simple duct through which water is made to flow with uniform velocity over the length of the channel. The uniform flow portion can be used to visualize flow past objects. The channel is made of Stainless-Steel sheet of 3 mm thickness, the sheet is bent into the shape as shown in Fig. 1. The width of the test section is about 280mm, and the depth of water will be maintained at 8mm. A pictorial view of a water flow channel, of rectangular cross-section is shown in Fig. 2. Water from the chamber spills over the inclined plate and is conditioned using an array of wire-meshes, before reaching the test-section. It is essential to ensure that the flow quality is uniform in the test section. For this a color dye may be injected at specified locations across the test-section width. If the dye streaks in the test-section are parallel and smooth, the quality of flow can be taken as good for any visualization study. The velocity of the flow may be measured using the floating-particle method technique, over the length of the test-section. The uniformity of the flow may also be checked by observing a floating body (say, a tiny bit of paper) moving from the beginning to the end of test-section of the channel. If the test-section flow is uniform, a tiny floating object will travel straight and parallel to the channel floor and sidewalls.

Calculation of Reynold's Number:

In the floating particle method, time taken for the travel of floating particle over a distance is estimated. The velocity can be calculated by dividing the distance with the time. After finding out the velocity the Reynolds number can be calculated with reference to the width of the model by using the following formula.

$$Re = \rho V D / \mu$$

Where

Re is the Reynolds number,

ρ is the density of the fluid (for Water $\rho=1000\text{Kg/m}^3$),

D is the characteristic dimension of the body. (Diameter for circle and width for flat plate).

' μ ' is the dynamic viscosity of the fluid (At atmospheric conditions).

Table of Observations:

S. No	Reynolds Number	Vortex size (mm)		Reverse flow distance (mm)	
		Object 1	Object 2	Object 1	Object 2
1					
2					
3					
4					

Theory:

Vortex motion is a fluid flow in which the streamlines are concentric circles. The flow process behind the object establishes the twin-vortex. On reaching the forward stagnation point, the flow encounters a positive pressure zone at the face of the body, popularly termed **pressure uphill**. The streamlines on either side of the stagnation streamline take an outward deflection, negotiating the pressure uphill, before reaching the edges of the body. On reaching the edge, these curved streamlines on either side of the body find a large space at the back of the body, and therefore, on reaching the edges, turn towards the body base. This process of turning towards the base results in the formation of **two vortices** of opposite nature soon after starting the flow. The formation of these vortices creates a low-pressure region at the base body. Suction at the base body increases with an increase of free stream velocity. This low-pressure region at the base attracts flow from the zone just downstream of the vortices towards the base, thus establishing a **reverse flow** towards the base.

Von Karman Vortex Street: A very interesting sequence of events begins to develop when the Reynolds number is increased beyond **40**, at which point the **wake** behind the cylinder becomes unstable. It is observed that the wake develops a slow oscillation in which the **velocity** is periodic in time and downstream distance, with the amplitude of the oscillation increasing downstream. The oscillating wake **rolls** up into two staggered rows of vortices with opposite sense of rotation.

Experimental Procedure:

- 1) Place the channel on a flat surface which has no inclination.
- 2) Now give the water supply to the channel at the water chamber so that the fairly uniform flow will be entering into the test section due to the effect of wedge and screens.
- 3) Test for the uniformity of flow i.e., whether the streamline going in a straight line from starting to the end of the channel, otherwise make some adjustments to get the uniform flow.
- 4) In this step flow velocity to be set in the channel according to the requirement, by using the floating particle method calculate the corresponding Reynolds number based on the characteristic dimension of the model.
- 5) Place the experimental model at the center of the test-section
- 6) Now inject enough dye at the second screen which helps to get the streamline flow and take the video from starting to the end of flow pattern over the object for further investigation.
- 7) Repeat the above step for different Reynolds numbers and models.
- 8) Repeat the experiment for another model.
- 9) From the video extract the frames and necessary information for the analysis.

Space for Sample Calculation:

Precautions:

- 1) Before starting of the experiment, make sure that the flow is uniform.
- 2) Do not inject too much amount of dye.
- 3) Flow should not be disturbed by any means.
- 4) The flow should be left to right in the video graph.
- 5) Take care while measuring the velocity of the flow.

Plots: Reynolds number Vs Reverse flow Distance

Observation made from results:

POTENTIAL FLOW VISUALIZATION OVER OBJECTS BY USING HELE-SHAW APPARATUS

Aim: To generate the potential flow and to visualize the potential flow pattern over given objects.

Apparatus: Hele-Shaw apparatus, measuring beaker, dye and stopwatch.

Apparatus Description:

The Hele-Shaw apparatus produces a flow pattern which is similar to that of potential flow. It is an analogy experiment known as Hele-Shaw analogy. The flow in the apparatus is a highly viscous flow between two parallel plates with a very small gap between them. In this flow the inertia force is negligible compared to the viscous force. Under this condition the flow equation has the same form as that of Euler's potential flow, however it does not satisfy the no-slip wall boundary condition. There is a slip at the wall. Many interesting phenomena pertaining to potential flow can be observed using this apparatus. The flow through Hele-Shaw apparatus is two-dimensional and incompressible. In Hele-Shaw apparatus, the viscous flow of a liquid between two closely spaced plates may be shown to simulate the streamlines in the flow of a frictionless inviscid fluid. The Hele-Shaw flow is a low Reynolds number flow which has wide application in flow visualization apparatus because of its surprising property of reproducing the streamlines of potential flows (i.e., infinite Reynolds number flows).

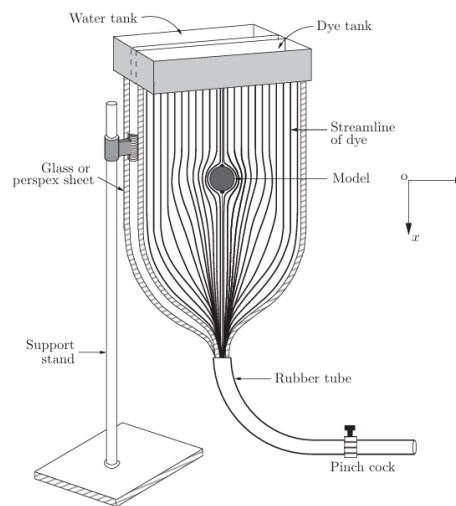


Fig. 1 Schematic of Helw-Shaw apparatus

This equipment basically consists of two parallel plates made of thick, trans parent (glass or plastic) plates clamped together along the edges with a narrow (about 1 mm) space in between them. The assembly of these two transparent sheets kept parallel with a uniform narrow space between them is provided with two small tanks of rectangular cross-section at the top end, as shown in Figure 1. The tanks are connected to the rectangular slit formed by the trans parent sheets by a set of small holes (about 1-mm diameter) arranged in a row, as

Calculation of Reynolds number:

The velocity can be calculated by measuring the amount of water collected in the beaker over a period of time. After finding out the velocity the Reynolds number can be calculated with reference to the width of the model by using the following formula

$$Re = \rho V D / \mu$$

Where

Re is the Reynolds number,

ρ is the density of the fluid, (Density of water is 1000 kg/m^3)

D is the characteristic dimension of the body, (diameter for circle, width for square and rectangle and side length for triangle)

μ is the dynamic viscosity of the fluid.

OBSERVATIONS:

S. No.	Reynolds number	Object 1		Object 2		Object 3	
		Stagnation point	Zone of influence	Stagnation point	Zone of influence	Stagnation point	Zone of influence

shown in Figure 1. The holes from the two tanks are arranged to occupy alternate locations for communication. The other end of the rectangular slit is made to terminate in a circular hole, by gradually narrowing it after a specified distance from the tanks at the top. One of the tanks is filled with water and the other with dye (say potassium permanganate). Initially the circular passage at the bottom end of the apparatus is closed and the apparatus is kept vertical. Once the passage is opened, flow of water and dye takes place through the rectangular passage. Within a short duration, a flow field of water with uniform streamlines of dye in it is established in the narrow passage between the transparent plates.

Theory:

Potential Flow:

Potential flow is an ideal flow. It is an irrotational, incompressible and inviscid flow. It does not include many of the characteristics that are encountered in real flow problems. The Hele-Shaw apparatus produces a flow pattern similar to that of potential flow.

Basic equations of Hele-Shaw Analogy:

The flow in the apparatus is actually a highly viscous flow between two parallel plates with a very small gap between them. In this flow inertia force is negligible compared to the viscous force. Under these conditions flow equations has same form that of the Euler's potential flow, however, it does not satisfy the no-slip wall boundary condition. There is slip at wall. Many interesting phenomena pertaining to potential flow can be observed using this apparatus.

Navier-Stokes equations:

$$\begin{aligned}u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} &= -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right] \\u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} &= -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] \\u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} &= -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right] \quad (1)\end{aligned}$$

Where, u, v and w are the velocity components along the x, y and z directions. The x -coordinate is along the flow, z is in the direction of the gap between the two plates, and y is the transverse direction as shown. The velocity component $w=0$, in as much as there is no flow normal to the side wall plates, and hence the z -component in the Navier Stokes equation vanishes. In the Hele-Shaw flow the inertia force is negligible compared to the viscous force and hence the left hand side of the x -and y -momentum equations can be neglected. Similarly, in the viscous terms only $\frac{\partial^2 u}{\partial z^2}$ and $\frac{\partial^2 v}{\partial z^2}$ are significant because the other gradients are comparatively small. Therefore, the x -and y -momentum equations

$$\frac{\partial p}{\partial x} = \mu \frac{\partial^2 u}{\partial z^2}$$

Space for sample calculations:

$$\frac{\partial p}{\partial y} = \mu \frac{\partial^2 v}{\partial z^2} \quad (2)$$

Also, we know that for a two dimensional laminar flow in a channel, the velocity profile is given by the relative

$$u = u_{max} \left(1 - \frac{z^2}{c^2} \right)$$

Where $2c$ is the channel width, the distance between the two plates in the Hele-Shaw apparatus and u_{max} is the maximum of the x-component of velocity u . The above velocity profile may also be written as

$$v = \frac{3}{2} u_m \left(1 - \frac{z^2}{c^2} \right) \quad (3)$$

Where u_m is the mean component value of the velocity u . Similarly the velocity along the transverse direction can be expressed as

$$v = \frac{3}{2} v_m \left(1 - \frac{z^2}{c^2} \right) \quad (4)$$

Substituting equations (3) and (4) into equations (2), we get

$$\begin{aligned} \frac{\partial p}{\partial x} &= -\frac{3\mu}{c^2} u_m \\ \frac{\partial p}{\partial y} &= -\frac{3\mu}{c^2} v_m \end{aligned} \quad (5)$$

Differentiating the x- and y- components of equation (5) with respect to y and x, respectively and comparing the equal terms, we obtain

$$\frac{\partial u_m}{\partial y} = \frac{\partial v_m}{\partial x} \quad (6)$$

In terms of stream function, the velocity components are

$$\begin{aligned} u &= \frac{\partial \psi}{\partial y} \\ v &= -\frac{\partial \psi}{\partial x} \end{aligned}$$

Assume (for the time being) that $u = u_m$ and $v = v_m$. From equation (6) we get that is

$$\frac{\partial^2 \psi}{\partial y^2} = -\frac{\partial^2 \psi}{\partial x^2};$$

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0 \quad (7)$$

Equation (7) represents a steady two dimensional, irrotational and incompressible flow. We arrived at Eq. (7) starting from full Navier-Stokes equations and simplifying them for Hele-Shaw flow. Hence, it may be concluded that the viscous incompressible flow through the Hele-Shaw apparatus is equivalent to potential flow.

Experimental procedure:

For flow visualizing with the Hele-Shaw apparatus the procedure described below may be employed,

- 1) Mount the Hele-Shaw apparatus in the vertical position.
- 2) Clean the holes which are made in the two tanks.
- 3) Place the model of interest having the thickness of same as the gap between two rectangular plates, in the middle of the test section.
- 4) Close the gate valve completely.
- 5) Fill one of the tanks with clean water and the other with the dye.
- 6) Connect the drain tube to the measuring beaker.
- 7) Open the valve slowly. Water and dye start flowing through the slit, establishing a potential flow field in the test section over the model of interest.
- 8) The flow field with the dye streaks as the streamlines flowing around the model may be taken the video for further use.
- 9) Calculate the Reynolds number.
- 10) Repeat the same procedure for different models and for different Reynolds numbers.

Precautions:

1. Clean the tank holes properly.
2. Do not mix more amount of dye.
3. Place the model at the middle of the test section.
4. Operate the gate valve gently.

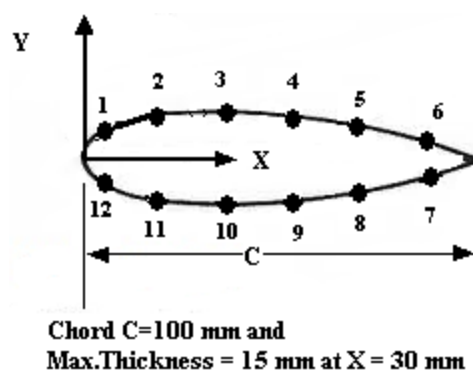
Summary:

PRESSURE DISTRIBUTION OVER A SYMMETRICAL AEROFOIL

Aim: To measure the pressure over a symmetrical airfoil (NACA 0015) at various angles of attack and for different test section velocities.

Apparatus: Low speed wind tunnel, symmetrical airfoil with pressure taps, multi tube manometer.

Note: Apparatus Description is same as given in the previous experiments.



X-Location	1	2	3	4	5	6	7	8	9	10	11	12
X/C	0.05	0.2	0.35	0.5	0.7	0.9	0.9	0.7	0.5	0.35	0.2	0.05
-----Upper surface-----							-----Lower surface-----					

Fig 1: Symmetrical aerofoil with pressure tap locations

Tabular column:

S. No.	parameter	Static Pressure Port No												Static Pressure Port
		1	2	3	4	5	6	7	8	9	10	11	12	
	Initial Reading													
	Final Reading													
	Δh (m)													
	Measured pressure (p) (N/m ²)													
	Pressure Coefficient ($C_p = \frac{p - p_\infty}{\frac{1}{2} \rho_\infty V_\infty^2}$)													

Experimental procedure:

The following procedure must be employed for estimation of pressure coefficient for symmetrical airfoil using wind tunnel.

- 1) Switch ON the console, and observe whether the velocity and pressure indicators are on.
- 2) Place the symmetrical airfoil in the test section along the direction of the flow.
- 3) Connect the outlets of pressure tapings to the multi tube manometer.
- 4) Set the required velocity in the test section by adjusting the knob.
- 5) The air will start to flow over airfoil with the given velocity.
- 6) The air will flow through the tapings provided on the surface and causes the deflection in the manometer.
- 7) Note down the manometer reading corresponding to tap number in the tabular column.
- 8) Calculate the dynamic pressure and test section static pressure by taking the head difference in the corresponding manometers.
- 9) Calculate the pressure coefficients at different locations of the airfoil and tabulate the readings.
- 10) Repeat steps 7, 8 & 9 for different positive and negative angles of attack.

Space for sample Calculations:

Precautions:

- 5) Make sure that there are no loose parts in the test section before the start of experiment.
- 6) Operate the fan speed control knob gently.
- 7) Make sure that pressure taps are connected to corresponding tube of multi tube manometer.
- 8) Do not place any devices behind the driving unit.
- 9) Avoid parallax error while placing the airfoil at different angles.

Graphs: Draw the graph between pressure coefficient (C_p) and distance along the chord (X/C) for different angles of attack.

Result:

PRESSURE DISTRIBUTION OVER AN UNSYMMETRICAL AEROFOIL

Aim: To measure the pressure over an unsymmetrical airfoil at various angles of attack and for different test section velocities.

Apparatus: Low speed wind tunnel, unsymmetrical airfoil with pressure taps, multi tube manometer.

Note: Apparatus Description is same as given in the previous experiments.

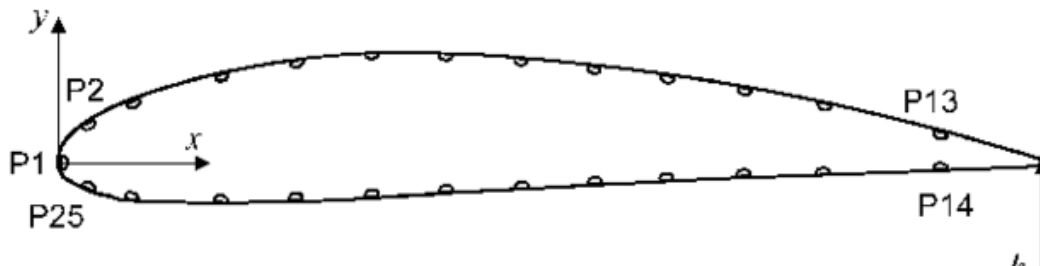


Fig 1: Unsymmetrical aerofoil with pressure tap locations

X-Location	1	2	3	4	5	6	7	8	9	10	11	12
X/C	0.05	0.2	0.35	0.5	0.7	0.9	0.9	0.7	0.5	0.35	0.2	0.05
-----Upper surface-----							-----Lower surface-----					

S. No.	parameter	Static Pressure Port No											Static Pressure Port
		1	2	3	4	5	6	7	8	9	10	11	
	Initial Reading												
	Final Reading												
	Δh (m)												
	Measured pressure (p) (N/m ²)												
	Pressure Coefficient ($C_p = \frac{P - P_\infty}{\frac{1}{2} \rho_\infty V_\infty^2}$)												

Experimental procedure:

The following procedure has to be employed for estimation of pressure coefficient for symmetrical airfoil using wind tunnel.

- 1) Switch ON the console, and observe whether the velocity and pressure indicators are on.
- 2) Place the unsymmetrical airfoil in the test section along the direction of the flow.
- 3) Connect the outlets of pressure tapings to the multi tube manometer.
- 4) Set the required velocity in the test section by adjusting the knob.
- 5) The air will start to flow over airfoil with the given velocity.
- 6) The air will flow through the tapings provided on the surface and causes the deflection in the manometer.
- 7) Note down the manometer reading corresponding to tap number in the tabular column.
- 8) Calculate the dynamic pressure and test section static pressure by taking the head difference in the corresponding manometers.
- 9) Calculate the pressure coefficients at different locations of the airfoil and tabulate the readings.

Repeat the steps 7, 8 & 9 for different positive and negative angles of attack.

Space for Calculations:

Precautions:

- 1) Make sure that there are no loose parts in the test section before the start of the experiment.
- 2) Operate the fan speed control knob gently.
- 3) Make sure that pressure taps are connected to corresponding tube of multi tube manometer.
- 4) Do not place any devices behind the driving unit.
- 5) Avoid parallax error while placing the airfoil at different angles.

Graphs: Draw the graph between pressure coefficient (C_p) and distance along the chord (X/C) for different angles of attack.

Result:

EXPERIMENTAL STUDIES ON CONVERGENT NOZZLE

Aim: To conduct experiment on the flow through convergent nozzle to understand and verify the flow physics through convergent nozzle.

Apparatus: Open jet facility, traverse mechanism, U-Tube manometer, and convergent nozzle.

Description: Open jet facility is the one which is used to study the characteristics of jets and flow thorough nozzles. The schematic diagram of open jet facility is shown in the Fig 1. Initially the air is compressed in the compressor and then it is sent into the reservoirs for storage through a high-pressure pipeline. The high-pressure air from the reservoirs tanks enters the settling chamber through a pressure regulating valve (PRV). The stagnation chamber is connected to the mixing length by a wide-angle diffuser. The flow is further conditioned inside the settling chamber by closely placing stainless steel screens. The test models are fixed to the stagnation chamber with “O” ring sealing to avoid leaks. During the experimentation the temperature in the stagnation chamber is atmospheric.

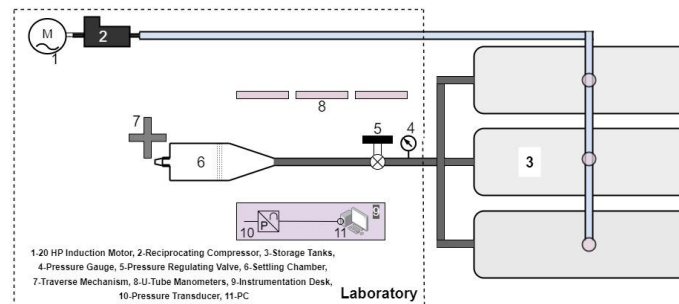


Fig. 1 Schematic of open jet facility

Physics to be Verified through the Experimentation:

(i) Verification of Stagnation Principle:

To ensure the stagnation in the chamber, we need to prove that the pressure at different locations must be same which ensures that the flow is stagnant. Measure the pressure at port 1 and port 1. If the pressure at both the ports is the same, then the stagnation is ensured in the stagnation chamber.

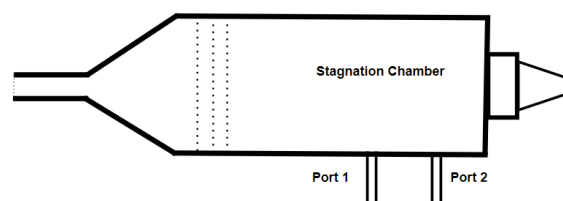


Fig. Schematic for verification of stagnation principle

Tabular column:**(i) Stagnation Chamber Proof:**

S. No.	Port 1				Port 2			
	Manometer Reading			Pressure (P_1) (N/m ²)	Manometer Reading			Pressure (P_2) (N/m ²)
	Left	Right	Δh (cm)		Left	Right	Δh (cm)	

(ii) Flow through nozzle is isentropic:

S. No.	Stagnation Chamber				Pitot Probe			
	Manometer Reading			Pressure (P_{01}) (N/m ²)	Manometer Reading			Pressure (P_{02}) (N/m ²)
	Left	Right	Δh (cm)		Left	Right	Δh (cm)	

(iii) Mach number at the exit of the nozzle:

S. No.	Pitot Pressure				Static Pressure at the Exit				Po/P	Exit Mach number (Me)
	Manometer Reading			Pressure (P0) (N/m²)	Manometer Reading			Pressure (P) (N/m²)		
	Left	Right	Δh (cm)		Left	Right	Δh (cm)			

(iv) Convergent Nozzle can Deliver Infinite Subsonic Mach numbers:

S. No.	Pitot Pressure				Static Pressure at the Exit				Po/P	Exit Mach number (Me)
	Manometer Reading			Pressure (P0) (N/m²)	Manometer Reading			Pressure (P) (N/m²)		
	Left	Right	Δh (cm)		Left	Right	Δh (cm)			

(ii) Flow through nozzle is isentropic:

To prove that the flow through the nozzle is isentropic, the stagnation pressure at the exit of the nozzle must be equal to stagnation pressure in the stagnation chamber. Measure the pitot pressure at the exit of the nozzle and compare with the inlet stagnation pressure.

(iii) Mach number at the exit:

Before conducting any sort of experiment using the nozzle, we should verify that the nozzle is delivering the flow with the Mach number that is calculated based on the isentropic relations and as per the isentropic theory. Initially set the stagnation pressure corresponding to an exit Mach number. This must be verified by calculating the Mach number at the exit of the nozzle by measuring the pitot pressure and static pressure at the exit of the nozzle.

(iv) Convergent Nozzle can Deliver Infinite Subsonic Mach numbers:

The Mach number that is delivered by a convergent nozzle depends on the pressure ratio across the nozzle. As the pressure ratio (P_0/P) increases the Mach number at the exit of the nozzle will also increase and vice versa.

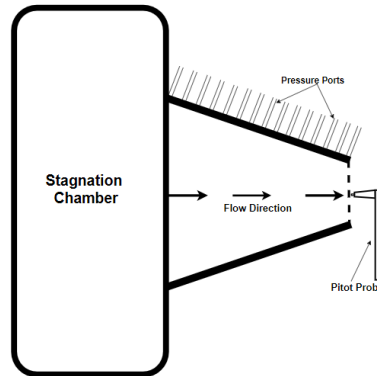
(v) Pressure and Mach number variation along the nozzle:

A nozzle is a device that is commonly used to accelerate or decelerate flow by virtue of its varying cross-section. The flow accelerates from entry to the exit of the convergent nozzle by virtue of pressure drop, the pressure energy associated with the flow is converted into kinetic energy at the exit of the nozzles. The pressure drops and the corresponding acceleration in the flow in terms of Mach number must be calculated and must be plotted against the axial distance along the nozzle length.

(vi) Convergent nozzle can't deliver supersonic flow at the exit:

The maximum Mach number that can be generated by a convergent nozzle is 1. As per the isentropic theory the Mach number generated by a convergent nozzle increases as the pressure ratio increases (Stagnation pressure (P_0) to Back Pressure (P_b) Ratio). But the convergent nozzle can't generate Mach number more than one at the exit even though the pressure ratio is sufficiently high (i.e., $P_0/P_b > 1.89$, $P_0/P_b = 1.89$ to deliver the Mach number one at the exit of the convergent nozzle). This operating condition of the nozzle is said to be the underexpanded condition of the nozzle. To prove that the Mach number is only one at the exit of the nozzle, the pressure ratio P_0/P_e (P_e is the static pressure at the exit pressure) must be 1.89 when the $P_0/P_b > 1.89$.

(v) Pressure and Mach number variation along the nozzle:



S.No	Port No	Manometer Reading (cm)		Δh (cm)	P_{gauge} (N/m ²)	P_{abs} (N/m ²)	Mach number	x/L
		Left	Right					

x- axial distance of the port from the inlet and L-Length of the nozzle

(vi) Convergent nozzle can't deliver supersonic flow at the exit:

S. No.	Stagnation Chamber				Back Pressure (P _b)	Static Pressure at the Nozzle exit				P ₀ /P _b	P ₀ /P _e	Mach number corresponding to P ₀ /P _e
	Manometer Reading			Pressure (P ₀) (N/m ²)		Manometer Reading			Pressure (P _e) (N/m ²)			
	Left	Right	Δh (cm)			Left	Right	Δh (cm)				

Experimental Procedure:

- 1) Initially fill the reservoirs with compressed air.
- 2) Mount the convergent nozzle to the stagnation chamber.
- 3) Open the pressure regulating valve for the flow.
- 4) Control the flow rate, so that the head difference in the stagnation chamber manometer shows the difference required to generate the flow with a speed of required Mach number.
- 5) Connect the ports to manometers to measure the pressure.
- 6) Calculate the corresponding Mach number(s).
- 7) Analyze the results by plotting necessary graphs.

Precautions:

- 1) Check for the stagnation principle
- 2) Operate the regulating valve gently.
- 3) Take care while taking readings of manometer.
- 4) There should be no leaks in the connections from the probe to the manometer.

Space for Calculations:

Observations from Results:

EXPERIMENTAL STUDIES ON CONVERGENT-DIVERGENT (C-D) NOZZLE

Aim: To conduct experiment on the flow through convergent-divergent nozzle to understand and verify the flow physics through nozzle.

Apparatus: Open jet facility, traverse mechanism, U-Tube manometer, pressure transducer, and convergent-divergent nozzle.

Description: Open jet facility is the one which is used to study the characteristics of jets and flow thorough nozzles. The schematic diagram of open jet facility is shown in the Fig 1. Initially the air is compressed in the compressor and then it is sent into the reservoirs for storage through a high-pressure pipeline. The high-pressure air from the reservoirs tanks enters the settling chamber through a pressure regulating valve (PRV). The stagnation chamber is connected to the mixing length by a wide-angle diffuser. The flow is further conditioned inside the stagnation chamber by closely placing stainless steel screens. The test models are fixed to the stagnation chamber with “O” ring sealing to avoid leaks. During the experimentation the temperature in the stagnation chamber is atmospheric.

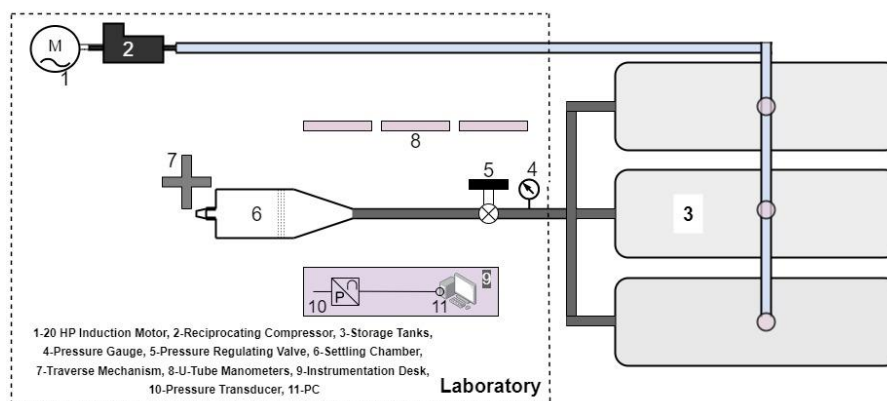


Fig. 1 Schematic of open jet facility

Theory:

The usual configuration for a converging diverging (CD) nozzle is shown in the figure. Gas flows through the nozzle from a region of high pressure (usually referred to as the chamber) to one of low pressure (referred to as the ambient or tank). The chamber is usually big enough so that any flow velocities here are negligible. The pressure here is denoted by the symbol p_c . Gas flows from the chamber into the converging portion of the nozzle, past the throat, through the diverging portion and then exhausts into the ambient as a jet. The pressure of the ambient is referred to as the 'back pressure' and given the symbol p_b .

Tabular Column:**Case 1: Subsonic Flow throughout the Nozzle**

S.No	Port No	Manometer Reading (cm)		Δh (cm)	P_{gauge} (N/m ²)	P_{abs} (N/m ²)	P/P_0	x/L
		Left	Right					

Case 2: Nozzle under just Chocking Condition

S.No	Port No	Manometer Reading (cm)		Δh (cm)	P_{gauge} (N/m ²)	P_{abs} (N/m ²)	P/P_0	x/L
		Left	Right					

Case 3: Normal Shock in the Divergent Portion of the Nozzle

S.No	Port No	Manometer Reading (cm)		Δh (cm)	P_{gauge} (N/m ²)	P_{abs} (N/m ²)	P/P_0	x/L
		Left	Right					

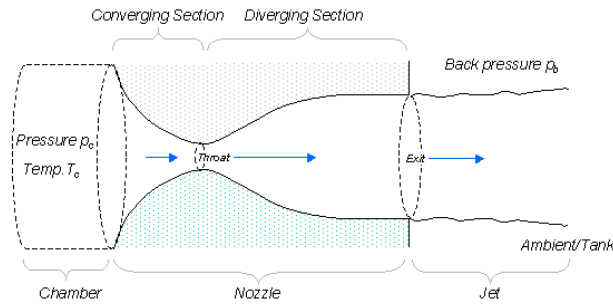


Figure 1: Converging Diverging Nozzle Configuration

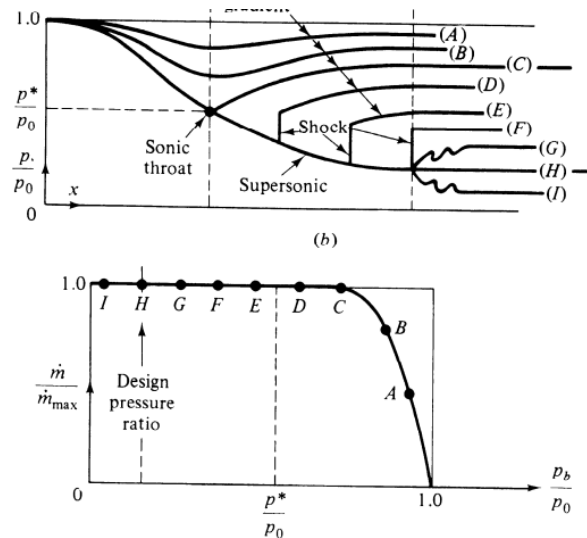


Fig. 2 Pressure distribution and mass flow rate variation curve for the flow through C-D nozzle

Figure 2 is a summary of the various possible flow regimes in a converging-diverging nozzle. Consider our experimental apparatus, where stagnation pressure P_0 (upstream of the converging-diverging nozzle) is held constant by a pressure regulator, and back pressure P_b (downstream the nozzle) is adjustable by a valve. If the downstream valve is closed, the entire nozzle is at a pressure equal to P_0 , and there is no flow. As the valve is slowly opened, the flow is at first subsonic, and follows a pressure distribution like that of curve (A) in Figure 2. Namely the pressure decreases through the converging portion of the nozzle, and then increases through the diverging nozzle. Mass flow rate is very small under these conditions. As the valve is opened further, mass flow rate increases, but the overall behavior of the pressure is the same, as seen in curve (B). The flow is still subsonic, but the velocity is high, and the Mach number near the throat (the location of minimum area) is approaching unity. Curve (C) represents the case where Mach number just reaches 1.0 at the throat. The Mach number increases in the converging part of the nozzle from nearly zero far upstream to 1.0 at the throat, then decreases again in the diverging portion of the nozzle, where the flow is again subsonic. Notice from Figure 1c that mass flux reaches a maximum value at this point. Why? Because now that the flow velocity is equal to the speed of sound at the throat, flow disturbances downstream of the throat, such as acoustic noise or fluctuations in back pressure, cannot travel upstream past the throat. The flow is referred to as choked, because

Case 4: Supersonic flow in the divergent portion and at the exit of the nozzle

S.No	Port No	Manometer Reading (cm)		Δh (cm)	P_{gauge} (N/m ²)	P_{abs} (N/m ²)	P/P_0	x/L
		Left	Right					

Space for Sample Calculations:

further decreases in back pressure have no effect on $m\dot{}$. In other words, for the given upstream (supply) conditions, this is the maximum amount of mass flow that can be forced through the nozzle, regardless of the back pressure.

As P_b is reduced even further, the flow becomes supersonic just downstream of the throat in the diverging part of the nozzle. A normal shock wave forms somewhere downstream of the throat, as illustrated in curve (D). The flow jumps from supersonic to subsonic across this normal shock. Mass flux, of course, remains fixed since the flow is choked, and upstream conditions have not changed. In curve (E), the back pressure is reduced even further, causing the shock wave to move downstream. Eventually, the shock moves to the exit plane of the nozzle, as shown in curve (F). At this point, the flow in the entire diverging part of the nozzle is supersonic but jumps to subsonic just downstream of the normal shock at the exit plane. Further decreases in P_b no longer affect the flow in the nozzle, since the entire diverging part of the nozzle is supersonic. Curve (G) represents a case where P_b is so low that the shock wave has moved downstream of the nozzle. In fact, a very complicated diamond-shaped pattern of shock waves forms outside of the nozzle. None of these has any effect on mass flow rate. Curve (H) represents the special case where P_b exactly matches P_e , the pressure at the exit plane. For this case, the flow exits the nozzle cleanly without any shock wave pattern outside the nozzle. This is commonly referred to in the literature as a fully expanded nozzle, or a nozzle running at design conditions.

Design of convergent divergent nozzle:

To design a convergent divergent (CD) nozzle the following process must be employed.

The CD nozzle must be designed for a particular Mach number

Let the Mach number be 2.

From the isentropic relation

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

Substituting $M=2$ and $\gamma=1.4$ in the above equation we get,

$$A_e/A^* = 1.668$$

Initially fix the diameter of the throat, converging angle and diverging angle as per the convenience and calculate the rest of the dimensions and make the nozzle.

Space for Sample Calculations:

EXPERIMENTAL PROCEDURE:

For conducting the experiment for the calibration of stagnation chamber the procedure described below to be employed,

- 1) Initially fill the reservoirs with the compressed air.
- 2) Mount the convergent divergent nozzle to the settling chamber.
- 3) Connect the ports of the nozzle to the pressure transducer.
- 4) Open the regulating valve to put up some flow rate.
- 5) Control the flow rate, so that the head difference in the stagnation chamber manometer shows the difference required to generate the flow with a speed of $M = 2$.
- 6) Note down the reading and calculate the gauge pressure for all the ports.
- 7) Draw the graph between X/D and wall pressure.

Observations from Results:

VISUALIZATION OF SHOCK WAVES USING SHADOWGRAPH TECHNIQUE

Aim: To visualize and capture the shock waves using shadowgraph technique.

Apparatus: Open jet facility, Nozzle, Shadowgraph set up.

Description:

For visualizing compressible flows, optical flow visualization techniques are commonly used. Interferometer, schlieren, and shadowgraph are the three popularly employed optical flow visualization techniques for visualizing shocks and expansion waves in supersonic flows. They are based upon the variation of the refractive index, which is related to the fluid density by the Gladstone–Dale formula and consequently to the pressure and velocity of the flow. For making these variations visible, three different classes of methods mentioned above are generally used. With respect to a reference ray, that is, a ray which has passed through a homogeneous field with refractive index ‘n’, the

- Interferometer makes visible the optical phase changes resulting from the relative retardation of the disturbed rays.
- Schlieren system gives the deflection angles of the incident rays.
- Shadowgraph visualizes the displacement experienced by an incident ray which has crossed the high-speed flowing gas.

Shadowgraph:

Shadowgraph visualizes the displacement experienced by an incident ray which has crossed the high-speed flowing gas.

The shadow Effect:

If the screen is placed at a position close to the test-section, the effect of ray deflection will be visible. This effect, termed shadow effect, is illustrated in Figure below.

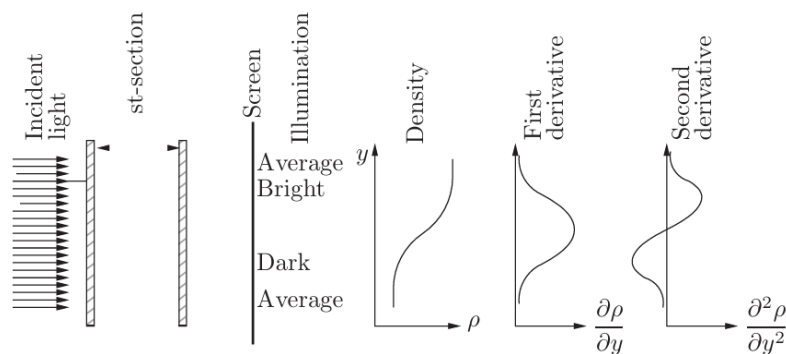


Fig. The shadow effect

A shadowgraph consists of a light source, a collimating lens, and viewing screen, as shown in the figure below. Let us assume that the test-section has stagnant air in it and that the illumination on the screen is of uniform intensity. When flow takes place through the test-

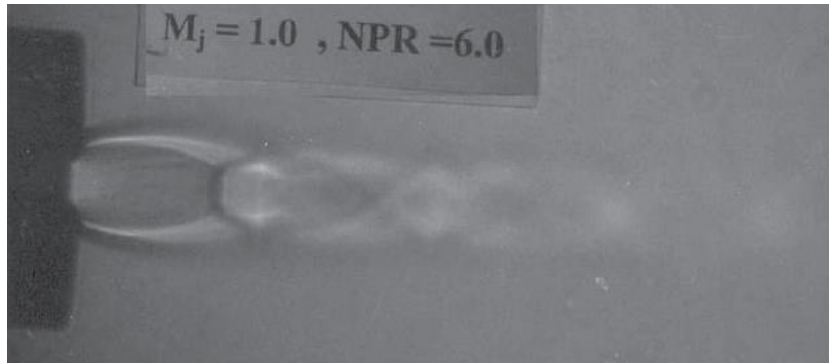


Fig. Shadowgraph of an underexpanded sonic jet operating at nozzle pressure ratio 6

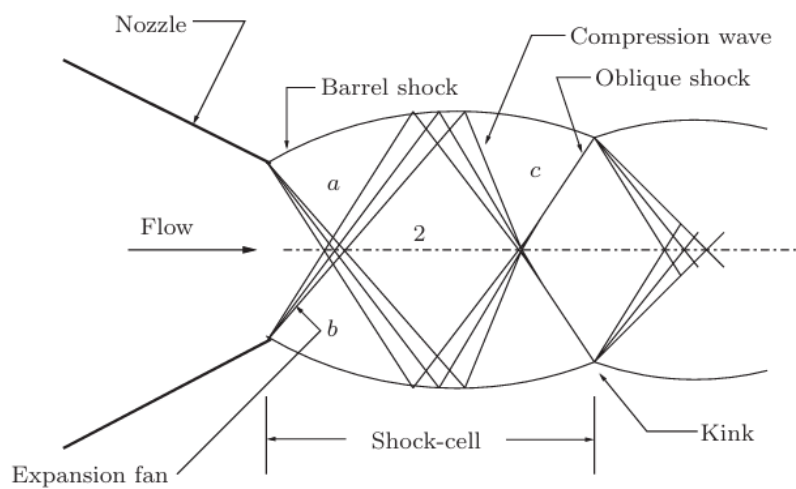


Fig. Waves in underexpanded sonic jet

section the light beam will be refracted wherever there is a density gradient. However, if the density gradient everywhere in the test-section is constant, all light rays would deflect by the same amount, and there would be no change in the illumination of the picture on the screen. Only when there is a gradient in density gradient will there be tendency for light rays to converge or diverge. In other words, the variations in illumination of the picture on the screen are proportional to the second derivative of the density.

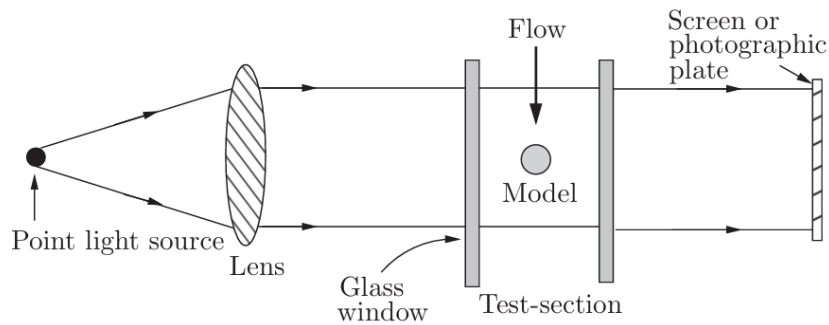


Fig. A Shadowgraph System

For a two-dimensional flow the increase of light intensity can be expressed as

$$\Delta I = k \left(\frac{\partial^2 \rho}{\partial x^2} + \frac{\partial^2 \rho}{\partial y^2} \right)$$

where k is a constant and x and y are the coordinates in a plane normal to the light path. Therefore, the shadowgraph is best suited only for flow fields with rapidly varying density gradients. A typical shadowgraph of a highly underexpanded circular sonic jet is shown in Figure 4.15. Since the jet is underexpanded, the waves present in the field would be strong enough to result in a large density gradient across them. One such wave termed Mach disk, normal to the jet axis, is seen in the field. The Mach disk is essentially a normal shock and hence, the shock has positive and negative rate of change of density gradient across it. Therefore, the shock is made up of a dark line followed by a bright line in the shadow picture, in accordance with the shadow effect.

EXPERIMENTAL PROCEDURE:

- 1) Initially fill the reservoirs with compressed air.
- 2) Mount a nozzle to the settling chamber.
- 3) Setup the shadowgraph system.
- 4) Ensure that the mirror light source and other parts are firmly placed.
- 5) Open the regulating valve to put up some flow rate.
- 6) Control the flow rate, so that you can see shock waves of the jet emanating from the nozzle.
- 7) Capture the shock waves with a camera.
- 8) Process the video and get the necessary frames from the video.

Result:

VIVA-VOICE QUESTIONS FOR AERODYNAMICS LABORATORY

1. What is a Wind tunnel?
2. What is the need of Honeycomb in wind tunnel?
3. Why are the screens used in the wind tunnel?
4. What are the types of wind tunnels?
5. What is the difference between open circuit and closed-circuit wind tunnels?
6. What does the calibration of wind tunnel mean to?
7. What is airfoil?
8. What are the forces acts on a body which is moving?
9. What are lift and drag?
10. Name the advisory committee which gives the numbering to airfoils?
11. What is the difference between symmetrical and cambered airfoil?
12. What is stall angle?
13. What is pitot probe?
14. What is pitot static probe?
15. What is static probe?
16. What is stagnation condition mean to?
17. Differentiate incompressible and compressible flows?
18. The maximum possible Mach number for convergent nozzle is?
19. What is the need of visualization?
20. The only means to generate supersonic flow is?
21. To obtain supersonic flow in the CD nozzle the velocity at the throat should be?
22. Define potential flow?
23. What is a vortex?
24. Define wake region?
25. Explain pressure uphill zone?
26. What is the separation point?
27. Zone of influence in the Hele-Shaw apparatus mean to?
28. What is bluff body?
29. What is the need of pressure distribution over the airfoil?
30. What is a shock wave?
31. What is difference between normal shock and oblique shock?
32. The shock will form only in what flows?
33. Differentiate over expanded, under expanded and correctly expanded flows?
34. Which probe is used to measure stagnation pressure?
35. The probe used to measure dynamic pressure is?
36. How the yaw effect will be on the pressure probe?
37. The exit Mach number of the convergent nozzle varies with.
38. How the pressure and Mach number varies in convergent nozzle?
39. What is the range of contraction ratio for convergent cone of a wind tunnel?
40. What is traverse mechanism? What for it is used?
41. The velocity in wind tunnel can be measured by measuring what pressure?
42. What is potential core region?
43. The region up to which the exit velocity of jet remains constant is?
44. What is a shock cell?
45. Explain the behavior of oblique shock when it is exposed to different boundaries?