EXPERIMENT SEVEN: FLOW VISUALIZATION AND ANALYSIS

I OBJECTIVE OF THE EXPERIMENT:

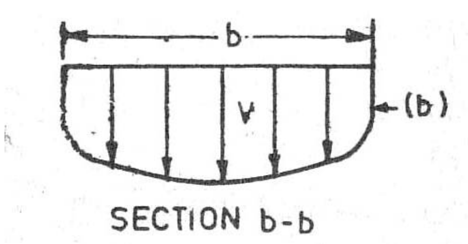
Visualization of flow pattern over or around immersed objects in open channel flow.

II THEORY AND EQUATION:

Open channel:
An open channel is a passage in which liquid flows with its upper surface exposed to atmosphere. In comparison to pipe flow, where flow occurs in closed passages under pressure, the flow in open channels takes place under the influence of gravity in open channels.
The primary purpose of this piece of apparatus is to demonstrate visually a wide range of hydraulic effects associated with flow in open channels.

Velocity Distribution over Channel Cross Section:
Typical velocity distribution curves in a straight reach of a rectangular channel are shown in figure below.

It is clear from the figure that the velocity diminishes towards the sides and the base of the channel because of frictional resistance. Theoretically the velocity of flow should be maximum at the topmost point on the vertical centre line. However due to effect of surface tension and resistance offered by the air the velocity is reduced at the free water surface.



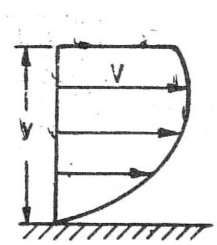


Figure: Horizontal velocity curve *Figure: Vertical Velocity curve*

Source: F1-00, Armfield Ltd. England, April 1996

Vertical velocity curve is a representation of velocity measured along a vertical line of channel cross section. Horizontal velocity curve is a representation measured along a horizontal line of channel cross section.

Reynolds Number:
It is the ratio of inertia force to the viscous force acting in any flow phenomenon.

Inertia force = mass X acceleration
Viscous force = viscous shear stress X area of flow
For open channel,
Reynolds Number (Re) = $\rho VR/\mu$

Where, V = avg. velocity of flow in channel
R = Hydraulic radius (ratio of area of flow to wetted parameter)

Reynolds number is very useful in predicting whether the flow is laminar or turbulent and for finding out the coefficient of friction in order to determine the frictional loss of head accurately.

Laminar flow:

When the fluid particle move in layers called lamina, then the flow is called laminar flow. Laminar flow occurs when velocity of flow is small and viscous forces are predominant. It is smooth and regular and thus also known as stream-line flow. There is practically no influence of fluid particles of one layer over those of the adjacent layer. Velocity at any point remains nearly constant in magnitude and direction. Flow through circular pipes with Reynolds number < 2000 is always laminar. In case of channel flow, if Reynolds number < 500, then the flow is laminar.

Turbulent flow:

When the velocity of flow reaches a certain limit the fluid particles no longer move in layers or lamina. Violent mixing of fluid particles takes place due to which they move in chaotic and random manner. As a result the velocity at any point varies both in magnitude and direction from instant to instant. Such a flow is known as turbulent flow. Flow through circular pipes with Reynolds number > 4000 is turbulent. In case of channel flow, flow with Reynolds number > 2000 is turbulent.

Steady and unsteady flow:

If the condition of flow e.g. depth of flow, velocity etc. do not vary with respect to time, then the flow is said to be steady. Otherwise it is unsteady flow.

Uniform and non uniform flow:

If the velocity, depth, cross-section area of flow remains constant over a given length of the channel, then the flow is said to be uniform flow. If the flow conditions change from section to section along the length of channel, then it is non uniform flow.

Boundary Layer:

When a real fluid flows past a solid surface, a fluid particle on the surface will have the same velocity as that of the surface because of the viscosity of the fluid. The fact is generally known as the no-slip condition at the boundary. If the boundary is static, the fluid particle on it will also have zero velocity. Further away from the boundary, the fluid velocity gradually increases. This gives rises to large shear stresses at the boundary. At the outer edge of boundary layer, the fluid velocity is very nearly the same as the local main stream velocity. In the region outside of the boundary layer, the velocity variation at any section is small and hence the shear stresses are practically negligible. Thus the flow in this region can be regarded as frictionless.

Boundary layer thickness:

The velocity u at any section approaches the local free stream velocity U asymptotically i.e. $u \rightarrow U$, as $y \rightarrow \infty$. For practical purposes, however, the boundary layer thickness δ is defined as that distance from the plate at which u=0.99 U.

Separation of boundary layer flow:

When flow takes place over a curved body under adverse or positive pressure gradient i.e. pressure increasing in the direction of flow, then the flow near the boundary is retarded much and, very soon, a point is reached where it separates from the boundary. This is known as separation point. The reason of this phenomenon is the adverse pressure gradient which tends to reduce the momentum of flow within the boundary layer due to higher viscous stresses.

Forces on immersed body:

Whenever there is relative motion between a real fluid and a body, the fluid exerts a force on a body and the body exerts equal and opposite force on the fluid. A body wholly immersed in a real fluid may be subjected to two kinds of forces.

a) **<u>Drag force</u>**: The component of force in the direction of flow on a submerged body is called drag force (F_D).

b) **<u>Lift force</u>**: The component of force in the perpendicular to the flow is called the lift force (F_L).

$$F_D = \int_A p\,dA \sin\theta + \int_A \tau_0\,dA \cos\theta$$

$$F_L = \int_A \tau_0\,dA \sin\theta - \int_A p\,dA \cos\theta$$

Where P = Pressure

T = Shear stress

A = Cross sectional Area.

The component due to pressure is known as pressure drag while the component due to shear stress is known as friction drag / shear drag. The relative contribution of pressure drag and friction drag to the total drag F_D in any particular case depends upon the shape of the body.

In the symmetrical body moving through an ideal fluid (no viscosity) at a uniform velocity, the pressure distribution around a body is symmetrical and hence the resultant force acting on the body is zero. However real fluids such as air, water, posses viscosity and if the is moved through these fluid at a uniform velocity, it is observed that the body does experience a resistance to motion.

For the body moving through a fluid density ρ at a uniform velocity U, the mathematical expression for the calculation of the drag and the lift forces are given by

$$F_D = C_D A \frac{\rho U^2}{2}$$

$$F_L = C_L A \frac{\rho U^2}{2}$$

Where,

C_D = coefficient of drag

C_L = coefficient of lift

A = characteristics area

= area projected on a plane perpendicular to the relative motion of the fluid, in the case of calculating F_D.

= area projected on a plane perpendicular to the direction of lift force, in the case of calculating F_L.

For the symmetrical body such as sphere and cylinder facing the flow is symmetrical, here is no lift force. The production of lift force requires asymmetry of flow, while drag force exists always. It is possible to create drag without lift but impossible to create lift without drag. The fluid viscosity affects the flow around the body causes the force on the body accordingly; at low Reynolds' Number the fluid is deformed in very wide zone around the body causing pressure force & friction force and as Reynolds' Number increases, viscous effects are confined to the boundary layer causes predominant the friction force on the boundary.

In case of real fluid flowing past a sphere, the varying pressure distribution present due to viscosity creates drag force. In case of body with sharp corners set normal to the flow, the wake extends across the full projected width of the body resulting in the fairly constant value of C_D. If the body has curved sides, it is laminar/turbulent boundary layer. This determines the size of wake and in term the amount pressure drag. For very low Reynolds number ($DV/\nu < 1$) the flow about the sphere is completely viscous & the friction drag is given by strokes law: $F_D = 3\pi\mu VD$.

The main objective of s streamline body is to move point of separation as back as possible to minimize size of turbulent wake. The optimization of SL is to minimize the sum of the friction and pressure drag

III DESCRIPTION OF EQUIPMENT SET-UP:

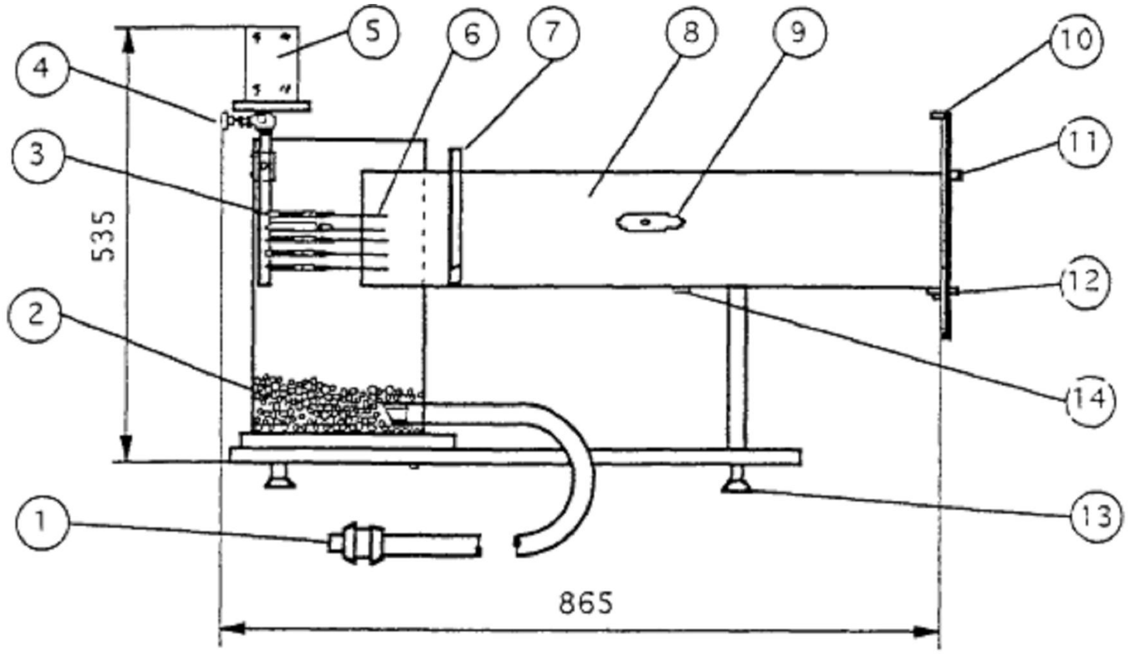


Figure: Flow Visualization Apparatus
Source: F1-00, Armfield Ltd. England, April 1996

The inlet pipe (1) is connected direct to the Hydraulics Bench outlet by a quick release connector. *The* model is positioned on the side channels of the bench top, with the overshot weir adjacent to the volumetric tank. Adjustable feet (13) are provided for leveling the apparatus.

Water is fed to the streamlined channel entry via a stilling tank which incorporates marbles (2) to reduce turbulence. The channel (8) consists of a Perspex working section of large depth to width ratio incorporating an undershot weir (7), and an overshot weir (10) at the Inlet and discharge ends respectively.

Water discharging from the channel is collected in the volumetric tank of the Hydraulics Bench and returned to the sump for recirculation. A dye injection system, consisting of a reservoir (5), flow control valve (4), manifold (3) and hypodermic **A** tubes *(6),* is incorporated at the inlet to the channel and permits flow visualization in conjunction with a graticule on the rear face of the channel. The overshot weir (10) is fully raised for low visualisation experiments, this is achieved by releasing thumb screw (11) and weir support (12), moving the weir to the desired position and locking the screws.

Before use the packet dye supplied must be diluted with 1 liter of deionised/distilled water. Open the *3gm* packet of Blue Dye and pour the contents along with 1 liter of deionised or distilled water into the 1 l k e bottle (supplied), "shake well". The 1 liter bottle can be used to store the unused Blue Dye.

V QUESTION AND ANSWER:

1. What are the parameters those affect the transformation of laminar flow to turbulent flow?

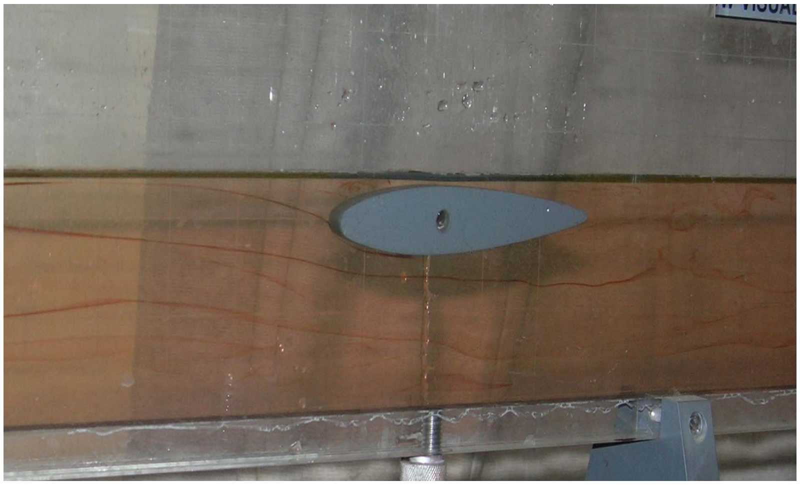
VI. OBSERVATION AND RESULT TABLE:

The flow visualization technique involves the use of dye injected at the hypodermic tubes. In operation, the overshot weir should be raised fully and the undershot weir should be removed. With the overshot weir in the raised position, the channel run full of water enabling flow patterns around and over submerged objects to be demonstrated.

CASE I: Immersed object: Broad crested weir

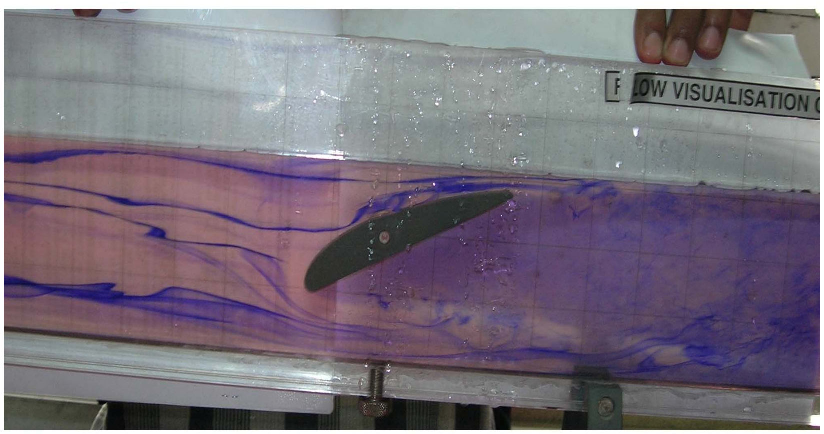
Flow is laminar at the upstream of the weir and converted into turbulent at the downstream. Turbulence occurs near the weir only. Boundary layer can be clearly viewed. At the top of the downstream of weir, separation of flow occurs.

CASE II: Immersed object: Symmetrical aerofoil placed horizontal



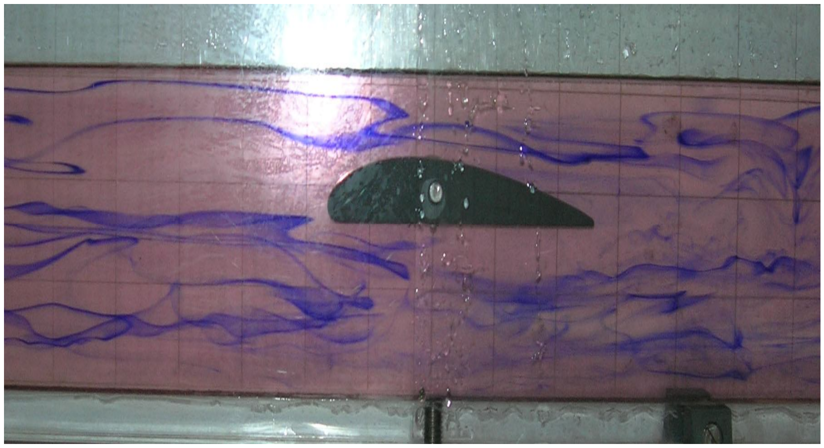
Boundary layer separation is not clear. Flow is nearly laminar around the body except the formation of small wake on the upper side of the airfoil.

CASE III: Immersed object: Asymmetrical airfoil inclined to horizontal



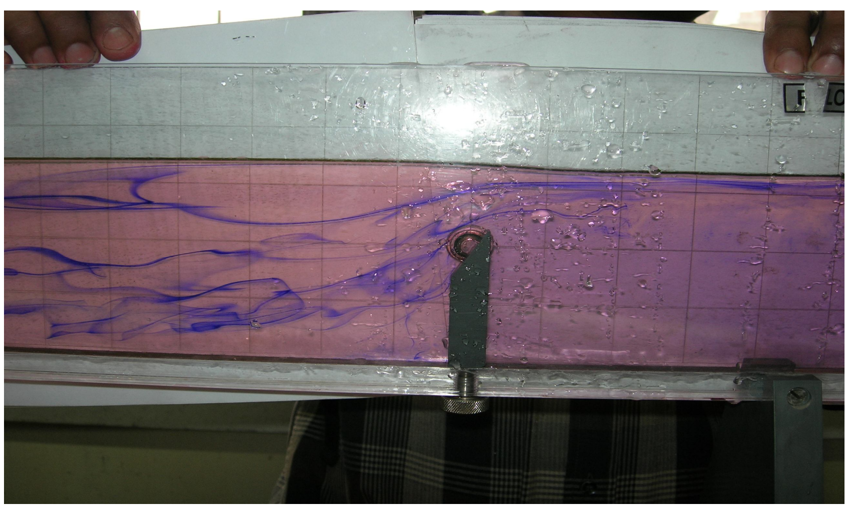
The flow is laminar above the airfoil, though some wakes are present, while the flow below the airfoil is turbulent. The flow is turbulent around the airfoil only. Flow far from the airfoil remains laminar.

CASE IV: Immersed object: Asymmetrical aerofoil



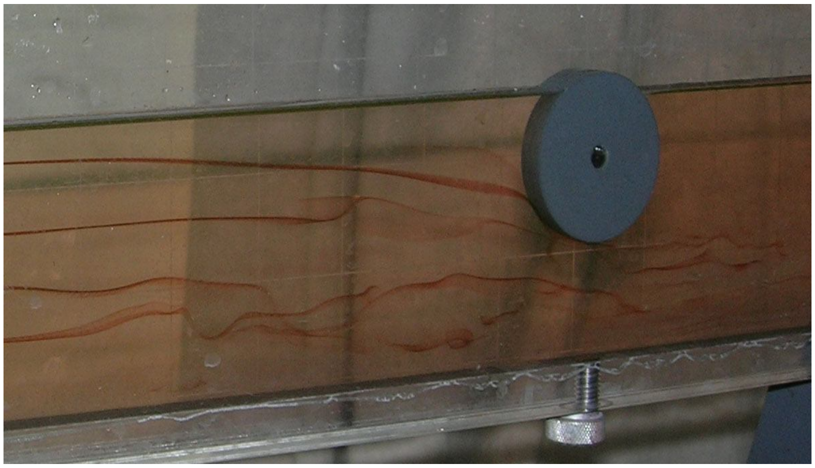
Flow is laminar through out the object except formation of small wake at the trailing edge of the airfoil. The boundary layer separation is clear.

CASE V: Immersed object: Narrow crested weir



The laminar flow converted into turbulent flow after passing through the narrow crested weir. The flow separation took place at the tip of the weir edge. The boundary layer separation is almost absent.

CASE VI: Immersed object: Circular object



Boundary layer can be viewed clearly around the boundary of circular object. No vortexes were found. The laminar characteristic of flow was not disturbed significantly.