Important note!

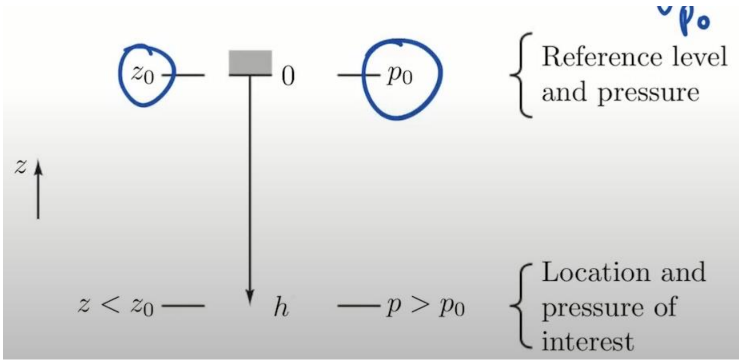
Når man tager krydsproduktet $\vec{V} \cdot \vec{A}$ så tager det højde for retningen af $\vec{V}$!! Så ved et outlet f.eks. får man positive $\vec{V} \cdot \vec{A} = VA$, så skal man stadig bare indsætte størrelsen af V og ikke retningen. Den retning ER DER taget højde for! Men skal man f.eks. have u af en vector som er negativ, så er $u = -V$ hvor V er størrelen af vektoreren! Med andre ord så vil u gerne have retning med

Lecture 2 - Fluid statics

Pressure variation in a static fluid

We have

$$p - p_0 = \rho g (z_0 - z)$$



The more commonly used is

$$\Delta p = \rho g h$$

Assumptions

- Static flow
- Incompressible flow

This is positive when going down! And positive when going up.

Fluid pressure in multiple liquid manometer

Multiple-Liquid Manometer:

1. Any two points at the same elevation in a continuous region of the same liquid are at the same pressure.
2. Pressure increase as one goes down a liquid column (pressure increase in a swimming pool).

$$\Delta p = g \sum_i \rho_i h_i$$

Hydrostatic force on submerged surfaces

How do we calculate forces from pressures?

One draws the coordinate system along the surface you want to calculate

The integral method

$$F_R = \int_A p \, dA$$

Where p is the pressure acting at the infinitesimal area

Location of force:

$$y' F_R = \int_A yp \, dA$$

Algebraic form:

$$F_R = p_c A$$

p_c = pressure at centroid of area

Her findet man midten som

$$y' = y_c + \frac{I_{\hat{x}\hat{x}}}{A y_c}$$

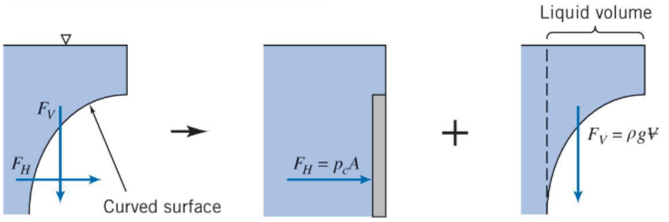
Midten i ”dybden” er ud fra geometrien i guess

$$x' = x_c + \frac{I_{\hat{x}\hat{y}}}{A y_c}$$

Curved sumberged surfaces:

The resultant hydrostatic force on a curved submerged surface is specified in terms of its components.

$$\boxed{F_H = p_c A \quad \text{and} \quad F_V = \rho g \forall}$$



p_c and A are the pressure at the center and the area of a vertical plane surface of the same projected area

$\forall$ is the volume of fluid above the curved surface

Det han mener, er hvis man ligesom kigger på ”siden af curven” - husk macdonald eksemplet

Bouyancy force

$$\boxed{F_{\text{buoyancy}} = \rho g \forall}$$

Assumptions

- Incompressible
- Static
- Submerged body (ikke en assumption, mere bare sådan et krav)

If an object is immersed in a fluid, the net vertical force acting on it due to fluid pressure is termed buoyancy

For a submerged body (can also be a bubble), the buoyancy force of the fluid is equal to the weight of the displaced fluid.

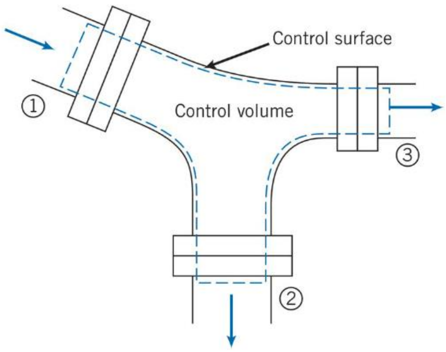
Lecture 3 - Control Volume analysis 1: Basic laws and mass conservation

System and control volume

In fluid, we don't do systems we do control volumes. The formal definition of a control volume is:

- A control volume is an arbitrary volume in space through which fluid flows. The geometric boundary of the control volume is called the control surface. The control surface may be real or imaginary. It may also be a t rest or in motion.

En example of a control volume:



For a control volume one can express the rate of change of a property in a control volume using the **Reynolds transport theorem**

$$\left.\frac{dN}{dt}\right)_{\text{system}} = \frac{\partial}{\partial t}\int_{\text{CV}} \eta\,\rho\,d\forall + \int_{\text{CS}} \eta\,\rho\vec{V}\cdot d\vec{A}$$

Depending on what you want, replace N and η accordingly

$$\begin{aligned} N &= M,\ \eta = 1 \\ N &= \vec{P},\ \eta = \vec{V} \\ N &= \vec{H},\ \eta = \vec{r}\times\vec{V} \\ N &= E,\ \eta = e \\ N &= S,\ \eta = s \end{aligned}$$

Lets break down the terms.

$\frac{dN}{dt}$ = the rate of change of the system extensive property N

$\frac{\partial}{\partial t}\int_{CV}\eta\rho d\forall$ = The rate of change of N inside the control volume that is caused by time.

$\int_{CS}\eta\rho\,\vec{V}\cdot d\vec{A}$ = The net flux of N across the control surface. This accounts for the movement of N in and out of the control volume due to fluid flow.

Conservation of mass:

This is the case where in Reynolds ttransport theorem we get $N = M => \eta = 1$. Conservation of mass / mass continuity is given by

$$\boxed{\frac{\partial}{\partial t}\int_{\mathrm{CV}} \rho \, d\forall + \int_{\mathrm{CS}} \rho \vec{V} \cdot d\vec{A} = 0}$$

For uniform flow one uses a sum for the control surface integral (Assume uniform flow)

$$\boxed{\frac{\partial}{\partial t}\int_{\mathrm{CV}} \rho \, d\forall + \sum_{\mathrm{CS}} \rho \vec{V} \cdot \vec{A} = 0}$$

Note: Uniform flow means the means that for any given cross section, the velocity and other flow properties are the same for the entire cross section.

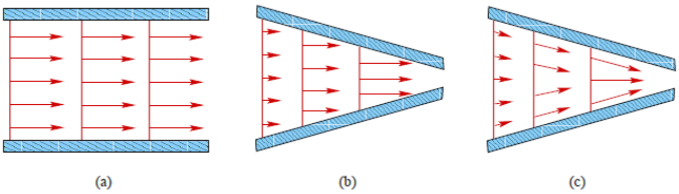
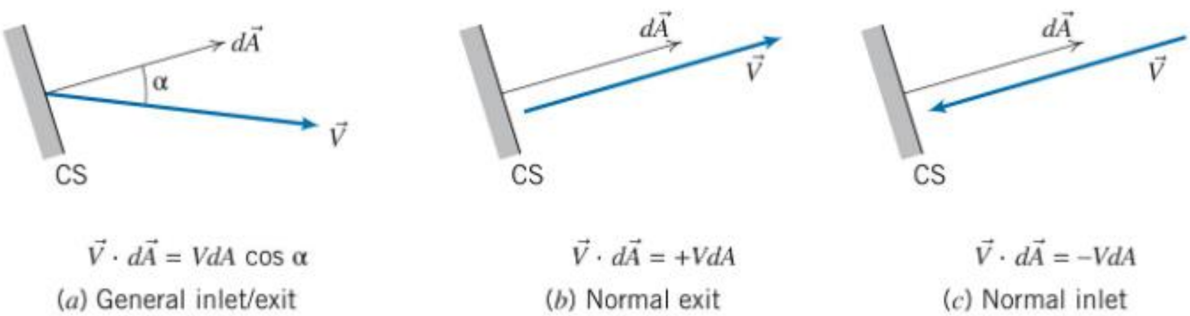


Figure 2.15: Uniform and non-uniform flows; (a) uniform flow, (b) non-uniform, but "locally uniform" flow, (c) non-uniform flow.

The continuity equation basicaly states, that the rate of change of mass in the control volume plus the net out flow must be 0.

When taking this cross product in the integral, one has to account for if its an inlet or an outlet. An inlet yields a negative result, and outlet a positive one.



There are several special cases of the continuity equation.

Fixed control volume incompressible flow (even though its unsteady) since $\rho = const$ the rate of change of mass through the CV is 0)

This means that the volume of the density doesn't change over time. We then get

$$\int_{CS} \vec{V} \cdot d\vec{A} = 0$$

Steady flow (compressible)

$$\int_{CS} \rho \vec{V} \cdot d\vec{A} = 0$$

Remember!: When there is uniform flow at each inlet and outlet, these two also become sums instead of integrals!

Another ting

The volume flow rate is given as:

$$\boldsymbol{Q} = \vec{\boldsymbol{V}} \cdot \vec{\boldsymbol{A}}$$

Lecture 4 - Control volume analysis 2: Momentum & Energy equation

Momentum equation for nonaccelerating control volume

I will present the equation, and then I will explain what each part is

$$\boxed{\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{\mathrm{CV}} \vec{V}\,\rho d\forall + \int_{\mathrm{CS}} \vec{V}\,\rho \vec{V}\cdot d\vec{A}}$$

$\vec{F}_S$ denotes surface forces. (forces from pressure, reaction forces, external forces and so on)

$\vec{F}_B$ denotes body forces (Typically just gravity). From the slides its explained as

These two can be calculated as

$$\vec{F}_B = \int_{\mathrm{CV}} \rho\vec{g}\,d\forall = \vec{W}_{\mathrm{CV}} = M\vec{g}\,,\ \vec{F}_S = \int_A -p\,d\vec{A}$$

Minus sign because pressure forces act ***onto*** CS!

Momentum equation for nonaccelerating control volume (Uniform flow)

$$\boxed{\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t}\int_{\mathrm{CV}} \vec{V}\,\rho d\forall + \sum_{\mathrm{CS}} \vec{V}\,\rho \vec{V}\cdot \vec{A}}$$

Recall! $\dot{m} = \rho\vec{V}\cdot d\vec{A}$ (Assume uniform flow)

In component form this equation becomes

$$\boxed{\begin{aligned}
F_x &= F_{S_x} + F_{B_x} = \frac{\partial}{\partial t}\int_{\mathrm{CV}} u\,\rho\,d\forall + \int_{\mathrm{CS}} u\,\rho\vec{V}\cdot d\vec{A}\\
F_y &= F_{S_y} + F_{B_y} = \frac{\partial}{\partial t}\int_{\mathrm{CV}} v\,\rho\,d\forall + \int_{\mathrm{CS}} v\,\rho\vec{V}\cdot d\vec{A}\\
F_z &= F_{S_z} + F_{B_z} = \frac{\partial}{\partial t}\int_{\mathrm{CV}} w\,\rho\,d\forall + \int_{\mathrm{CS}} w\,\rho\vec{V}\cdot d\vec{A}
\end{aligned}}$$

Remember to always reduce your equations to something that's easier by using assumptions

Momentum equation for a CV with rectilinear acceleration

$$\boxed{F_{S_x} + F_{B_x} - \int_{\mathrm{CV}} a_{rf_x}\,\rho d\forall = \frac{\partial}{\partial t}\int_{\mathrm{CV}} u_{xyz}\,\rho d\forall + \int_{\mathrm{CS}} u_{xyz}\,\rho\vec{V}_{xyz}\cdot d\vec{A}}$$

(The last term is what is known as thrust)

The control volume is moving, so we need to account for that. XYZ denotes the absolute coordinates of the system, whereas xyz denotes the coordinate if one is "standing" on the controlvolume. In component form

$$F_{S_x} + F_{B_x} - \int_{\mathrm{CV}} a_{rf_x} \rho \, d\forall = \frac{\partial}{\partial t} \int_{\mathrm{CV}} u_{xyz} \rho \, d\forall + \int_{\mathrm{CS}} u_{xyz} \rho \vec{V}_{xyz} \cdot d\vec{A}$$

$$F_{S_y} + F_{B_y} - \int_{\mathrm{CV}} a_{rf_y} \rho \, d\forall = \frac{\partial}{\partial t} \int_{\mathrm{CV}} v_{xyz} \rho \, d\forall + \int_{\mathrm{CS}} v_{xyz} \rho \vec{V}_{xyz} \cdot d\vec{A}$$

$$F_{S_z} + F_{B_z} - \int_{\mathrm{CV}} a_{rf_z} \rho \, d\forall = \frac{\partial}{\partial t} \int_{\mathrm{CV}} w_{xyz} \rho \, d\forall + \int_{\mathrm{CS}} w_{xyz} \rho \vec{V}_{xyz} \cdot d\vec{A}$$

Jeg har lavet et raket eksempel, cool nok eksempel. Forstået meget lidt, men det er i bogen.

Igen -> antag ting

Conservation of angular momentum (for inertial CV)

$$\boxed{\vec{r} \times \vec{F}_s + \int_{\mathrm{CV}} \vec{r} \times \vec{g} \, \rho \, d\forall + \vec{T}_{\mathrm{shaft}} = \frac{\partial}{\partial t} \int_{\mathrm{CV}} \vec{r} \times \vec{V} \rho \, d\forall + \int_{\mathrm{CS}} \vec{r} \times \vec{V} \rho \vec{V} \cdot d\vec{A}}$$

What does it all mean? We get

Angular-momentum principle for a **nonaccelerating (inertial) CV**: The total torque exerted on a system due to surface and body forces and shaft torque acting on the CV is equal to the rate of change of angular momentum within the CV and the net rate of flux of angular momentum from the CV through the CS.

$\vec{r}$ = position vector for each volume element with respect to coordinate system

$\int \vec{r} \times \vec{g} d\forall$ = the moment of the body forces acting on the control volume

$\vec{T}_{shaft}$ = External torques or shaft moments. For example the torque from a pump, turbine or other mechanical systems interacting with the CV:

$\frac{\partial}{\partial t} \int_{CV} \vec{r} \times \vec{V} \rho d\forall$ = Time rate of change of angular momentum within the control volume

$\int_{CS} \vec{r} \times \vec{V} \rho \vec{V} \cdot d\vec{A}$ = the net flux angular momentum across the control surface.

For uniform velocity (if one assumes that)

$$\boxed{\vec{F} = \vec{F}_S + \vec{F}_B = \frac{\partial}{\partial t} \int_{\mathrm{CV}} \vec{V} \rho d\forall + \sum_{\mathrm{CS}} \vec{V} \rho \vec{V} \cdot \vec{A}}$$

Lecture 5 - Differntial analysis 1: Mass conservation & kinematics

Key difference between integral equations and differential forms:

Integral equations are useful when we are interested in the gross behaviour of a flow field and its effect on various devices

Differential forms of the equations of motion are needed to analyse the local details of a flow

Differential mass conservation

$$\boxed{\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} + \frac{\partial \rho}{\partial t} = 0}$$

Alternatively written as

$$\boxed{\nabla \cdot \rho \vec{V} + \frac{\partial \rho}{\partial t} = 0}$$

For incompressible flows (density does not change with time)

$$\boxed{\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \nabla \cdot \vec{V} = 0}$$

For steady flows

$$\boxed{\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = \nabla \cdot \rho \vec{V} = 0}$$

Stream function for 2D incompressible flow

$$d\psi = \frac{\partial \psi}{\partial x} dx + \frac{\partial \psi}{\partial y} dy$$

Assume: Incompressible flow & 2D flow

This automatically satisfies mass conservion. Best explained by an example:

Example: Obtain the stream function for the following velocity field

$$\vec{V} = 2y\hat{\imath} + 4x\hat{\jmath}$$

Solution:

$$u \equiv \frac{\partial \psi}{\partial y} \quad \text{and} \quad v \equiv -\frac{\partial \psi}{\partial x}$$

$$\left.\begin{aligned} u = \frac{\partial \psi}{\partial y} &= 2y \Rightarrow \psi = y^2 + f(x) \\ v = -\frac{\partial \psi}{\partial x} &= 4x \Rightarrow \psi = -2x^2 + g(y) \end{aligned}\right\} \quad \begin{aligned} g(y) &= y^2 \\ f(x) &= -2x^2 \end{aligned}$$

$$\Longrightarrow \psi = -2x^2 + y^2$$

Lecture 6 - Differntial analysis 2: Kinematics & Navier-Stokes equation

Total acceleration of a fluid particle in a velocity field

Can e used to find acceleration of fluid particle

$$\boxed{\frac{D\vec{V}}{Dt} \equiv \vec{a}_p = u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z} + \frac{\partial \vec{V}}{\partial t}}$$

This is called the substantial derivative to remind us that it is computed for a substance.

How is this to be understood?

$$\underset{\substack{\text{total}\\\text{acceleration}\\\text{of a particle}}}{\vec{a}_p = \frac{D\vec{V}}{Dt}} = \underbrace{u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z}}_{\substack{\text{convective}\\\text{acceleration}}} + \underset{\substack{\text{local}\\\text{acceleration}}}{\frac{\partial \vec{V}}{\partial t}}$$

Convective acceleration is also what is present in steady flow. Its when particles are cnvected toward a low velocity region like a corner, and then away to a high acceleration.

If the flow field is undsteady, then the flow will change with time meaning the particle undergoes some local acceleration, since the velocity field is a function of time. In scalar components this function is

$$a_{x_p} = \frac{Du}{Dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t}$$
$$a_{y_p} = \frac{Dv}{Dt} = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t}$$
$$a_{z_p} = \frac{Dw}{Dt} = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} + \frac{\partial w}{\partial t}$$

For 2D:

$$\frac{D\vec{V}}{Dt} = u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + \frac{\partial \vec{V}}{\partial t}$$

For steady flow:

$$\frac{D\vec{V}}{Dt} = u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z}$$

Rotation of a fluid

A fluid particle is in general in a 3D flow field may rotate about all three coordinate axes. The particle rotation is a vector quantity and is given as

$$\vec{\omega} = \hat{\imath}\omega_x + \hat{\jmath}\omega_y + \hat{k}\omega_z$$

Rotation will always occur for fluids in which we have shear stresses, which is present in every viscous fluid. So rotation only occurs in viscous flows unless the particles are initially rotating.

Positive is CCW, ω_x denotes rotation about the x-axis. The rotation can be calculated directly from the curl of the velocity field

$$\boxed{\vec{\omega} = \frac{1}{2}\underbrace{\left[\hat{i}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) + \hat{j}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) + \hat{k}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\right]}_{\text{curl } \vec{V} = \nabla \times \vec{V}}}$$

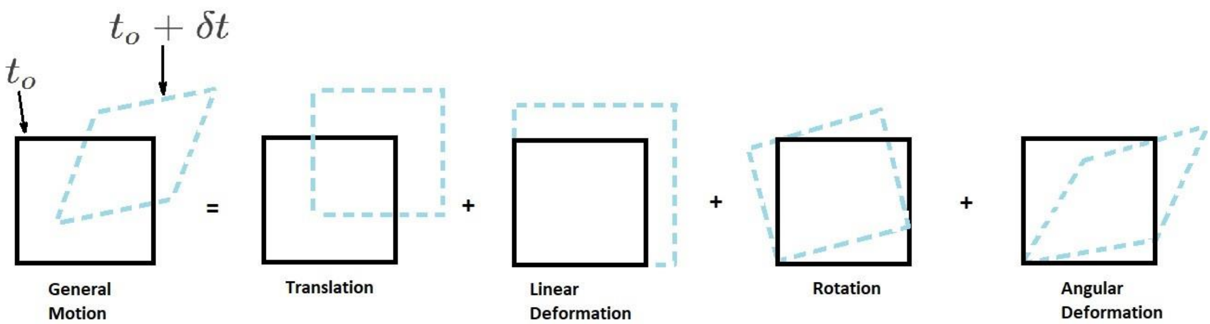
There is also the quantity known as "Vorticity" defined as

$$\vec{\zeta} \equiv 2\vec{\omega} = \nabla \times \vec{V}$$

It's a measure of the rotation of a fluid element as it moves in the flow field.

Deformation

There are to tipes of deformation. Linear and angular. These can be seen here



We can calculate the rates of **angular deformation** as

$$\text{Rate of angular deformation in the } xy\text{-plane} = \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)$$

$$\text{Rate of angular deformation in the } yz\text{-plane} = \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right)$$

$$\text{Rate of angular deformation in the } xz\text{-plane} = \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)$$

Linear deformation is when the angles in the fluid element remains unchanged. The element will change length in the $x, y, or\ z$-direction only if $\frac{\partial u}{\partial x}, \frac{\partial v}{\partial y}, \frac{\partial w}{\partial z}$ are other than zero.

Changes in length of the sides may change the volume of the element. For this once can calculate the volume dilation rate.

$$\text{Volume dilation rate} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \nabla \cdot \vec{V}$$

For **incompressible flows this volume dilation rate is 0!!**

The momentum equation (The navier stokes equation)

Involves particle acceleration, pressure gradient and viscous terms.

General navier stokes:

x –direction:

$$\rho \frac{Du}{Dt} = \rho g_x - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left[\mu\left(2\frac{\partial u}{\partial x} - \frac{2}{3}\nabla \cdot \vec{V}\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right] + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right]$$

y –direction:

$$\rho \frac{Dv}{Dt} = \rho g_y - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(2\frac{\partial v}{\partial y} - \frac{2}{3}\nabla \cdot \vec{V}\right)\right] + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)\right]$$

z –direction:

$$\rho \frac{Dw}{Dt} = \rho g_z - \frac{\partial p}{\partial z} + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial w}{\partial z} + \frac{\partial w}{\partial y}\right)\right] + \frac{\partial}{\partial z}\left[\mu\left(2\frac{\partial w}{\partial z} - \frac{2}{3}\nabla \cdot \vec{V}\right)\right]$$

Navier stokes for Newtonian fluid **(Incompressible with constant viscosity) This is the one we use!!**

x −direction: $$\boxed{\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)}$$

y −direction: $$\boxed{\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=\rho g_y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)}$$

z −direction: $$\boxed{\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\rho g_z-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)}$$

Vector notation: $$\boxed{\rho\frac{\partial\vec{V}}{\partial t}+\rho(\vec{V}\cdot\nabla)\vec{V}=\rho\vec{g}-\nabla p+\mu\nabla^2\vec{V}}$$

This along with differential continuity can be used to solve a lot of shit

Shear stress distrubtion -> Newtons law of viscosity

Euler and Bernoulli equation (Incompressible Inviscid flow)

Navier stokes for frictionless and inviscid flow (and imcompressible) (Momentum equation)

Includes pressure gradient and acceleration. Recall $\frac{D\vec{V}}{Dt} = \vec{a}_p$

$$\boxed{\rho \frac{D\vec{V}}{Dt} = \rho\vec{g} - \nabla p}$$

This is eulers equation. It states that for an inviscid flow, the change in momentum of a fluid particle is caused by the body force (assumed to be only gravity) and the net pressure force.

Assumptions

- Frictionless flow (Inviscid fluid)
- Only body force is gravity

In component form:

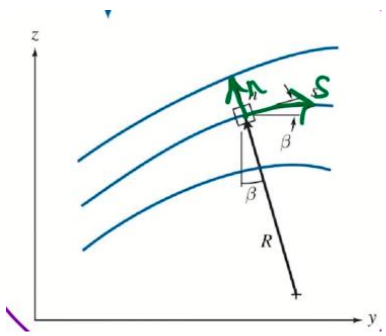
$$x\text{-direction:} \quad \rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \rho g_x - \frac{\partial p}{\partial x}$$

$$y\text{-direction:} \quad \rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \rho g_y - \frac{\partial p}{\partial y}$$

$$z\text{-direction:} \quad \rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = \rho g_z - \frac{\partial p}{\partial z}$$

Eulers equation in streamline coordinates.

Remember, the streamline direction is the direction the flow is flowing. The normal direction is the direction tangent



The euler equation along the streamline is

Along the streamline;

$$\boxed{-\frac{1}{\rho}\frac{\partial p}{\partial s}-g\frac{\partial z}{\partial s}=\frac{\partial V}{\partial t}+V\frac{\partial V}{\partial s}}$$

Normal to the streamline:

$$\boxed{\frac{1}{\rho}\frac{\partial p}{\partial n}+g\frac{\partial z}{\partial n}=\frac{V^2}{R}}$$

Now the ones **we'll be using** are the ones neglecting body forces for steady flow

For streamline direction:

$$\boxed{\frac{1}{\rho}\frac{\partial p}{\partial s}=-V\frac{\partial V}{\partial s}}$$

For normal direction:

$$\boxed{\frac{1}{\rho}\frac{\partial p}{\partial n}=\frac{V^2}{R}}$$

What is this telling us?

For the streamwise direction

- The change in pressure in the s direction is accompanied by a change in velocity but with a negative. So if pressure increases along streamline, then velocity decreases and vice versa.
- I.E Flow accelerates towards low pressure region, and decelerates

For the normal direction

- Since R is curvature, the change in pressure in the normal direction decreases for straighter lines I.E, if we have a straight line, then the curvature is infinity and then we have no change in pressure in the normal direction. So there is no pressure change normal to straight streamlines.

Bernoulli's equation

Its basically an energy equation

What assumptions are to be used?

- Flow along a streamline
- Steady flow
- Inviscid flow
- Incompressible flow

$$\boxed{\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant}}$$

Or

$$\boxed{\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2}$$

(The p in this equation is the static pressure btw)

Static, stagnation and dynamic pressures

Static pressure

- The pressure experienced by a fluid particle as it moves

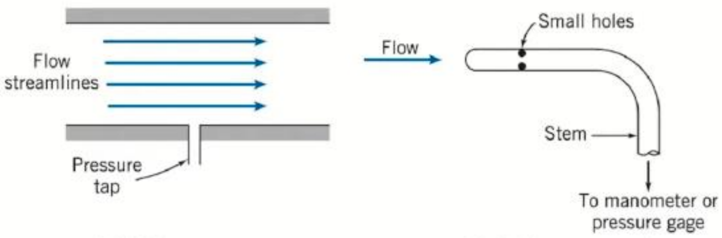
Stagnation pressure

- The pressure obtained when a flowing fluid is decelerated to zero

Dynamic pressure

- Pressure due to the flow velocity. The difference between stagnation and static pressure

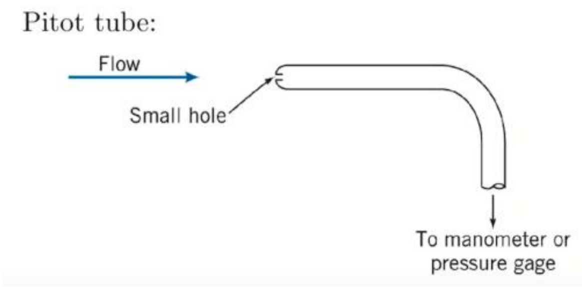
Examplified static pressure



(a) Wall pressure tap (b) Static pressure probe

Two difference ways to measure static pressure. You basically drill a hole, and the pressure you can measure from that, is the static pressure, as the flow has to move

Examplified stagnation pressure



This is the pressure you get when the flow is decelerated to 0. So one would have a hole facing the flow, that's causing the flow to decelerate.

How to calculate?

Static pressure same as before

Stagnation pressure:

$$\boxed{p_0 = p + \frac{1}{2}\rho V^2}$$

The latter term $\frac{1}{2}\rho V^2$ is called the dynamic pressure. p is static pressure, p_0 is the stagnation

Pressure.

Isolated for velocity:

$$\boxed{V = \sqrt{\frac{2(p_0 - p)}{\rho}}}$$

Again to use these

- Flow along a streamline
- Steady flow
- Inviscid flow
- Incompressible flow

Energy Grade Line and Hydraulic Grade line

Head of flow:

Dividing the Bernoulli equation by g gives

$$\frac{p}{\rho g} + \frac{V^2}{2g} + z = H$$

This is the total head of the flow measured in $meters$. The total head represents the TOTAL energy of the flow. So its also known as the EGL (energy grade line)

$$\boxed{EGL = \frac{p}{\rho g} + \frac{V^2}{2g} + z}$$

(Height of column) (Stangation pressure) (Pitot)

We also have the the hydraulic gradeline

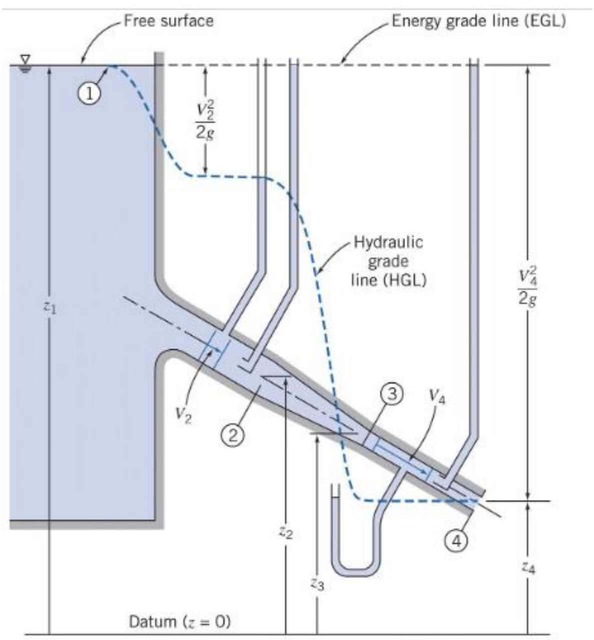
$$\boxed{HGL = \frac{p}{\rho g} + z}$$

This does not account for dynamic pressure (Height for static pressure)

The dynamic pressure term is then

$$\boxed{EGL - HGL = \frac{V^2}{2g}}$$

This is a visualization tool basically. Lets look at it



At point 1 we don't have any velocity, so $EGL = HGL$

Going to point 2 first some of our pressure and potential energy gets converted to both pressure and kinetic energy, so we the HGL drop. Then it statys constant for a bit, until the pipe starts to get smaller on its way to point 3, where the decrease in area means and increase in velocity, and thus a decrease in pressure. So the HGL falls. It then remains flat again. Its flat because the z term is both in HGL and EGL

Lecture 8 - Bernoulli continued and irrotational flow

Meget af det får faktisk under lectureren før

Eulers as energy equation

$$\boxed{\underbrace{\frac{p}{\rho}}_{\text{Pressure energy}} + \underbrace{\frac{V^2}{2}}_{\text{Kinetic energy}} + \underbrace{gz}_{\text{Potential energy}} = \text{constant}}$$

FLUID MECHANICS | ULRICH DOLL

Bernoullis equation for unsteady flow

Bernoulli no longer holds when you have an unsteady flow, but one can show what there is an **Bernoulli equation for Unsteady flows** given as

Unsteady Bernoulli Equation

$$\boxed{\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2 + \int_1^2 \frac{\partial V}{\partial t} ds}$$

Assumptions are

- Inviscid flow
- Flow along a streamline
- Incompressible flow.

The term $\int_1^2 \frac{\partial V}{\partial t} ds$ accounts for the unsteady flow component. $\frac{\partial V}{\partial t}$ is the local acceleration of the fluid, ds is an infitesimal distance along the streamline between points 1 and 2. The integral sums the effect of unsteady acceleration along the streamline between the two points.

Rotational flow

Note about bernoullis equation

In an irrotational flow, Bernoullis equation is valid between any two points (not just along a streamline) if flow is also steady compressible & inviscid

Lecture 9 - Dimensional analysis.

Noter I onenote

Lecture 10/11 : Internal incompressible Vicous flows: Fully developed flows (Laminar) + Flows in pipes and ducts

Internal = Flows completely bounded

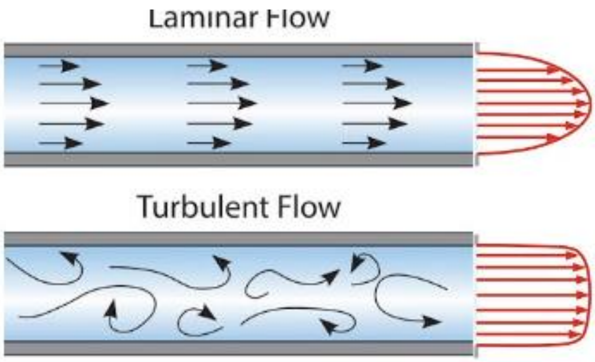
Incompressible = $\rho = const$ (Liquids / gasses with no big change in temp, or a $M > 0.3$ (mach number

Viscous = We have viscosity. Shear stresses are present. We're going to have friction.

We're going to be working a lot with Reynolds numbers!

$$Re = \frac{\rho \bar{V} D}{\mu}$$

Laminar vs turbulent flows



Laminar flow:

Layed flow. No mixing along the layers other than molecular diffusion

Turbulent flow

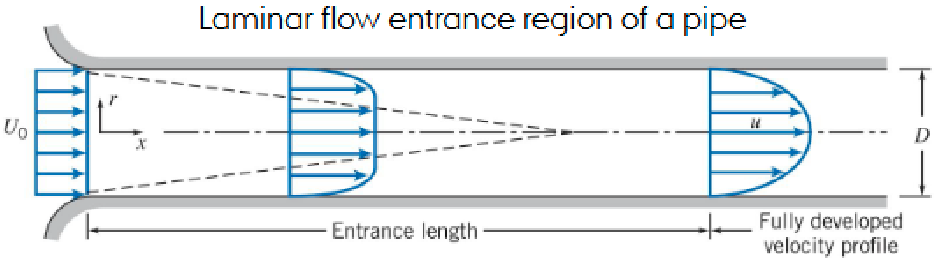
- Mixing due to eddies or coherent structures
- $\vec{V}(x,y,z,t) = (\bar{u} + u')\hat{\imath} + (\bar{v} + v')\hat{\jmath} + (\bar{w} + w')\hat{k}$

The Reynolds number is an indicator for when the transition to turbulence occurs.

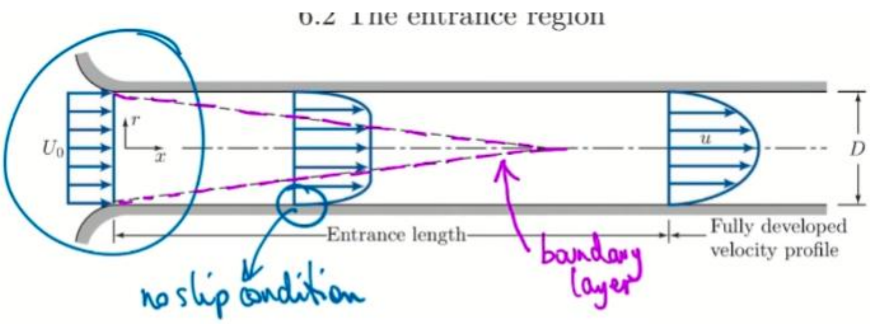
Pipes (Continued further down)

Transition to turbulence occurs at $Re \approx 2300$

If we look at the entrance region of a pipe



- Uniform velocity at entrance
- No-slip condition at the walls
- Developing boundary layer slows fluid near the surface (This devoloping boundary layer is marked with purple in drawing below
- The flow is **fully** developed when the profile stops changing



The **Entrance length L_e** is the distance from inlet to where the flow becomes fully developed.

The **laminar flow entrance length for a pipe** is approximately $\boldsymbol{L_e = 138D}$ $(80D\ for\ turbulent)$

We have the relation

$$\frac{L_e}{D} \approx 0.06 \underbrace{\frac{\rho \bar{V} D}{\mu}}_{Re}$$

For the flow above then $U_0 = \bar{V}$

Fully developed flow between infinitely parallel plates

- Simplest geometry
- Applies to small gaps.
- $u = u(y)$. Velocity only changes when going up and down between the plates

Assumptions:

These apply to all formulas for fully developed flow between infinite parallel playes

- Steady flow $\frac{\partial}{\partial t} = 0$
- Fully developed flow $\frac{\partial u}{\partial x} = 0$ (flow doesn't change in direction of flow
- Incompressible fluid
- 2D flow in $x - y$ plane (consequence of infinte length in z)
- Gravity in y-direction
- Laminar flow!!

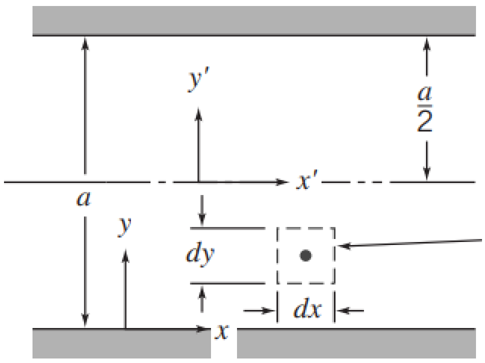
Reynolds number for parralel playes

$$Re = \rho\bar{V}(2\delta)/\mu,$$

$$2\delta = the\ height\ of\ the\ plates\ (or\ a\ on\ the\ drawing\ below)$$

Both plates stationary

The geometry of the plates:



One can reduce the navier stokes equation with a lot of assumptions, then integrate twice to get an expression for the velocity u

$$\underbrace{\frac{\partial p}{\partial x}}_{f(x)} = \underbrace{\mu \frac{d^2u}{dy^2}}_{f(y)}$$

Since they're functions of two different variables, they must both be constant to. Integration twice and then using boundary conditions that $u = 0$ at $y = a$ and $y = 0$. (a is the height of the channel)

$$u = \frac{1}{2\mu}\left(\frac{dp}{dx}\right)y^2 - \frac{1}{2\mu}\left(\frac{dp}{dx}\right)ay = \frac{a^2}{2\mu}\left(\frac{dp}{dx}\right)\left[\left(\frac{y}{a}\right)^2 - \left(\frac{y}{a}\right)\right]$$

And remember! $\frac{dp}{dx}$ is constant! So that's something we can figure out.

We can then find the **shear stress distribution for fully developed flow between parallel plates**.

We know $\tau_{yx} = \mu\frac{du}{dy}$

$$\tau_{yx} = \left(\frac{dp}{dx}\right)y + c_1 = \left(\frac{dp}{dx}\right)y - \frac{1}{2}\left(\frac{dp}{dx}\right)a = a\left(\frac{dp}{dx}\right)\left[\frac{y}{a} - \frac{1}{2}\right]$$

Flow rate between stationary infinitely parallel plates (Per unit depth)

$$\frac{Q}{l} = -\frac{1}{12\mu}\left(\frac{dp}{dx}\right)a^3$$

Where l is the depth in the z direction (so into the picture above)

Since $\frac{\partial p}{\partial x}$ is constant, We then have

$$\frac{\partial p}{\partial x} = \frac{p_2 - p_1}{L}$$

So now we can calculate the volumetric flow rate per unit depth as a function of pressure drop

$$\frac{Q}{l} = -\frac{1}{12\mu}\left[\frac{-\Delta p}{L}\right]a^3 = \frac{a^3\Delta p}{12\mu L}$$

Average velocity stationary infinitely parallel plates

$$\overline{V} = \frac{Q}{A} = -\frac{1}{12\mu}\left(\frac{dp}{dx}\right)\frac{a^3 l}{la} = -\frac{1}{12\mu}\left(\frac{dp}{dx}\right)a^2$$

Maximum velocity infinitely parallel plates

$$y = \frac{a}{2}, \qquad u = u_{\max} = -\frac{1}{8\mu}\left(\frac{dp}{dx}\right)a^2 = \frac{3}{2}\overline{V}$$

Transformation of coordinates (making y start at the middle of the plate instead of the bottom)

$$u = \frac{a^2}{2\mu}\left(\frac{dp}{dx}\right)\left[\left(\frac{y'}{a}\right)^2 - \frac{1}{4}\right]$$

IMPORTANT NOTE!!

The transition number for two parallel plates stationary **is $Re \approx 1400$.** So the above formulae are only valid in this region. You can always do the calculations and then check afterwards. Their Reynolds is defined as

$$Re = \frac{\rho u_m H}{\mu}$$ a

Where:

- ρ is the fluid density $(\mathrm{kg/m^3})$.
- u_m is the mean velocity of the fluid $(\mathrm{m/s})$.
- H is the distance between the two plates (m).
- μ is the dynamic viscosity of the fluid $(\mathrm{Pa}\backslash\mathrm{cdotps})$.

Moving upper plate

Now there are different BC

$$\begin{aligned} &\text{at} \quad y = 0 \qquad u = 0 \\ &\text{at} \quad y = a \qquad u = U \end{aligned}$$

The only difference ins the solution is a term that accounts for a plate moving

$$u = \frac{Uy}{a} + \frac{a^2}{2\mu}\left(\frac{dp}{dx}\right)\left[\left(\frac{y}{a}\right)^2 - \left(\frac{y}{a}\right)\right]$$

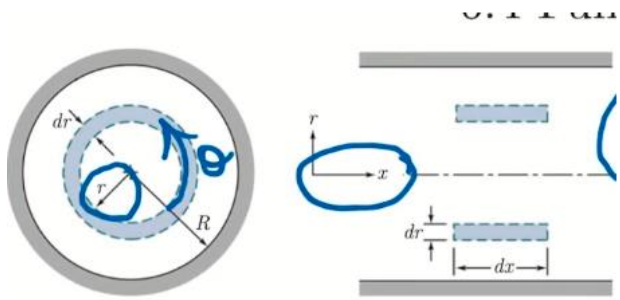
Pipes continued (Laminar flow in pipe - analytical solutions)

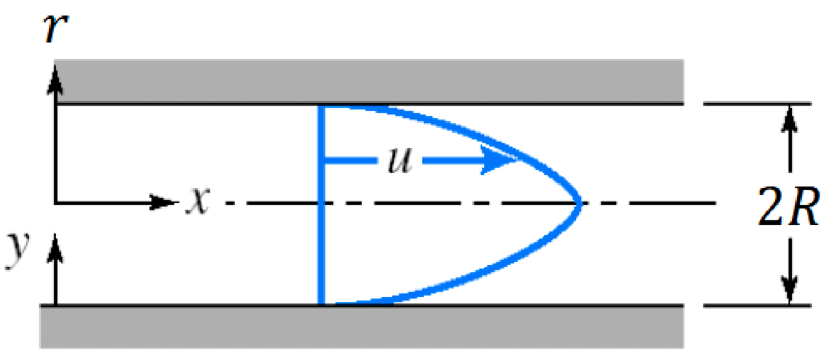
Remember, laminar flow is for $Re > 2300$ for pipes)

We want to express the velocity in a pipe. This is done by using the navier stokes in cylindrical coordinates and reducing the equations using the assumptions:

Assumptions:
1. Steady flow (given)
2. Fully developed flow (given) – no flow variation in the x direction
3. No flow in the r direction (mass balance)
4. No flow or variation of flow in the θ direction (symmetry)
5. Low Reynold's number (laminar)
6. Gravity is negligible

Geometry of pipe:





Navier stokes simplifies to

$$\underbrace{\frac{\mu}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right)}_{f(r)} = \overbrace{\frac{\partial p}{\partial x}}^{f(x)} = \text{constant}$$

Again **noting** $\frac{\partial p}{\partial x} = \boldsymbol{const.}$ Integrating twice and using boundary conditions that $u = 0\ at\ r = R$. And another thing that is weird.

This gives us **the velocity profile for a laminar flow in a pipe**

$$\boxed{u = -\frac{R^2}{4\mu}\frac{\partial p}{\partial x}\left(1-\left(\frac{r}{R}\right)^2\right)}$$

shear stress distribution for fully developed laminar flow in pipe

$$\tau_{rx} = \mu\frac{du}{dr} = \frac{r}{2}\left(\frac{\partial p}{\partial x}\right)$$

Flow rate for fully developed laminar flow in pipe

$$Q = -\frac{\pi R^4}{8\mu}\left[\frac{-\Delta p}{L}\right] = \frac{\pi \Delta p R^4}{8\mu L} = \frac{\pi \Delta p D^4}{128\mu L}$$

Average velocity for fully developed laminar flow in pipe

$$\overline{V} = \frac{Q}{A} = \frac{Q}{\pi R^2} = -\frac{R^2}{8\mu}\left(\frac{\partial p}{\partial x}\right)$$

Maximum velocity fully developed laminar flow in pipe

$$u = u_{\max} = U = -\frac{R^2}{4\mu}\left(\frac{\partial p}{\partial x}\right) = 2\overline{V}$$

$$@r = 0$$

Pressure difference fully developed laminar flow in pipe

$$\Delta p = \frac{128\mu L Q}{\pi D^4}$$

Normalized velocity profile fully developed laminar flow in pipe

$$\boxed{\frac{u}{U} = 1 - \left(\frac{r}{R}\right)^2}$$

Wall shear stress in fully developed laminar flow in pipe

$$\tau_w = -\left[\tau_{rx}\right]_{r=R} = -\frac{R}{2}\frac{\partial p}{\partial x}$$

Lecture 10B (Actually lecture 11) - Flow in pipes and ducts

This section is more focused on when we have turbulence! Lecture 10 was primarily laminar, so we could look at analytical solutions, which we wont be able to as much here.

The wish is to find out how to find pressure drop in turbulent flows.

Types of losses in pipes and ducts (and other places for that matter)

Major losses (h_l)

- Losses due to friction

Minor losses (h_{l_m})

- Losses due to geometry. (Pipe, fitting, valves, elbows etc)

Bernoulis for flows with friction.

Recall the Bernoulli equation. Since we now have friction, this equation is no longer equal to constant, since the friction creates a loss of mechanical energy.

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \cancel{\text{constant}} \downarrow$$

Friction leads to a pressure drop. Both in laminar flows and in turbulent flows (turbulent has larger pressure drop)

Turbulent velocity profiles in fully developed pipe flows

We cannot find an analytical solution, so through experiments we can apporixmate it using the power law

$$\frac{\bar{u}}{U} = \left(\frac{y}{R}\right)^{1/n} = \left(1 - \frac{r}{R}\right)^{1/n}$$

One can calculate this n using Reynolds number

$$n = -1.7 + 1.8 \log \mathrm{Re}_U$$

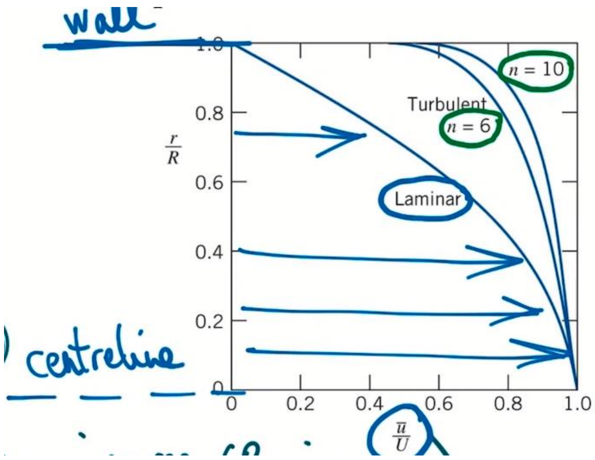
One can also approximate the average velocity

$$\frac{\overline{V}}{U} = \frac{2n^2}{(n+1)(2n+1)}$$

Often we use the **seventh** powers law is good approximation generally

$$\frac{\bar{u}}{U} = \left(\frac{y}{R}\right)^{1/7} = \left(1 - \frac{r}{R}\right)^{1/7}$$

A turbulent flow is often more “blunt”



Energy considerations in pipe flow

We cannot use the old benoulli equation. But we introduce a kinetic energy coefficient α and set up our energy equation as

$$\boxed{\left(\frac{p_1}{\rho} + \alpha_1 \frac{\overline{V}_1^2}{2} + gz_1\right) - \left(\frac{p_2}{\rho} + \alpha_2 \frac{\overline{V}_2^2}{2} + gz_2\right) = h_{l_T}}$$

Here h_{l_T} denotes the total head loss of the system.

(sometimes H_{l_T} is given, that’s just $H_{l_T} = \frac{h_{l_t}}{g}$)

This kinetic energy coefficient α can be calculated as using the blow integral, but this is **not** very accurate

$$\alpha = \frac{\int_A \rho V^3 \, dA}{\dot{m}\overline{V}^2}$$

What we **instead do** is use approximate values for α.

For laminar $\alpha = 2.0$. For turbulent $\alpha = 1.0$ usually

Calculation of headloss

We have that the totalt headloss is equal to the sum of the minor and major

$$h_{l_T} = h_l + h_{l_m}$$

For **fully developed pipe flows through constant area**, then

$$\frac{p_1 - p_2}{\rho} = g(z_2 - z_1) + h_l$$

And if it's a horizontal pipe

$$\boxed{\frac{p_1 - p_2}{\rho} = \frac{\Delta p}{\rho} = h_l}$$

So a head loss actually causes a pressure drop. We here have a **relationship between head loss and pressure drop**

How to find headlosses in flows using exeperimental data

One way is using the darcy friction factor ***f*** which is determined experimentally

$$\boxed{h_l = f \frac{L}{D} \frac{\overline{V}^2}{2}}$$

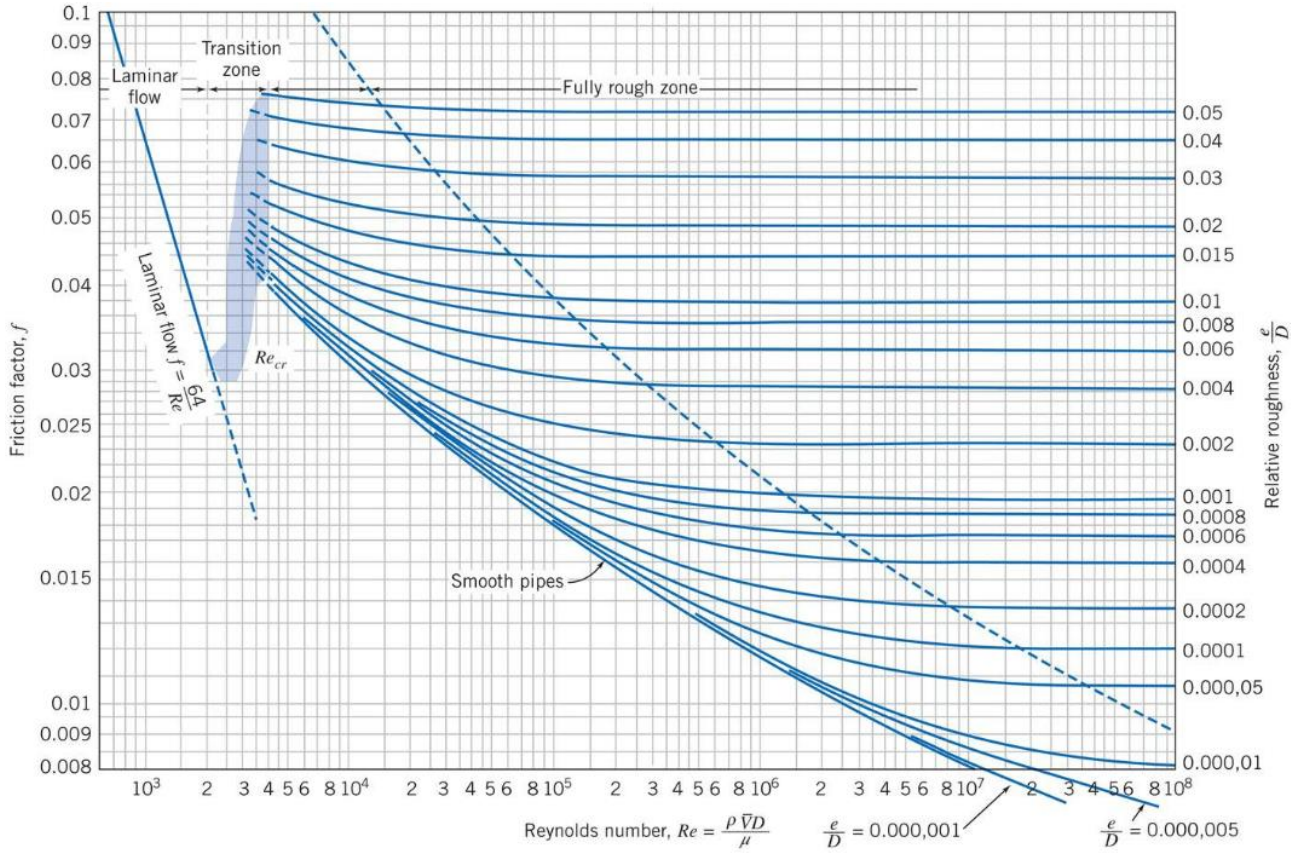
Altnerativly:

For **laminar flows in pipes** the darcy friction factor can be found using the relation:

For $Re < 2300$

$$f = \frac{64}{Re}$$

The **turbulent** friction factor is found using a moody diagram:



For turbulent flow $Re > 4000$. This has an accuracy of $\pm 10\%$

There are a bunch of **turbulent friction factor general relations** depending on the Reynolds number.

Turbulent friction factor – general correlations

Colebrook formula

$$\frac{1}{\sqrt{f}} = -2.0 \log\left(\frac{e/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right)$$

Colebrook formula $Re > 3000$

$$\frac{1}{\sqrt{f}} = -1.8 \log\left[\left(\frac{e/D}{3.7}\right)^{1.11} + \frac{6.9}{Re}\right]$$

Turbulent friction factor – smooth pipe correlations

Blasius correlation for smooth pipes $Re < 10^5$

$$f = \frac{0.316}{Re^{0.25}}$$

Modified Blasius correlation for smooth pipes $Re > 10^5$

$$f = \frac{0.184}{Re^{0.2}}$$

DET ER $LOG10$ PÅ DEM ALLE!

A table over roughness

Table 8.1
Roughness for Pipes of Common Engineering Materials

Pipe	Roughness, e	
	Feet	Millimeters
Riveted steel	0.003–0.03	0.9–9
Concrete	0.001–0.01	0.3–3
Wood stave	0.0006–0.003	0.2–0.9
Cast iron	0.00085	0.26
Galvanized iron	0.0005	0.15
Asphalted cast iron	0.0004	0.12
Commercial steel or wrought iron	0.00015	0.046
Drawn tubing	0.000005	0.0015

Source: Data from Moody [8].

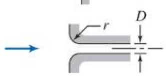
How to find minor headlosses

- These typically come from flow separation. We have

$$\boxed{h_{l_m} = K\frac{\overline{V}^2}{2}} \quad \left[\frac{\text{J}}{\text{kg}}\right]$$

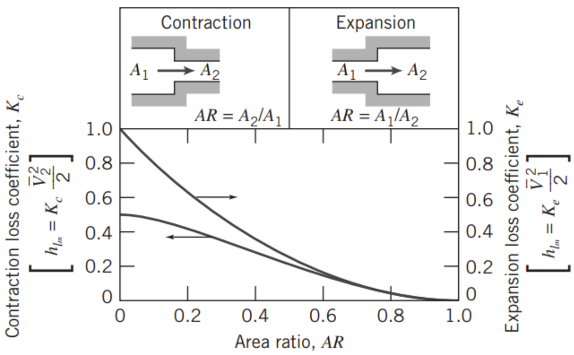
Where K is the **loss coefficient**. These are found experimentally

Table 8.2
Minor Loss Coefficients for Pipe Entrances

Entrance Type		Minor Loss Coefficient, K^a
Reentrant		0.5 – 1.0 (depending on length of pipe entrance)
Square-edged		0.5
Rounded		r/D: 0.02, 0.06, ≥ 0.15; K: 0.3, 0.2, 0.04

[a] Based on $h_{l_m} = K(\overline{V}^2/2)$, where $\overline{V}$ is the mean velocity in the pipe.
Source: Data from Reference [12].

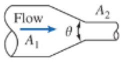
Loss coefficient when areas is changing: (Inlets and exists)



Se pilene for Pil frem og tilbage. Right pil expansion, left contraction

And

Table 8.3
Loss Coefficients (K) for Gradual Contractions: Round and Rectangular Ducts

		Included Angle, θ, Degrees						
	A_2/A_1	10	15–40	50–60	90	120	150	180
	0.50	0.05	0.05	0.06	0.12	0.18	0.24	0.26
	0.25	0.05	0.04	0.07	0.17	0.27	0.35	0.41
	0.10	0.05	0.05	0.08	0.19	0.29	0.37	0.43

Pumps, fand and blowers in fluid sections

Calculate added from pump

$$\dot{W}_{\text{pump}} = \dot{m}\left[\left(\frac{p}{\rho} + \frac{\overline{V}^2}{2} + gz\right)_{\text{discharge (outlet)}} - \left(\frac{p}{\rho} + \frac{\overline{V}^2}{2} + gz\right)_{\text{suction (inlet)}}\right]$$

Alternatively

$$\dot{W}_{\text{pump}} = Q\Delta p_{\text{pump}}$$

Effenciency

$$\eta = \frac{\dot{W}_{\text{pump}}}{\dot{W}_{\text{in}}}$$

$\dot{W}_{in}$ = how much power provided to the pump

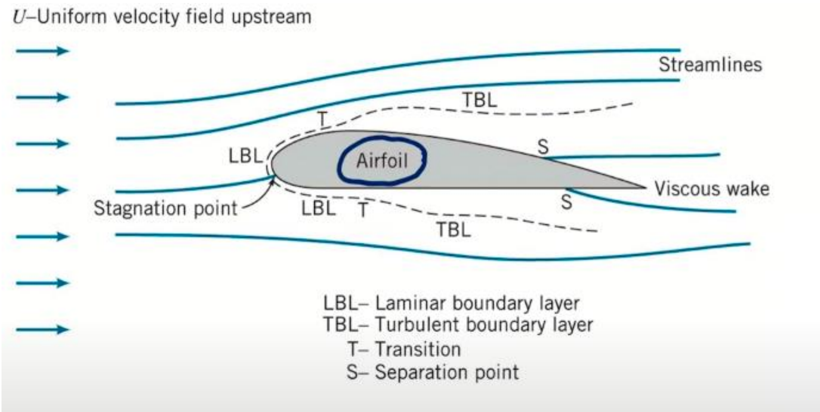
$\dot{W}_{pump}$ = Power from pump to system. So for a pipe system we use

$$\left(\frac{p_1}{\rho} + \alpha_1\frac{\overline{V}_1^2}{2} + gz_1\right) - \left(\frac{p_2}{\rho} + \alpha_2\frac{\overline{V}_2^2}{2} + gz_2\right) = h_{l_T} - \Delta h_{\text{pump}}$$

There must be some head gain from the pump.

Lecture 11 - External Incompressible Viscous flow pt 1. Boundary layer theory.

Introduction to boundary layer.

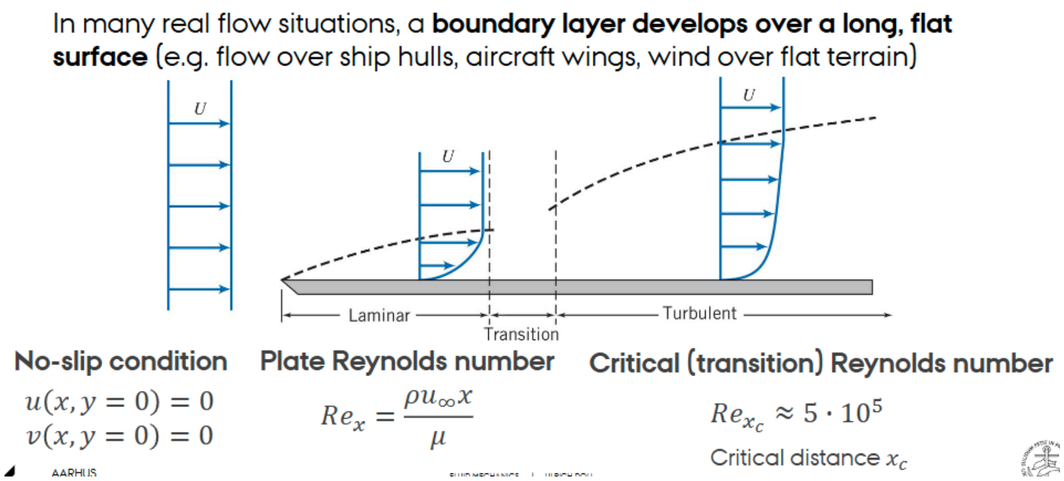


We have a free stream velocity hitting and airfoil. It has to change the around the airfoil, we have the stagnation point, where the velocity hits 0. We then have some air boing up and usm under. Due to the no-slip, we have frictional effects grow larger and larger along the wing. We see first we have a laminar boundary layer LBL, then we hit the point called "T", where the boundary layer transitions to a turbulent boundary layer TBL.

TBL grows faster than LBL. Finally, we arrive at s which is the separation point. Here the flow separates from the body. This creates a wake, which is going to cause pressure effects.

Far away from the airfoil the viscous effects don't matter, so we can just use bernoullis, but down at the airfoil we CANNOT

Boundary layer development



The dashed line is the boundary layer. Above the boundary layer there are no viscous effects.

The transition Reynolds number is defined as

$$Re_x = \frac{\rho U x}{\mu}$$

U = Freestream. This is approx. equal to $Re_x \approx 5 \cdot 10^5$

Boundary layer thickness

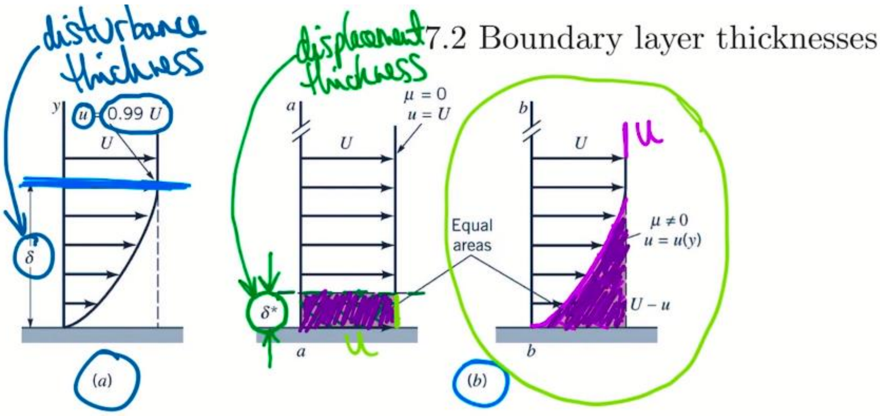
There are multiple ways of defining boundary layer thickness

Disturbance thickness δ = where the velocity within the boundary layer is 99% of the free stream, so $u = 0.99U$

Displacement thickness δ^* = The distance the plate would have to be moved so that the loss of mass flux (due to reduction of area) is equivalent to the loss the boundary layer causes. This can be calculated as! Now one can just use our old equations! Epic.

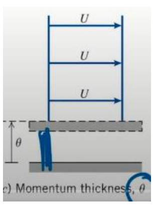
$$\boxed{\delta^* = \int_0^\infty \left(1 - \frac{u}{U}\right) dy \approx \int_0^\delta \left(1 - \frac{u}{U}\right) dy}$$

Visualised from mr. macdonald. The shaded purple areas are equal



Momentum thickness θ: the distance the plate would have to be moved so that the loss of momentum flux is equivalent to the loss the boundary layer causes.

$$\boxed{\theta = \int_0^\infty \frac{u}{U}\left(1 - \frac{u}{U}\right) dy \approx \int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right) dy}$$



Laminar flat plate boundary layer formulas:

99%-boundary layer thickness for laminar flat plate boundary layer

$$\frac{\delta}{x} = \frac{5.0}{\sqrt{Ux/\nu}} = \frac{5.0}{\sqrt{Re_x}}$$

Wall shear stress

$$\tau_w = \frac{0.332\rho U^2}{\sqrt{Re_x}}$$

Friction coefficient

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho U^2} = \frac{0.664}{\sqrt{Re_x}}$$

Also

$$\frac{\delta^\star}{\delta} = 0.344$$

Turbulent flat plate boundary layer formulas

All these are semianalytical using $\frac{1}{7}$ power law profile

99%-boundary layer thickness for turbulent flat plate boundary layer

$$\frac{\delta}{x} \approx \frac{0.382}{Re_x^{1/5}}$$

Wall shear stress

$$\tau_w = \frac{0.0297\rho U^2}{Re_x^{1/5}}$$

Friction coefficient

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho U^2} = \frac{0.0594}{Re_x^{1/5}} \quad \text{for} \quad 5 \times 10^5 < Re < 10^7$$

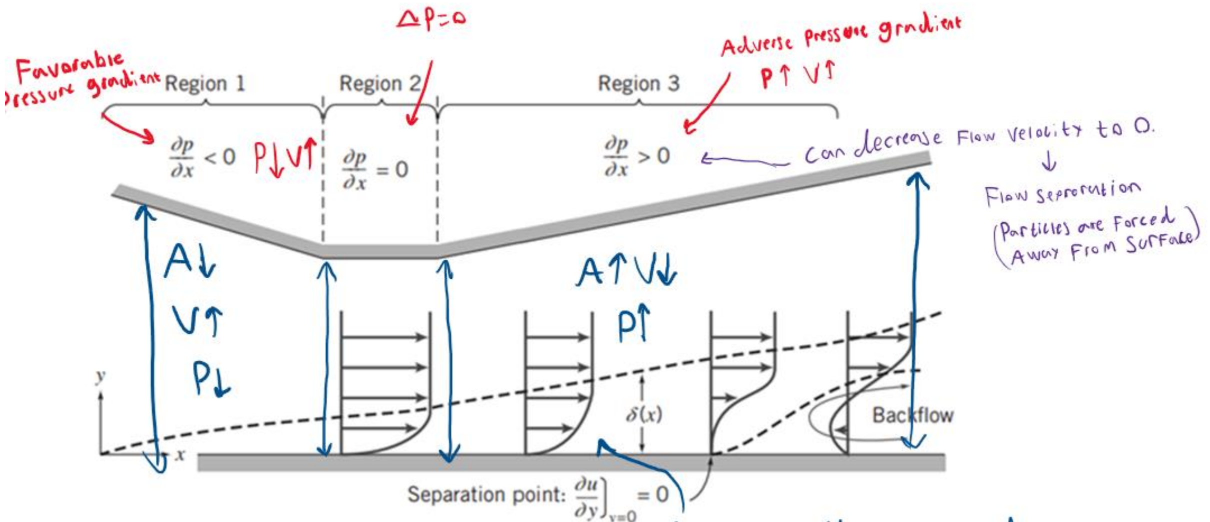
Table

Table 9.2
Results of the Calculation of Laminar Boundary-Layer Flow over a Flat Plate at Zero Incidence Based on Approximate Velocity Profiles

Velocity Distribution $\frac{u}{U}=f\left(\frac{y}{\delta}\right)=f(\eta)$	$\beta \equiv \frac{\theta}{\delta}$	$\frac{\delta^*}{\delta}$	$H \equiv \frac{\delta^*}{\theta}$	Constant a in $\frac{\delta}{x}=\frac{a}{\sqrt{Re_x}}$	Constant b in $C_f=\frac{b}{\sqrt{Re_x}}$
$f(\eta)=\eta$	$\frac{1}{6}$	$\frac{1}{2}$	3.00	3.46	0.577
$f(\eta)=2\eta-\eta^2$	$\frac{2}{15}$	$\frac{1}{3}$	2.50	5.48	0.730
$f(\eta)=\frac{3}{2}\eta-\frac{1}{2}\eta^3$	$\frac{39}{280}$	$\frac{3}{8}$	2.69	4.64	0.647
$f(\eta)=2\eta-2\eta^3+\eta^4$	$\frac{37}{315}$	$\frac{3}{10}$	2.55	5.84	0.685
$f(\eta)=\sin\left(\frac{\pi}{2}\eta\right)$	$\frac{4-\pi}{2\pi}$	$\frac{\pi-2}{\pi}$	2.66	4.80	0.654
Exact	0.133	0.344	2.59	5.00	0.664

Pressure gradients in boundary layer flow

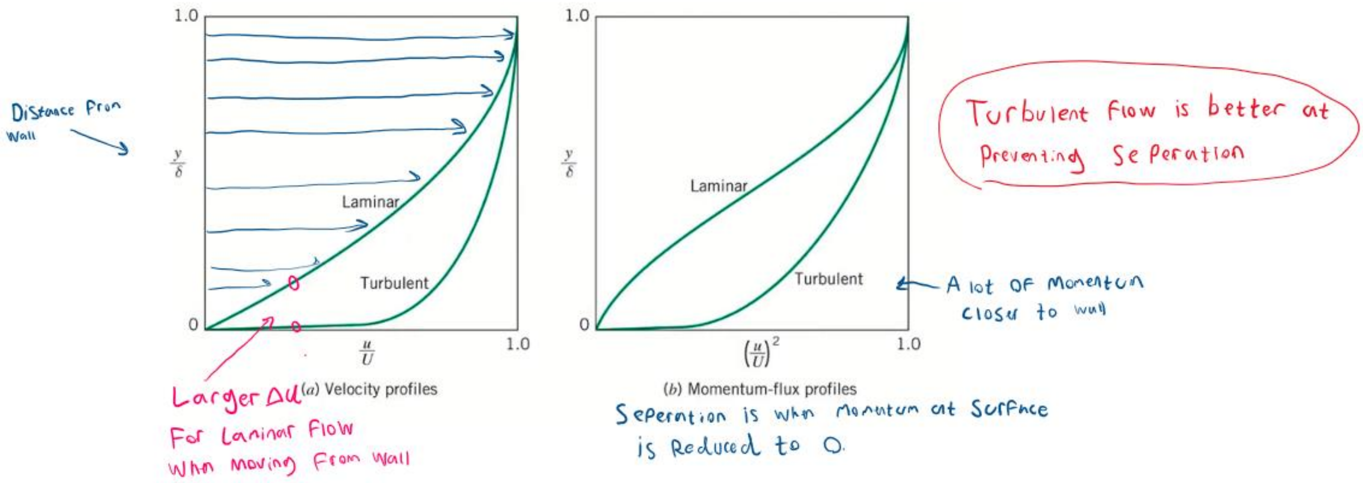
Flow separation is the detatchment of a boundary layer from a surface into a wake



Low velocity gets slowed further by Adverse pressure gradient ($\frac{\partial P}{\partial x}>0$). Pressure is causing Back Flow, But mass has to be conserved

$\rightarrow$ Flow seperates

How to delay flow separation



(a) Velocity profiles
(b) Momentum-flux profiles

Lecture 17 - External Incompressible Viscous flow pt 2. - Flow about immersed bodies - Drag, Lift and airfoils.

Drag

Drag is parallel to the direction of motion. Total drag can have two contributions. **Friction drag** and/or **pressure drag**

The drag coefficient is used to determine drag

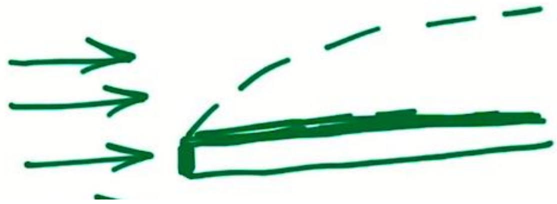
$$\boxed{C_D \equiv \frac{F_D}{\frac{1}{2}\rho V^2 A}}$$

The area is the ”wetted area”

So looking at this, F_D will vary based on if we have friction drag and or pressure drag.

Pure friction drag (Drag over a plate)

The area “height” of the plate is so small compared to its length, that we simply just neglect it



The drag coefficient in this case given depending on flow type

Laminar boundary layer

$$\boxed{C_D = \frac{1.33}{\sqrt{Re_L}}}$$

Turbulent boundary layer

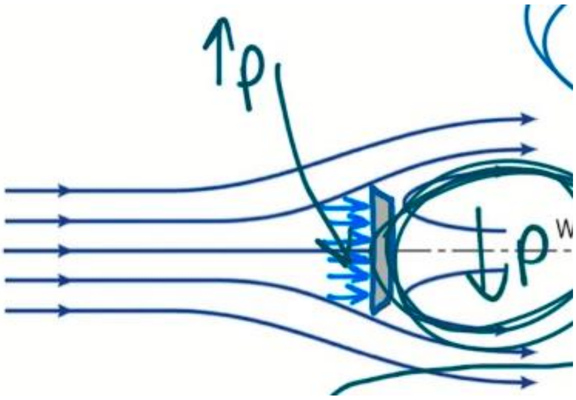
$$\boxed{C_D = \frac{0.0742}{Re_L^{1/5}}} \qquad (5 \times 10^5 < Re_L < 10^7)$$

$$\boxed{C_D = \frac{0.455}{(\log Re_L)^{2.58}} - \frac{1610}{Re_L}}^{*} \qquad (5 \times 10^5 < Re_L < 10^9)$$

Turbulent boundary layers have higher friction drag.(Antager det er log 10 😊)

Pure pressure drag

We now turn the plate around compared to before



Now all the drag is from pressure drag.

We're going to have a high pressure from the velocity hitting the plates (They have dynamic pressure since they have velocity, and then they get stopped) and we'll have low pressure in the wake. So we have a pressure difference pulling the plate to the right.

Doing this, we use the planeform area. So the area then looking on the "front of the car" forexample.

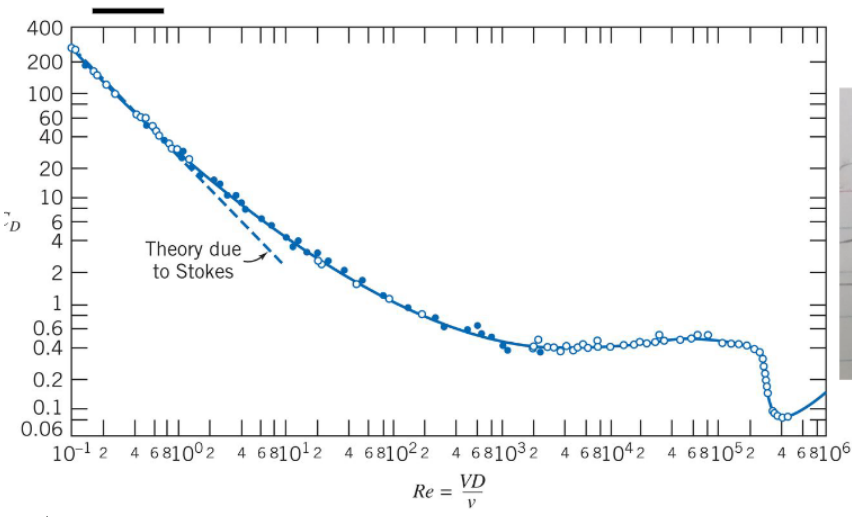
Table 9.3
Drag Coefficient Data for Selected Objects ($Re \gtrsim 10^3$)[4]

Object	Diagram	$C_D(Re \gtrsim 10^3)$
Square prism	$b/h = \infty$	2.05
	$b/h = 1$	1.05
Disk		1.17
Ring		1.20^b
Hemisphere (open end facing flow)		1.42
Hemisphere (open end facing downstream)		0.38
C-section (open side facing flow)		2.30
C-section (open side facing downstream)		1.20

Friction and pressure drag

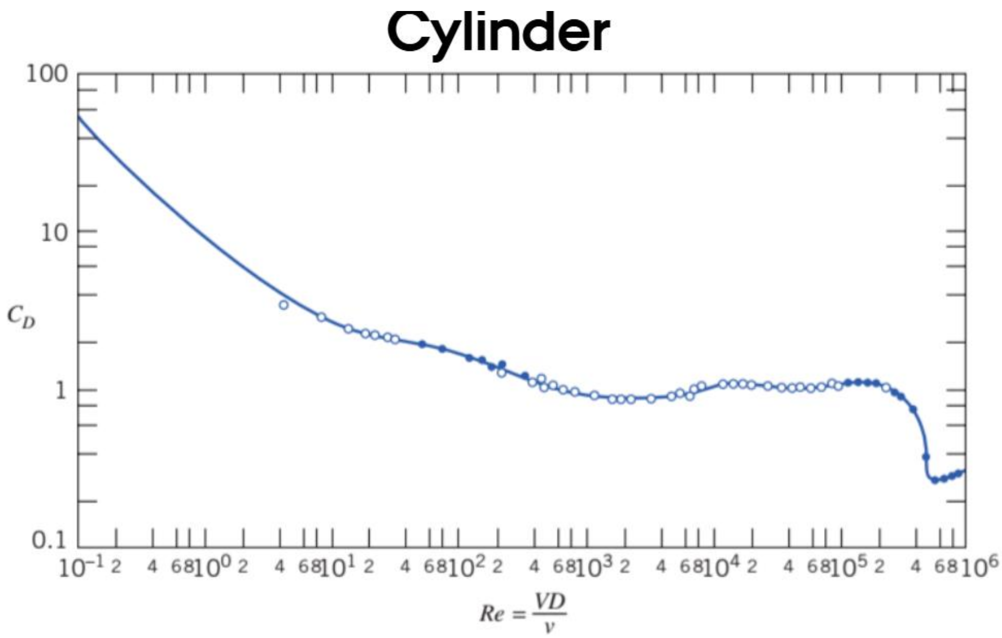
Note: Friction drag depends on Re, pressure does not! Vi tager udgangspunkt I

Flow and pressure drag: Flow over a sphere



After Reynolds 10^3 we have like 95% of all drag due to pressure drag. We're laminar from 0 to Around 10^5 where we go turbulent. As we know the turbulent boundary layer resists separation better, and therefore the separation point is pushed back. Since the drag is almost only from pressure, the extra friction added doesn't matter - what really matters is that the wake gets smaller, causing the drag to drop.

Graph for a cylinder

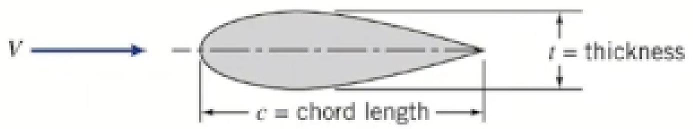


Drag reduction

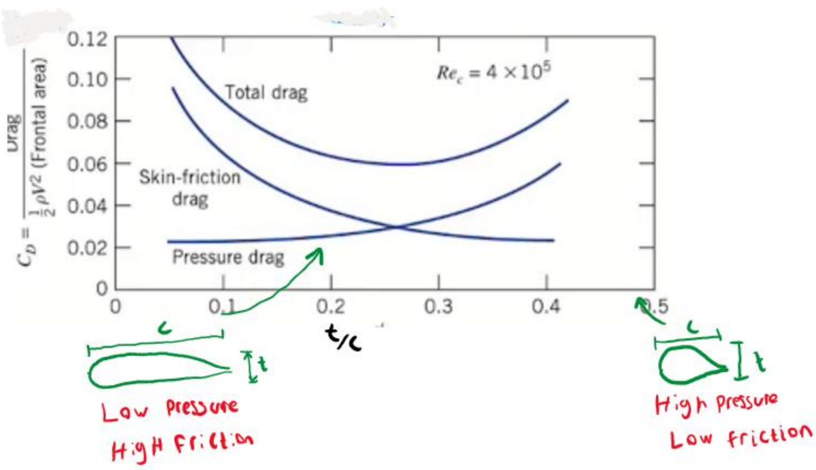
Simple.

- Flow over a flat plate (Pure friction)
  - Reduce drag by having smooth surface so less friction
- Flow over sphere and cylinder (Primarily pressure drag)
  - Reduce drag by keeping flow turbulent so wake gets smaller. Keep flow turbulent by having rough surface

For aerodynamic shapes we use **streamlining**



What we can do is streamline. So to keep the flow attached, we make a more tear-drop like shape, so geometry doesn't change so drastically. This makes the wake smaller and reduces the adverse pressure gradient. **However! Skin drag increases** so one has to find a balance



Lift!

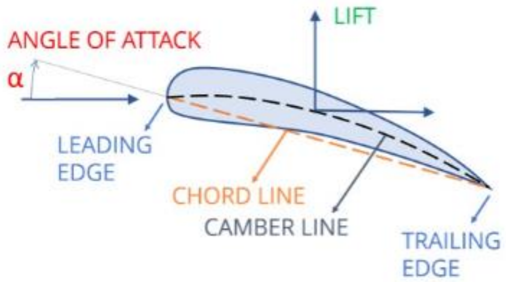
Lift is the component of a pressure force on a body acting perpendicular to the direction of motion.

$$\boxed{C_L \equiv \frac{F_L}{\frac{1}{2}\rho V^2 A_p}}$$

$A_p = bc$ = maximum projected wing area

No lift for symmetric shapes with zero angle of attack

Wing terminology



- **Chord of an airfoil:** straight line joining the leading edge and the trailing edge
- **Angle of attack (α):** angle between the airfoil chord line and the freestream velocity vector.

How does it work?

We know from Bernoulli, that high velocity = lower pressure. So the wing forces the air over its top faster than its bottom. This results in a pressure difference, with higher pressure on top and lover on the bottom, so its going to lift.

Lecture 26 - Turbo machinery

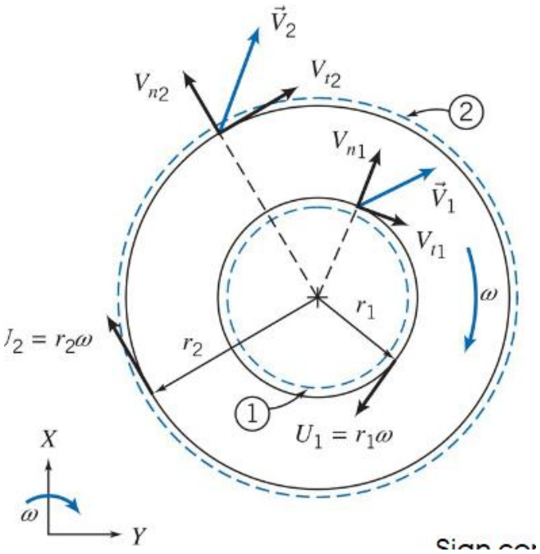
Important equations:

Euler turbine equation

$$\boxed{T_{\text{shaft}} = (r_2 V_{t_2} - r_1 V_{t_1})\dot{m}}$$

Euler Turbomachine Equation
(steady, frictionless flow, uniform flow at inlet and exit, and negligible pressure effects)

Assumptions are as given!



Mechanical power

$$\dot{W}_m = \omega T_{\text{shaft}} = \omega(r_2 V_{t_2} - r_1 V_{t_1})\dot{m}$$

Or

$$\boxed{\dot{W}_m = (U_2 V_{t_2} - U_1 V_{t_1})\dot{m}}$$

Theoretical head

$$\boxed{H = \frac{\dot{W}_m}{\dot{m}g} = \frac{1}{g}(U_2 V_{t_2} - U_1 V_{t_1})}$$

Turbine efficiency

$$\eta_t = \frac{\dot{W}_m}{\dot{W}_h} = \frac{\omega T}{\rho Q g H_t}$$

Pump efficiency

$$\eta_p = \frac{\dot{W}_h}{\dot{W}_m} = \frac{\rho Q g H_p}{\omega T}$$

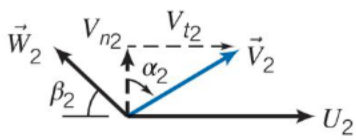
Euler turbomachiner equations for centrifugal pumpts

Purely radial velocity at pump inlet → $V_{t_1} = 0$

Theoretical head: $H = \frac{U_2 V_{t_2}}{g}$

with

$$V_{t_2} = U_2 - W_2 \cos\beta_2 = U_2 - \frac{V_{n_2}}{\sin\beta_2}\cos\beta_2 = U_2 - V_{n_2}\cot\beta_2$$



(c) Velocity components at outlet

and

$$Q = \pi D_2 w V_{n_2}$$

$$\boxed{H = \frac{U_2^2}{g} - \frac{U_2 \cot\beta_2}{\pi D_2 w g} Q} \qquad \boxed{H = C_1 - C_2 Q}$$

$$\frac{kg \cdot m^2}{s^3} = W$$