Subject : FLUID DYNAMICS 2
HYDRAULICS MACHINES

5th Semester, BTech (Mechanical)

Prepared by

Dr. Hrushikesh Barik
Assistant Professor (Reader)
Mechanical Engg. Department
VSS University of Technology, Burla

FLUID DYNAMICS & HYDRAULICS MACHINES

Module - I

1. Dimensional Analysis and Principles of Model Testing
2. Navier stokes equation and its solution

Module - II

3. Boundary layer theory
4. Drag and lift

Module - III

5. Hydraulic Turbines

Module - IV

6. Centrifugal Pump
7. Reciprocating Pump
8. Miscellaneous machines.

Dimensional Analysis and Principle of Model Testing

⇒ There are several instances in which we claim a simplified flow exist

⇒ For example, we have stated that a flow with typical speed V will be essentially incompressible, if the Mach number, $M = \frac{V}{c}$ (where c is the speed of sound) is less than about 0.3 and that we can neglect viscous effects in most of the flow if the Reynolds number $Re = \frac{\rho V L}{\mu}$ (L is a typical or characteristic size scale of flow) is large.

⇒ We will also make extensive use of the Reynolds number based on the pipe diameter, D. ($Re = \frac{\rho V D}{\mu}$) to predict with a high degree of accuracy wheater the pipe flow is laminar and turbulent

⇒ It turns out that there are many such interesting dimensionless groupings in engineering science — for example, in heat transfer, the value of Biot number, $Bi = \frac{hL}{K}$ of a hot body, size 'L' and conductivity 'K' indicates wheater that body will tend to cool on the outside surface ~~surface~~ first or will basically cool uniformly when it's ~~is~~ plunged into a cool fluid with convection coefficient 'h'. (Can you figure out what a high Bi number predicts?)

⇒ How do we obtain these groupings, and why do their values have such powerful predictive power?

⇒ The answer to these questions will be provided in this chapter when we introduce the method of dimensional analysis.

⇒ This is a technique for gaining insight into the fluid flows (in fact, into many engineering and scientific phenomena) before we ~~either~~ do either extensive theoretical analysis or experimentation.

⇒ It also enables us to extract trends from data that would otherwise remain disorganized and incoherent.

* We will also discuss Model Testing modeling. For example, how do we correctly perform a 3/8-scale model test on drag of an automobile in a wind tunnel to predict what the drag would be on a full-size automobile at the same speed?

⇒ Must we use the same speed for the model and the full size automobile?

⇒ How do we scale up the measured model drag to find the automobile drag?

Nondimensionalizing the Basic Differential Equations:

⇒ Before describing dimensional analysis let us see what we can learn from our previous analytical description of fluid flow.

⇒ Consider, for example, a steady incompressible two-dimensional flow of a Newtonian fluid with constant viscosity (already quite a list of assumption!)

The mass conservation equation becomes

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

The Navier-Stokes eq^n

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \quad -(2)$$

and

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = -\rho g - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \quad -(3)$$

These equations form a set of coupled nonlinear partial differential equation u, v, and p and are difficult to solve for most flows.

Eq^n (1) has dimension of 1/time and Eq^n (2) & (3) have dimensions of force/volume.

→ Let us see what happens when we convert them into dimensionless equations.

→ To nondimensionalize these equations, divide all lengths by a reference length L, and all velocity by reference speed V_∞ which usually is taken as the freestream velocity. Make the pressure nondimensional by dividing by ρV_∞^2 (twice the freestream dynamic pressure)

→ Denoting nondimensional quantities with asterisks, we obtain

$$x^* = \frac{x}{L}, \quad y^* = \frac{y}{L}, \quad u^* = \frac{u}{V_\infty}$$

$$v^* = \frac{v}{V_\infty}, \text{ and } p^* = \frac{p}{\rho V_\infty^2} \quad \text{— (4)}$$

so that $x = x^* L$

$y = y^* L$

$u = u^* V_\infty$ and so on.

We can then substitute into equation (1) through eqⁿ (3), below we show two representative substitutions.

$$u\frac{\partial u}{\partial x} = u^* V_\infty \frac{\partial (u^* V_\infty)}{\partial (x^* L)}$$

$$= \frac{V_\infty^2}{L} \; u^* \frac{\partial u^*}{\partial x^*}$$

and

$$\frac{\partial^2 u}{\partial x^2} = \frac{\partial^2 (u^* V_\infty)}{\partial (x^* L)^2}$$

$$= \frac{V_\infty}{L^2} \; \frac{\partial^2 u^*}{\partial x^{*2}}$$

Using ~~this~~ procedure, the equations become

$$\frac{V_\infty}{L} \frac{\partial u^*}{\partial x^*} + \frac{V_\infty}{L} \frac{\partial v^*}{\partial y^*} = 0 \quad — (5)$$

$$\frac{\rho V_\infty^2}{L} \left(u^* \frac{\partial u^*}{\partial x^*} + v^* \frac{\partial u^*}{\partial y^*} \right)$$

$$= - \frac{\rho V_\infty^2}{L} \frac{\partial p^*}{\partial x^*} + \frac{\mu V_\infty}{L^2} \left(\frac{\partial^2 u^*}{\partial x^{*2}} + \frac{\partial^2 u^*}{\partial y^{*2}} \right)$$

$$— (6)$$

$$\frac{\rho V_\infty^2}{L}\left(u^* \frac{\partial v^*}{\partial x^*} + v^* \frac{\partial v^*}{\partial y^*}\right)$$

$$= -\rho g - \frac{\rho V_\infty^2}{L}\frac{\partial p^*}{\partial y^*} + \frac{\mu V_\infty}{L^2}\left(\frac{\partial^2 v^*}{\partial x^{*2}} + \frac{\partial^2 v^*}{\partial y^{*2}}\right) \quad \text{— (7)}$$

Dividing eq^n (5) by $\frac{V_\infty}{L}$

and eq^n (6) & eq^n (7) by $\frac{\rho V_\infty^2}{L}$ gives.

$$\frac{\partial u^*}{\partial x^*} + \frac{\partial v^*}{\partial y^*} = 0 \quad \text{— (8)}$$

$$u^* \frac{\partial u^*}{\partial x^*} + v^* \frac{\partial u^*}{\partial y^*} = -\frac{\partial p^*}{\partial x^*} + \frac{\mu}{\rho V_\infty L}\left(\frac{\partial^2 u^*}{\partial x^{*2}} + \frac{\partial^2 u^*}{\partial y^{*2}}\right) \quad \text{— (9)}$$

$$u^* \frac{\partial v^*}{\partial x^*} + v^* \frac{\partial v^*}{\partial y^*} = -\frac{gL}{V_\infty^2} - \frac{\partial p^*}{\partial y^*} + \frac{\mu}{\rho V_\infty L}\left(\frac{\partial^2 v^*}{\partial x^{*2}} + \frac{\partial^2 v^*}{\partial y^{*2}}\right) \quad \text{— (10)}$$

Eq^n (8), (9) & (10) are the nondimensional forms of our original equations (1), (2) & (3).

Eq^n 9 contains a dimensionless coefficient $\frac{\mu}{\rho V_\infty L}$ (which we recognize as the inverse of the Reynolds number) in front of the second order (viscous) terms.

Eq^n 10 contains this and another dimensionless coefficient $\frac{gL}{V_\infty^2}$ (which we will discuss shortly) for the gravity force term.

(*) We recall from the theory of differential equations that the mathematical form of the solution of such equations is very sensitive to the values of the coefficients (e.g. certain second-order partial differential equations can be elliptical, parabolic or hyperbolic depending on coefficient values)

(*) These equations tell us that the solution and hence the actual flow pattern they describe, depends on the values of the two coefficients.

(*) For example.

if $\frac{\mu}{\rho V_\infty L}$ is very small (i.e. we have a high Reynolds number), the second-order differentials, representing viscous forces can be neglected, at least in most of the flow, and we end up with a form of Euler's equations.

(*) We say "in most of the flow" because we have already learned that in reality for this case we will have a boundary layer in which there is significant effect of viscosity, in addition, from a mathematical point of view, it is always dangerous to neglect higher-order derivatives, even if their coefficients are small, because reduction to a lower-order ~~derivatives~~ equation means we lose a boundary condition (specifically the no-slip condition)

(*) We can predict that if $\dfrac{\mu}{\rho V_\infty L}$ is large or small, then viscous forces ~~are~~ will be significant or not, respectively.

(*) We can thus gain insight even before attempting a solution to the differential equation.

(*) Note that for completeness, we would have to apply the same non-dimensionalizing approach to the boundary conditions of the problem, which often introduce further dimensionless coefficients.

⇒ Writing nondimensional forms of the governing equations, then, can yield insight into the underlying physical phenomena, and indicate which forces are dominant.

⇒ If we had two geometrically similar but different scale flows satisfying eqn 8, 9 & 10 (for example model & prototype), the equation would only yield the same mathematical results if the two flows had the same values for the two coefficients.

⇒ This non-dimensional form of the equations is also the starting point in numerical methods, which is very often the only way of obtaining their solution.

⇒ Method of dimensional analysis can be used to find appropriate dimensionless grouping of physical parameters.

Dimensional Homogeneity :-

The method of dimensional analysis ~~homogeneity~~ are based on Fourier's principle of dimensional homogeneity."

It states that an equation which express a physical phenomenon of fluid flow must be algebraically correct and dimensionally homogeneous.

$t = 2\pi\sqrt{\frac{L}{g}} = [T]$

consider the eqn $V = \sqrt{2gH}$

Dimension of L.H.S = Dimension of R.H.S = $[LT^{-1}]$

So this eqn is dimensionally homogeneous. So it can be used in any system of units.

Method of Dimensional Analysis :-

If the number of variable involved in a physical phenomenon are known, then the relation among the variables can be determined by the following two method.

(i) Rayleigh's method
(ii) Buckingham's π - theorem.

Rayleigh's method

This method is used for determining the expression for a variable which depends upon maximum of 3 or 4 variable only. If the number of independent variables becomes more than four, then it is very difficult to ~~to~~ find the expression for the dependent variable.

Let X is a variable, which depends on X_1, X_2 and X_3 variable.

Then accordg to Rayleight method

X is a function of X_1, X_2, X_3

Mathematically $X = f[X_1, X_2, X_3]$

X is a variable which depends on x_1, x_2, x_3 variable

This can also be written as

$$X = K\, x_1^a x_2^b x_3^c$$

where K is a constant and a, b, c are arbitary power.
The values of the a, b & c are obtained by comparing the fundamental dimension ~~as~~ on both the sides.

Problem

Consider the flow of a fluid through a small orifice discharging freely into the atmosphere under a constant head.

Let Q = be the discharge through a small orifice of diameter '$\underline{d}$' under a constant head '$\underline{H}$'

ρ = mass density
μ = dynamic viscosity } of the fluid flowing through the orifice

g = Acc^n due to gravity.

From dat given.

$$Q = f(\mu, \rho, d, H, g) \quad -①$$

which by Rayleigh's method may be expressed in an exponential form

$$Q = K(\mu^a \rho^b d^c H^d g^e) \quad -②$$

where K is a constant.

Substitute the proper dimensions for each variable in this exponential eq^n in M-L-T system.

$$F = \mu \frac{du}{dy} A$$

$$\mu = M L^{-1} T^{-1}$$

$$\frac{L^3}{T} = [M^0 L^0 T^0]\left[\left(\frac{M}{LT}\right)^a \left(\frac{M}{L^3}\right)^b (L)^c (L)^d \left(\frac{L}{T^2}\right)^e\right]$$

For dimensional homogeneity the expont of each dimension on both sides of the eq^n must be identical.

Power of M : $0 = a + b$

L : $3 = -a - 3b + c + d + e$

T : $-1 = -a - 2e$

Since there are 5 unknowns in three eqn.

3 unknowns must be expressed in terms of other two.

$$a = -b$$

$$e = \frac{1}{2} - \frac{a}{2}$$

$$c = \frac{5}{2} - \frac{3a}{2} - d$$

$$\therefore Q = K\left[\mu^a \rho^{-a} d^{\left(\frac{5}{2} - \frac{3a}{2} - d\right)} H^d g^{\left(\frac{1}{2} - \frac{a}{2}\right)}\right]$$

$$= K\left[(d^{\frac{5}{2}} g^{\frac{1}{2}})\left(\mu^a \rho^{-a} d^{\frac{-3a}{2}} g^{\frac{-a}{2}}\right)\left(H^d d^{-d}\right)\right]$$

$$= K\left[(d^2 d^{\frac{1}{2}} g^{\frac{1}{2}})\left(\frac{\mu}{\rho d^{3/2} g^{\frac{1}{2}}}\right)^a \left(\frac{H}{d}\right)^d\right]$$

$$= K\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right)^a \left(\frac{H}{d}\right)^{d-\frac{1}{2}} d^2 H^{1/2} g^{1/2}$$

$$= \frac{K}{\sqrt{2}}\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right)^a \left(\frac{H}{d}\right)^{d-\frac{1}{2}} d^2\sqrt{2gH}$$

$$= (a\sqrt{2gH})\, f_1\left[\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right), \left(\frac{H}{d}\right)\right]$$

$$Q = C_d\, a\sqrt{2gH}$$

C_d = coefficient of discharge

$$= f_1\left[\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right), \left(\frac{H}{d}\right)\right]$$

& both term are dimension-less

$C_d \rightarrow$ is also dimension less.

Power developed by a pump

P depend on

Head H

Discharge Q

Speed ω

$P = K H Q \omega$

Buckingham π - Method :-

The Rayleigh's method of dimensional analysis becomes more laborious if the variables are more than the number of fundamental dimensions (M, L, T). This difficulty is overcame by using Buckingham's π - theorem.

It states that " If there are 'n' variables (independent and dependent variables) in a physical phenomenon and if these variables contain 'm' fundamental dimensions (M, L, T) then the variables are arranged into (n-m) dimensionless terms. Each term is called π - term.

n = 7
m = 4
(n-m) = 3 - n-m dimensionless param
π₁, π₂, π₃

Let $x_1, x_2, x_3 \cdots x_n$ are the variable involved in a physical phenomena.

Let x_1 be the dependent variable and $x_2, x_3 \cdots x_n$ are the independent variables on which x_1 depends may be written as. Then the functional eqn

$$\therefore x_1 = f(x_2, x_3, \cdots x_n) \quad \text{—} (1)$$

which can be transformed to another functional relation

$$f_1(x_1, x_2, x_3 \cdots x_n) = C \quad \text{—} (2)$$

where C is a dimentionless const.

$x = f(y) = y^2 + C$
$x - y^2 = C$
$f(x, y) = C$

In accordance with the π - theorem, a non-dimensional eqn can thus be obtained in the following form.

$$f_2(\pi_1, \pi_2, \pi_3, \cdots \pi_{n-m}) = C_1 \quad \text{—} (3)$$

Each π term is dimensionless and independent of the system. Division or multiplication by a constant does not change the character of the π-term.

Each π-term contains $m+1$ variables, where m is the number of fundamental dimensions and is also called repeating variables.
(These m variables which appear repeatedly in each of the π-terms, are consequently called repeating variables.)

Let in the above case X_2, X_3 & X_4 are repeating variables if the fundamental dimension $m\ (M, L, T) = 3$.
Then each π-term is written as

$$\left.\begin{aligned}
\pi_1 &= X_2^{a_1} \cdot X_3^{b_1} \cdot X_4^{c_1}\ X_1 \\
\pi_2 &= X_2^{a_2}\ X_3^{b_2}\ X_4^{c_2}\ X_5 \\
&\cdots \\
\pi_{n-m} &= X_2^{a_{n-m}}\ X_3^{b_{n-m}}\ X_4^{c_{n-m}}\ X_n
\end{aligned}\right\} \quad -④$$

Each individual eqn is dimensionless and the exponents $a_1, b_1, c_1 \cdots$ etc. are obtained by considering dimensional homogeneity for each equation.

These values are substituted in eqn 4 & values of $\pi_1, \pi_2, \cdots \pi_{n-m}$ are obtained.

These values of π are substituted in eqn ③ to get the final equation for the phenomenon may be obtained by expressing any one of the π-terms as a function of the other as

$$\left.\begin{aligned} \pi_1 &= \phi_1[\pi_2, \pi_3, \pi_4 \cdots \pi_{n-m}] \\ \pi_2 &= \phi_2[\pi_1, \pi_3, \pi_4 \cdots \pi_{n-m}] \end{aligned}\right\} \text{—(5)}$$

—o—

Now we will solve the same problem (Flow of fluid through a small orifice discharging freely into the atmosphere) as discussed under Rayleigh's method) the following steps may be adopted to solve it by Buckingham's π-method.

Step 1 :- The physical quantity involved in the phenomenon are Q, d, H, g, μ & ρ. The functional equation for the discharge Q may be expressed as

$$Q = f(d, H, g, \mu, \rho)$$

which in most general form can be written as

$$f_1(Q, d, H, g, \mu, \rho) = C$$

Total no. of variables $n = 6$

These variables may be completely described by either the M-L-T or F-L-T the three fundamental dimensions of systems. Here $m = 3$ (no. of fundamental dimensions)

$\therefore (n-m) = 3$ dimensionless π-terms.

$$f_2(\pi_1, \pi_2, \pi_3) = C_1$$

Step 2. Each π-term contains $(m+1)$ variable.

Since here $m = 3$ (no. of fundamental dimension)

So we have to choose 3 repeating variables. These variable should be such that they, among them contain all the three fundamental dimensions and they themselves do not form a dimensionless parameter.

Thus let us choose, $\rho\left(\frac{M}{L^3}\right)$, $d(L)$ & $g(L/T^2)$ as the repeating variables, since the above noted requirements are fulfilled by these.

Step 3. Each π-term is written as

$$\left.\begin{aligned} \pi_1 &= \rho^{a_1} d^{b_1} g^{c_1} Q \\ \pi_2 &= \rho^{a_2} d^{b_2} g^{c_2} \mu \\ \pi_3 &= \rho^{a_3} d^{b_3} g^{c_3} H \end{aligned}\right\}$$

Step 4. Expressing π_1 dimensionally in terms of M-L-T system.

$$\pi_1 = M^0 L^0 T^0 = \left(\frac{M}{L^3}\right)^{a_1} (L)^{b_1} \left(\frac{L}{T^2}\right)^{c_1} \left(\frac{L^3}{T}\right)$$

Equating the exponents of M, L & T, we get

For M : $0 = a_1$

For L : $0 = -3a_1 + b_1 + c_1 + 3$

For T : $0 = -2c_1 - 1$

From these eqn we get

$$a_1 = 0, \quad b_1 = -\frac{5}{2} \quad c_1 = -\frac{1}{2}.$$

Hence $\pi_1 = \left(\frac{Q}{d^{5/2} g^{1/2}}\right)$

Scanned by CamScanner

Similarly, we have

$$\pi_2 = \rho^{a_2} d^{b_2} g^{c_2} \mu$$

$$M^0 L^0 T^0 = \left(\frac{M}{L^3}\right)^{a_2} (L)^{b_2} \left(\frac{L}{T^2}\right)^{c_2} \left(\frac{M}{LT}\right)$$

Equating the exponents of M, L and T, we get

For M : $0 = a_2 + 1$

For L : $0 = -3a_2 + b_2 + c_2 - 1$

For T : $0 = -2c_2 - 1$

For which $a_2 = -1$ $b_2 = -\frac{3}{2}$ $c_2 = -\frac{1}{2}$

Hence $\pi_2 = \left(\frac{\mu}{\rho g^{\frac{1}{2}} d^{3/2}}\right)$

Lastly, we have

$$\pi_3 = \rho^{a_3} d^{b_3} g^{c_3} H$$

$$M^0 L^0 T^0 = \left(\frac{M}{L^3}\right)^{a_3} (L)^{b_3} \left(\frac{L}{T^2}\right)^{c_3} (L)$$

Equating the exponents of M, L and T. we get

For M : $0 = a_3$

For L : $0 = -3a_3 + b_3 + c_3 + 1$

For T : $0 = -2c_3$

For which $a_3 = 0$ $b_3 = -1$ $c_3 = 0$

Hence $\pi_3 = \left(\frac{H}{d}\right)$

Step 5 since $f_2(\pi_1, \pi_2, \pi_3) = C_1$

By substitution

$$f_2\left(\frac{Q}{g^{\frac{1}{2}} d^{\frac{5}{2}}}, \frac{\mu}{\rho g^{\frac{1}{2}} d^{3/2}}, \frac{H}{d}\right) = C_1$$

or, $$\frac{Q}{g^{\frac{1}{2}} d^{\frac{5}{2}}} = c_2 f_3\left(\frac{H}{\rho g^{\frac{1}{2}} d^{3/2}}, \frac{H}{d}\right)$$

From this we can ~~get~~ write the expression in its usual form.

$$Q = C_d a\sqrt{2gH} \; f_3\left[\left(\frac{H}{\rho g^{\frac{1}{2}} d^{3/2}}\right), \left(\frac{H}{d}\right)\right]$$

~~where C_d~~

or $$\boxed{Q = C_d a\sqrt{2gH}}$$

Scanned by CamScanner

MODEL ANALYSIS

The engineers associated with design, construction and efficient working of various types of hydraulic structure or hydraulic m/c (dam, spillways) (pump, turbine etc) usually try to findout in advance, how the structure or m/c would behave when it is actually constructed.

For this purpose they have to depend on the experimental results. Such experiments are also necessary ~~where~~ in case of problem which can not be solved completely by simply by theoretical analysis.

Obviously the experiments can not be carried out on the full size hydraulic structures or machines which are proposed to be ~~erected~~ erected. It is then ~~the~~ essential to ~~test are~~ construct ~~on the~~ a small scale replica of the m/c or structure and the test are preformed on it to obtain the desired information.

<u>Model</u> : → The small scale replica of the actual structure or the machine.

<u>Model Analysis</u> → Analysis of model to get some information about actual m/c.

<u>Prototype</u> : → Actual structure or m/c is called prototype.

Modeal test are quite economical and convenient because the design, construction and operation of the model may be alter several times if necessary till all the defects ~~are~~ of model are eliminated & the model ~~of~~ suitable design is obtained.

Similitude – Types of similarities

There are in general 3 types of similarities to be established ~~to~~ complete similarity to exist between the model and its prototype. They are

(i) Geometric similarity
(ii) Kinematic similarity
(iii) Dynamic similarity

Geometric similarity :→ (similarity of shape).

Geometric similarity exist between the model and prototype if the ratios of corresponding length dimensions in model and the prototype are equal. ~~Such a ratio is~~ ~~call~~ defined as ~~scale ratio and may be written as~~

~~Length scale rat~~

Let L_m = Length of model
b_m = breadth of model
D_m = Diameter of model
A_m = Area of model
V_m = volume of model

and L_p, b_p, D_p, A_p and V_p = Corresponding value of the prototype.

For geometric similarity,

Length scale ratio $= L_r = \dfrac{L_m}{L_p} = \dfrac{b_m}{b_p} = \dfrac{D_m}{D_p}$ etc.

Area scale ratio $= A_r = \dfrac{A_m}{A_p} = \left(\dfrac{L_m \times b_m}{L_p \times b_p}\right) = L_r^2$

Volume scale ratio $= V_r = \dfrac{V_m}{V_p} = \left(\dfrac{L_m \times b_m \times D_m}{L_p \times b_p \times D_p}\right) = L_r^3$

Kinematic Similarity :- (similarity of motion)

Kinematic similarity exist betn the model and prototype if the ratios of the kinematic quantities representing the flow characteristics such as the time, velocity, acceleration and discharge must be same at all corresponding points in the model and prototype.

$$\text{Time scale ratio} = T_r = \frac{T_{m_1}}{T_{p_1}} = \frac{T_{m_2}}{T_{p_2}}$$

$$\text{velocity scale ratio} = V_r = \frac{V_{m_1}}{V_{p_1}} = \frac{V_{m_2}}{V_{p_2}}$$

$$= \frac{\frac{L_m}{T_m}}{\frac{L_p}{T_p}} = \frac{L_r}{T_r}$$

$$\text{Acc}^n \text{ scale ratio} = a_r = \frac{a_m}{a_p} = \frac{\frac{L_m}{(T_m)^2}}{\frac{L_p}{(T_p)^2}} = \frac{L_r}{T_r^2}$$

$$\text{Discharge scale ratio} = Q_r = \frac{Q_m}{Q_p} = \frac{\frac{(L_m)^3}{T_m}}{\frac{(L_p)^3}{T_p}} = \frac{L_r^3}{T_r}$$

Dynamic similarity :- (similarity of forces)

Dynamic similarity exist betn model & prototype if the ratio of corresponding forces acting at the corresponding points are equal. (direction of forces at corresponding point should ~~also~~ be same)

Let $(F_i)_p$ = Inertia forces for prototype. (at a point)

$(F_v)_p$ = viscous " " "

$(F_g)_p$ = gravity " " "

and $(F_i)_m, (F_v)_m, (F_g)_m$ = corresponding values of forces at corresponding point in model.

Then for dynamic similarity

$$\frac{(F_i)_p}{(F_i)_m} = \frac{(F_v)_p}{(F_v)_m} = \frac{(F_g)_p}{(F_g)_m} = \dots = F_r = \text{Force ratio}$$

Model Laws :-

For the dynamic similarity betn the model and prototype the ratio of the ~~forces~~ corresponding forces acting at corresponding points in model and prototype should be equal. The ratio of forces are dimensionless numbers. It ~~The ratio of forces are dimensionless number.~~ means for dynamic similarity betn model and prototype the dimensionless number should be same for model and prototype. but it is quite difficult to satisfy the condn that all the dimensionless number are the same for model & prototype. Hence models are designed on the basis of ratio of forces which are dominating in the phenomenon. The law on which the models are designed for dynamic similarity are called model laws or laws of similarity. The various model laws are

1. Reynold's model law.
2. Froude model Law
3. Euler model Law.
4. Weber model Law
5. Mach model Law.

Reynold's model law :- is the law in which Model is designed on the basis of Reynold's number. ~~number~~ ~~law~~. It sates that the Reynold number for the model must be equal to the Reynold number for the prototypes.

(i.e) $(Re)_m = (Re)_p$

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_p V_p L_p}{\mu_p}.$$

~~Scale ratio~~ for vel. ... which =>

$$\frac{\rho_p}{\rho_m} \times \frac{V_p}{V_m} \times \frac{L_p}{L_m} \times \frac{1}{\left(\frac{\mu_p}{\mu_m}\right)} = 1$$

Scale ratio for

$$\rho_r \times V_r \times L_r \times \frac{1}{\mu_r} = 1$$

Types of model

(I) UnDistorted Model

(II) Distorted model

(1) Undistorted model → if the model is geometrically similar to its prototype then the model is called undistorted model.

h → depth

Ratio of linear dimension

$$\frac{L_p}{L_m} = \frac{B_p}{B_m} = \frac{D_p}{D_m} = \frac{h_p}{h_m} \rightarrow \text{undistorted model}$$

(2) Distorted model. → $\frac{L_p}{L_m} \neq \frac{D_p}{D_m} \neq \frac{h_p}{h_m}$

(geometrically not similar)

In this case two scale ratio are taken.

(I) Horizontal scale ratio $= \frac{L_p}{L_m} = \frac{B_p}{B_m}$.

(II) Vertical scale ratio $= \frac{h_p}{h_m}$ — Linear vertical dimension.

* results from the distorted model can not be directly transfered to its prototype.

Scanned by CamScanner

Module I (Ch 2)

Derivation and its exact solution to plane and circular Couette flow, fully developed flow between infinite parallel plates, axial flow through circular pipe and annulus.

Classical Exact Solutions to N.–S. Equations

As we have previously indicated, there are very few exact solutions to the Navier–Stokes equations—it is sometimes claimed there are 87 known solutions to this system of equations that represents all possible fluid phenomena within the confines of the continuum hypothesis. In this section we will derive two of the easiest and best-known of these exact solutions. These will correspond to what is known as Couette flow and plane Poiseuille flow. In what follows we will devote a section to the treatment of each of these. Before doing this, for ease of reference, we again present the 3-D incompressible N.–S. equations.

$$u_x + v_y + w_z = 0\,, \tag{4.25a}$$

$$u_t + uu_x + vu_y + wu_z = -\frac{1}{\rho}p_x + \nu\Delta u + \frac{1}{\rho}F_{B,x}\,, \tag{4.25b}$$

$$v_t + uv_x + vv_y + wv_z = -\frac{1}{\rho}p_y + \nu\Delta v + \frac{1}{\rho}F_{B,y}\,, \tag{4.25c}$$

$$w_t + uw_x + vw_y + ww_z = -\frac{1}{\rho}p_z + \nu\Delta w + \frac{1}{\rho}F_{B,z}\,. \tag{4.25d}$$

All notation is as used previously.

Couette flow

The simplest non-trivial exact solution to the N.–S. equations is known as *Couette flow*. This is flow between to infinite parallel plates spaced a distance h apart in the y direction, as depicted in Fig. 4.9. This is a shear-driven flow with the shearing force produced only by motion of the upper plate traveling in the x direction at a constant speed U, provided we ignore all body forces. The reader should recognize this as the flow situation in which Newton's law of viscosity was introduced in Chap. 2 and recall that it is the no-slip condition of the fluid adjacent to the upper plate that leads to viscous forces ultimately producing fluid motion throughout the region between the plates.

We obtain the solution to this problem as follows. First, since U is constant and represents the only mechanism for inducing fluid motion, it is reasonable to assume that steady flow will ensue, at least after some transient response to initiation of plate motion. Furthermore, because the plates

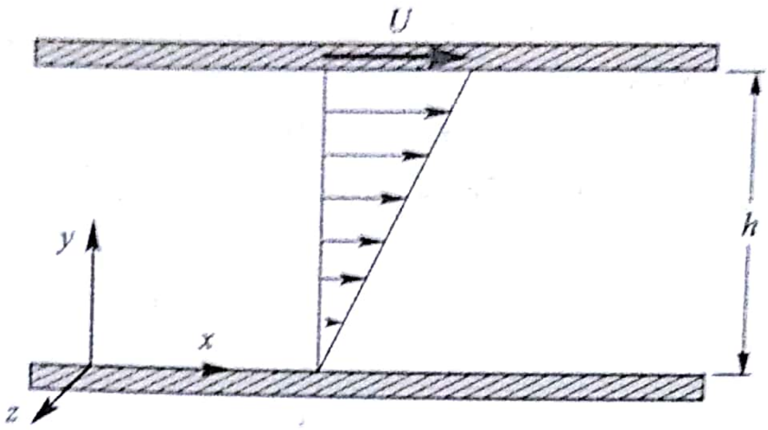


Figure 4.9: Couette flow velocity profile.

are taken to be infinite in the x and z directions there is no reason to expect x and z dependence in any flow variables since there is no way to introduce boundary conditions that can lead to such dependences. The lack of x and z dependence reduces the continuity equation to

$$v_y = 0\,,$$

as is clear from inspection of Eq. (4.25a). Thus, v is independent of y (as well as of x and z), and hence must be constant. But at the surface of either of the plates we must have $v = 0$ since fluid cannot penetrate a solid surface; then, *e.g.*, $v(0) = 0$, which implies that $v \equiv 0$.

If we now consider the y-momentum equation (4.25c) in this light we see that all that remains is

$$p_y = 0\,,$$

and again because there is no x or z dependence we conclude that $p \equiv$ const.

Next we observe that lack of x and z dependence in w, along with the constancy of pressure just demonstrated, leads to the z-momentum equation taking the simple form

$$w_{yy} = 0\,.$$

The boundary conditions appropriate for this equation are

$$w(0) = 0\,, \qquad \text{and} \qquad w(h) = 0\,,$$

both of which arise from the no-slip condition and the fact that neither plate exhibits a z-direction component of motion. Then integration (twice) of the above second-order differential equation leads to

$$w(y) = c_1 y + c_2\,,$$

and application of the boundary conditions yields $c_1 = c_2 = 0$, implying that $w \equiv 0$.

To this point we have shown that both v and w are zero, and p is identically constant. We now make use of these results, along with the steady-state and x-independence assumptions to simplify the x-momentum equation (4.25b). It is clear that this equation is now simply

$$u_{yy} = 0\,.$$

The corresponding boundary conditions arising from the no-slip condition applied at the bottom and top plates, respectively, are

$$u(0) = 0\,, \qquad \text{and} \qquad u(h) = U\,.$$

Integration of the above equation leads to a result analogous to that obtained earlier for the x-momentum equation, namely,

$$u(y) = c_1 y + c_2 ,$$

and application of the boundary conditions shows that

$$c_2 = 0 , \qquad \text{and} \qquad c_1 = \frac{U}{h} .$$

Thus, the Couette flow velocity profile takes the form

$$u(y) = \frac{U}{h} y , \tag{4.26}$$

exactly the same result we obtained from heuristic physical arguments when studying Newton's law of viscosity in Chap. 2.

We remark that despite the simplicity of this result it is very important and widely used. As we mentioned in Chap. 2 the case when h is small arises in the analysis of lubricating flows in bearings. Furthermore, this linear profile often provides a quite accurate approximation for flow near a solid boundary even in physical situations for which the complete velocity profile is far more complicated.

4.4.2 Plane Poiseuille flow

The next exact solution to the N.–S. equations we consider is *plane Poiseuille flow*. This is a pressure-driven flow in a duct over a finite length L, but of infinite extent in the z direction, as depicted in Fig. 4.10. For the flow as shown we assume $p_1 > p_2$ with p_1 and p_2 given, and that pressure is constant in the z direction at each x location. We again start with Eqs. (4.25) and

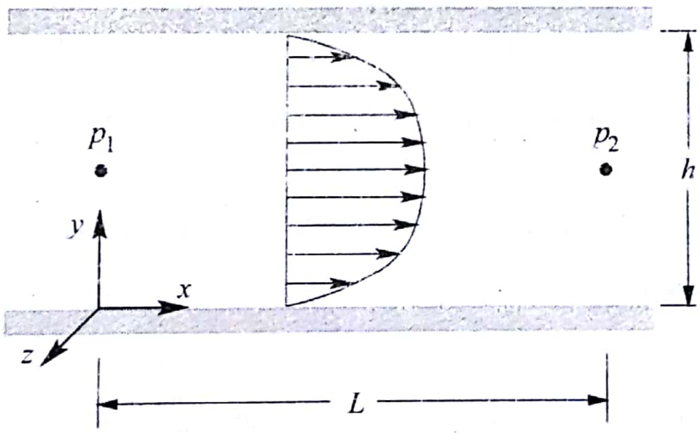


Figure 4.10: Plane Poiseuille flow velocity profile.

assume the flow to be steady, that body forces are negligible, and that velocity does not change in the x direction. It is not obvious that this last assumption should hold because pressure is changing in the x direction; but it will be apparent that it leads to no physical or mathematical inconsistencies, and without the assumption it would not be possible to obtain a simple solution as we will do. We will examine this further when we study pipe flow in the next section.

We begin with arguments analogous to those used in the Couette flow analysis. In particular, since the flow varies only in the y direction, the continuity equation collapses to $v_y = 0$ from which it follows that $v \equiv 0$ must hold. The y-momentum equation, Eq. (4.25c), becomes simply

$$p_y = 0\,,$$

which implies that the pressure p does not depend on y.

Similarly, because w is zero at both the top and bottom plates, and the z-momentum equation can again be reduced to $w_{yy} = 0$ by utilizing the preceding assumptions and results, it follows that $w \equiv 0$ also.

Now applying the steady-state assumption with $v = 0$ and $w = 0$ simplifies the x-momentum equation (4.25b) to

$$uu_x = -\frac{1}{\rho}p_x + \nu(u_{xx} + u_{yy})\,,$$

and with the assumption that the flow velocity does not vary in the x direction, we have $u_x = 0$ (and $u_{xx} = 0$). So the above becomes

$$u_{yy} = \frac{1}{\mu}p_x\,. \tag{4.27}$$

Next we note, since u is independent of x and z, that $u = u(y)$ only. This in turn implies from the form of Eq. (4.27) that p_x cannot depend on x and must therefore be constant. We can express this constant as

$$p_x = \frac{\Delta p}{L} = \frac{p_2 - p_1}{L}\,. \tag{4.28}$$

Then (4.27) becomes

$$u_{yy} = \frac{1}{\mu}\frac{\Delta p}{L}\,. \tag{4.29}$$

The boundary conditions to be provided for this equation arise from the no-slip condition on the upper and lower plates. Hence,

$$u(0) = 0\,, \qquad \text{and} \qquad u(h) = 0\,. \tag{4.30}$$

One integration of Eq. (4.29) yields

$$u_y = \frac{1}{\mu}\frac{\Delta p}{L}y + c_1\,,$$

and a second integration gives

$$u(y) = \frac{1}{2\mu}\frac{\Delta p}{L}y^2 + c_1 y + c_2\,.$$

Application of the first boundary condition in Eq. (4.30) shows that $c_2 = 0$, and from the second condition we obtain

$$0 = \frac{1}{2\mu}\frac{\Delta p}{L}h^2 + c_1 h\,,$$

which implies

$$c_1 = -\frac{1}{2\mu}\frac{\Delta p}{L}h\,.$$

Substitution of this into the above expression for $u(y)$ results in

$$u(y) = \frac{1}{2\mu}\frac{\Delta p}{L}y(y - h)\,, \tag{4.31}$$

the plane Poiseuille flow velocity profile. We observe that since the definition of Δp implies $\Delta p < 0$ always holds, it follows that $u(y) \geq 0$ as indicated in Fig. 4.10.

We can also provide further analysis of the pressure. We noted earlier that p_x could not be a function of x. But this does not imply that p, itself, is independent of x. Indeed, the fact that $p_1 \neq p_2$ requires x dependence. We can integrate Eq. (4.28) to obtain

$$p(x) = \frac{\Delta p}{L} x + C \,,$$

and from the fact that

$$p(0) = p_1 = C \,,$$

we see that

$$p(x) = \frac{p_2 - p_1}{L} x + p_1 \,. \tag{4.32}$$

This is simply a linear interpolation formula between the known pressures p_1 and p_2 over the distance L.

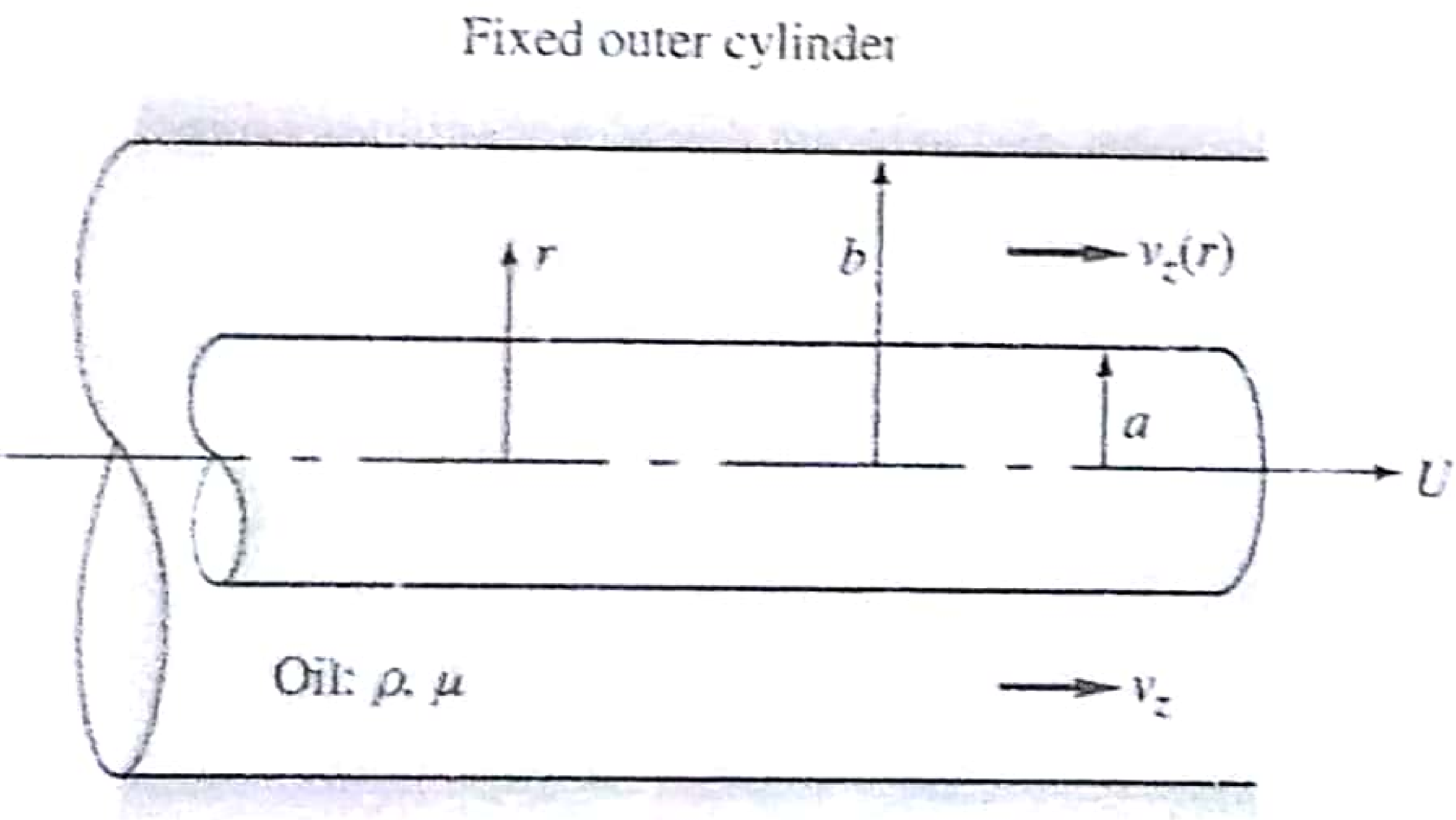
4.5 Pipe Flow

Analysis of pipe flow is one of the most important practical problems in fluid engineering, and it provides yet another opportunity to obtain an exact solution to the Navier–Stokes equations, the Hagen–Poiseuille flow. We will derive this solution in the present section of these notes. We begin with a physical description of the problem being considered and by doing this introduce some important terminology and notation, among these some basic elements of boundary-layer theory. Following this we present the formal solution of the N.–S. equations that provides the Hagen–Poiseuille velocity profile for steady, fully-developed flow in a pipe of circular cross section, and then we use this to produce simple formulas useful in engineering calculations. In particular, we will see how to account for pressure losses due to skin-friction effects, thus providing a simple modification to Bernoulli's equation that makes it applicable to viscous flow problems. Then we extend this to situations involving pipes with arbitrary cross-sectional shapes and other geometric irregularities including expansions and contractions, bends, tees, *etc.*

4.5.1 Some terminology and basic physics of pipe flow

In this subsection we consider some basic features of pipe flow that allow us to solve the N.–S. equations for a rather special, but yet quite important, case corresponding to steady, fully-developed flow in a pipe of circular cross section. We have schematically depicted this in Fig. 4.11. What is apparent from the figure is a uniform velocity profile entering at the left end of the region of pipe under consideration and then gradually evolving to a velocity profile that is much smoother and, in fact, as we will later show is parabolic in the radial coordinate r. As indicated in the figure, the distance over which this takes place is called the *entrance length*, denoted L_e, and this corresponds to the distance required for merging of regions starting at the pipe walls within which the originally uniform velocity adjusts from zero at the walls (imposed by the no-slip condition) to a free-stream velocity ultimately set by mass conservation.

4.88 The viscous oil in Fig. P4.88 is set into steady motion by a concentric inner cylinder moving axially at velocity U inside a fixed outer cylinder. Assuming constant pressure and density and a purely axial fluid motion, solve Eqs. (4.38) for the fluid velocity distribution $v_z(r)$. What are the proper boundary conditions?



4.88 The viscous oil in Fig. P4.88 is set into steady motion by a concentric inner cylinder moving axially at velocity *U* inside a fixed outer cylinder. Assuming constant pressure and density and a purely axial fluid motion, solve Eqs. (4.38) for the fluid velocity distribution $v_z(r)$. What are the proper boundary conditions?

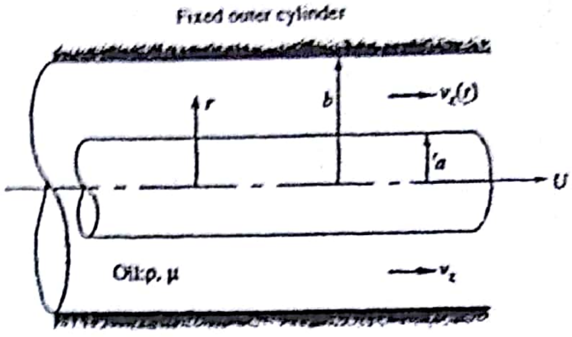


Fig. P4.88

Solution: If $v_z = fcn(r)$ only, the *z*-momentum equation (Appendix E) reduces to:

$$\rho\frac{dv_z}{dt} = -\frac{\partial p}{\partial z} + \rho g_z + \mu\nabla^2 v_z, \quad \text{or:} \quad 0 = 0 + 0 + \frac{\mu}{r}\frac{d}{dr}\left(r\frac{dv_z}{dr}\right)$$

The solution is $v_z = C_1\ln(r) + C_2$, subject to $v_z(a) = U$ and $v_z(b) = 0$

Solve for $C_1 = U/\ln(a/b)$ and $C_2 = -C_1\ln(b)$

The final solution is: $$v_z = U\frac{\ln(r/b)}{\ln(a/b)}$$ *Ans.*

4.89 Modify Prob. 4.88 so that the outer cylinder also moves to the *left* at constant speed *V*. Find the velocity distribution $v_z(r)$. For what ratio *V/U* will the wall shear stress be the same at both cylinder surfaces?

Solution: We merely modify the boundary conditions for the known solution in 4.88:

$v_z = C_1\ln(r) + C_2$, subject to $v_z(a) = U$ and $v_z(b) = -V$

Solve for $C_1 = (U + V)/\ln(a/b)$ and $C_2 = U - (U + V)\ln(a)/\ln(a/b)$

The final solution is $$v_z = U + (U+V)\frac{\ln(r/a)}{\ln(a/b)}$$ *Ans.*

The shear stress $\tau = \mu(U + V)/[r\ln(a/b)]$ and is <u>never equal at both walls for any ratio of V/U</u> unless the clearance is vanishingly small, that is, unless $a \approx b$. *Ans.*

Module II (chp. 3)

Boundary Layer

1904
Ludwig Prandtl
German scientist.

①

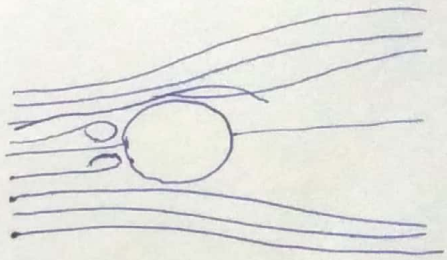
- External flows.

In potential flow → which assume an ideal fluid without viscosity, only pressure and inertia forces determine the flow dynamics.

ideal flow

Real fluid do have viscosity, and the flow field can be very different.

Potential flow streamline about a thin plate attached to a cylinder.
(fig 1)



water flowing past the object in fig 1.
(Hydrogen bubble visualization)

Boundary layers, thin layers of fluid in which viscosity effects are significant, are formed along solid boundaries. In some cases these boundary layers under the influence of pressure gradients significantly affect the entire flow field.

* When a real fluid flows past a solid body or a solid wall, the fluid particle adhere to the boundary and condition of no slip occurs.

* [The velocity of the fluid close to the boundary will be same as that of the boundary. If the boundary is stationary, the velocity of the fluid at the surface/boundary will be zero. This is called the no-slip boundary condition of viscous flow.]

* Further away from the boundary, the velocity will be higher and as a result of this variation of velocity, the velocity gradient $\frac{du}{dy}$ will exist.

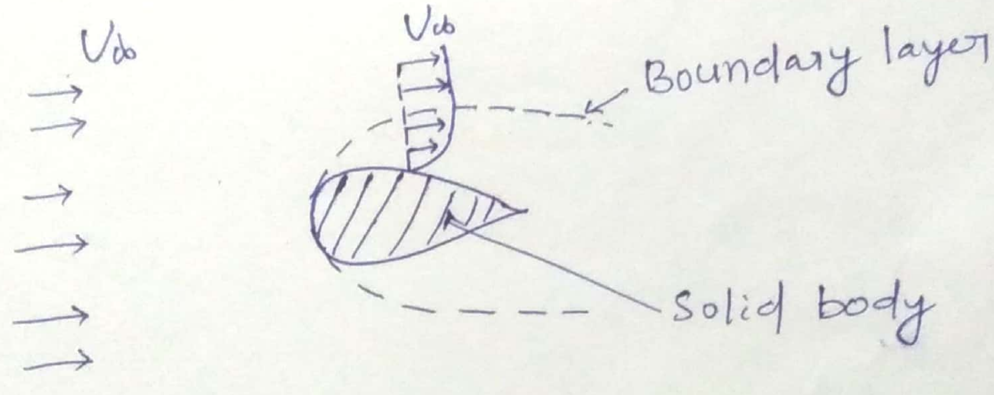
* The velocity of fluid increases from zero velocity on the stationary boundary to the free-stream velocity (V_∞) of the fluid in the direction normal to the boundary.

* This variation of velocity from zero to free-stream velocity in the direction normal to the boundary takes place in a narrow region in the vicinity of the solid boundary. This narrow region of fluid is called boundary layer.

* The thickness of the boundary layer increases along the length of the solid body.

* The theory dealing with boundary layer flow is called boundary layer theory.

* According to the boundary layer theory, the flow of fluid in the neighbourhood of the solid boundary may be divided into two regions as shown in figure

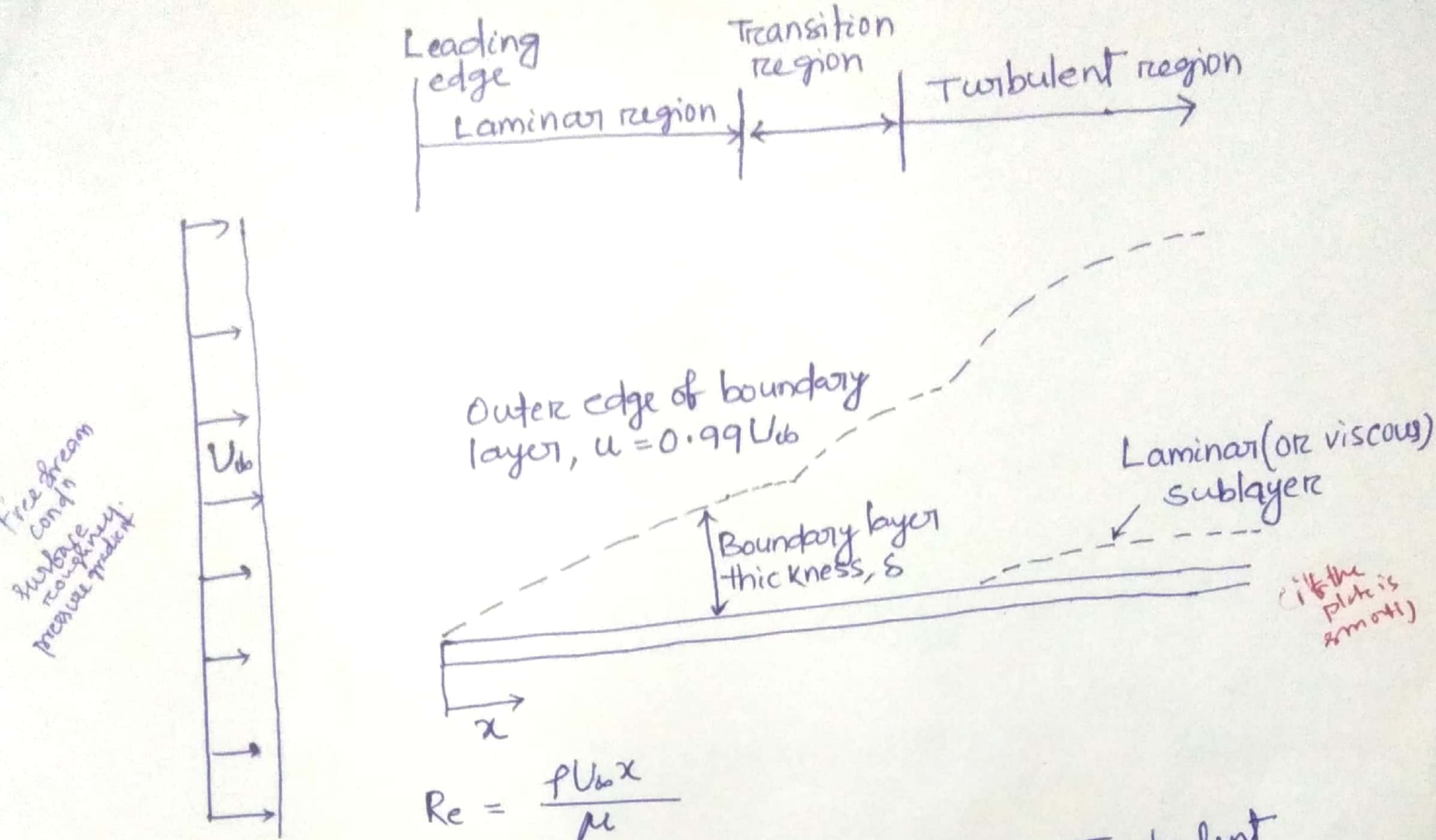


(flow over a solid body)
Fig 1

(1) A very thin layer of fluid, called the boundary layer, in the immediate neighbourhood of the solid boundary, where the variation of velocity from zero at the solid boundary to free-stream velocity in the direction normal to the boundary takes place. In this region velocity gradient $\frac{du}{dy}$ exists and hence the fluid exerts a shear stress on the wall in the direction of motion.

$$\tau = \mu \frac{du}{dy}$$

Description of the boundary layer



$$Re = \frac{\rho U_\infty x}{\mu}$$

$V_\infty \rightarrow$ Free stream velocity

$\tau = \mu \frac{du}{dy}$

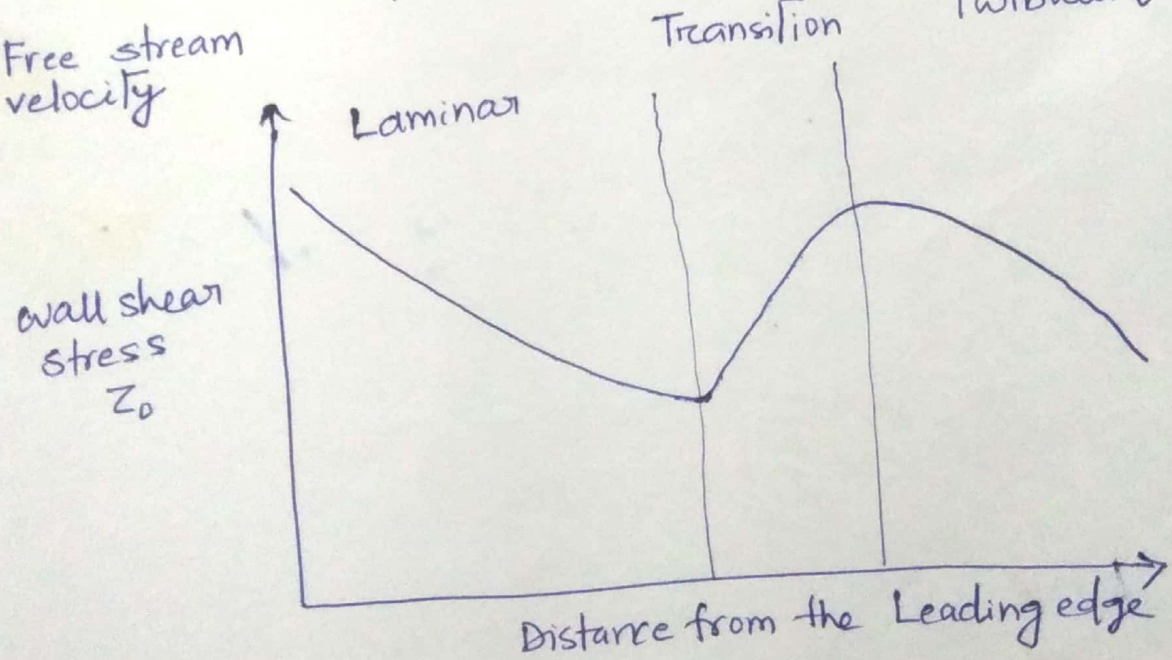


Fig 2

(Development of the boundary layer along a flat plate, illustrating variation in layer thickness and wall shear stress)

* The boundary layer is taken as that region of fluid close to the surface immersed in the flowing fluid.

* figure shows a flat plate in a free fluid stream.

* Only the top surface boundary layer is shown but there will, in practice, by symmetry between the upper and lower surface boundary layers, provided both the surface are identical in nature.

* The fluid in contact with the plate surface has zero velocity, 'no-slip'. and the velocity gradient exist between the fluid in free-stream and the plate surface.

* shear stress may be defined as

$$\tau = \mu \frac{\partial u}{\partial y} \quad -①$$

$\tau \to$ shear stress, $\mu \to$ fluid viscosity

and $\frac{du}{dy} \to$ the velocity gradient.

* This shear stress acting at the flat surface sets up a shear force which opposes the fluid motion, and fluid close to the wall is decelerated.

Factors affecting Transition from Laminar to Turbulent flow regims

(*) The transition from laminar to turbulent boundary layer conditions may be considered as Reynolds number dependent.

$$Re = \frac{\rho U_\infty x}{\mu}$$

$Re = 5 \times 10^5$ ← quoted figure

(*) If the surface is rough → Reynold number considerably reduced

For $Re < 10^5$, the laminar layer is stable

However Re near 2×10^5 it is difficult to prevent Transition.

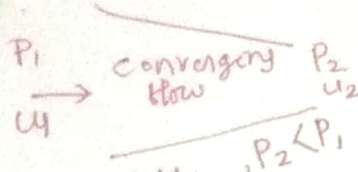
(*) The presence of pressure gradient $\frac{dp}{dx}$ can also be a major factor.

divergent flow.

fluid decelerate, thicken the b.l.

If $\frac{dp}{dx} > 0$ (positive) the transition Reynolds number is reduced.

→ If $\frac{dp}{dx} < 0$ (negative)



transition Reynolds number increases

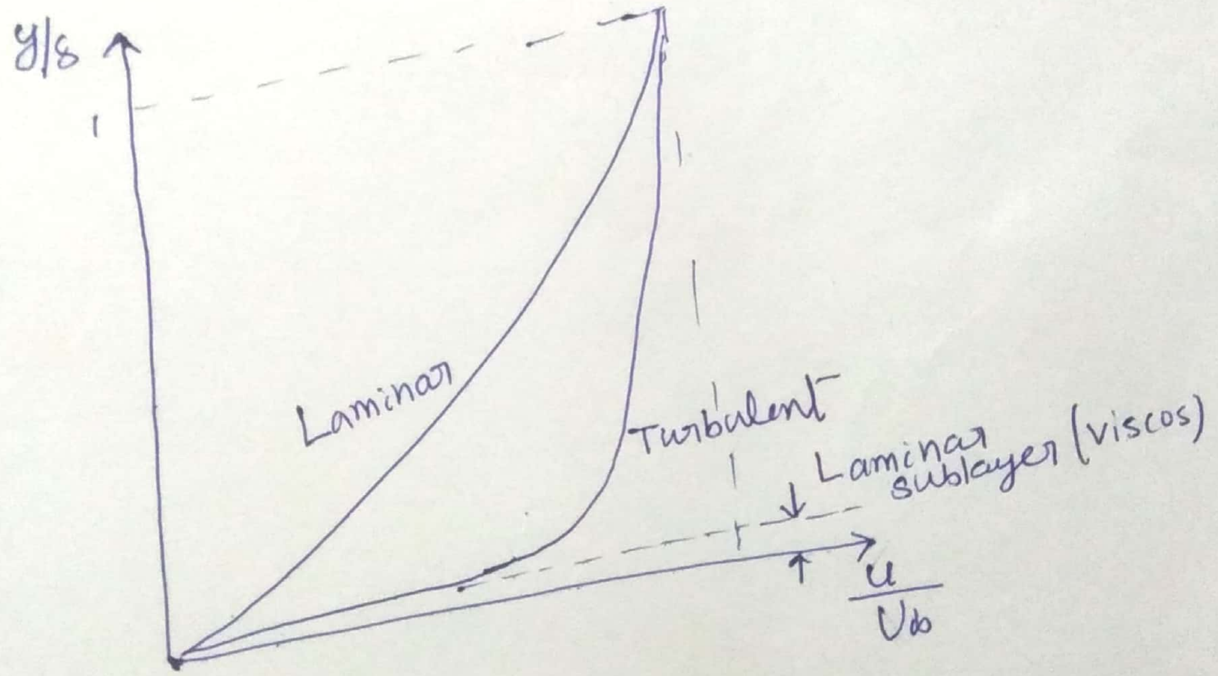
$u_2 > u_1$, $P_2 < P_1$

accelerate the flow
b.l. growth retarded.

(This effect forms the basis of suction high-lift device designed for aircraft wing)

$\frac{\partial p}{\partial x} = 0$ (in flat plate)

Velocity profiles in laminar and turbulent boundary layer regions



* The growth of the boundary layer thickness is more rapid in the turbulent region ~~th~~ (roughly varying $x^{0.8}$ compared to $x^{0.5}$ in the laminar region)

→ ⊙ The velocity profile across the laminar sublayer is assumed linear and tangential to the velocity profile up through the turbulent boundary layer.

Boundary layer Thickness (δ)

It is defined as the distance from the boundary of the solid body measured in the y-direction to the point, where the velocity of the fluid is approximately equal to 0.99 times the free stream velocity (V_∞) of the fluid.

$$\delta = y(u = 0.99\, V_\infty)$$

⇒ This defination of boundary layer thicknes is some what arbitrary, a physically more meaningful measure of the boundary layer estimation is expressed through displacement thickness (δ^*), momentum thickness (θ) and energy thickness (δ^{**})

⇒ Boundary layer thickness in terms of the effect of the flow.

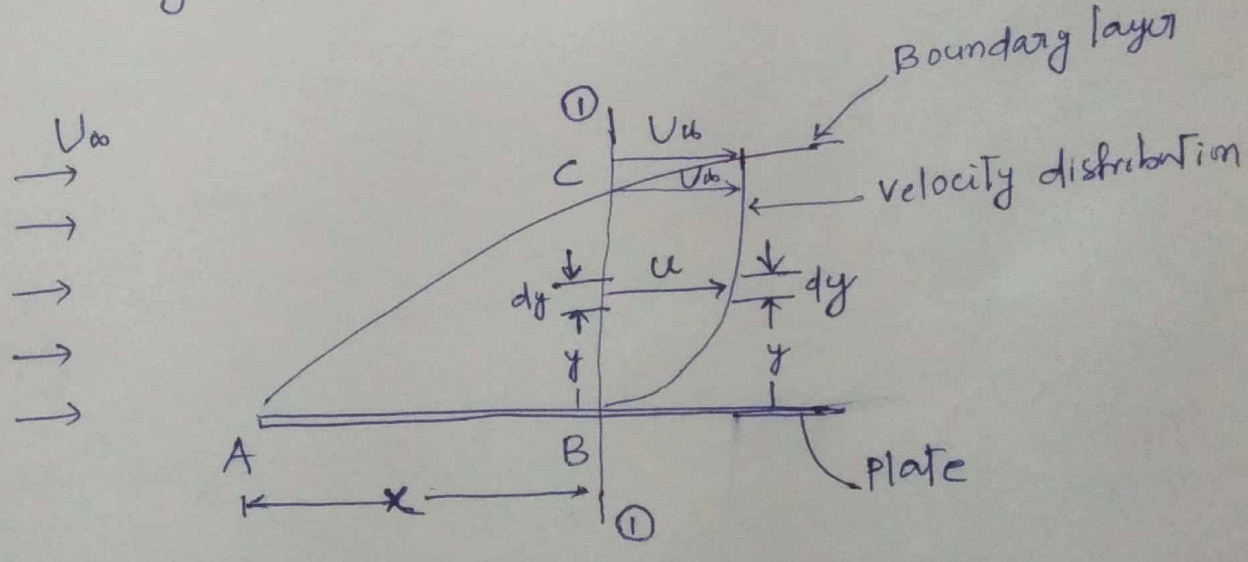
Displacement thickness (δ^*)

It is defined as the distance, measured perpendicular to the boundary of the solid body, by which the boundary should be displaced to compensate for the reduction in the mass flow rate on account of the boundary layer formation.

(mass flux)

rate of mass flow per unit area.

(The distance the surface ~~should~~ would have to move in the y-direction to reduce the flow ~~rate~~ passing by a volume equivalent to the real effect of the boundary layer)



(Displacement thickness)

Momentum Thickness (θ)

Momentum thickness is defined as the distance, measured perpendicular to the boundary of the solid body, by which the boundary should be displaced to compensate for the reduction in momentum of the flowing fluid on account of boundary layer formation.

Energy Thickness (δ^{**})

It is defined as the distance measured perpendicular to the boundary of the solid body, by which the boundary should be displaced to compensate for the reduction in kinetic energy of the flowing fluid on account of the boundary layer formation.

Kinetic energy of the fluid $= \frac{1}{2} m \times velocity^2$

$$= \frac{1}{2} (\rho u b dy) u^2$$

Kinetic energy of this fluid in the absence of the boundary layer $= \frac{1}{2} (\rho u b dy) U_\infty^2$

Loss of K.E through strip

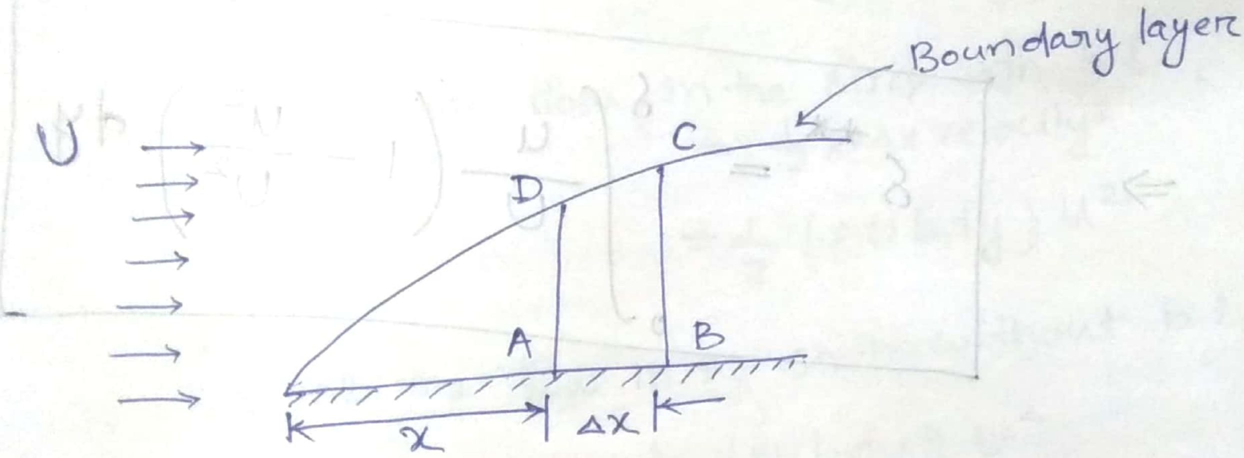
$$= \frac{1}{2} (\rho u b dy) U_\infty^2 - \left(\frac{1}{2} (\rho u b dy) u^2\right)$$

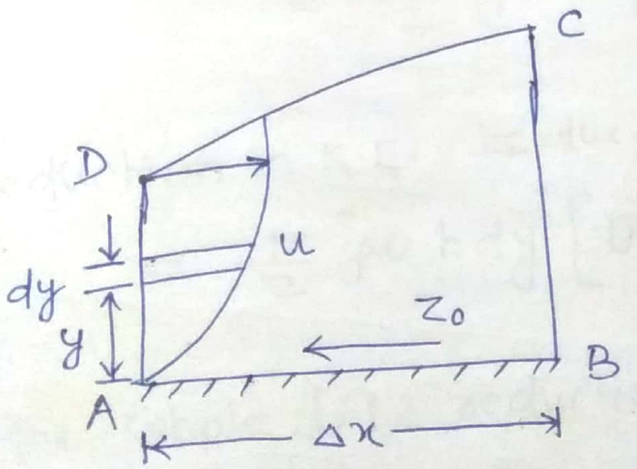
$$= \frac{1}{2} \rho u b \left[U_\infty^2 - u^2 \right] dy$$

Momentum integral Equation of Boundary layer:

Von Karman integral equation

Momentum integral equation of boundary layer represent the relation that must exist between rate of flux of momentum of boundary layer, the shear stress develop at the boundary surface and the pressure gradient in the direction of flow.





(Enlarge view of ABCD

Consider the flow of fluid having free stream velocity equal to U over a thin plate.

Consider an element Δx of the boundary layer formed along a flat plate at a distance x from the leading edge. (section AD and BC) shear stress $\tau_0 = \mu \left(\frac{du}{dy}\right)_{y=0}$

External force in the direction of rate of change of momentum $= -\tau_0 \times \Delta x \times b$

According to momentum principle.

$$-\tau_0 \times \Delta x \times b = \rho b \frac{\partial}{\partial x}\left[\int_0^\delta (u^2 - uU)\,dy\right] \times \Delta x$$

$$\Rightarrow \quad \tau_0 = -\rho \frac{\partial}{\partial x}\left[\int_0^\delta (u^2 - uU)\,dy\right]$$

$$\Rightarrow \quad \boxed{\tau_0 = \rho \frac{\partial}{\partial x}\left[\int_0^\delta (uU - u^2)\,dy\right]}$$

$$= \rho U^2 \frac{\partial}{\partial x}\left[\int_0^\delta \left(\frac{u}{U} - \frac{u^2}{U^2}\right) dy\right]$$

$$= \rho U^2 \frac{\partial}{\partial x}\left[\int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right) dy\right]$$

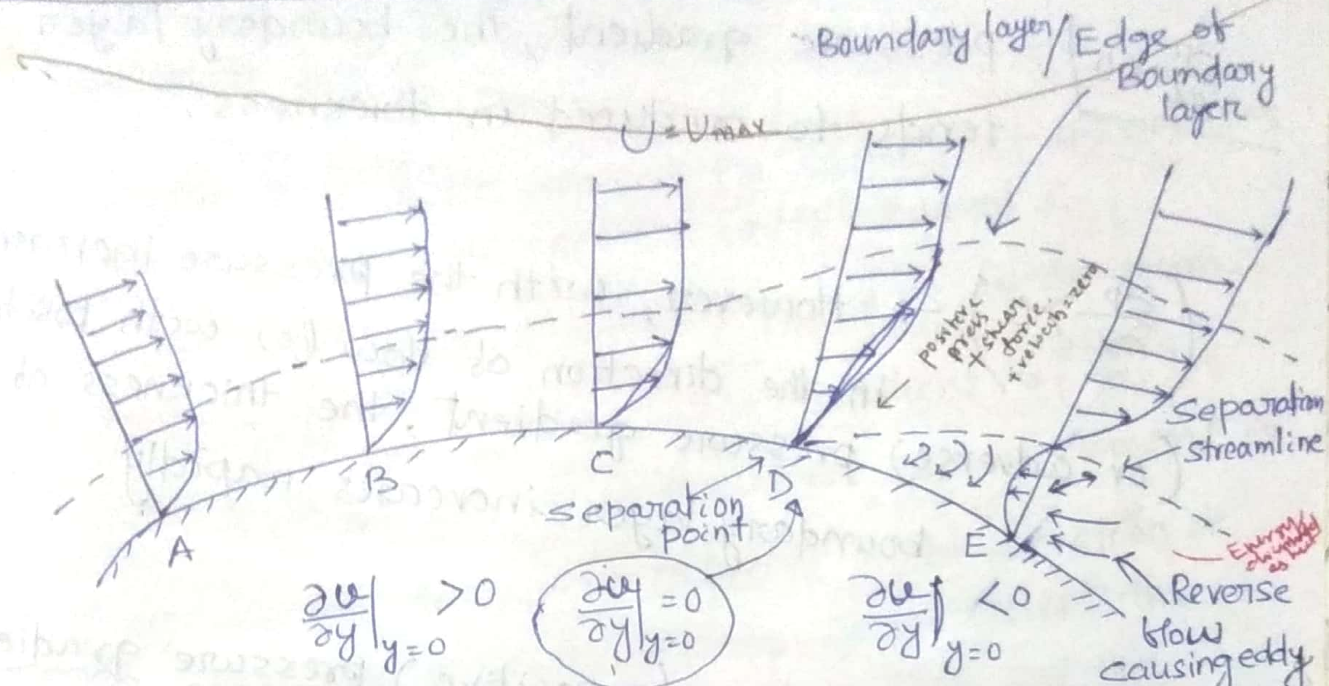
$$\Rightarrow \quad \boxed{\frac{\tau_0}{\rho U^2} = \frac{\partial \theta}{\partial x}}$$

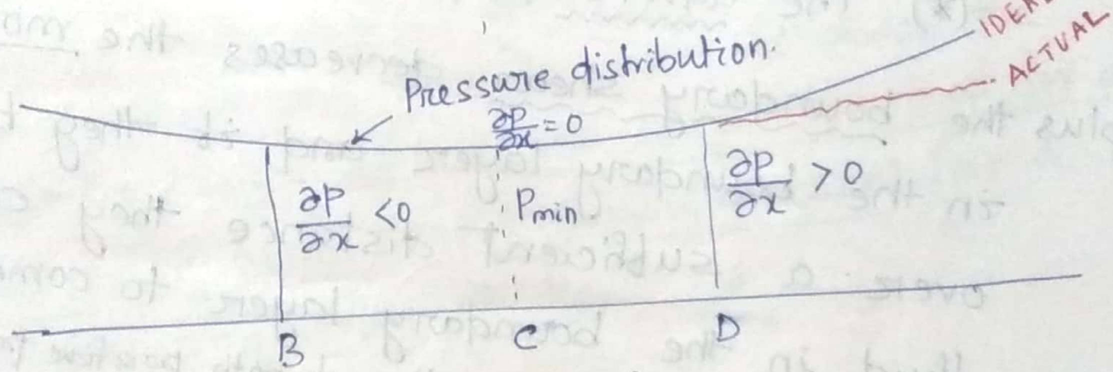
$\theta \rightarrow$ momentum thickness

This eqn is known as Von Karman momentum integral eqn for boundary layer flow.

Scanned by CamScanner

Separation of Boundary Layer





Pressure distribution along the boundary

* The boundary layer thickness is considerably affected by the pressure gradient in the direction of flow.

$\left(\frac{\partial p}{\partial x} = 0\right) \rightarrow$ If the pressure gradient is zero, the boundary layer continues to grow in thickness along a flat plate.

$\left(\frac{\partial p}{\partial x} < 0\right) \rightarrow$ With the decreasing pressure in the direction of flow (ie) with negative

$\frac{\partial p}{\partial x} < 0$ pressure gradient, the boundary layer tends to reduced in thickness.

$\left(\frac{\partial p}{\partial x} > 0\right) \rightarrow$ However, with the pressure increasing in the direction of flow (i.e.) with positive (or adverse) pressure gradient, the thickness of the boundary layer increases rapidly.

(*) The adverse (or positive) pressure gradient plus the boundary shear decreases the momentum in the boundary layer and if they both act over a sufficient distance they cause the fluid in the boundary layer to come to rest (i.e) (By the action of both positive pressure gradient and shear force the fluid on the b.l. comes to stand still)

the retarded fluid particle, cannot move too far into the region of increased pressure due to small kinetic energy.

Thus the boundary layer is deflected sideways from the boundary, separate from it and moves into the main stream.

This phenomena is called separation.

Scanned by CamScanner

* Consider flow of fluid over a curved surface as shown in figure.

* As the fluid flows round the surface it accelerates over the left hand section. until at point C.
* At this section the (point C) pressure is minimum.

upto C

* From A to C the pressure gradient is $\left(\frac{\partial p}{\partial x}\right)$ is negative (the net pressure force on an element of fluid in the boundary layer is in the direction of flow — which counteracts to some extent the slowing down effect of the boundary on the flowing fluid and thus the rate at which the boundary layer thickens is less than that for a flat plate with zero pressure gradient (at a corresponding value of Re_x)

rate of growth of b.l.

(*) Beyond C the pressure increases, hence the net pressure force on the element of the fluid in the boundary layer opposes the forward flow. Thus at a certain distance on the downstream of point C, the fluid near the boundary surface soon brought to stand-still.

* The value of the velocity gradient $\left(\frac{\partial u}{\partial y}\right)$ at the boundary surface is then zero as at point D.
* The fluid is no longer able to follow the contour of the curved surface and it separates from it.

Scanned by CamScanner

(*) The separation of the flowing fluid from the boundary first occurs at a point where $\left(\frac{\partial u}{\partial y}\right)$ at the boundary becomes zero and this point is known as separation point (D)

* On the downstream of the separation point 'D' a further retardation of the fluid close to the boundary can even have a reverse or a back flow as at point E, shown in figure.

* If all the point below which a reverse flow occurs are joined by a smooth curve, a line dividing the forward and reverse flow obtained which is known as separation streamline.

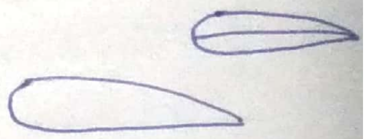
* In a region in between the boundary surface and separation streamline, as a result of reverse flow, large irregular eddies are formed in which much energy is dissipated as heat.

(*) This region of disturbed fluid usually extends for some distance on the downstream. Since the energy of the eddies is dissipated as heat the pressure downstream ~~seem~~ remains approximately the same as the separation point.

Separation of the boundary layer gives rise to additional resistance to the flow so it should be avoided by some means.

→ one of the method is developing such boundary shape for which separation is as far as possible eliminated

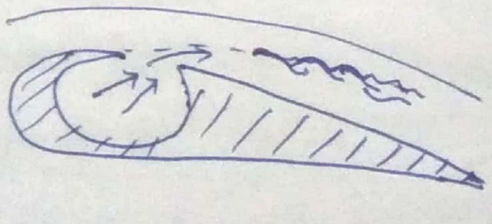
*example :- airfoil



Methods of preventing the separation of boundary layer / method of controlling b.l.

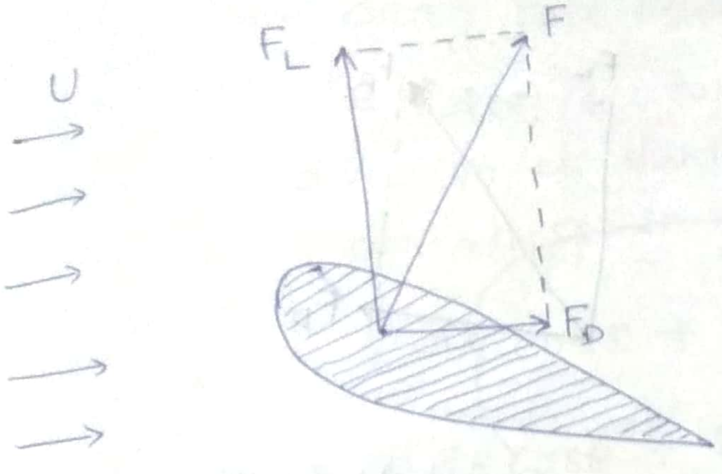
1. Rotating boundary in the direction of flow (flow past a rotating cylinder)

2. Injecting fluid into the boundary layer.



Module II (Ch 4)

Drag & Lift : Fluid flow around submerged objects



Forces on an immersed body

The force exerted by the fluid on the moving body may in general be inclined to the direction of motion.

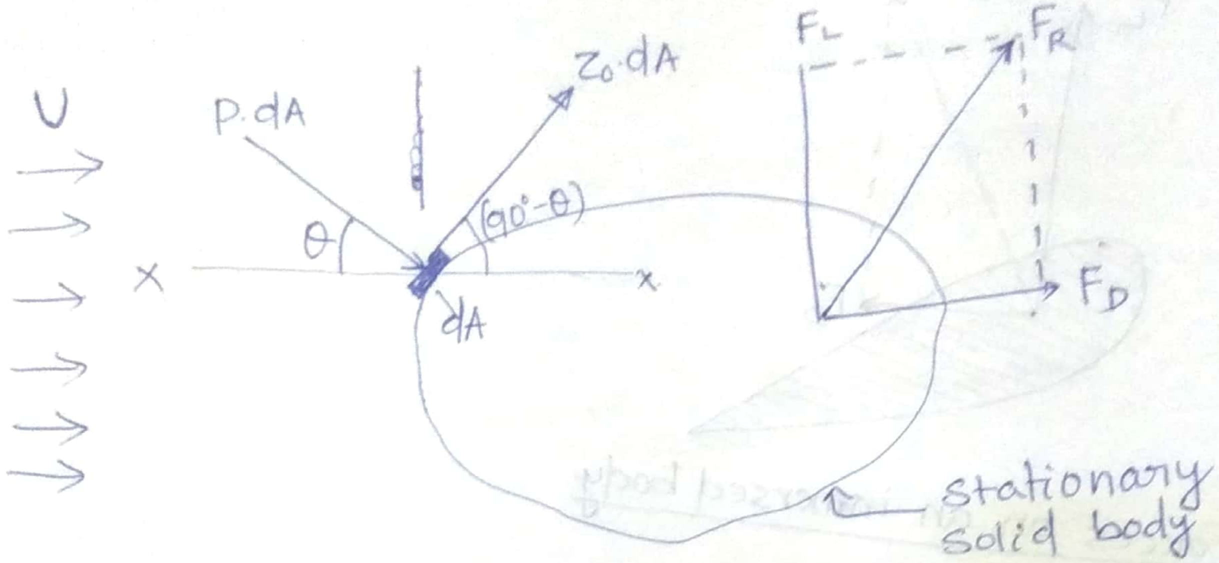
Hence the force (resultant force) has a component in the direction of motion as well as one perpendicular to the direction of motion.

The component of this force in the direction of motion is called drag (F_d).

and the component perpendicular to the direction of motion is called lift (F_L)

For symmetrical body, such as sphere or a cylinder, facing the flow symmetrically, there is no lift and the total force exerted by the fluid is equal to the drag on the body.

Expression for drag and Lift



Consider an arbitrary shaped body held stationary in a stream of real fluid moving at a uniform velocity U.

Consider a small elemental area dA on the surface of the body.

The forces acting on the surface area dA are.

(i) pressure force equal to $\underline{P \cdot dA}$, acting perpendicular to the surface area (dA)

(ii) shear force equal to $\underline{\tau_0 \cdot dA}$, acting parallel to the surface area (dA).
[along tangential direction to the surface].

Let θ be the angle made by pressure force with horizontal direction.

(a) Drag Force (F_D) :

The drag force on the elemental area

= Force due to ~~the~~ pressure in the direction of fluid motion + Force due to shear stress in the direction of fluid motion

$$= p\,dA\cos\theta + \tau_0\,dA\cos(90^\circ-\theta)$$

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

∴ For the whole arbitrary body, total drag

$$F_D = \int p\,dA\cos\theta + \int \tau_0\,dA\sin\theta \quad \text{—} \; (1)$$

$$= \int p\cos\theta\,dA + \int \tau_0\sin\theta\,dA$$

$\int p \cos\theta\, dA$ is called pressure drag or form drag

$\int \tau_0 \sin\theta\, dA$ is called shear drag or skin drag or friction drag.

(b) Lift Force (F_L)

The lift force on the elemental area.

The lift force in the direction

= Force due to the pressure in the direction perpendicular to the direction of motion + Force due to the shear stress in the direction perpendicular to the direction of motion

$$= -p\,dA\sin\theta + \tau_0\,dA\sin(90^\circ-\theta)$$

$$= -p\,dA\sin\theta + \tau_0\,dA\cos\theta$$

(-ve ~~sing~~ sign taken pressure is acting downward while shear stress is acting upward)

∴ Total lift acting on whole arbitrary body

$$F_L = \int \tau_0\, dA\, \cancel{\sin\theta}\cos\theta - \int p\, dA\, \sin\theta$$

$$= \int \tau_0 \cos\theta\, dA - \int p \sin\theta\, dA$$

Mathematical equation for drag and lift

Total drag $F_D = C_D \frac{1}{2} \rho A U^2$

A = projected area of the body $\perp^r$ to the flowing fluid.

ρ = density of the fluid

C_D = co-efficient of drag.

Lift $F_L = C_L \frac{1}{2} \rho A U^2$

C_L = co-efficient of lift

The resultant force on the body

$$F_R = \sqrt{F_D^2 + F_L^2}$$

Pressure drag and Friction drag :-

The total drag on a body is given by equation

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

where $\int p\cos\theta\, dA$ = Pressure drag or form drag

$\int \tau_0 \sin\theta\, dA$ = Friction drag or skin drag or shear drag

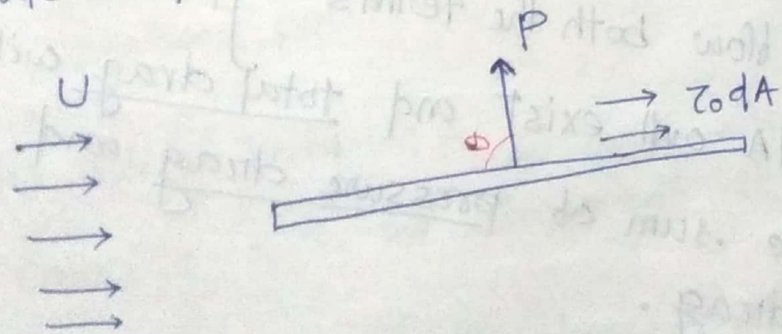
The contribution of pressure drag and friction drag depends on

(i) shape of the immersed body

(ii) position of the body immersed in the fluid

(iii) characteristic of the fluid

(*) Consider the flow of fluid over a flat plate, when the plate is placed parallel to the direction of flow



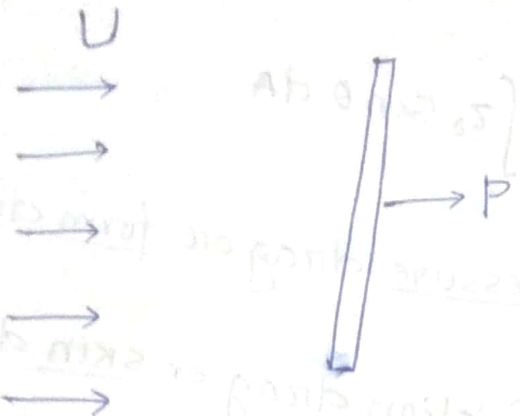
θ = Pressure force with horizontal direction.

$\cos\theta = \cos 90^\circ = 0$

$$F_D = \underbrace{\cancel{\int p\cos\theta\, dA}}_{0} + \int \tau_0 \sin\theta\, dA$$

$\therefore$ Total drag = friction drag (shear drag)

If the plate is placed perpendicular to the flow



θ = pressure force with horizontal direction.

$\theta = 0^\circ$

$\sin\theta = \sin 0 = 0$

$$F_D = \int p \cos\theta \, dA + \int \tau_0 \cancel{\sin\theta} \, dA$$

0

Total drag will be due to the pressure difference between the upstream and downstream side of the plate

* If the plate is held at an angle with the direction of flow both the terms $\int p\cos\theta\, dA$ and $\int \tau_0 \sin\theta\, dA$ will exist and total drag will be equal to the sum of pressure drag and friction drag.

(*) shape of the body.

(i) <u>stream-lined body</u> :-

A body whose ~~surface~~ shape coincide with the stream lines is called streamlined body

example :- Aeroplane

No wake formation zone

Incase of streamline body the separation of boundary layer takes place at the end of the body. (trailing edge)

so the pressure force is neglected.

For streamline body the force acting is due to friction drag (skin drag) only.

A body may be stream-lined

(i) at low velocities but may not be so at higher velocities.

(ii) when placed in a particular position in the flow but may not be so when placed in another position.

(ii) <u>Bluff body</u> :- A body whose shape does not coincide with the streamline is called bluff body

Large wake formation zone.

In this case the force acting is due to pressure drag only. (Friction drag is negligible) (pressure drag is very large compared to friction drag)

Scanned by CamScanner

Drag on a sphere

G.G. Stokes, developed a mathematical equation for the total drag on a sphere ~~immersed~~ immersed in a flowing fluid (For which Renolds number is upto 0.2, so that inertia forces may be assumed negligible. (viscous forces are much more predominate)) $\left[Re = \frac{UD\rho}{\mu} < 0.2 \right]$

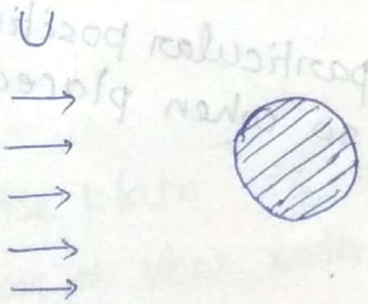
The total drag is given by

$$F_D = 3\pi \mu D U \quad \text{——(1)}$$

where $\mu \rightarrow$ co-efficient of dynamic viscosity

$D \rightarrow$ Diameter of the sphere

$U \rightarrow$ velocity of fluid over sphere



Stokes further observed that out of the total drag given by equation (1), two-third is contributed by skin friction drag and one-third by pressure drag.

$\therefore$ Skin friction drag $F_{D_f} = \frac{2}{3} F_D$

$$= \frac{2}{3} \times 3\pi \mu D U$$

$$= 2\pi \mu D U$$

pressure drag $F_{D_p} = \frac{1}{3} F_D$

$$= \frac{1}{3} \times 3\pi \mu D U$$

$$= \pi \mu D U$$

(i) **Expression of C_d for sphere when Reynolds number is less than 0.2.** ($Re < 0.2$)

$$F_D = C_D \times \frac{1}{2} \rho A U^2$$

For sphere $F_D = 3\pi \mu D U$

$$\therefore \quad 3\cancel{\pi} \mu \cancel{D} \cancel{U} = C_D \times \frac{1}{2} \rho \times \frac{\cancel{\pi}}{4} D^{\cancel{2}} \times U^{\cancel{2}}$$

$A = \frac{\pi}{4} D^2$ (Projected area of sphere)

$$\Rightarrow \quad C_D = \frac{3\mu}{\frac{1}{2} \rho \frac{DU}{4}}$$

$$= \frac{24 \mu}{\rho D U} = \frac{24}{\frac{\rho D U}{\mu}}$$

$$\boxed{C_D = \frac{24}{Re}} \quad \text{--- (2)}$$

This is called Stoke's law

(ii) Value of C_D ~~when~~ for sphere when Re is between 0.2 and 5

$(0.2 < Re < 5)$

With increase in Reynolds number the inertia forces increases and must be taken into account.

When Re lies between 0.2 and 5.

Oseen, a swedish physicist improved stoke's law as

$$C_D = \frac{24}{Re}\left[1 + \frac{3}{16\,Re}\right] \quad \text{---} (2)$$

Eq^n (3) is called Oseen formulae and is valid for Re between 0.2 and 5

(iii) Value of C_D for sphere where Re from 5.0 to 1000. $(5.0 < Re < 1000)$

$C_D = 0.4$

[Drag coefficient for Re from 5 to 1000 is 0.4]

Scanned by CamScanner

(iv) Value of C_D for sphere for Re from 1000 to 100,000. $(1000 < Re < 100{,}000)$

$$C_D = 0.5$$ [Independent of the Reynolds number]

(v) Value of C_D for sphere for Re more than 10^5. $(Re > 10^5)$

$$C_D = 0.2$$

Scanned by CamScanner

Terminal velocity of a body :-

Terminal velocity is defined as the maximum constant velocity of a falling body with which the body will be travelling. (sphere or a composite body such as parachute together with man)

(*) When the body is allowed to fall from rest in the atmosphere, the velocity of the body increases due to acceleration of gravity.

(*) With the increase of the velocity, the drag force, opposing the motion of body also increases.

(*) A stage is reached when the upward drag force acting on the body will be equal to the weight of the body.

(*) Then the net external forces acting on the body will be zero and the body will be travelling at constant speed. This constant speed is called the terminal velocity of the falling body.

If the body is drops in fluid, at the instant it acquired terminal velocity, the net forces acting on the body will be zero. The forces acting on the body at this stage will be acting on the body

1. weight of the body (W) acting downward
2. Drag force (F_D), acting vertically upward
3. Buoyant Force (F_B) acting vertically up

The net force on the body will be zero, (ie) $\boxed{W = F_D + F_B}$

Drag on a Cylinder

(*) Consider a real fluid flowing over a circular cylinder of diameter D and length L, when the cylinder is placed in the fluid such that its length is perpendicular to the direction of flow.

(*) If the Reynolds number of the flow is less than 0.2 (i.e. $\frac{\rho U D}{\mu} = \frac{U D}{\nu} < 0.2$), the inertia force is negligibly small as compared to viscous force and hence the flow pattern about the cylinder will be symmetrical.

(*) As the Reynolds number is increased, inertia forces increases and hence they must be taken into consideration for analysis of flow over a cylinder.

(*) With the increase of the Reynolds number, the flow pattern becomes unsymmetrical with respect to an axis perpendicular to the direction of flow

$Re = \frac{\rho U D}{\mu}$

The drag force (i.e) the force exerted by the flowing fluid on the cylinder in the direction of flow depends upon the Reynolds number of the flow.

From experiment it has been observed that

$F_D \propto vel$
$C_D \propto \frac{1}{Re}$

(i) When the Reynodls number $(Re) < 1$, the drag force is directly proportional to the velocity and hence ~~drag~~ drag co-efficient (C_D) is inversely proportional to Reynolds number

$Re \to$ increase from 1 to 2000
$C_D \to$ decreases

(ii) With increase in Reynolds number from 1 to 2000, the drag co-efficient decreases and reaches a minimum value of 0.95 at $Re = 2000$.

$Re \to 1$ to 2000 increase

C_D decreases and $C_D = 0.95$ at $Re = 2000$

$Re \to$ increases
$C_D \to$ increases.

(iii) With the further increase of Reynolds number from 2000 to 3×10^4, the co-efficient of drag increases and attains maximum value of 1.2 at $Re = 3 \times 10^4$

$C_D = 1.2$ at $Re = 3 \times 10^4$

(iv) The value of co-efficient of drag (C_D) decreases if the Reynolds number is increased from 3×10^4 to 3×10^5. $C_D = 0.3$ at $Re = 3\times10^5$

Re increases
Cd decreases

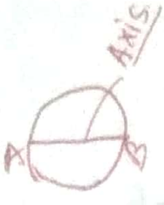
(v) If the Reynolds number increased beyond 3×10^6 the value of C_D increases and it becomes equal to 0.7 in the end.

Re increase
Cd increase

$F_D = C_D \frac{1}{2}\rho A U^2$

F_D $C_D \frac{1}{2}\rho A U^2$

Development of Lift on a Circular cylinder :-



(*) When a body is placed in a fluid in such a way that its axis is parallel to the direction of fluid flow and body is symmetrical, the resultant force acting on the body is in the direction of flow.

(*) There is no force component on the body perpendicular to the direction of flow. Hence in this case Lift will be zero.

(*) The Lift will be acting on the body when the axis of the symmetrical body is inclined to the direction of flow or body is unsymmetrical.

Stationary

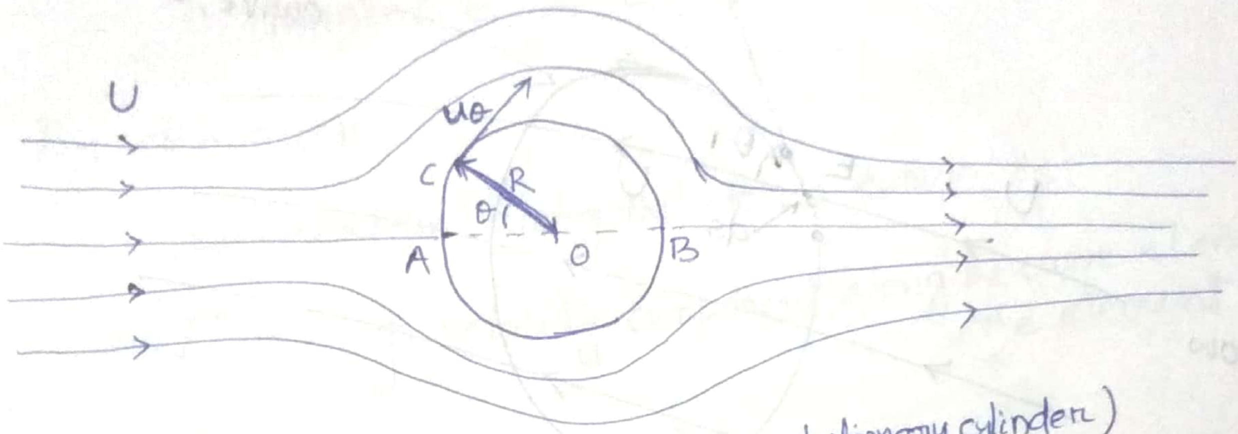
(*) In case of circular cylinder, the body is symmetrical and the axis is parallel to the direction of flow when cylinder is stationary. Hence the lift will be zero.

rotation

(*) If the cylinder is rotated, the axis of the cylinder is no longer parallel to the direction of flow and hence lift will be acting on the rotating cylinder.

This is explained by considering the following cases.

1.(*) Flow of Ideal fluid over a stationary cylinder :-



(Flow of ideal fluid over a stationary cylinder)

Consider the flow of an ideal fluid over a cylinder, which is stationary as shown in the figure.

Let U = free stream velocity of fluid

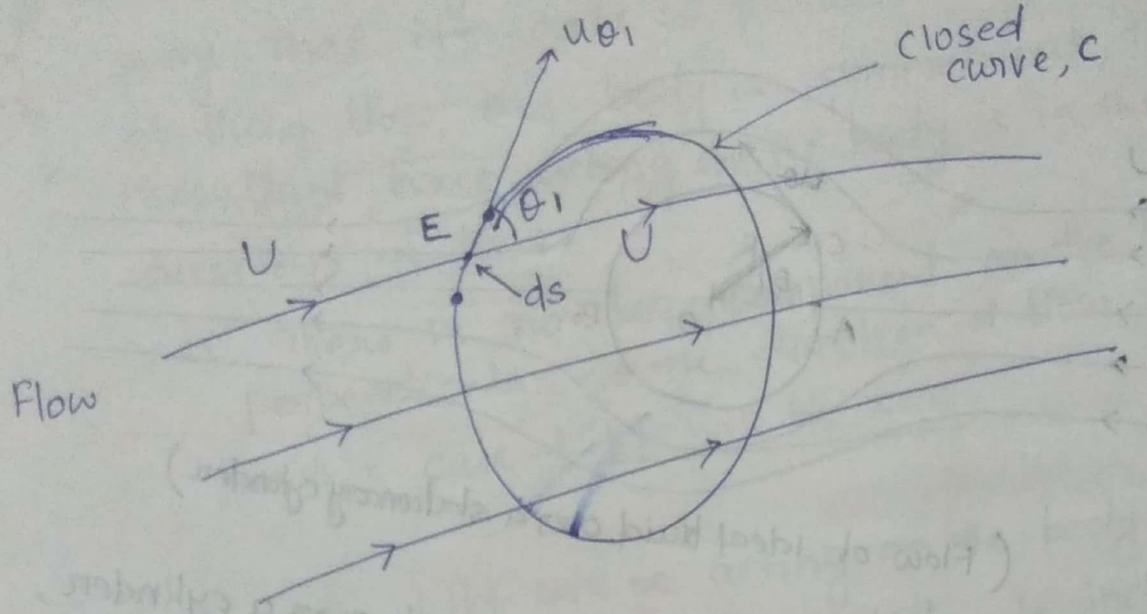
R = Radius of the cylinder

θ = Angle made by any point (say) C on the circumference of the cylinder with the direction of flow

The flow pattern will be symmetrical and the velocity at any point C on the surface of the cylinder is given by $u_\theta = 2U\sin\theta$

The velocity distribution on the upper half of the cylinder from the axis AB of and lower half of the cylinder are identical and hence the pressure distributions will ~~be~~ also be same. Hence the lift acting on the cylinder will be zero.

Flow Pattern around the cylinder when a constant Circulation Γ is imparted to the cylinder :-



Circulation

* Circulation is defined as the flow along a closed curve.

Mathematically,

Circulation is obtained if the product of the velocity component along the curve at any point and the length of the small element containing that point is integrated around the curve

Consider a fluid flowing with a free stream velocity equal to U.

Consider a closed curve within the fluid as shown in figure.

Let E is any point on the closed curve and 'ds' is a small length of the closed curve containing point E.

Let θ_1 = Angle made by the tangent at E with the direction of flow.

u_{θ_1} = Component of free stream velocity along the tangent E = $U\cos\theta_1$

By definition,

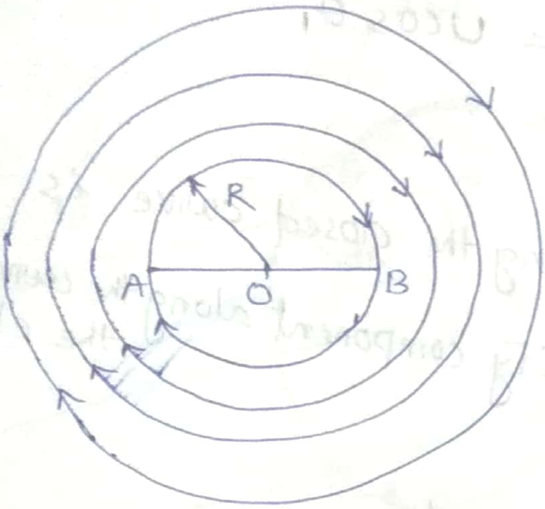
Circulation along the closed curve is

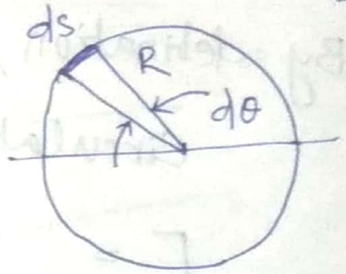
$\Gamma = \oint$ velocity component along the curve × Length of the element

$= \oint U\cos\theta_1 \times ds$

where $\oint$ = Integral for the complete closed curve

Circulation for the Flow-field in a free-Vortex :-





(b)

$\theta = \frac{1}{R}$

$\theta = \frac{ds}{R}$

(a)

stream-lines for free vortex

The equation for free vortex flow is given by

$$u_{\theta_1} \times r = \text{Constant} = K \text{ (say)}$$

where u_{θ_1} = Velocity of the fluid in a free-vortex flow

r = Radius, where velocity is u_{θ_1}

The flow-pattern for free-vortex flow consists of streamlines which are series of concentric circle as shown in figure.

In case of free-vortex flow, the stream velocity at any point on a circle of radius R is equal to the tangential velocity at that point

Scanned by CamScanner

This means that angle between the stream-lines and tangent on the stream is zero.

From figure (b)

the length of the element 'ds' is given as

$$ds = R d\theta$$

$\therefore$ For a free vortex flow

$V = u_{\theta_1}$, $\cos\theta_1 = 1$ and $ds = R d\theta$

$\cos\theta = 1$

Substituting these value

$\therefore$ We get the circulation for a free vortex is

$$\Gamma = \oint V\cos\theta_1 \times ds$$

$$= \oint u_{\theta_1} \cos\theta_1 \, ds$$

$$= \int u_{\theta_1} \times 1 \times \cancel{ds} \, R d\theta$$

But $u_{\theta_1} \times R = K$

$\therefore$ $\oint d\theta = 2\pi$

$\therefore$ $K = u_{\theta_1} \times R$

$$\therefore \Gamma = \oint K \, d\theta$$

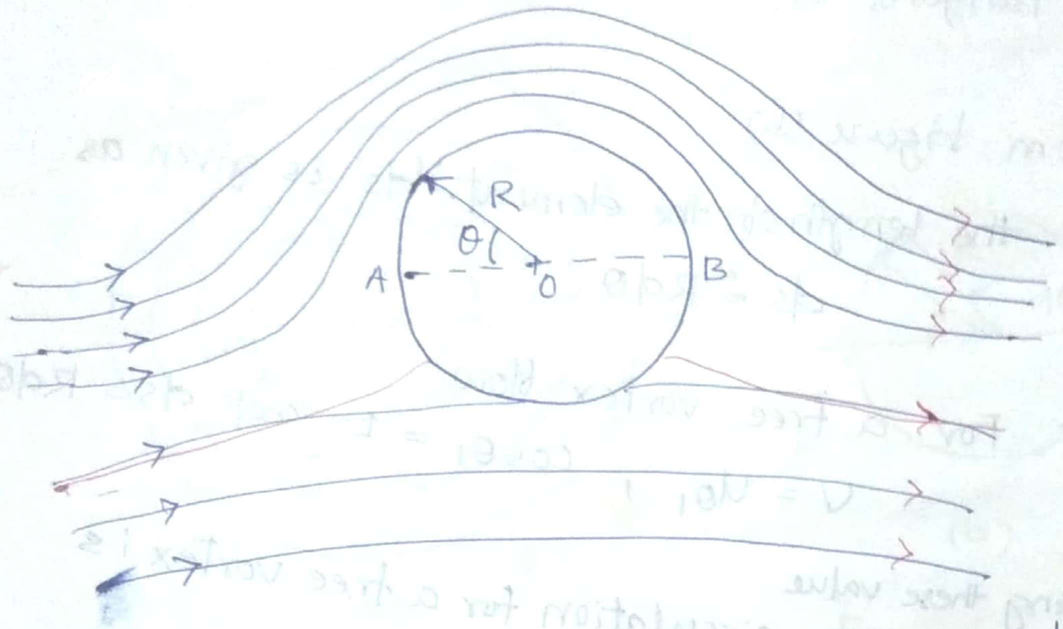
$$= 2\pi K$$

$$= 2\pi \, u_{\theta_1} \times R$$

$$\Rightarrow \boxed{u_{\theta_1} = \frac{\Gamma}{2\pi R}}$$

Scanned by CamScanner

Flow over Cylinder due to Constant Circulation :-



(Flow pattern over a rotating cylinder)

(*) The flow pattern over a cylinder to which constant circulation (Γ) is imparted is obtained by combining the flow patterns

(i) flow of fluid over stationary cylinder

(ii) Stream lines for free vortex.

∴ The resultant flow pattern is shown in figure

(*) The velocity at any point on the surface of the cylinder is obtained by adding eqn for (i) + (ii)

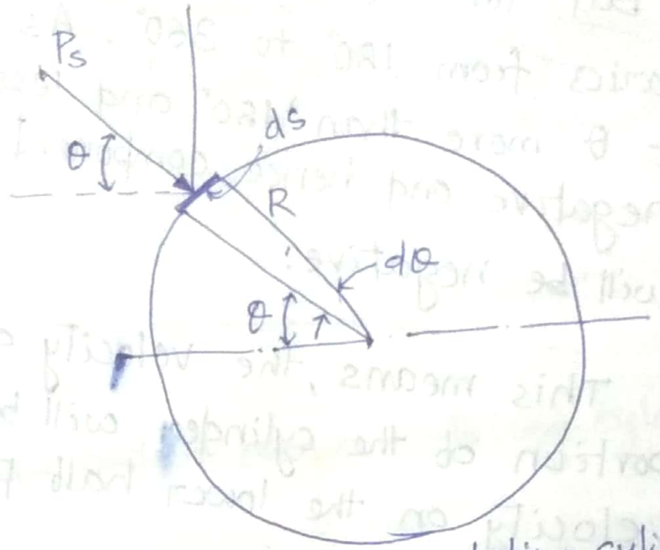
$$u = u_\theta + u_{\theta_1}$$

$$= 2U\sin\theta + \frac{\Gamma}{2\pi R}$$

Scanned by CamScanner

Expression of Lift force acting on Rotating Cylinder :-

Let a cylinder is rotating in a uniform flow field.



(Lift on a rotating cylinder)

Consider a small length on the surface of the cylinder.

Let P_s = Pressure on the surface of the element on cylinder

ds = Length of the element

R = Radius of cylinder

$d\theta$ = Angle made by the length ds at the centre of the cylinder

p = pressure of the fluid far away from cylinder

U_s = Velocity of the fluid on the surface of the cylinder.

$$= \int_0^{2\pi} -P_s \times R \times L \times \sin\theta \, d\theta$$

substituting the value of P_s from eq^n (2), we get

$$F_L = \int_0^{2\pi} -\left[P + \frac{\rho g U^2}{2g}\left(1 - 4\sin^2\theta - \frac{\Gamma}{4\pi^2 R^2 U^2} - \frac{4\sin\theta\,\Gamma}{U \times 2\pi R}\right)\right] \times RL \times \sin\theta \, d\theta$$

$$= -RL \int_0^{2\pi} \left[p\sin\theta + \frac{\rho g U^2}{2g}\left(\sin\theta - 4\sin^3\theta - \frac{\Gamma\sin\theta}{4\pi^2 R^2 U^2} - \frac{4\sin^2\theta\,\Gamma}{U \times 2\pi R}\right)\right] d\theta$$

But $\int_0^{2\pi} \sin\theta \, d\theta = \int_0^{2\pi} \sin^3\theta \, d\theta = 0$

$$\therefore F_L = -R \times L \int_0^{2\pi} \frac{\rho g U^2}{2g}\left(-\frac{4\sin^2\theta\,\Gamma}{U \times 2\pi R}\right) d\theta$$

$$= R \times L \times \frac{\rho g U^2}{2g} \times \frac{4\Gamma}{U \times 2\pi R} \int_0^{2\pi} \sin^2\theta \, d\theta$$

$$= \frac{L}{g} \frac{\rho g U \Gamma}{\pi} \int_0^{2\pi} \sin^2\theta \, d\theta$$

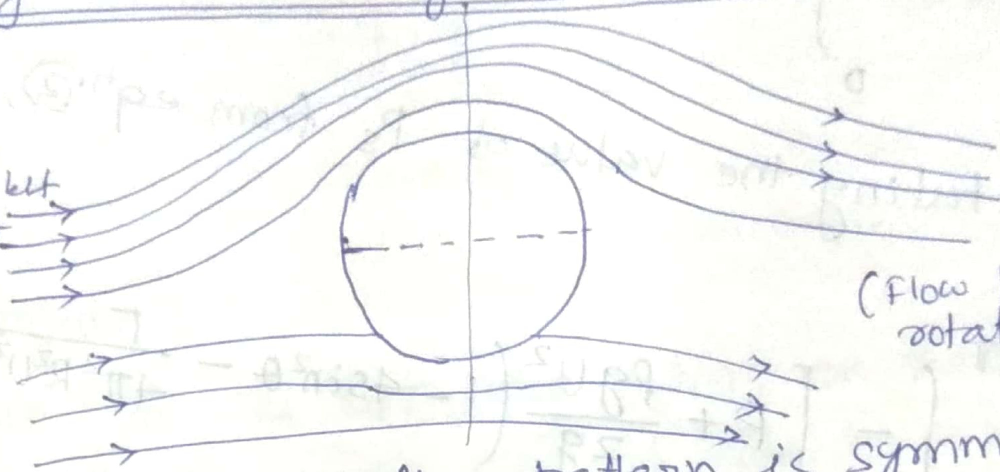
But $\int_0^{2\pi} \sin^2\theta \, d\theta = \left[\frac{\theta}{2} - \frac{\sin 2\theta}{4}\right]_0^{2\pi} = \frac{2\pi}{2} - \frac{\sin 4\pi}{4} = \pi$

$$\therefore F_L = \frac{L}{g} \times \frac{\rho g U \Gamma}{\pi} \times \pi = \frac{L}{g} \rho g U \Gamma = \rho L U \Gamma \quad \text{--- (4)}$$

Eq^n (4) is known as Kutta-Joukowski equation

Drag Force Acting on a Rotating cylinder

vel. distribution & pressure distribution on both left & right are similar.



(Flow pattern over a rotating cylinder)

* The resultant flow pattern is symmetrical about the vertical axis of the cylinder. Hence the velocity distribution and also pressure distribution is symmetrical about the vertical axis and as such there will be no drag on the cylinder.

Expression for Lift Co-efficient for Rotating cylinder

The lift co-efficient is defined by the equation

$$F_L = C_L \frac{1}{2} \rho A U^2 \quad \text{--- (1)}$$

where C_L = Lift co-efficient

A = Projected area

U = free stream velocity or uniform velocity of flow.

Kutta-Joukowski eqn

For a rotating cylinder, the lift force is given by $F_L = \rho L U \Gamma$ --- (2)

$L \rightarrow$ Length of cylinder

A = Projected area of the cylinder = $2RL$

Substituting these values in equation (1)

$$\rho L U \Gamma = C_L \frac{1}{2} \rho \times 2RL \times U^2$$

$$\Rightarrow C_L = \frac{\rho L U \Gamma}{\rho R L U^2}$$

$$= \frac{\Gamma}{RU} \quad \text{--- (3)}$$

$u_{\theta 1}$ = Vel. of fluid in a free-vortex flow

We know that $u_{\theta 1} = \frac{\Gamma}{2\pi R} \Rightarrow \frac{\Gamma}{R} = 2\pi u_{\theta 1}$

Substituting this value of $\frac{\Gamma}{R}$ in eqn (3)

$$\boxed{C_L = \frac{2\pi u_{\theta 1}}{U}} \quad \text{--- (4)}$$

where $u_{\theta 1}$ = velocity of rotation of the cylinder in the tangential direction.

Location of stagnation Points for a Rotating cylinder in a uniform flow-field :-

stagnation points are those points on the surface of the cylinder, where velocity is zero.

For a rotating cylinder, the resultant velocity is given by equation as

$$u = 2U\sin\theta + \frac{\Gamma}{2\pi R} \quad \text{--- (1)}$$

For stagnation point $u = 0$

$$\therefore \quad 2U\sin\theta + \frac{\Gamma}{2\pi R} = 0$$

$$\Rightarrow \quad 2U\sin\theta = -\frac{\Gamma}{2\pi R}$$

$$\Rightarrow \quad \boxed{\sin\theta = -\frac{\Gamma}{4\pi U R}} \quad \text{--- (2)}$$

The solution of equation (2) gives the location of stagnation points on the surface of the cylinder.

* There are two values of θ, which satisfy equation (2).

As $\sin\theta$ is negative in equation (2), it means θ is more than 180° but less than 360°.

Scanned by CamScanner

* The two values of θ are such that one value is between 180° and 270° and the other value is between 270° and 360°.

For a single stagnation point, $\theta = 270^\circ$, Eqⁿ ② becomes as.

$[\because \sin 270^\circ = -1]$

$$\sin 270^\circ = -\frac{\Gamma}{4\pi UR}$$

$$\Rightarrow \quad -1 = -\frac{\Gamma}{4\pi UR}$$

$$\Rightarrow \quad \boxed{\Gamma = 4\pi UR}$$

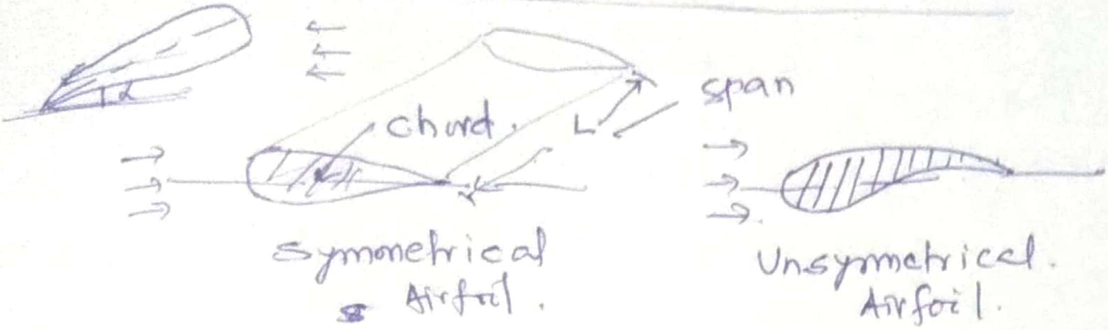
Scanned by CamScanner

Magnus Effect.

When a cylinder is rotated in a uniform flow, a lift force is produced on the cylinder. This phenomenon of the lift force produced by a rotating cylinder in a uniform flow is known as Magnus Effect.

(*) This fact was investigated by German physicist H. G. Magnus and hence the name is given as Magnus Effect.

Development of Lift on an Airfoil.



Airfoil is characterized by its chord length (C) & span (L) & Angle of attack (α)

$\alpha \leftarrow$ Angle betn dirn of the fluid flow and chord line.

Lift on the airfoil is due to negative pressure created on the upper part of the airfoil

The drag force on the airfoil is always small due to the design of the shape of the body, which is stream-line.

Circulation Γ developed on the airfoil so that the stream-line at the trailing edge of the airfoil is tangential to the airfoil is given by.

$$\Gamma = \pi C U \sin\alpha$$

Lift force $F_L = \rho U L \Gamma$

$$= \rho U L \times \pi C U \sin\alpha$$

$$= \pi \rho C U^2 L \sin\alpha. \quad — ①$$

$$F_L = C_L \times A \times \frac{\rho U^2}{2} = C_L \times C \times L \times \frac{\rho U^2}{2} \quad ②$$

A = projected area $= C \times L$ for airfoil.

Equating ① & ②

$$\boxed{C_L = 2\pi \sin\alpha}$$

Co-efficient of lift depends on angle of attack.

Module III (Ch 5)

Hydraulic Machine / Fluid Machine

1. Fluid m/cs are energy conversion devices in which mechanical energy is either produced (Turbine) or absorbed (pump) due to the momentum change of stream of fluid.

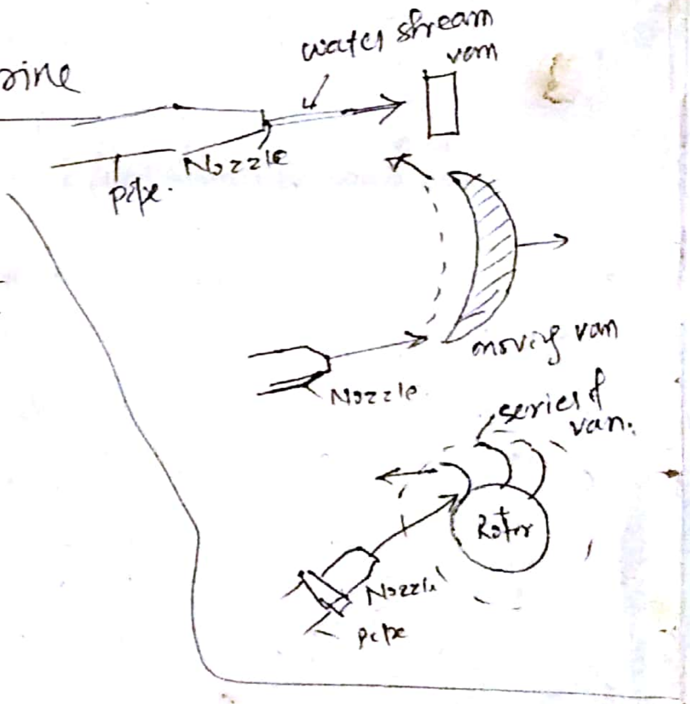
[(Turbine) → Hydraulic E → Mechanical work → Electr

(pump) → mechanical E → Hydraulic energy

Principle of working of Turbine

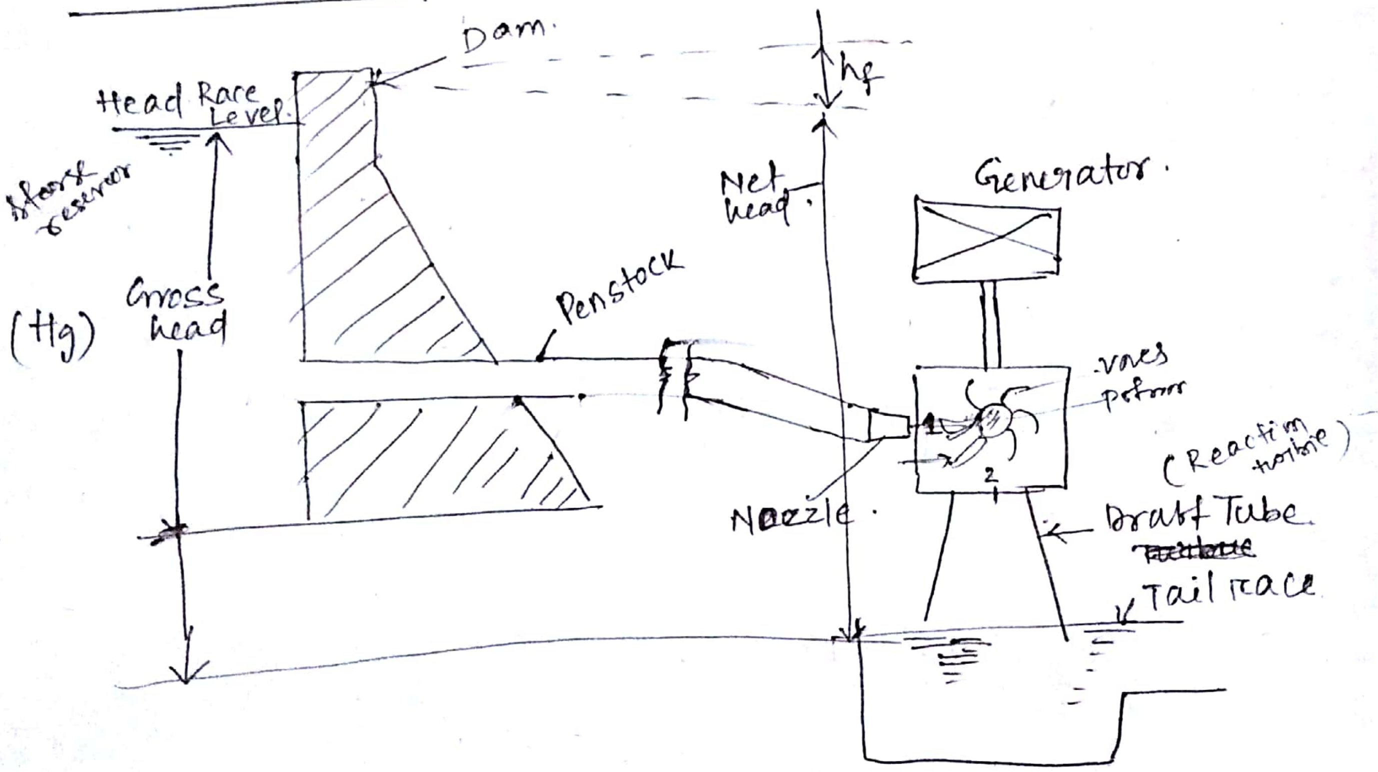
Consider the

* Action of a jet of water gliding along the moving vane



In general a hydraulic ... of a wheel called runner (or rotor) having a number of specially designed vanes or blades or buckets. The water possessing a large amount of hydraulic enrsy when strikes the runner, it does work on the runner and causes it so to rotate. The mechanical energy developed is supplied to generator which is coupled to runner to generate electrical enrsy.

General Layout of Hydroelectric power plant



(1)* essential requirements of hydroelectric power generation is → availability of ~~large~~ continuous source of water with a large amount of hydraulic energy.

(2) * A dam is constructed across a river to store water.

(3) * ~~Pipe of large~~ water from the storage reservoir is carried through penstock to the power house., Penstocks are pipe of large diameter made of (steel, wood or reinforced concrete) which carry water under pr. from the storage reservoir to the turbine.

(4) The water surface in the storage reservoir is known as head race level. or simply head race.

(5) The water passing through the turbine is discharge to the tail race. through draft tube. It is channel which carries water away from the power house

Definations of Heads and Efficiencies of a Turbines

1. Gross head → (H_g) = Difference betn the head race level & tail race level when no water is flowing ~~is known~~

2. Net Head $H = H_g - h_f$

H_g → gross head

h_f → Head loss due to friction in the penstock betn HRL & TRL

$$h_f = \frac{4 f L V^2}{D \times 2g}$$

f → Friction factor (0.01 to 0.03)
L → length of penstock
D → diameter of penstock
V → vel of water in penstock

Head available at the entrance of the turbine
H = (Net head) $= H_g - h_f$

Efficiencies of Turbines :-

1. Hydraulic Efficiency η_h
2. Mech. Efficiency η_m
3. volumetric efficiency η_v
4. Overall efficiency.

Hydraulic efficiency (η_h) :-

output / i/put

$$\eta_h = \frac{\text{Power developed by the runner of a turbine}}{\text{Power supplied by the water at the inlet of the turbine.}}$$

$$= \frac{H \cdot P}{W \cdot H \cdot P}$$

water horse power

$$H \cdot P = \frac{W}{g} \frac{[V_{w1} \pm V_{w2}] \times u}{75}$$ ... For pelton Turbine.

$$= \frac{W}{g} \left[\frac{V_{w1} u_1 \pm V_{w2} u_2}{75} \right]$$ — for radial flow Turbine.

$u \rightarrow$ tangential vel. of van

$V_{w1} \rightarrow$ velocity of whirl at inlet.

$$W H P = \frac{W \times H}{75}$$

Mechanical efficiency (η_m)

$$= \frac{\text{Power at the shaft of the turbine}}{\text{Power developed by the runner}} = \frac{S \cdot H \cdot P}{H \cdot P}$$

Volumetric efficiency (η_v)

$$= \frac{\text{Vol. of water actually striking the runner}}{\text{Vol. of water supplied to the turbine.}}$$

Overall efficiency = η_o

$$= \frac{\text{Power available at the shaft of the turbine}}{\text{Power supplied at the inlet of turbine}}$$

$$= \frac{S \cdot H \cdot P}{W H \cdot P}$$

power developed by runner

$$= \frac{SHP}{HP} \times \frac{HP}{WHP}$$

$$= \eta_m \times \eta_h .$$

CLASSIFICATION OF TURBINES

Hydrulic Turbines ~~are~~ may be classified according to several consideration

① According to action of water flowing through the turbine runners

(Pelton wheel turbine) ⓐ Impulse Turbine :- In this case all the available energy of water is converted into K.E. or velocity head by passing it through a contracting nozzle provided at the end of penstock.

* The water coming out ~~from~~ of the nozzle in the form of free jet is made to strike on a series of buckets (mounted on the rotor) ~~and~~ of runner thus causing it to revolve.
* The runner revolve freely in air.
* The water is in contact with only a part of the runner at a time. and throughout its action on the runner.

(it strikes few bucket)

* There is no difference of pressure in water at the inlet to the runner and the discharge. Therefore the casing of an impulse turbine has no hydraulic function to perform. It is necessary only to prevent splashing and to guide the water discharge to tail race. and also acts as a safe guard against accident.

(pr is atmosphere)

* This turbine is also known as free jet turbine.

In such ~~turn~~ a type of turbine

$P_1 = P_2$, $V_1 >>> V_2$ ← absolute velocity of jet leaving the bucket

↳ absolute vel. of jet before striking the bucket

$V_{r1} = V_{r2}$ → velocity of the jet leaving the bucket relative to the bucket

→ velocity of incoming jet relative to the bucket

eg. Pelton Turbine, Turgo-impulse, Girard, Banki, Jonval
(Predominantly used)

Reaction Turbine

* In this case at the entrance to the runner only a part of the available energy of water is converted into kinetic energy and a substantial part remains in the form of pressure energy.

* The reaction turbine operates with its runner sub-merged in water. The ~~moment on~~ movement on the runner is produced by both kinetic and pr. energies.

* The water leaving the turbine has still some of the pr. as well as K.E.

* The pr. at the inlet to the turbine is much higher than the pr. at outlet. Thus there is possibility of water flowing through some passage other than the runner and escape without doing any work.

* Hence a casing is absolutely essential due to the difference of pr. in reaction turbine.

For this type $P_1 >> P_2$, $V_1 > V_2$ and $V_{r1} < V_{r2}$

eg. - Francis, Kaplan, Thomson, propeller
(predominantly used)
Fourneyron turbine.

② Classification according to the main dir^n of flow of water in the runner

Hydr. Turbine

- Tangential flow (Pelton turbine)
- Radial flow.
  - outward radial flow turbine (Fourneyron turbine)
  - Inward radial flow turbine (Old Francis Turbine) Thomson turbine Girard radial flow turbine.
- Axial flow turbine (Kaplan turbine) Propeller Jonval Girard axial flow T.
- Mixed flow. (Radial & Axial) turbine (Morden Francis turbine)

In Tangential flow turbine the water flow along the tangent to the path of rotation of the runner. (Eg. Pelton wheel is a tangential flow Turb.)

In radial flow turbine water flows along the radial direction and remains wholly and mainly in the plane normal to the axis of rotation. ((ie) → water enters radially & leaves radially)

↳ Two type

① inward radial flow → the water enters at the outer circumference and flows radially inward towards the centre of the runner (casing to runner centre) old Francis turbine

② outward radial flow → water enters at the centre and flows radially outwards towards the outer periphery of the runner (centre runner to casing)

es: Fourneyron turbine.

Axial flow turbine → the water flows parallel to the axis of turbine shaft

eg:- Jonval, Girard axial flow, Kaplan, propeller.

Mixed flow turbine → water enters readially and emerges out so that the discharge is parallel to the axis of the turbine shaft.

eg:- Mordon Francis turbine.

(3) **Classification on the basis of head and quantity of water availables.**

Impulse wheel. Pelton (i) High head turbine → less quantity of water → 250 m - 1770 m.

Mordon Francis — (ii) medium head turbine → 60 m - 250 m. → relatively large quantity of water.

reaction Kaplan or propeller (iii) Low head turbine. → less than 60 m. → large quantity of water

Impulse turbine requires high head & low rate of flow
Reaction turbine requires low head & high rate of flow

(4) **According to specific speed.** →

Specific speed of a turbine is defined as the speed of a geometrically similar turbine that would develope ~~unit~~ 1 kW power. ~~horse power~~ when working under ~~unit~~ head. of 1 m.

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

(i) speed varying from 10 to 35 → Pelton wheel with single jet
50 → Pelton wheel with double jet.

(ii) 60 to 300 → Francis turbine.

(iii) 300 to 1000 → Kaplan and Propeller turbine.

5. Classification according to the disposition of turbine shaft :-

turbine shaft may be either vertical or horizontal.

In morden turbine practice. pelton turbine usually have - horizontal shaft where the rest, especially the large unit have vertical shaft
↳ (commonly adopted)

6. Classification after the name of the originator :-

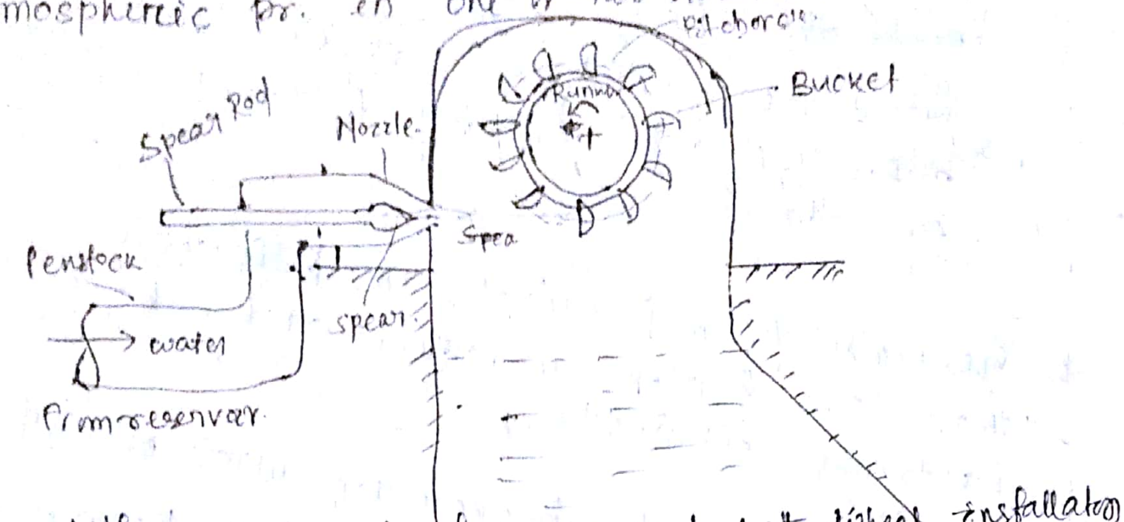
(a) Pelton turbine :— named in honour of Lester Allen Pelton (1829-1908) of California (USA)
is an impulse type turbine used for high head & low discharge

(b) Francis turbine — named after James Bichens Francis (1815-1892) ~~of Eng~~ (England — USA)
↳ is a reaction type turbine for medium high to medium low and medium small to medium large quantity of medium flow.

(c) Kaplan turbine. → named in honour of Dr. Victor Kaplan (1876-1934) (Bruenn, Germany)
is a reaction type of turbine for low head & large quantity of flow.

Pelton Wheel

The Pelton wheel is an impulse turbine where the hydrostatic head is expanded down to atmospheric pr. in one or two nozzles.

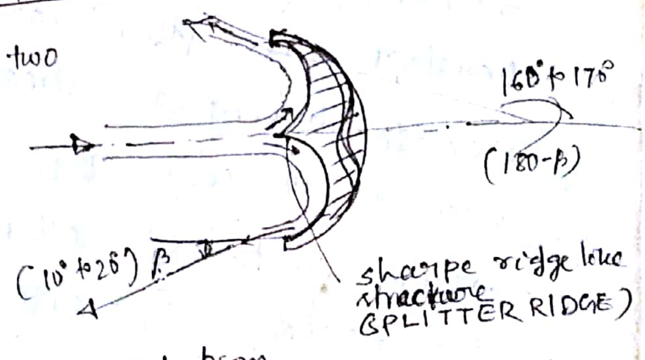


The different parts of a typical pelton wheel installation

(1) Runner with buckets :- (1) The rotor of the turbine is called runner. The runner consists of a circular disc with a number of buckets evenly spaced around its periphery.

(2) The bucket has a shape of ~~double semi~~ double semi-ellipsoidal cup

* Each bucket is divided into two symmetrical parts by a sharpe-edged ridge known as splitter



(3) ~~One or more~~ The jet of water coming out from the nozzle ~~...~~ strikes on the splitter, which divide the jet into two equal portion. each of which after flowing round the smooth inner surface of the bucket leaves it at its outer edge.

(4) The bucket are so shaped that angle at the outlet tip varies from 10° to 20° (usually kept as 15°) so that the jet of the water gets deflected through 160° to 170°.

(5) Advantage of having a double cup-shaped bucket is that the axial thrust neutralise each other being equal & opposite and hence the bearings supporting the wheel shaft are not subjected to any axial or end thrust.

* Generally the no. of buckets in ~~Pelton~~ rotor is more than 15. Buckets are either cast integrally or fastened separately..

* For low head buckets are made of cast iron

For high head ⟶ cast steel, bronze or stainless steel.

~~(4) Penstock~~

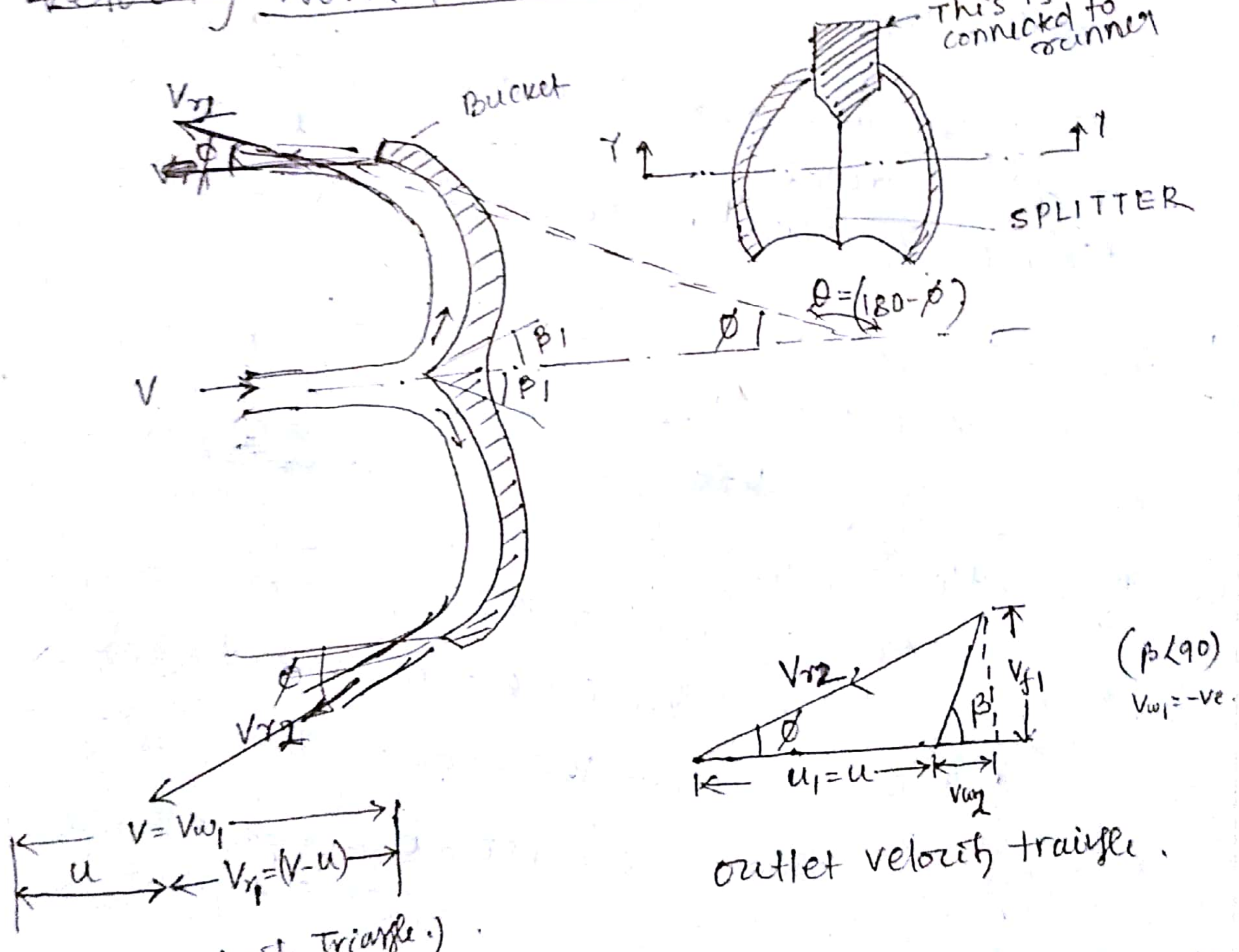
(2) Spear and nozzle :- In order to control the quantity of water striking the runner, the nozzle fitted at the end of the penstock is provided with a spear or needle having a streamline head which is fixed to the rod. The movement of the spear is ~~either~~ controlled either manually by a hand wheel (in case of small units) or automatically by governing mechanism (in case of bigger units)

when shaft of the pelton wheel is horizontal then not more than 2 jets are used.

If wheel is mounted on vertical shaft ⟶ upto 6 jet can be used.

Casing :→ A casing made of cast iron or fabricated steel plates is usually provided for a pelton wheel. It has no hydraulic function to perform. It is provided only to prevent splashing of water and to guide the water to tail race. and also it acts as a safe-guard against accidents.

~~Velocity~~ Work done & efficiencies of Pelton wheel :.



(Inlet velocity Triangle.)

outlet velocity triangle.

V = absolute vel of jet before striking the bucket
V_1 = absolute vel. of jet leaving the bucket
u = absolute vel. of bucket considered along the dirn tangential to the path circle.

V_{r_1} = velocity of incoming jet relative to the bucket
V_{r_2} = velocity of jet leaving the bucket relative to the bucket.

V_{w_1} = velocity of whirl at inlet tip of the bucket
V_{w_2} = velocity of outlet tip of the bucket.

θ = Angle ~~ϕ~~ through which the jet is deflected by the bucket $(= 180° - \phi)$

where ϕ is the angle of the bucket at the outlet tip.

Since the flow along the bucket is axial, the inlet and outlet jets are at the same radial distance for which the periferal speed $u_1 = u_2 = u$

The inlet relative velocity is $V_{r_1} = V - u$
& outlet $V_{r_2} = K(V_{r_1}) = K(V-u)$
~~K is~~ also $K < 1 \rightarrow$ bcoz of frictional effects

From outlet vel. triangle
$$V_{w_2} = V_{r_2}\cos\phi - u_1$$
$$= K V_{r_1} \cos\phi - u = K(V-u)\cos\phi - u$$
~~$= u - KV_{r_1}\cos\phi$~~

~~Also $V_{w_2} = u_1 - V_{r_2}\cos\phi$~~

~~where $\phi = 180 - \theta$ = Angle of bucket at outlet tip.~~

~~If losses are neglected K=1.~~

$\therefore$ Work done per unit mass of fluid
$$W = u(V_{w_2} - V_{w_1})$$
$$= u\left[K(V-u)\cos\phi - u - (-V)\right]$$

or $$W = \left[uV + Ku(V-u)\cos\phi - u^2\right]$$

Hydraulic efficiency

$$\eta_h = \frac{\text{Work done per second}}{\text{K.E of the jet per second}}$$

$$= \frac{2[V_{w_1} + V_{w_2}] \times u}{V^2}$$

$V_{w_1} = V$

$V_{r_1} = V - u$

If $K=1$ $V_{r_2} = K(V-u)$

$= (V-u)$ $\cancel{K[V-u]}$

$V_{w_2} = (V-u)\cos\phi - u.$

Substitute

$$\eta_h = \frac{2(V_1 - u)[1+\cos\phi]u}{V^2}$$

$$\frac{d}{du}(\eta_h) = 0$$

$$\eta_{max} = \frac{1+\cos\phi}{2}$$

Properties of pelton turbine

(1) The vel. of jet at inlet $V = C_v\sqrt{2gH}$

C_v = Co-efficient of vel $= .98$ or $.99$

$H \rightarrow$ Net head of turbine.

(2) velocity of wheel (u) is given by

$u = \phi\sqrt{2gH}$ ϕ = speed ratio ($.43$ to $.48$)

(3) Jet deflection Angle $165°$ (if no angle detection is nee)

(4) mean diameter of pelton wheel

$D = \frac{60u}{\pi N}$

(5) Jet ratio $m = \frac{D}{d}$ $d \rightarrow$ dia of jet.

(6) No. of bucket on the runner

$Z = 15 + \frac{D}{2d} = 15 + 0.5m$

(7) No. of jet (n) = $\frac{\text{Total rate of flow through turbine}}{\text{rate of flow of water through one jet}}$ any jet rate.

Radial Flow Reaction Turbine

Kaplan → Axial flow reaction turbine

Two types
1. Inward radial flow — old Francis.
2. Outward radial flow.

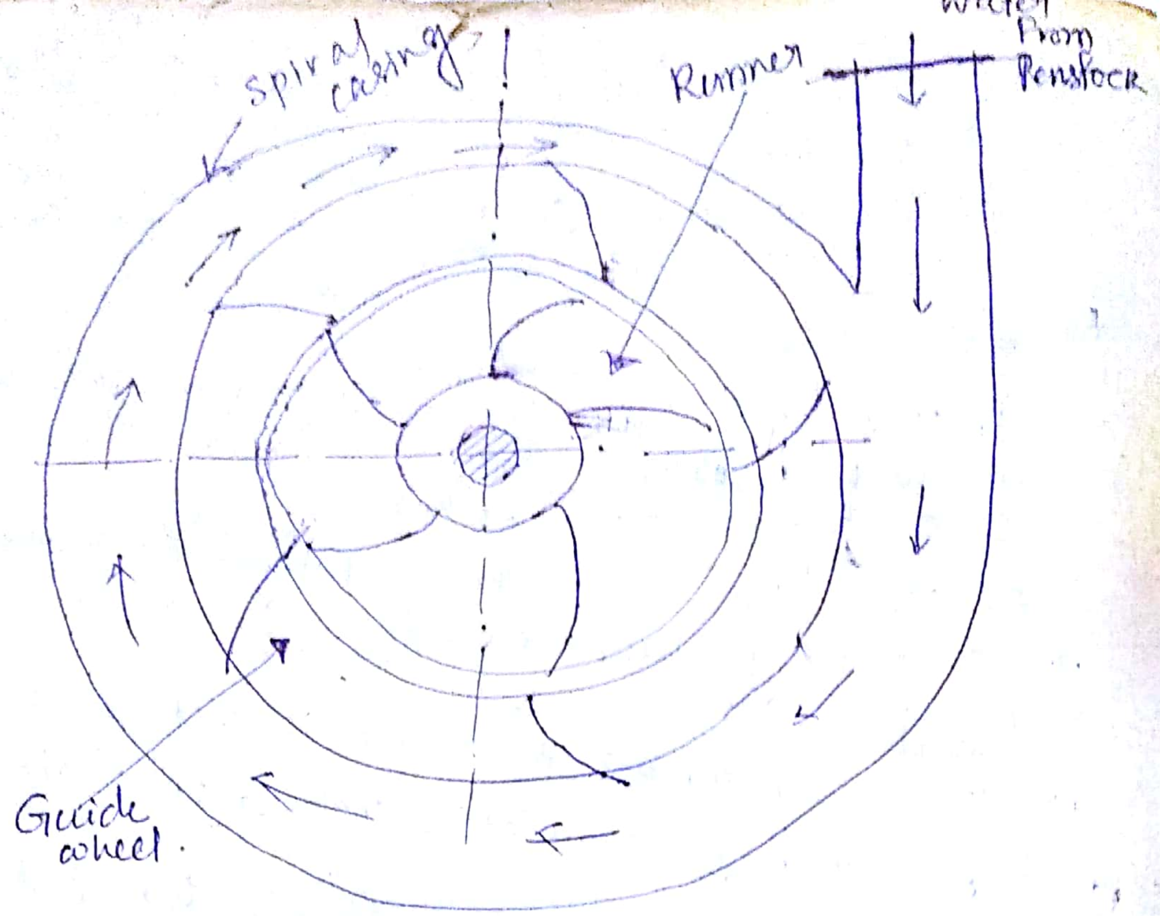
Mixed radial flow ↳ Modern turbine.

Reaction Turbine

1. The water at inlet of turbine posses K.E as well as pressure energy.
   So at inlet T.E = K.E + P.E.
2. When the water flows through the runner a part of the pressure energy goes on changing into K.E.
3. The water through the runner is under pressure.
4. The runner is completely enclosed in a air tight casing.
5. The casing and runner is always full of water.

Main ~~Component~~ parts of a ~~Radial~~ Radial flow Reaction Turbine

1. Casing.
2. Guide mechanism
3. Runner
4. Draft tube



1. <u>Casing</u>

① The water from the penstock enters the casing

② The casing is spiral shape in which area of c/s goes on decreasing gradually.

③ Due to spiral shape water enters the runner at const. velocity throughout the circumference of the runner.

④ The casing is made of concrete, cast steel or plate steel.

2. <u>Guide Mechanism</u> :-

① It consists of a stationary circular wheel all around the runner.

② Stationary guide ~~vaths~~ vanes are fixed on the guide mechanism.

③ The guide vanes allow the water to strike ~~which~~
the guide vanes fixed on the runner
without shock at inlet.

④ The width betn two adject vanes can
be ~~altered~~ changed so that the amount of
water striking the vain can be changed.

Runner :-

① It is a circular wheel on which a series
of radial curved vanes are fixed.

② Surfaces of vanes are made very smooth.

③ water enter and leaves the runner without
shock.

④ made of cast steel, cast iron or stainless steel.

Draft tube

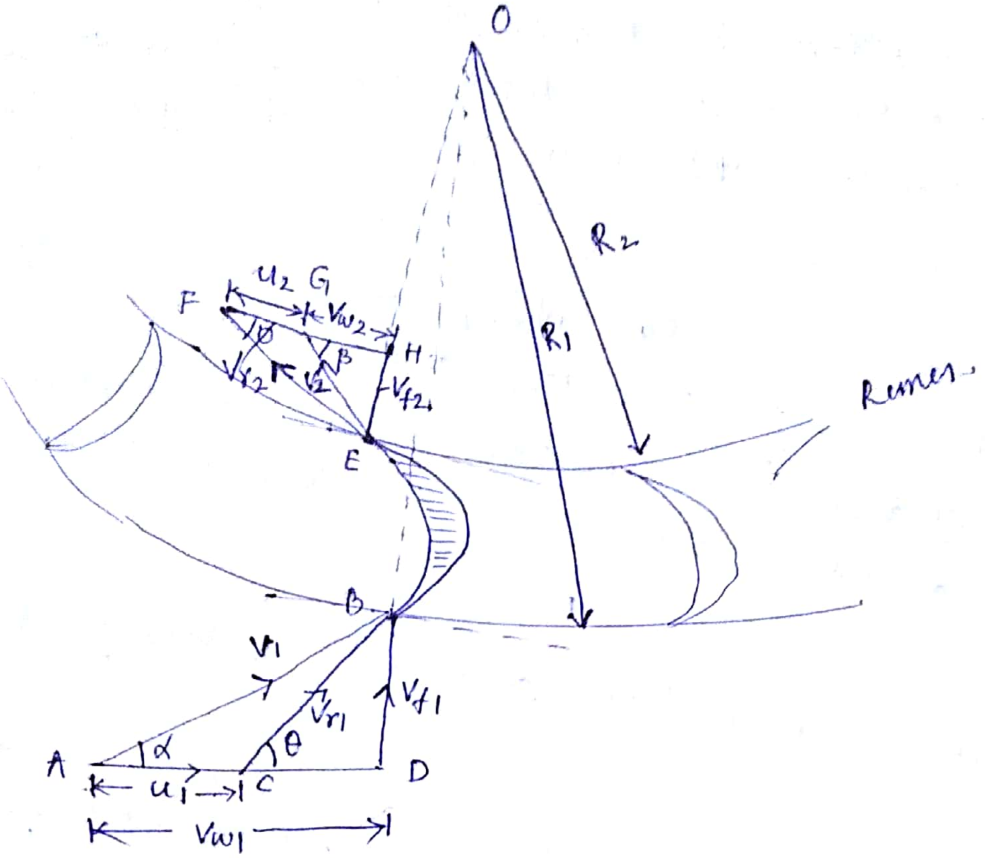
1. In reaction turbine the pressure at the
exit of runner is generally less than
atmospheric.

2. Water at exit can not be directly
discharge to the tailrace.

3. A tube or pipe of gradually increasing
area is used for discharging water from the
exit of the turbine to the tail race. This
tube of increasing area is called draft tube.

Inward Radial flow Reaction Turbine

(velocity triangle diagram)
& work done by runner



Work done/sec on the runner by water is given by eq^n

$$= \rho a V_1 \left[V_{w_1} u_1 \pm V_{w_2} u_2 \right]$$

$$= \rho Q \left[V_{w_1} u_1 \pm V_{w_2} u_2 \right] \qquad a V_1 = Q$$

where V_{w_1} = Vel. of whirl at inlet

V_{w_2} = vel. of whirl at outlet

u_1 = Tangential velocity at inlet

$$= \frac{\pi D_1 N}{60} \qquad D_1 = \text{outer dia of runner}$$

$$u_2 = \frac{\pi D_2 N}{60} \qquad D_2 = \text{inner dia of runner}$$

N = speed of runner in rpm

$$\text{Workdone/kg/sec} = \frac{\text{Workdone/sec}}{\text{Wt of water striking/sec}}$$

$$= \frac{\rho Q \left[V_{w_1} u_1 \pm V_{w_2} u_2 \right]}{\rho Q g}$$

$$= \frac{1}{g} \left[V_{w_1} u_1 \pm V_{w_2} u_2 \right]$$

+ve when β is acute

−ve β is obtuse.

~~9~~ ~~β~~ .

Axial Flow Reaction Turbine (Kaplan Turbine)

If the water flows parallel to the axis of the rotation of the shaft, the turbine is known as axial flow turbine.

Head at inlet of turbine (Total E) = K.E + P.E

During the flow of water through runner a part of pressure energy is converted into K.E., the turbine is known as a reaction turbine.

For axial flow reaction turbine

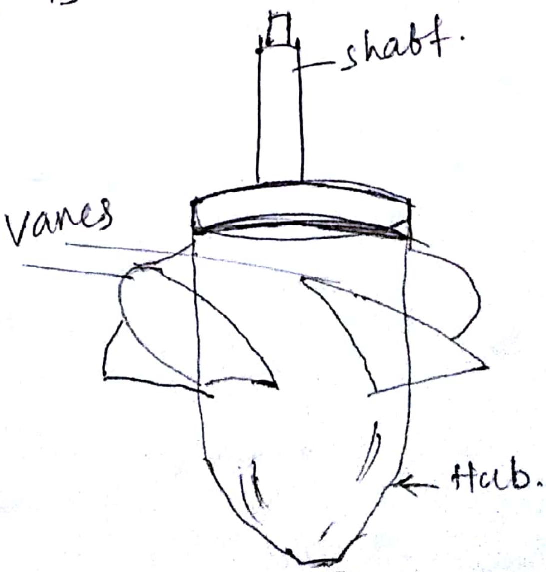
* the shaft of the turbine is vertical
* the lower end of the shaft is made larger which is known as hub or boss.
* the vanes are fixed on the hub and hence hub acts as a runner for axial flow reaction turbine.

eg: — ① Propeller Turbine → vanes fixed to the hub are not adjustable

② Kaplan turbine → If vanes on the hub are adjustable the turbine is known as Kaplan turbine.

→ This is suitable where a large quantity of water at low heads is available.

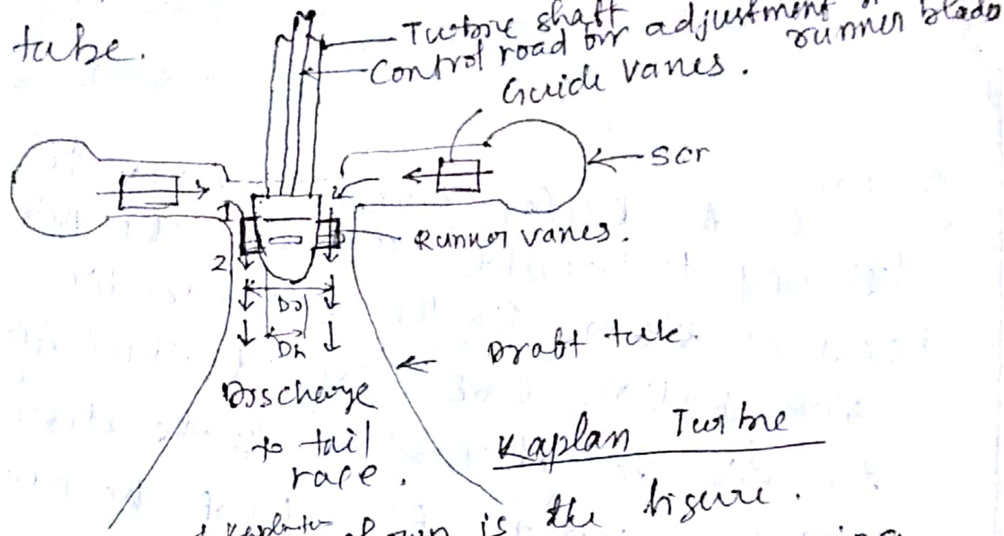
Kaplan turbine consists of a hub fixed to the shaft. On hub the adjustable vanes are fixed



Kaplan Turbine

b(ii) Main Parts of a Kaplan turbine are

1. Scroll casing
2. Guide vanes mechanism.
3. Hub with vanes or runner of the turbine
4. Draft tube.



Kaplan Turbine

All the main parts of Kaplan turbine are shown is the figure.

Water from the penstock enters the scroll casing and than move to the guide vanes. From the guide vanes, the waters runs through 90° and flow axially through runner. as shown in fige

Discharge through runner.

$$= \frac{\pi}{4}\left(D_o^2 - D_h^2\right) \times V_{f1}$$

$D_o \rightarrow$ outer dia of the runner.
$D_h \rightarrow$ Dia of hub.
$V_{f1} \rightarrow$ vel. of flow at inlet.

In this case the inlet & outlet vel. triangle are drawn at the extreme edge of the runner vane corresponding to the points 1 and to 2 as shown.

Important Relation (Kaplan Turbine)

① The Peripheral vel. at inlet & outlet are equal

$u_1 = u_2 = \frac{\pi D_o N}{60}$ $\quad D_o \rightarrow$ outer dia of runner.

② $V_{f1} = V_{f2}$ (i.e) vel. of flow inlet & outlet are equal.

③ Area of flow at inlet = Area of flow at outlet

$$= \frac{\pi}{4}\left(D_o^2 - D_h^2\right)$$

Centrifugal pumps Module IV

The hydraulic m/c which converts mechanical energy into hydraulic energy are called pumps.

The hydraulic enrgy is in the form of pressure energy.

* A device which transfer liquid at the expense of power input
* A m/c designed to elevate, deliver and move various liquids.

{pump} → belongs to the category of ~~power~~ power absorbing m/c.

C.P → If the mechanical enrgy is converted into pressure energy by means of centrifugal force acting on the fluid, the hydraulic m/c is called centrifugal pump.

pump → used to lift a fluid against pressure head

Turbine → works because of pressure differential.

Centrifugal pump acts as a reversed of an inward radial flow reaction Turbine.

That means → the flow in centrifugal pumps is in the radial outward direction.s.

Principle :- The C.P. acts on the principle forced vortex flow. which means that when a certain mass of liquid is rotated by means of a external torque, then rise is pressure head at any point of the liquid takes place.

Scanned by CamScanner

(Pressure head = $\frac{V^2}{2g} = \frac{\omega^2 r^2}{2g}$)

This rise is pressure head ~~is prof~~ at any point of the rotating fluid is proportional to the square of tangential velocity of the liquid at that point.

Thus at the outlet of the impeller where the radius is more, the rise in pressure head will be more and liquid will be discharged at the outlet with a high pressure head. Due to this high pressure ~~head~~ head liquid can be ~~at~~ lifted to high level.

Main Parts of a Centrifugal pump :-

The ~~c~~ (1) Impeller → rotor provided with a series of blade.
(a) shrouded or closed impeller
(b) Semi-open (c) open.

(2) Casing → Air tight chamber surrounds the impeller.

(3) Suction pipe with foot valve and strainer.
↳ non-return value or one-way type valve.
↳ opens only in upward direction.

(4) Delivery pipe.
↳ one end is connected to the outlet & other end deliver water at required height.

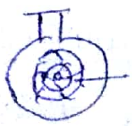
(i) volute casing. → spiral type. - flow are increasing gradually.

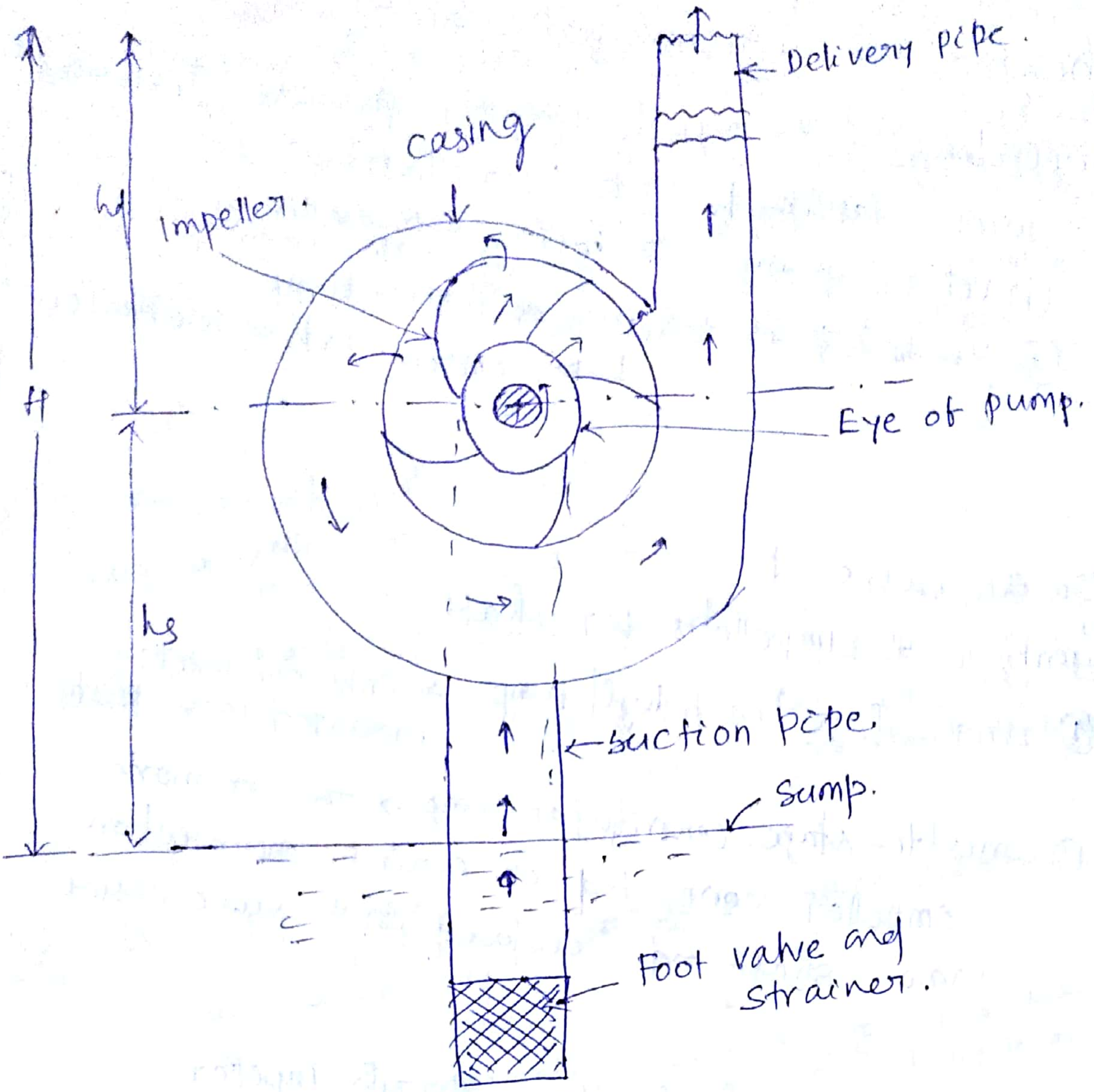
(ii) Vortex casing → circul chambe → loss due to eddies reduced.
→ effi is mr

(iii) Casing with guide blades.
↳ guide blade are mounted on a ring which is known as diffuser.

when water passes through guide vanes → vel. of flow reduces and pressure of water increases.

vortexchambr.





Delivery pipe.
casing
h_d
Impeller.
H
Eye of pump.
h_s
suction pipe.
sump.
Foot valve and strainer.

Classification of Centrifugal pump :-

① According to the type of casing provided, centrifugal pumps are classified into two classes

(i) Volute pump → impeller surrounded by a spiral shape casing

(ii) ~~Turbine pump~~ Diffuser or Turbine Pump.
↳ casing with guide blades.

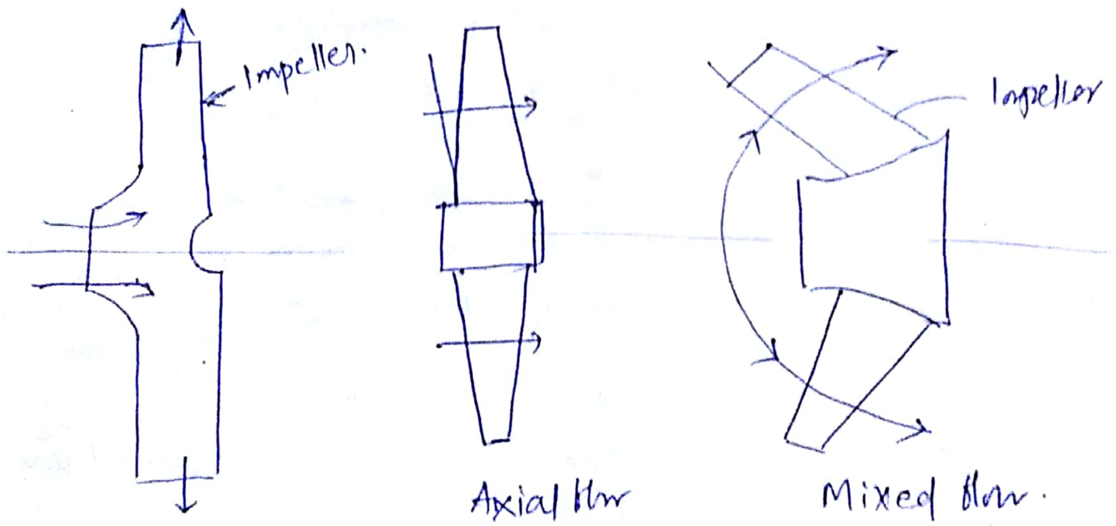
On the basis of.

② Number of impellers per shaft

ⓐ single stage centrifugal pump → only one impeller mounted on the shaft

ⓑ multi-stage centrifugal pump → two or more impeller connected in series mounted on same shaft and enclosed on the same casing.

③ Relative direction of flow through impeller.



Radial flow.

Axial flow
↓
large with relatively low heights.

Mixed flow.
↓
axially as well as radially ← liquid flow
↓
large quantity of water is discharged to low heights.

④ Number of entrances to the impeller.

ⓐ single suction pump. → liquid is admitted from the suction pipe on one side of impeller.

ⓑ Double suction pump. ↳ both side of the impeller.

[adv:-
↳ axial thrust is neutralised.

⑤ Disposition of shaft.

c.p designed with either horizont shaft →

vertical shaft → deep well & mines. more suitable. because the pump with vertically disposed shaft, occupy less space.

⑥ Working head

low → capable working against a total head of up to 15 m.

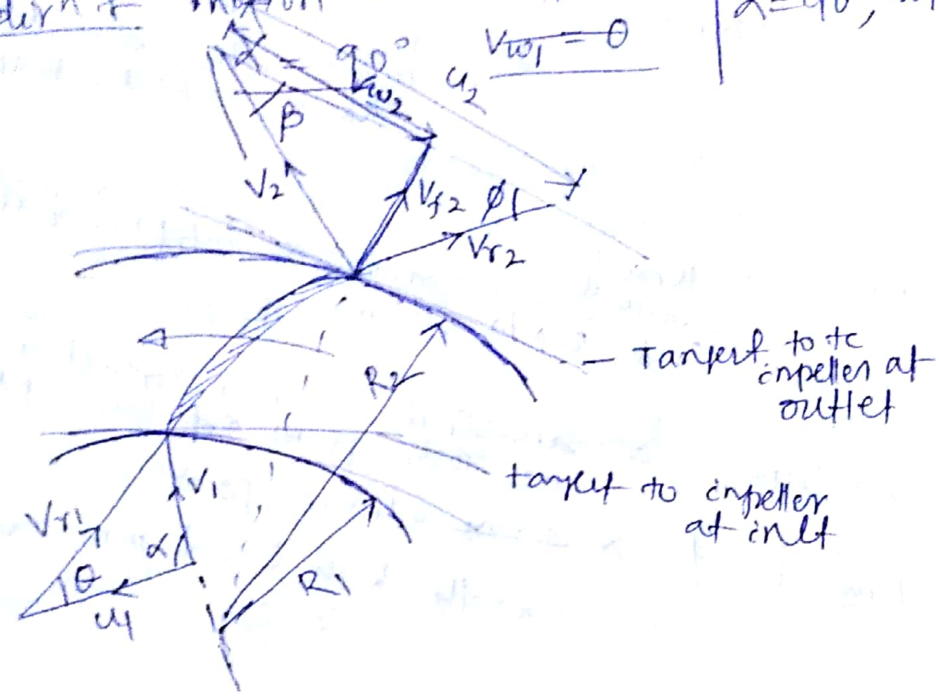
medium → capable of working against a total head more than 15 m ~~of~~ but up to 40.

high head → ~~40 m~~ above 40 m
↳ generally high head pum are multistage pumps.

Work done by the Centrifugal Pump (or by Impeller on water :-

This is obtained by drawing vel. triangles at inlet & outlet of the impeller in the same way as for a turbine.

For best efficiency → the water enters the impeller radially at inlet
(means) absolute vel. makes an angle 90° with dirn of motion of the impeller at inlet $| \alpha = 90°, V_{w_1} = 0$



$N \rightarrow$ speed of impeller in r.p.m.

u_1 = tangential vel. of impeller at inlet.

$$= \frac{\pi D_1 N}{60}$$

$$u_2 = \frac{\pi D_2 N}{60}$$

V_1 = Absolute vel. ~~of~~ water at inlet

V_{r_1} = Relative vel. " " " ".

α = Angle made by absolute vel. (V_1) at inlet with the direction of motion of vane

θ = Angle made by rel. vel. (V_{r_1}) at inlet
" " " " "

V_2, V_{r_2}, β & ϕ → corresponding value at outlet water enters impeller radially

$\alpha = 90^\circ \quad V_{w_1} = 0$

C.P → reverse of inward radial flow reaction turbine.

on turbine.

Work done by the water on the runner / sec / unit wt of water

$$= \frac{1}{g}\left(V_{w_1} u_1 - V_{w_2} u_2\right)$$

Work done by the impeller on the water per sec / unit wt of water

$= -$ (work done in case of turbine)

$$= -\left[\frac{1}{g}\left(V_{w_1} u_1 - V_{w_2} u_2\right)\right]$$

$$= \frac{1}{g}\left(V_{w_2} u_2 - \cancel{V_{w_1} u_1}\right)$$

$$= \frac{1}{g} V_{w_2} u_2$$

Work done by impeller on water/sec.

$$= \frac{W}{g} V_{w_2} u_2$$

W = Wt of water $= w \times Q$

Q = vol. of water $= \pi D_1 B_1 \times V_{f_1} = \pi D_2 B_2 \times V_{f_2}$

↓ width of impeller. ↓ vel of flow.

Scanned by CamScanner

Text Books/Reference

1. Fluid Mechanics & Hydraulic Machines
   By Modi and Seth (standard Book House)
2. Introduction to Fluid Mechanics and Fluid Machines
   By S.K. Som & G. Biswas (TMH Pub. Ltd)
3. Fluid Mechanics & Hydraulic Machines
   By Dr. R.K. Bansal (Laxmi Pub. Ltd)