Experiment 1

The Center of pressure

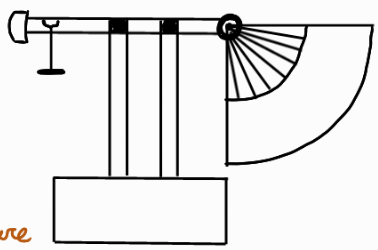
What did we do?

- The moment done by fluid on a partially/fully sumberged plane is measured and compared to theoretical analysis

What did we use?

- Center of pressure apparatus with two angular settings: $\theta = 0°$, $\theta = 20°$
- Two cases were studied: Partially Submerged $h > R_1 \cos\theta$

  Fully Submerged $h < R_1 \cos\theta$



Theory of the Experiment

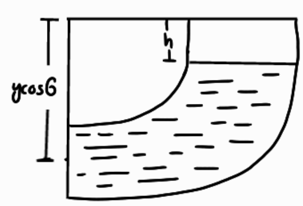
The hydrostatic force acts on a point calld **Center of pressure**

Force by the fluid $= \gamma h_f A$

γ: weight per unit volume

h_f: Distance to centroid of Area

A: Area of cross section

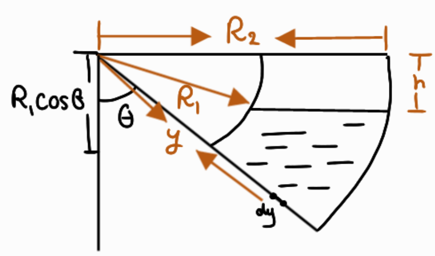


Force on the element dy:

$$F = \gamma (y\cos\theta - h) B dy$$

Moment on element

$$M = Fr = \gamma (y\cos\theta - h) B y\, dy$$



Submerged plane

Fully: $h < R_1 \cos\theta \rightarrow$ water level is higher than plane face height

$$M = \gamma B \int_{R_1}^{R_2} (\cos\theta\, y^2 - hy)\, dy = \frac{\gamma B \cos\theta}{3}(R_2^3 - R_1^3) - \frac{\gamma B}{2}(R_2^2 - R_1^2) h$$

Partially: $h > R_1 \cos\theta \rightarrow$ Elevation of water is smaller than the height of plane face

$$M = \gamma B \int_{h\sec\theta}^{R_2} (\cos\theta\, y^2 - hy)\, dy$$

$$M + \frac{\gamma B R_2^2 h}{2} = \frac{\gamma B \sec^2\theta}{6} h^3 + \frac{\gamma B \cos\theta}{3} R_2^3$$

Results

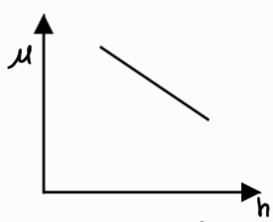
$\theta = 0°, 20°$

fully

$\frac{M}{h} = \text{slope} = \frac{\gamma B}{2}(R_2^2 - R_1^2)$

γ is calculated

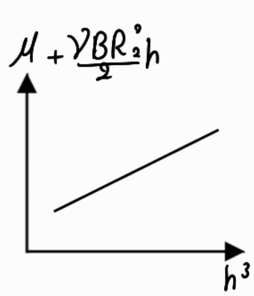
and inserted in equation of case 2



Moment from weights $= W \times R_{Arm}$

partially

$\frac{M + (\gamma B R_2^2 h)/2}{h^3} = \text{Slope} = \frac{\gamma B \sec^2\theta}{6}$



To find force

$$F = \gamma B \int_{R_1}^{R_2} (y\cos\theta - h)\, dy$$

$$= \gamma B \left(\frac{y^2}{2}\cos\theta - hy\right)\Big|_{R_1}^{R_2}$$

$$= \gamma B \left[\left(\frac{R_2^2 - R_1^2}{2}\right) - h(R_2 - R_1)\right]$$

$= -0.409$

$=$

To find Center of pressure

$y_{cp} = \frac{M}{F} =$

Experiment 2

The Stability of a floating body

What did we do?

A rectangular pontoon floats in water, a Jockey weight was used to change it's center of gravity

y and x values were changed and angle of tilt was recorded

Theory of the experiment

Types of stability of floating bodies:

1- Stable : forces provide a restoring couple

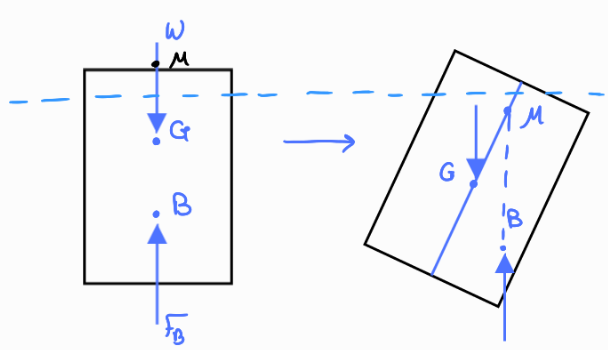
2- Unstable : No restoring moment is provided

3- Nuetral :

The forces acting on a floating body are:

1- it's weight which acts vertically downword through the center of gravity

2- The buoyancy force upwards through the center of buoyancy (centroid of displaced) volume

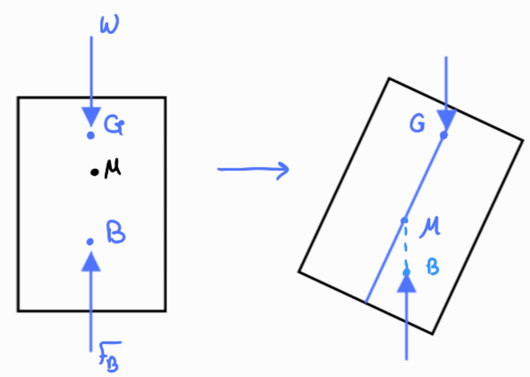


Stable Equilibrium

Metacenter ← M is above G

Means that the body will return to the equilibrium

M=G
neutral

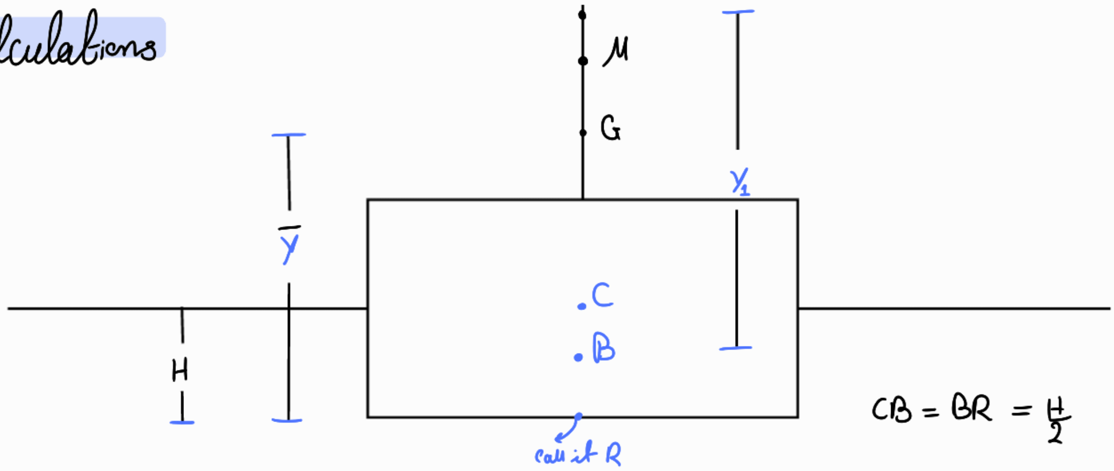


Unstable Equilibrium

M is Below G

Means that the body will take new equilibrium

Calculations



$CB = BR = \frac{H}{2}$

$BM = \text{Constant} = \frac{I}{V}$ inertia, volume of liquid displaced

$H = \frac{V}{LD}$

CM: Theo $= BM - \frac{H}{2}$

Exp $= CG + GM$

$CG = \bar{Y} - H$

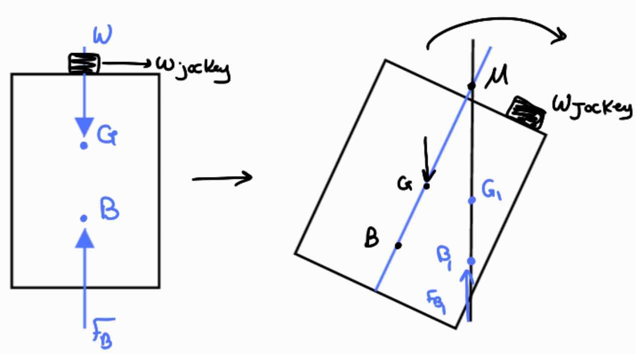
$GM = \frac{m_{jockey}}{m_{tot}} \left(\frac{dx}{dG} \right)$

A can be calculated by measuring $\bar{Y}$ for Y (first height)

$A = \bar{Y} - Y_1 \frac{m_{jockey}}{m_{tot}}$

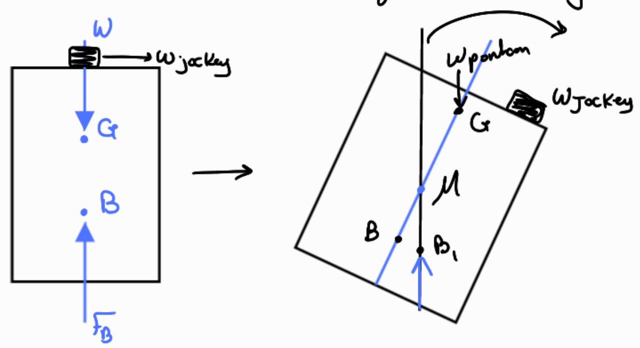
Results

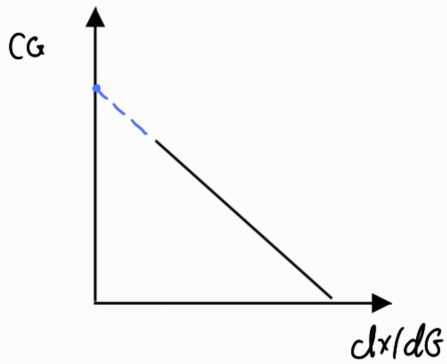
- Moving the jockey weight will shift center of gravity ($G \to G_1$). This will cause a moment (Disturbing moment). If the is in stable equilibrium, W and F_{B_1} will provide a restoring Couple.



For unstable equilibrium

G & F_B will provide a moment that is in the Direction of Disturbing moment





dx/dG : represents How much the pontoon tilts for an X movement of the Jockey

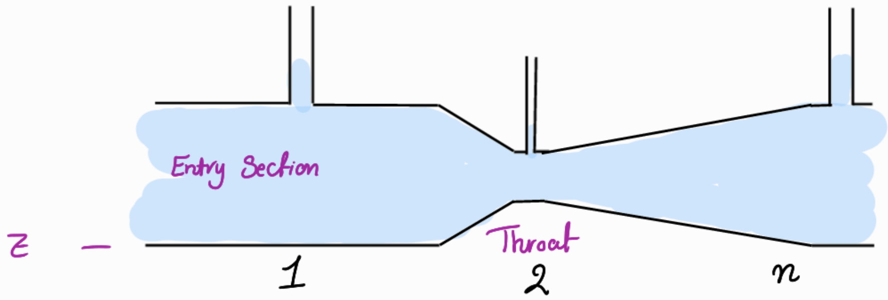
dx/dG = 0 : at any position in the x direction there will be tilting which means the body is unstable → CM = CG, GM = 0

If the fluid has higher density → more stable (BM increases → CM increases)

Experiment 3

Flow through a venturi meter

- Venturi meter: it is a convergent–divergent pipe used to measure flow rate of a fluid by gradually changing the cross sectional area and then measure Δhead
- Apparatus: Piezometer tubes are drilled into the wall at a number of points along the passage, at each point there is a manometer connected and attached to scale. The tubes are connected at their tops to a manifold and at which air is controlled by an air valve



$$\underbrace{\frac{P_1}{\rho g}}_{h_1} + \frac{u_1^2}{2g} + \cancel{z_1} = \underbrace{\frac{P_2}{\rho g}}_{h_2} + \frac{u_2^2}{2g} + \cancel{z_2}$$

Area is changing gradually and so energy losses are reduced meaning that it has a high efficiency

Calculations

$$\frac{u_1^2}{2g} + h_1 = \frac{u_2^2}{2g} + h_2 = \frac{u_n^2}{2g} + h_n$$

$$u_2^2 = \left(\frac{u_1^2}{2g} + h_1 - h_2\right)2g = u_1^2 + (h_1 - h_2)2g$$

$$Q = a_1 u_1 = a_2 u_2 \Rightarrow u_1 = \frac{a_2}{a_1} u_2$$

$$u_2^2 = \frac{a_2^2}{a_1^2} u_2^2 + (h_1 - h_2)2g$$

$$u_2^2\left(1 - \frac{a_2^2}{a_1^2}\right) = (h_1 - h_2)2g \longrightarrow u_2 = \sqrt{\frac{2g(h_1 - h_2)}{1 - \left(\frac{a_2}{a_1}\right)^2}}$$

$$\text{So } \underset{\sim}{\overset{Exp}{Q}} = \underset{\text{venturi meter coefficient}}{C} \; \underbrace{a_2 \sqrt{\frac{2g(h_1 - h_2)}{1 - \left(\frac{a_2}{a_1}\right)^2}}}_{\text{Ideal (calculated)}}$$

$$C = \frac{Q_{Exp}}{Q_{Ideal}} \qquad Q_{Exp} = \frac{12}{tP}$$

Always less than 1

Experimental pressure distribution

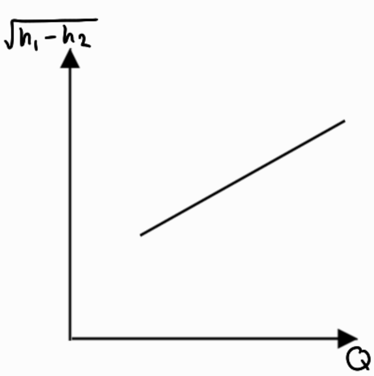
$$\frac{h_n - h_a}{u_2^2/2g}$$

Theoretical pressure distribution

$$\left(\frac{a_2}{a_1}\right)^2 - \left(\frac{a_2}{a_n}\right)^2$$

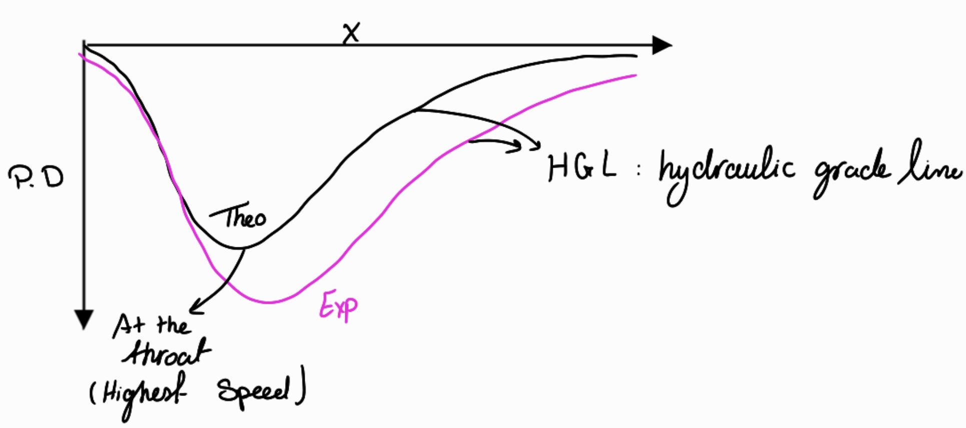
These are calculated at sections B, C, E, F, G, H, J, K, L

Results



$$\frac{1}{\text{slope}} = C \left(a_2 \sqrt{\frac{2g}{1 - \left(\frac{a_2}{a_1}\right)^2}} \right)$$

Pressure Distribution



If venturi was set vertically

$$\frac{u_1^2 - u_2^2}{2g} = (h_2 - h_1) + (z_1 - z_2)$$

$$\frac{Q^2\left(\frac{1}{a_1^2} - \frac{1}{a_2^2}\right)}{2g} = \Delta h + \Delta z$$

$$Q = \sqrt{\frac{2g(\Delta h + \Delta z)}{\left(\frac{1}{a_1^2} - \frac{1}{a_2^2}\right)}} = a_2\sqrt{\frac{2g(\Delta h + \Delta z)}{1 - \left(\frac{a_2}{a_1}\right)^2}} > Q_{\text{Horizontal}}$$

C is less since Q_{Theo} is encreased

pressure Distrubution

$$(h_n - h_1) = \frac{u_1^2 - u_n^2}{2g} - (z_n - z_1)$$

$$(h_n - h_1) = \frac{Q^2\left(\frac{1}{a_1}^2 - \frac{1}{a_n}^2\right)}{2g} - (z_n - z_1)$$

$u_2 a_2$

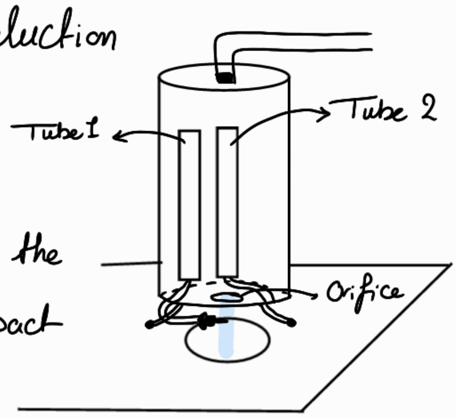
$$(h_n - h_1) + (z_n - z_1) = \frac{(u_2)^2\left(\left(\frac{a_2}{a_1}\right)^2 - \left(\frac{a_2}{a_n}\right)^2\right)}{2g}$$

$$\frac{(h_n - h_1) + (z_n - z_1)}{\frac{(u_2)^2}{2g}} = \left(\frac{a_2}{a_1}\right)^2 - \left(\frac{a_2}{a_n}\right)^2$$

Experiment 4

Discharge through an orifice meter

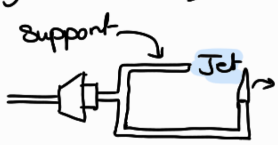
- The orifice meter is a device used to measure the flow rate using the pressure drop caused by diameter reduction



- **Apparatus**:

Tube 1: Connected to the pitot tube and it gives the total head of the jet (H_c) $\rightarrow$ Static pressure + impact pressure

Support at the beginning of the jet and the blade at the end



Blade moves by a lead screw of one thread (pitch/mm)
1 turn = 1 mm

Tube 2: Gives the head of the water above the orifice (H_o)

Calculations

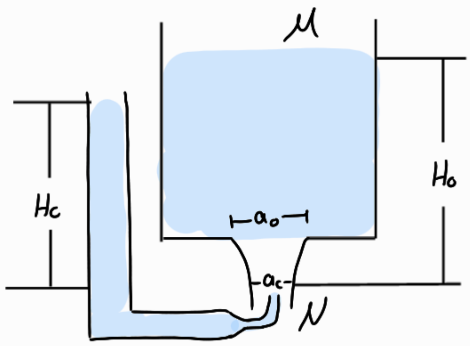
Bernoulli's equation:

$$\cancel{\frac{U_M^2}{2g}} + \frac{\cancel{P_M}^{\,0}}{\rho g} + Z_m = \frac{U_N^2}{\rho g} + \frac{\cancel{P_N}^{\,0}}{\rho g} + Z_N$$

$$Z_M - Z_N = \frac{U_N^2}{2g}$$

$$H_o = \frac{U_N^2}{2g}$$

$$H_c = \frac{U_c^2}{2g}$$



Assumptions of Bernoulli's equation:

1. Steady flow (V, P does not change)
2. Incompressible flow (ρ is constant)
3. Friction by viscous forces are neglected

Coefficients

Measured by knife

Coefficient of contraction = $C_c = \frac{a_c}{a_o} = \frac{d_c^2}{d_o^2}$ $\quad a_o$: No reduction $\quad C_c < 1$

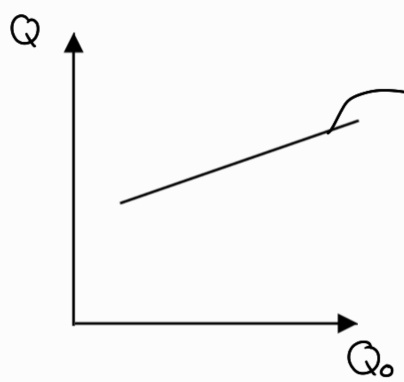
d_o: Given = 13

Coefficient of velocity = $C_u = \sqrt{\frac{H_c}{H_o}}$ $\quad C_u < 1$

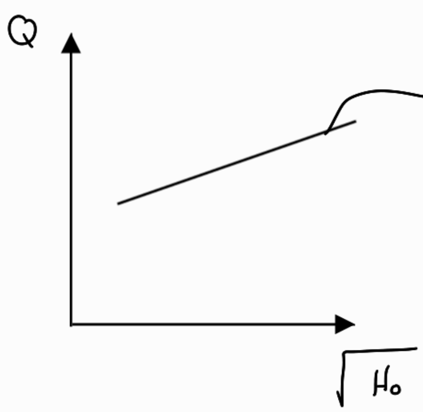
Coefficient of discharge $= C_d = \dfrac{Q}{Q_o} = \dfrac{\overset{\text{Actual}}{Q}}{\sqrt{2g}\, H_o\, a_o}$

$C_d = C_c \times C_v$

Results



Slope $= C_d$

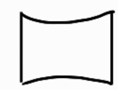


Slope $= \dfrac{Q}{\sqrt{H_o}} = C_d\left(\sqrt{2g}\; a_o\right)$

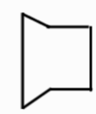
Note:

we used a sharp orifice

Sharp



Smooth



Conical

Experiment 5

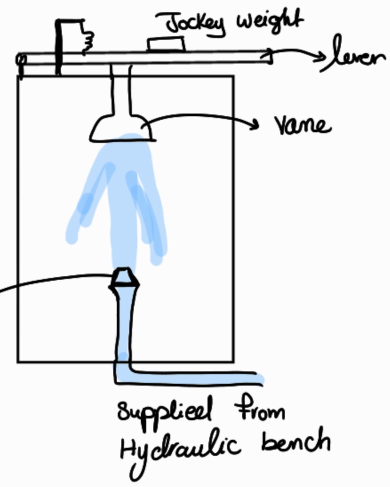
Impact of a Jet

Turbines are devices used to generate power, a fluid acts on the vanes and the jet causes torque on the vanes. (This is called a momentum exchange between high velocity jet and turbines vanes)

Types of turbines:

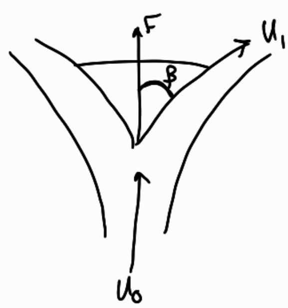
1. Impulse: water moves runner
2. Reaction: the runner is placed directly in the water stream

Apparatus



Vanes used: 1. Hemispherical cup

2. flat plate



Force on jet =

$F_j = \dot{m}\,(u_1 \cos\beta - u_0)$

Force on Vane =

$F = -F_j$

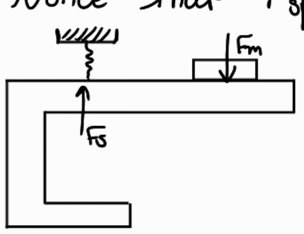
For flat plate $\beta = 90$

$F_p = -F_j = \dot{m}\,u_0$

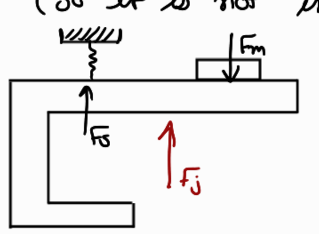
For Hemispherical cup $\beta = 180$

$F = -F_j = 2\dot{m}\,u_0$ ($u_1 = u_0$ when z, P are neglected)

Notice that $F_{spring} = F_{weight}$ (so it is not included)



No Jet



Jet is introduced

Calculations

$$u = \frac{\dot{m}}{\rho a_0} = \frac{m/t}{\rho a_0}$$

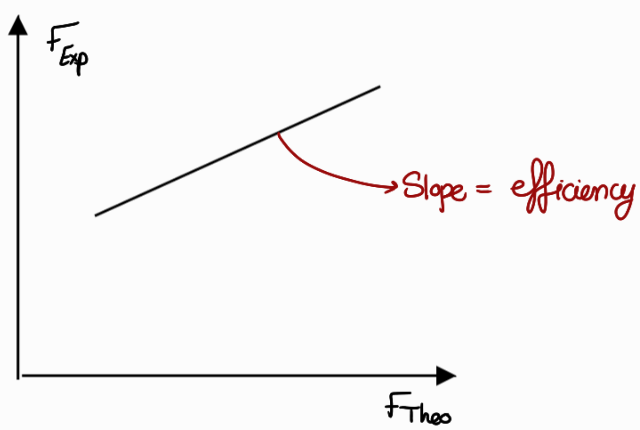
$$u_0 = u^2 - 2gs$$ height of vane above tip of Nozzle

Moment from weight = Moment from jet

$$(0.61)(g)(y) = 0.1525\, F_{Exp}$$ calculated from this equation

Mass of Jockey weight — measured — From vane to pivot lever

Efficiency of the vanes $= \dfrac{F_{C\,Exp}}{F_{C\,The}}$

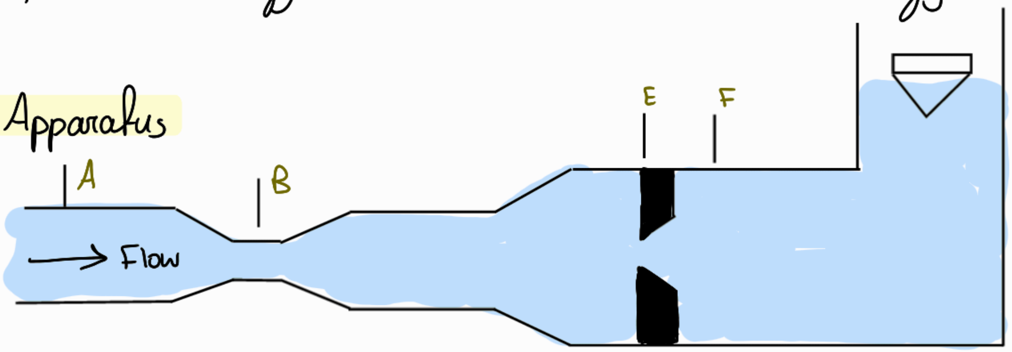


Experiment 6

Flow measuring apparatus

If a fluid flows through a tube and a change in cross sectional area occurs, the fluid's pressure changes and so does it's velocity
Potential energy is converted to kinetic energy but with losses.

Apparatus



Head is measured at A, B, E, F

$\dot{m} = u_A a_A \rho = u_B a_B \rho$

$\dot{m}_{actual} = \frac{m}{t}$

$\dot{m}_{venturi\ Exp} = \rho a_B \sqrt{\frac{2g}{1-\left(\frac{a_B}{a_A}\right)^2}(h_A - h_B)}$

$\dot{m}_{orifice\ Exp} = \rho a_F \sqrt{\frac{2g}{1-\left(\frac{a_F}{a_E}\right)^2}(h_E - h_F)}$

$Cd = \frac{\dot{m}_{actual}}{\dot{m}_{Exp}}$

Note

$Cd < 1$ Always

It elaborates losses occuring when energy is converted from potential to kinetic energy so energy obtained cannot be higher than energy given

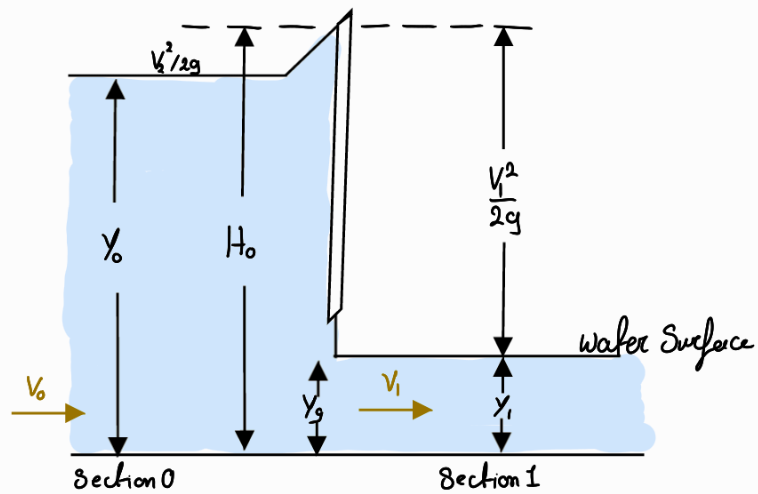
Orifice: Sudden contraction → high energy losses
Venturi: Gradual change in the area → less energy losses
Rotameter: best one (efficiency almost 100%)

Experiment 7

Discharge beneath a sluice gate

Sluice gates are used to control the flow of fluids in open channel
For free surface flow, the velocity distribution is uniform across the section and each fluid layer move at V $\longrightarrow$ velocity head indicated by the pitot tube for one layer of the fluid is assumed to be the same for every layer $\equiv$ Kinetic energy per unit weight of fluid

Calculations



Q (through the gate) $= C_d\, Y_g\, b\sqrt{2gY_0}$

$C_d = C_c C_v$ it is calculated to take friction losses in consideration

$C_v = \sqrt{\frac{H_c}{Y_0}}$ $\rightarrow$ Measured by Pitot tube

$C_c = \frac{Y_1}{Y_g}$

$Y_{0,1}$: fluid depth before and after gate

Y_g : gate opening

Velocity through the gate : $V = \frac{Q}{A}$

Forces on the gate :

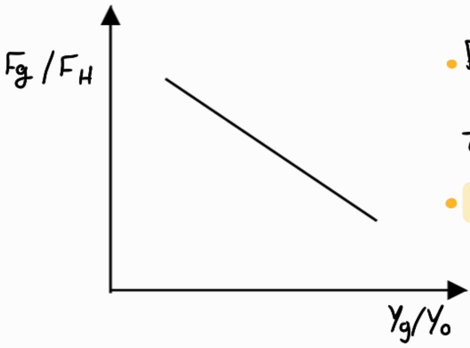
- Resultant gate thrust F_g : Reaction force that the gate exerts

$$F_g = \frac{1}{2}\rho g Y_1^2\left[\frac{Y_0^2}{Y_1^2} - 1\right] - \frac{\rho Q^2}{b^2 Y_1}\left[1 - \frac{Y_1}{Y_0}\right]$$

- Resultant hydrostatic thrust F_H : Reaction force that the fluid exerts

$$F_H = \frac{1}{2}\rho g (Y_0 - Y_g)^2$$

Results



- F_g keeps decreasing then becomes negative due to the hydraulic jump.
- Hydraulic jump occurs when a stream with high velocity strikes with a stream with lower velocity and that's because kinetic energy (velocity) converts to potential energy (water elevation)

F_H is always positive since it is always in direction of flow

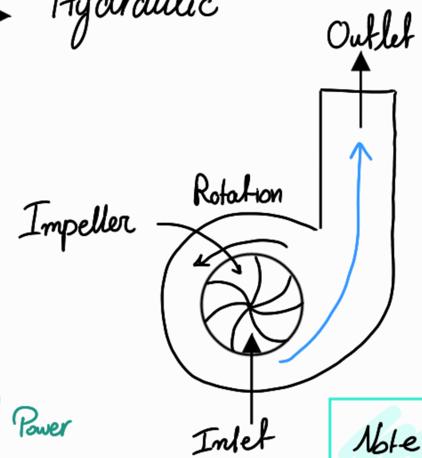
Experiment 8

Centrifugal Pump power measurements

A centrifugal pump is a device that transports mechanical rotational energy to hydraulic energy

It is driven by an electrical motor

$$\text{Electrical} \xrightarrow[\text{Impeller}]{} \text{Mechanical} \longrightarrow \text{Hydraulic}$$



Calculations

Efficiency:

$$\eta = \frac{\rho g Q H}{W_e} = \frac{\rho g Q H}{T\omega} \times \frac{T\omega}{W_e}$$

Hydraulic power

Mechanical Power

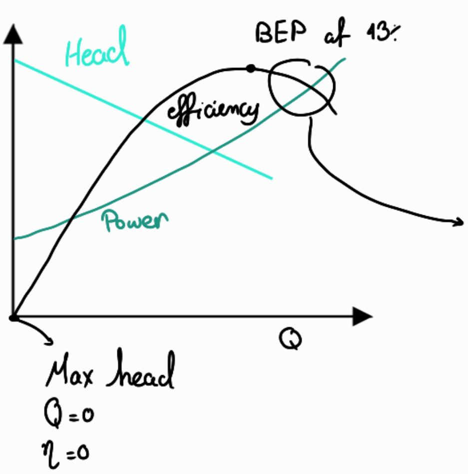
Electrical Power

Note

Flow rate is max at $Y_1 = \frac{2}{3} Y_0$ (See Report

$$W_m = \frac{2\pi}{60} N T \qquad N: 2500 \text{ RPM (constant)}$$

Results



- If the flow is increased it will need more electrical power to operate since this needs higher force (mechanical power)
- Q increases when head decreases
- Efficiency decreases due to turbulence which causes cavitation
- Cavitation: Vapor bubbles on the impeller that causes damages
- RPM
  - higher: I needed ↑, $P_{electrical}$ ↑, η ↓
  - lower: I needed ↑, $P_{electrical}$ ↑, η ↓

Experiment 9

Pressure losses in ductwork

The Ductwork is normally designed to pass fluid from point to point so that total pressure loss in the system is overcome by a pump or a fan system

Important parameter for design is pressure loss down the ductwork for a given flow

Calculations

$$\underbrace{\Delta P_{loss}}_{\text{loss in static pressure}} = \underbrace{f\,\frac{L_{eq}}{D}}_{\text{friction factor}} \underbrace{\left[\frac{1}{2}\rho V^2\right]}_{\text{Dynamic pressure (velocity pressure)}}$$

This is gained by a fan

Sections used

Highest pressure loss: Screen, straight duct, Orifice meter (lowest pressure loss)

Round elbow, venturi meter, heat bank

Right angle elbow.

Fluid velocity

$V = \frac{Q_v}{A}$ by venturi meter, $Q_v = 163.3\sqrt{h_v}$

$V = \frac{Q_o}{A}$ by orifice meter, $Q_o = 123.7\sqrt{h_o}$

mbar

$A = \frac{\pi D^2}{4}$, D constant

Equivalent length of a pipe:

The length of pipe of the same size as the pipe used that would give rise to the same pressure drop as the pipe chosen

يعني شو طول ال pipe اللي رح يعطيني نفس الخسارة اللي اعطاني ياها هاد النظام

- For the slow fan a longer pipe is needed to cause ΔP loss than for a fast fan

يعني معي مجال اقصر عشان اسبب لنفس الخسارة

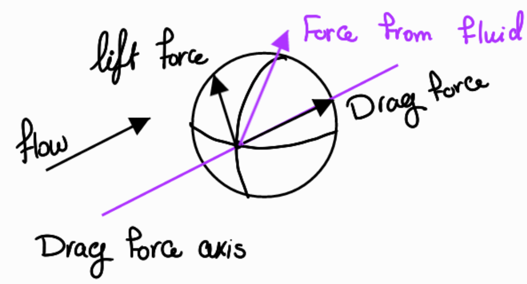
- $\Delta P_{orifice} > \Delta P_{venturi}$
- ΔP_{loss} for fast fan is higher because when V increases, collision between molecules increase and friction increases so more losses

Stream lines Separation

When an object is placed in a fluid flow it will experience a force exerted by the fluid. This force is divided into two components: Drag force in the direction of the flow and the lift force which is perpineluclar to the drag force

Drag force: considered to be a flow loss and the body needs to overcome it

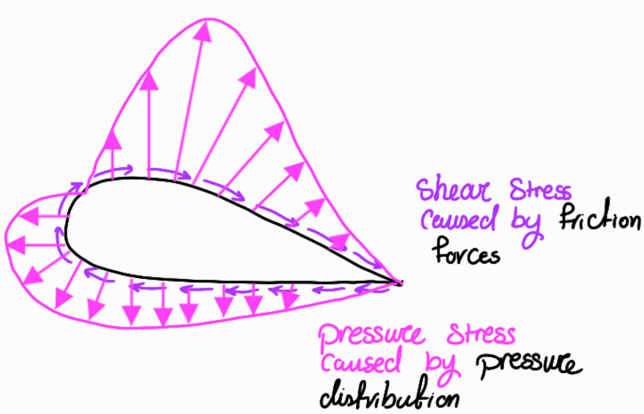
Lift force: provides the useful job and it opposites the weigh of the body.

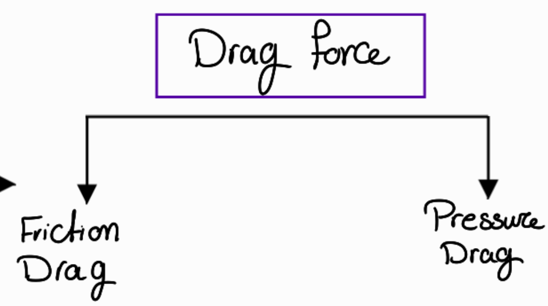


- In Aerodynamics the fluid flowing is a gas.

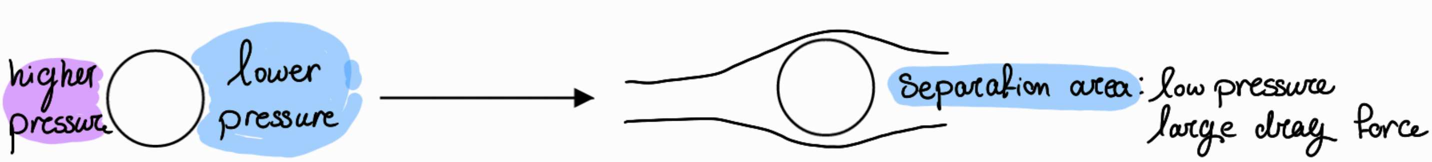
- It is important to reduce the drag force

Why is the drag force generated?





Pressure drag: Caused by differenc of pressure between front and rear of the object



Vortex are introduced causing vibrations and unstability

Drag coefficient C_D

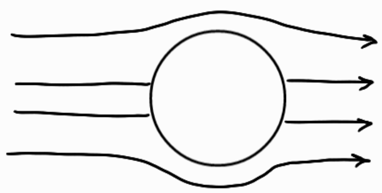
$C_D = \frac{D}{\frac{1}{2}\rho V^2 A}$ A: Characteristic area (in case of a sphere it is the frontal area)

D: Drag force

C_D is determined experimentally / Drag depends on: Velocity (Reynolds), Roughness, Shape of the object and Angle of atack

How is the drag affected by increasing Reynolds number by increasing flow speed?

① At low values of reynolds number $Re < 1 \rightarrow$ very slow flow (viscosity is neglected)

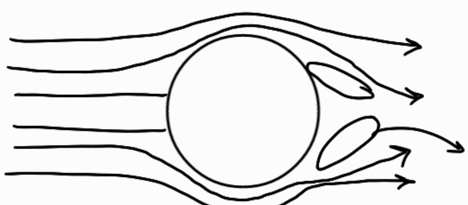


very low Reynolds – Ideal : no boundry layer and the flow is almost attached (no separation) and no viscous wak downstream of the cylinder

Flow is symmetric so no drag force is acting

This case does not exist since there is always small amount of viscosity and so drag

② At low values of reynolds number – Viscosity is not neglected

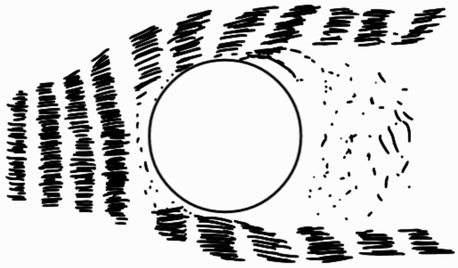


Vortices : these generate higher drag

The flow is steady but separated

③ At higher values of reynolds number

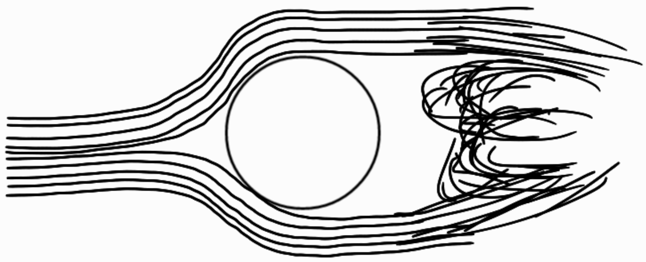
Velocity is increased $\rightarrow$ wake region is wide and generates high drag



The flow is unsteady and oscillating

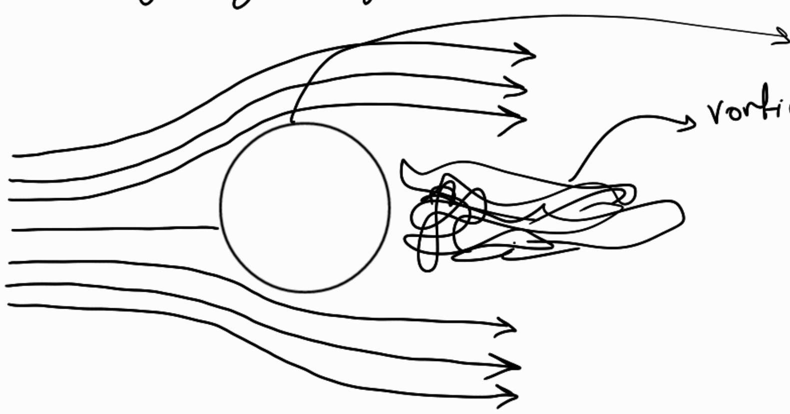
In this type a sound is produced which we hear when wind blows

④ At even higher values of reynolds number
wake is less than 3 and so is drag



The flow is laminar

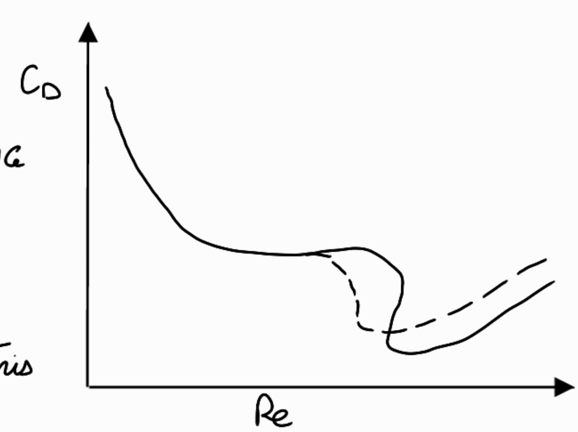
⑤ At very high reynolds number



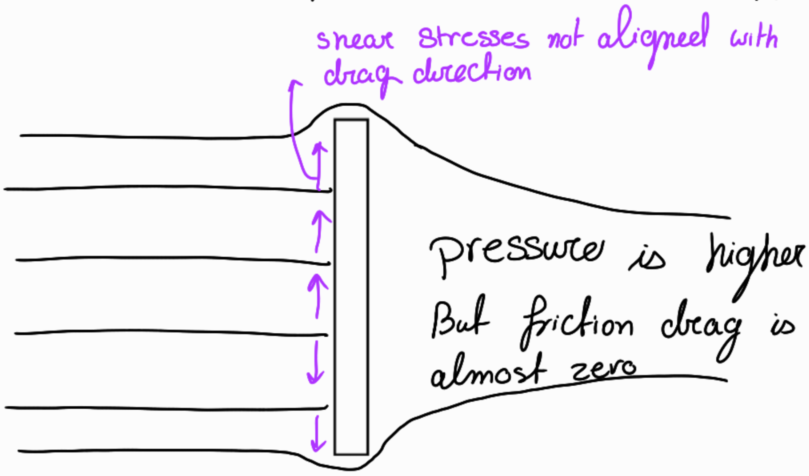
Chaotic turbulent flow, drag is less than in laminar since separation is delayed.
When speed is increased even more the drag ends to be higher than in laminar

What if the surface is not smooth?

For a rough surface, transition to turbulent flow will be at lower Re number
For example: a golf ball has dimples (rough) and so transition to turbulent flow is at lower speed since dimples generates turbulence and reduces pressure drag
This will delay separation and so drag is reduced (See cases 4 and 5). This allows the ball to go faster



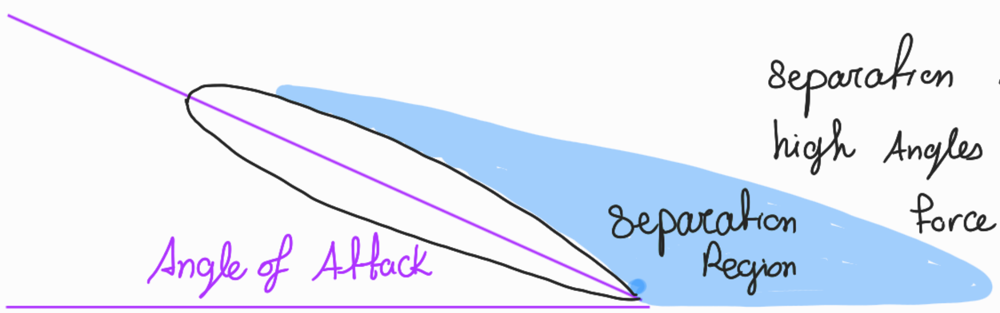
How does shape of the object affect the stream lines?



no separation Region so pressure drag is almost zero

But friction drag is high

How does angle of atack affect stream lines



separation occurs at high Angles and so drag force increases

why is airfoils used as a design for airplane wings?

pressure difference increase lift force and makes the plane fly