LOW SPEED AERODYNAMICS

II B. Tech IV semester (Autonomous IARE R-16)

by

Dr. MARUTHUPANDIYAN K

Professor

DEPARTMENT OF AERONAUTICAL ENGINEERING

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

DUNDIGAL, HYDERABAD - 500 043

UNIT-I
Introductory Topics for Aerodynamics

Conservation Laws

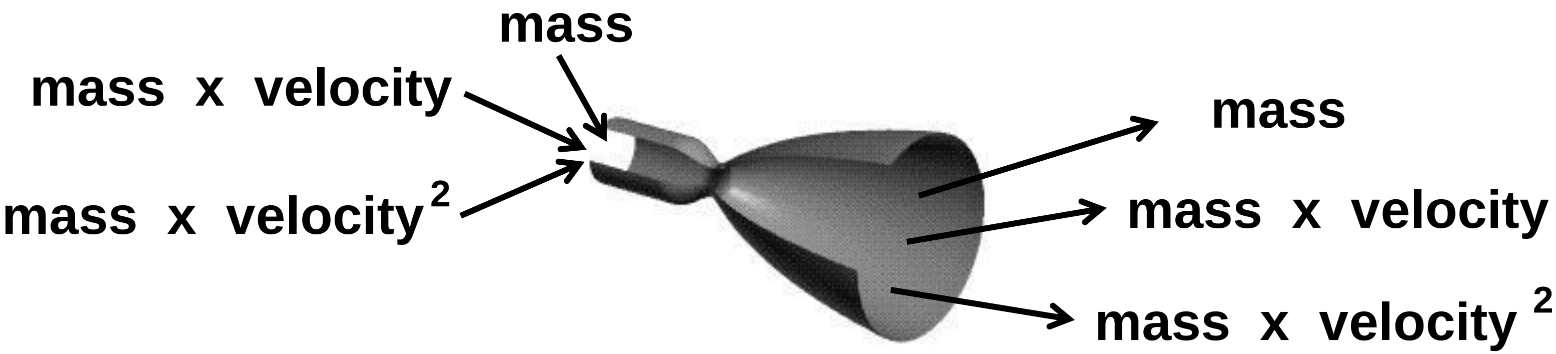
Observations of the Relations between Derived Quantities

For any fluid system:

1) Mass is neither created nor destroyed.
   Conservation of Mass - Continuity

2) Momentum is neither created nor destroyed.
   Conservation of Momentum (3 directions)

3) Energy is neither created nor destroyed.
   Conservation of Energy



Velocity Potential

Assume

$$V_x = \frac{\partial \phi}{\partial x}$$

$$\frac{\partial V_y}{\partial x} = \frac{\partial V_x}{\partial y} = \frac{\partial^2 \phi}{\partial y \partial x} = \frac{\partial^2 \phi}{\partial x \partial y}$$

$$V_y = \frac{\partial \phi}{\partial y}$$

$$\frac{\partial V_y}{\partial x} - \frac{\partial V_x}{\partial y} = 0$$

In 3D, similarly it can be shown that

$$V_z = \frac{\partial \phi}{\partial z}$$

f is the velocity potential

Stream Function & Velocity Potential

Stream lines/ Stream Function (Y)

- Concept
- Relevant Formulas
- Examples
- Rotation, vorticity

Velocity Potential(f)

- Concept
- Relevant Formulas
- Examples
- Relationship between stream function and velocity potential
- Complex velocity potential

Stream Lines

Consider 2D incompressible flow

Continuity Eqn

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}\left(\rho V_x\right) + \frac{\partial}{\partial y}\left(\rho V_y\right) + \frac{\partial}{\partial z}\left(\rho V_z\right) = 0$$

$$\frac{\partial}{\partial x}\left(V_x\right) + \frac{\partial}{\partial y}\left(V_y\right) = 0 \qquad V_y = \int\left(-\frac{\partial V_x}{\partial x}\right)dy$$

Vx and Vy are related

Can you write a common function for both?

Stream Function

Assume

$$V_x = \frac{\partial \psi}{\partial y}$$

Then

$$V_y = \int \left(-\frac{\partial V_x}{\partial x} \right) dy = \int \left(-\frac{\partial^2 \psi}{\partial x \partial y} \right) dy$$

$$= \int \left(-\frac{\partial^2 \psi}{\partial y \partial x} \right) dy = \left(-\frac{\partial \psi}{\partial x} \right)$$

Instead of two functions, Vx and Vy, we need to solve for only one function y - Stream Function

Order of differential eqn increased by one

Stream Function

What does Stream Function y mean?

Equation for streamlines in 2D are given by

y = constant

Streamlines may exist in 3D also, but stream function does not

Why? (When we work with velocity potential, we may get a perspective)

In 3D, streamlines follow the equation

$$\frac{dx}{V_x} = \frac{dy}{V_y} = \frac{dz}{V_z}$$

Velocity Potential vs Stream Function

	Stream Function (ψ)	Velocity Potential (ϕ)
Exists for	only 2D flow	all flows
	viscous or non-viscous flows	Irrotational (i.e. Inviscid or zero viscosity) flow
	Incompressible flow (steady or unsteady)	Incompressible flow (steady or unsteady state)
	compressible flow (steady state only)	compressible flow (steady or unsteady state)

- In 2D inviscid flow (incompressible flow OR steady state compressible flow), both functions exist
- What is the relationship between them?

Laplace equation

- We are going to be solving the Laplace equation in the context of electrodynamics
- Using spherical coordinates assuming azimuthal symmetry
  - Could also be solving in Cartesian or cylindrical coordinates
  - These would be applicable to systems with corresponding symmetry
- Begin by using separation of variables
  - Changes the system of partial differential equations to ordinary differential equations
- Use of Legendre polynomials to find the general solution

- Cartesian coordinates

$$\frac{\partial^2 V}{\partial x^2} + \frac{\partial^2 V}{\partial y^2} + \frac{\partial^2 V}{\partial z^2} = 0$$

  - V is potential
  - Harmonic!
- Spherical coordinates

$$\frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2 \frac{\partial V}{\partial r}\right) + \frac{1}{r^2 \sin\theta}\frac{\partial}{\partial \theta}\left(\sin\theta \frac{\partial V}{\partial \theta}\right) + \frac{1}{r^2 \sin^2\theta}\frac{\partial^2 V}{\partial \phi^2} = 0$$

  - r is the radius
  - θ is the angle between the z-axis and the vector we're considering
  - ϕ is the angle between the x-axis and our vector

UNIFORM FLOW

Definitions

- a) Open Channel: Duct through which Liquid Flows with a Free Surface - River, Canal

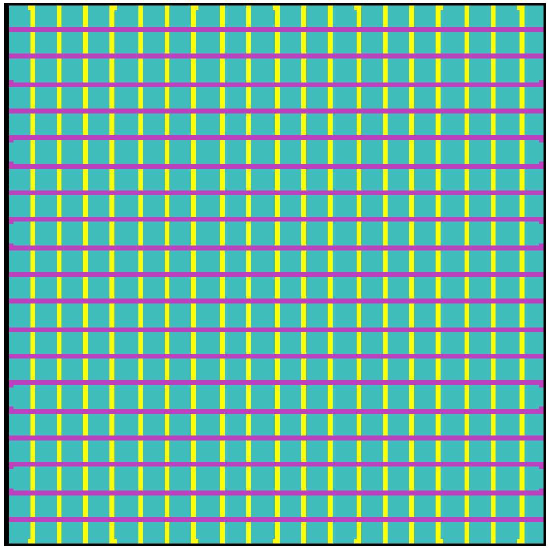
- b) Steady and Non- Steady Flow: In Steady Flows, all the characteristics of flow are constant with time. In unsteady flows, there are variations with time.

THE UNIFORM FLOW

The first and simplest example is that of a uniform flow with velocity *U* directed along the *x* axis.

In this case the complex potential is

$$W = \phi + i\psi = Uz$$



and the streamlines are all parallel to the velocity direction (which is the *x* axis).

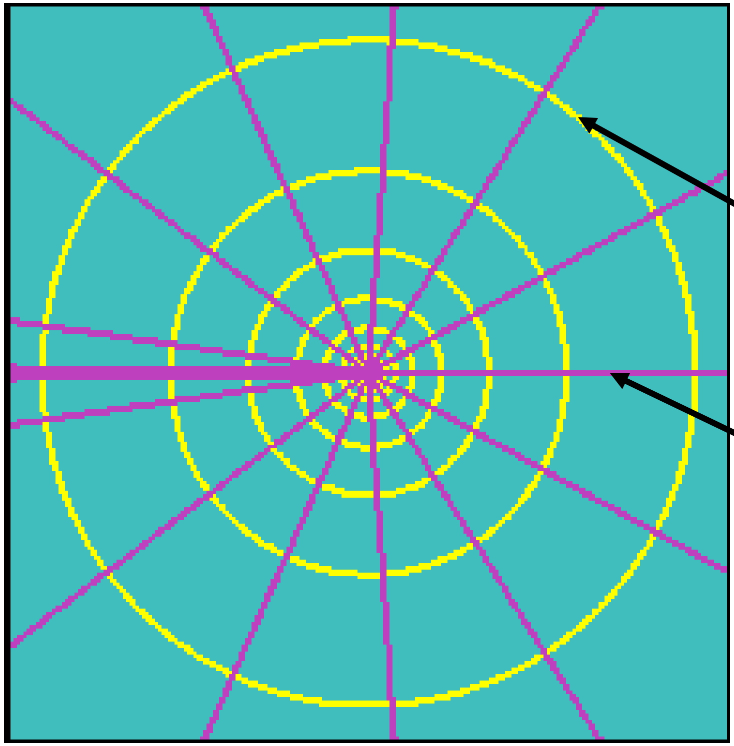
Equi-potential lines are obviously parallel to the *y* axis.

THE SOURCE OR SINK

- source (or sink), the complex potential of which is

$$W = \phi + i\psi = \frac{m}{2\pi} \ln z$$

- This is a pure radial flow, in which all the streamlines converge at the origin, where there is a singularity due to the fact that continuity can not be satisfied.
- At the origin there is a source, $m > 0$ or sink, $m < 0$ of fluid.
- Traversing any closed line that does not include the origin, the mass flux (and then the discharge) is always zero.
- On the contrary, following any closed line that includes the origin the discharge is always nonzero and equal to *m*.



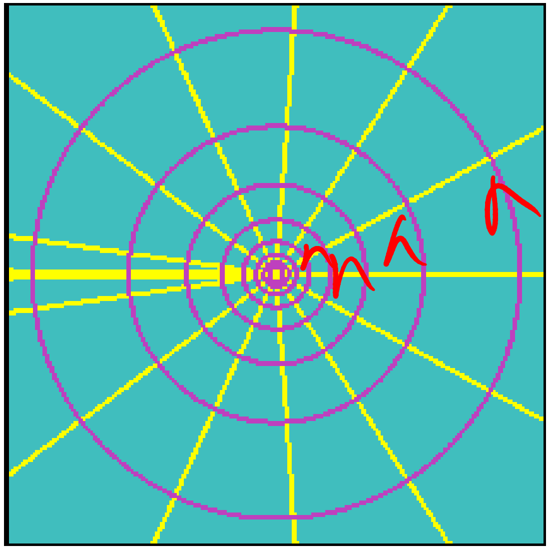
The flow field is uniquely determined upon deriving the complex potential *W* with respect to *z*.

$$W = \phi + i\psi = \frac{m}{2\pi}\ln z$$

THE VORTEX

- In the case of a vortex, the flow field is purely tangential.

The picture is similar to that of a source but streamlines and equipotential lines are reversed.
The complex potential is



$$W = \phi + i\psi = i\frac{\gamma}{2\pi}\ln z$$

There is again a singularity at the origin, this time associated to the fact that the circulation along any closed curve including the origin is nonzero and equal to g.

If the closed curve does not include the origin, the circulation will be zero.

Uniform Flow Past A Doublet with Vortex

- The superposition of a doublet and a uniform flow gives the complex potential

$$W = Uz + \frac{\mu}{2\pi z} + i\frac{\gamma}{2\pi}\ln z$$

$$W = \frac{2\pi U z^2 + \mu + iz\gamma \ln z}{2\pi z}$$

$$W = \frac{2\pi U(x+iy)^2 + \mu + i\gamma(x+iy)\times \ln(x+iy))}{2\pi(x+iy)}$$

Uniform Flow Past A Doublet

- The superposition of a doublet and a uniform flow gives the complex potential

$$W = Uz + \frac{\mu}{2\pi z}$$

$$W = \frac{2\pi U z^2 + \mu}{2\pi z}$$

$$W = \frac{2\pi U \left(x + iy\right)^2 + \mu}{2\pi \left(x + iy\right)}$$

$$W = \frac{2\pi U\left(x^3 + xy^2\right) + \mu x}{2\pi\left(x^2 + y^2\right)} + i\,\frac{\left[2\pi U\left(x^2 y + y^3\right) - \mu y\right]}{2\pi\left(x^2 + y^2\right)} = \phi + i\,\psi$$

$$\phi = \frac{2\pi U\left(x^3 + xy^2\right) + \mu x}{2\pi\left(x^2 + y^2\right)} \quad \& \quad \psi = \frac{\left[2\pi U\left(x^2 y + y^3\right) - \mu y\right]}{2\pi\left(x^2 + y^2\right)}$$

$$\psi = Uy - \frac{\mu y}{2\pi\left(x^2 + y^2\right)}$$

Find out a stream line corresponding to a value of steam function is zero

$$0 = Uy - \frac{\mu y}{2\pi\left(x^2 + y^2\right)}$$

$$0 = Uy - \frac{\mu y}{2\pi\left(x^2 + y^2\right)}$$

$$0 = 2\pi U y\left(x^2 + y^2\right) - \mu y$$

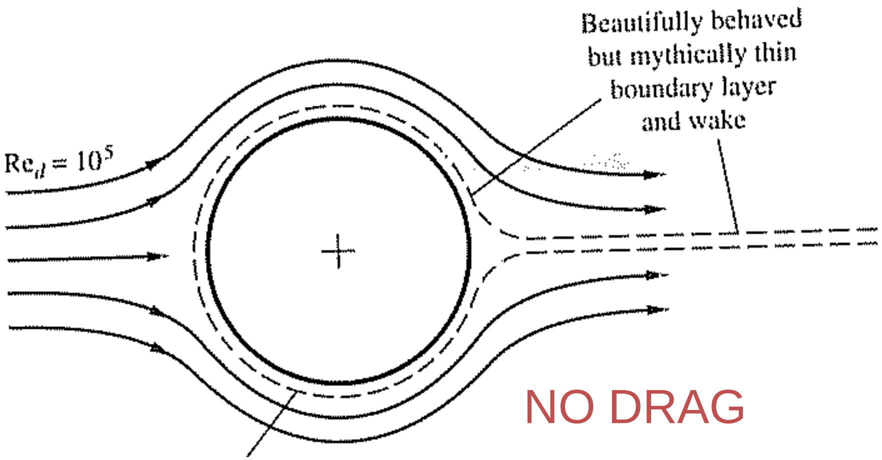
$$0 = 2\pi U\left(x^2 + y^2\right) - \mu$$

$$x^2 + y^2 = \frac{\mu}{2\pi U}$$

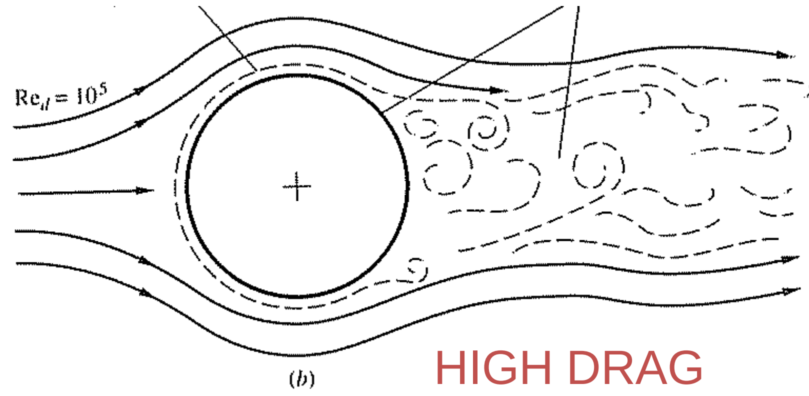
$$x^2 + y^2 = \frac{\mu}{2\pi U} = R^2$$

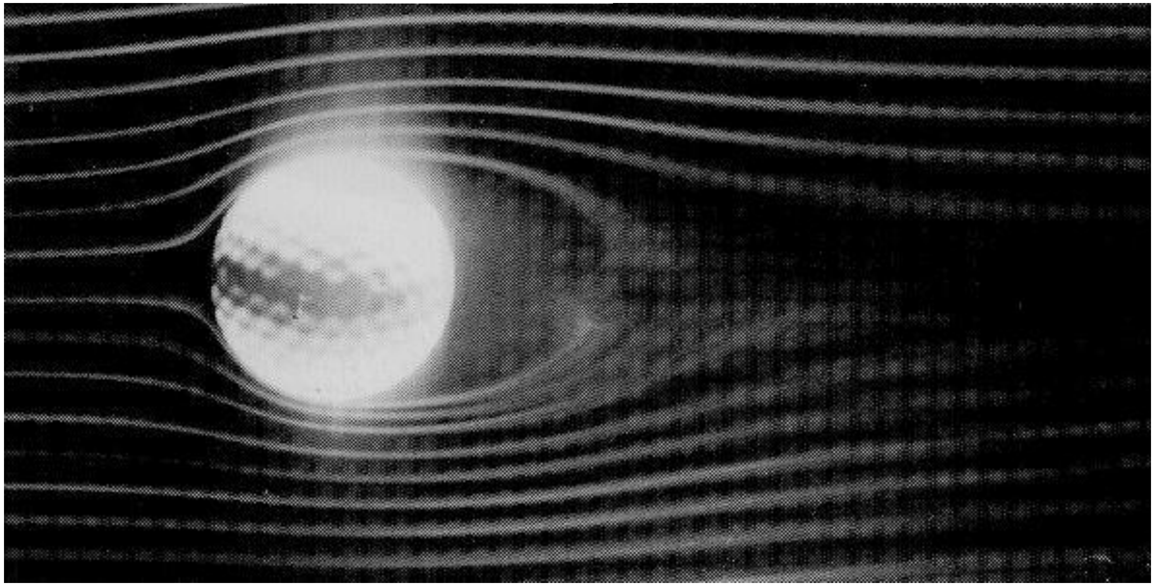
- There exist a circular stream line of radius R, on which value of stream function is zero.
- Any stream function of zero value is an impermeable solid wall.
- Plot shapes of iso-streamlines.

Non lifting flow over cylinder

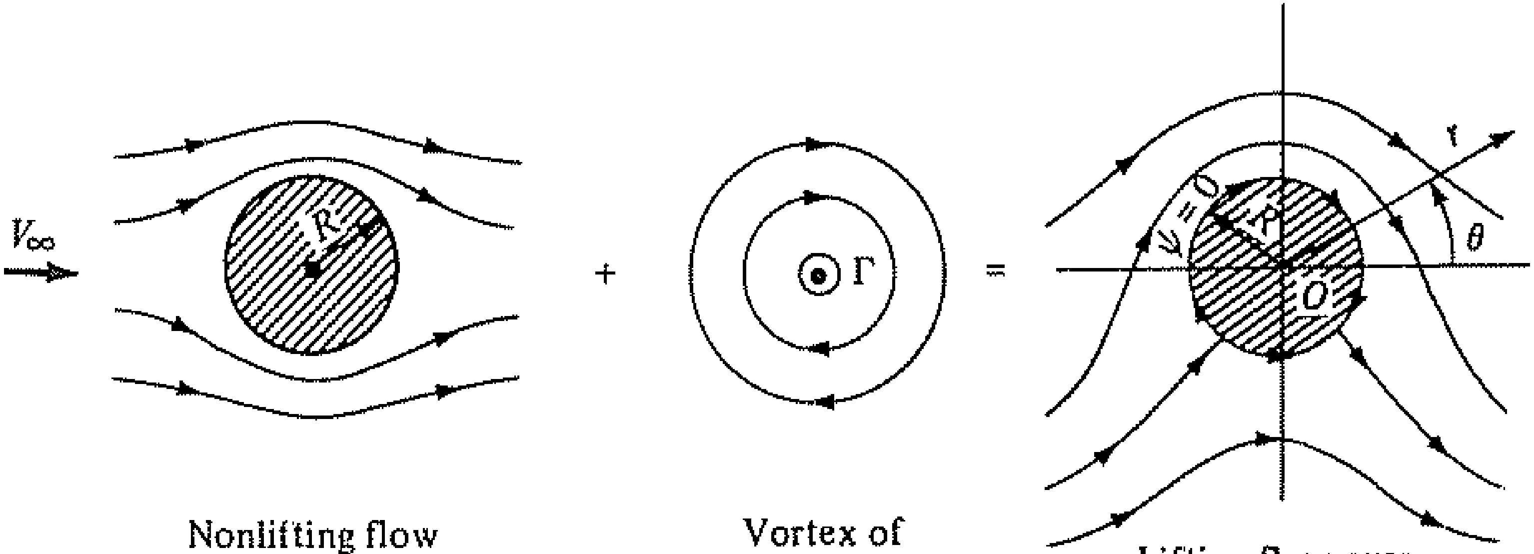


Actual: High separated
Flow and large wake region





Kutta-Joukowski Theorem



Nonlifting flow over a cylinder

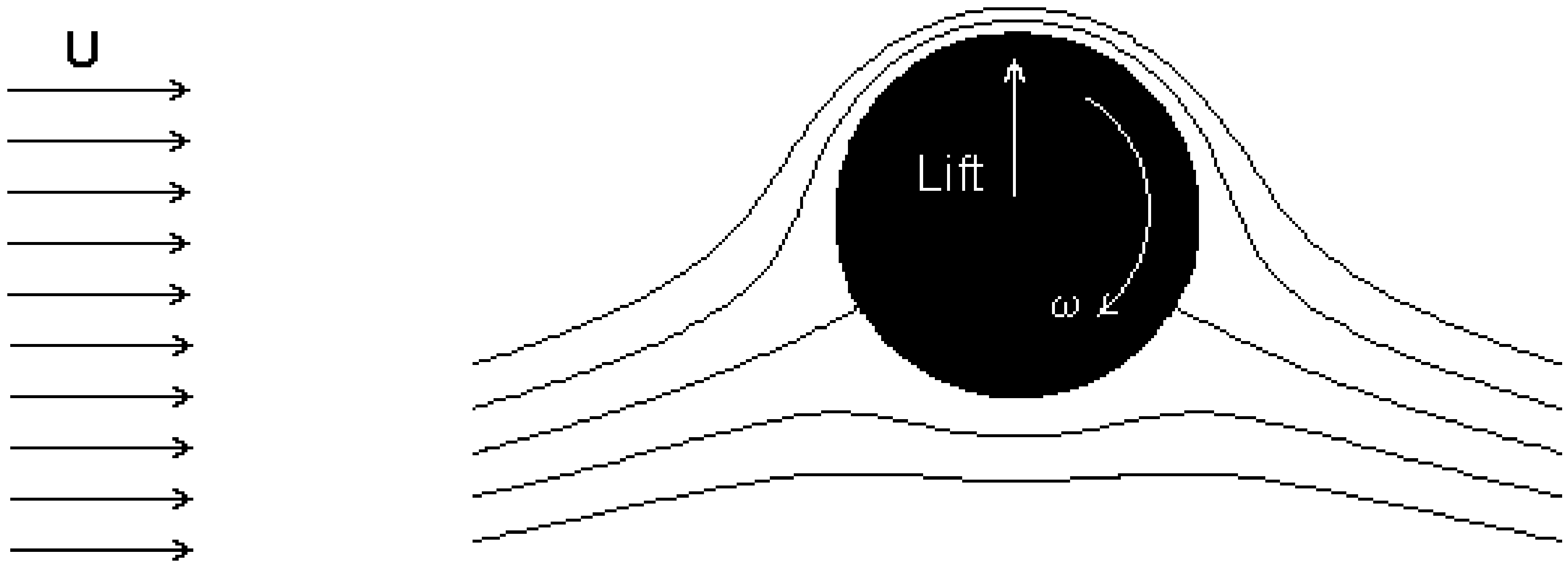
Vortex of strength Γ

Lifting flow over a cylinder

$$\psi = V_\infty r \sin\theta \left(1 - \frac{R^2}{r^2}\right) + \frac{\Gamma}{2\pi} \ln\left(\frac{r}{R}\right)$$

$$L' = \rho_\infty V_\infty \Gamma$$

SUMMARY OF ROTATING CYLINDER IN CROSS-FLOW

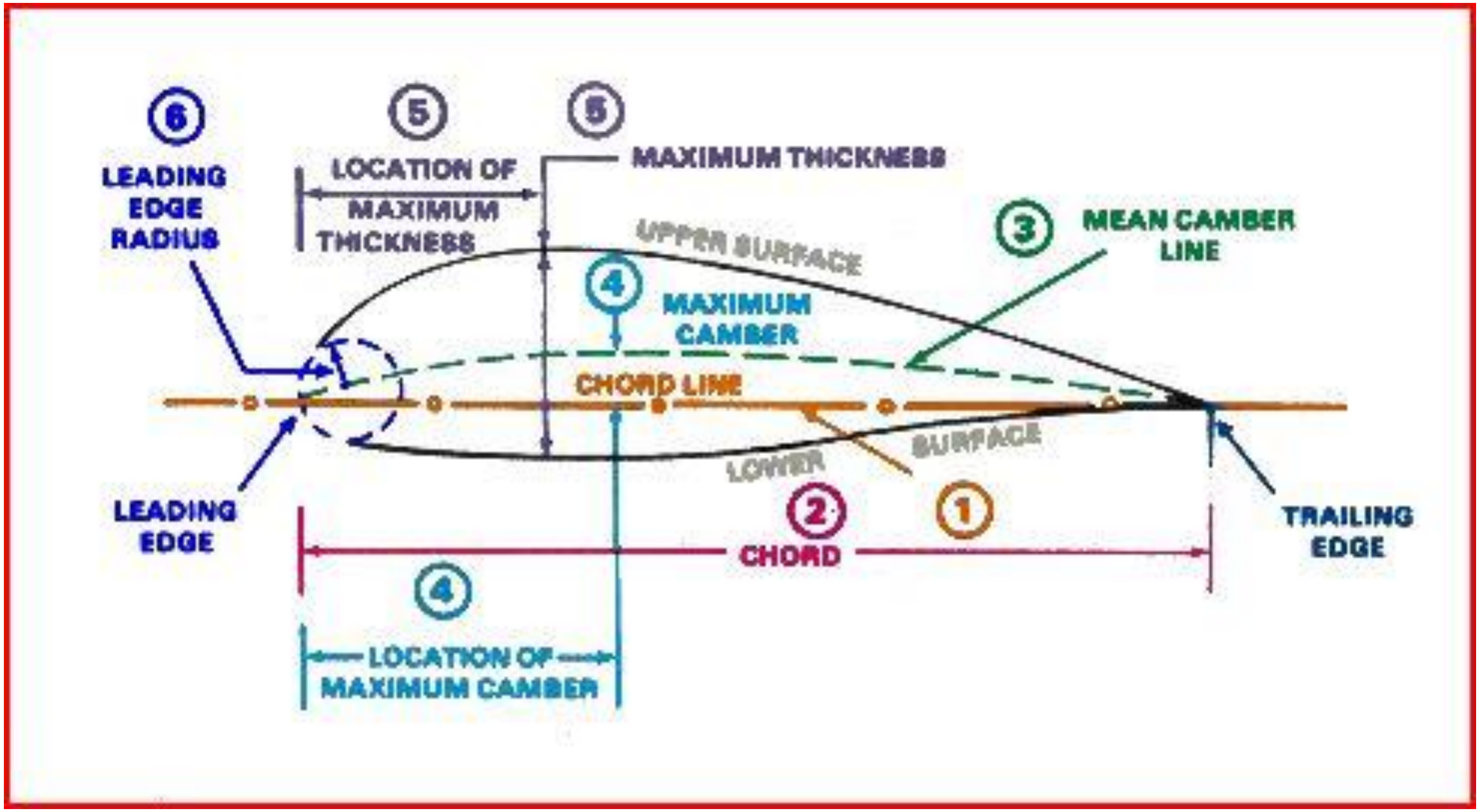


Rotating Cylinder

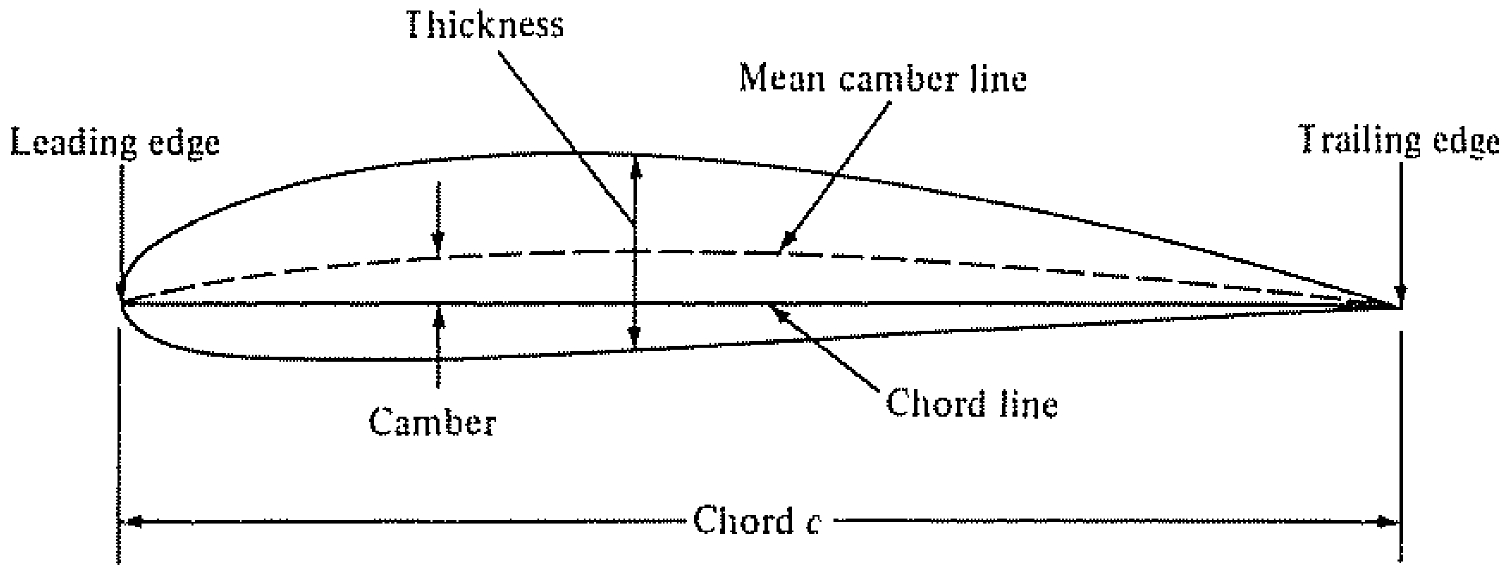
- Rotating Cylinder Generates Lift
  - Velocity is faster over the top of the cylinder than bottom
  - Pressure is higher on the bottom than over the top
  - lifting force is directed perpendicular to the cylinder velocity (or the free stream velocity if the cylinder is stationary)
- Predicts Zero Drag
  - Notice vertical plane symmetry
  - Inviscid flow approximation does not model drag physics

UNIT-II
Thin Airfoil Theory

Airfoil Nomenclature



AIRFOIL NOMENCLATURE



- Mean Chamber Line: Set of points halfway between upper and lower surfaces
  - Measured perpendicular to mean chamber line itself
- Leading Edge: Most forward point of mean chamber line

- Trailing Edge: Most reward point of mean chamber line
- Chord Line: Straight line connecting the leading and trailing edges
- Chord, c: Distance along the chord line from leading to trailing edge
- Chamber: Maximum distance between mean chamber line and chord line
  - Measured perpendicular to chord line

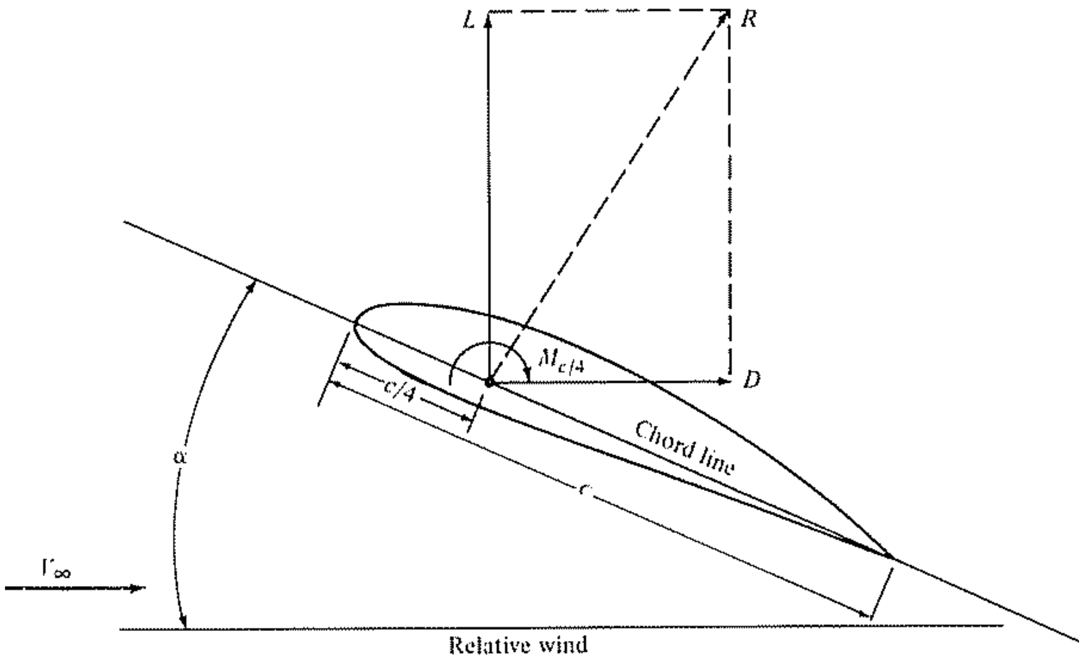
NACA FOUR-DIGIT SERIES

- First set of airfoils designed using this approach was NACA Four-Digit Series
- First digit specifies maximum camber in percentage of chord
- Second digit indicates position of maximum camber in tenths of chord
- Last two digits provide maximum thickness of airfoil in percentage of chord

Example: NACA 2415

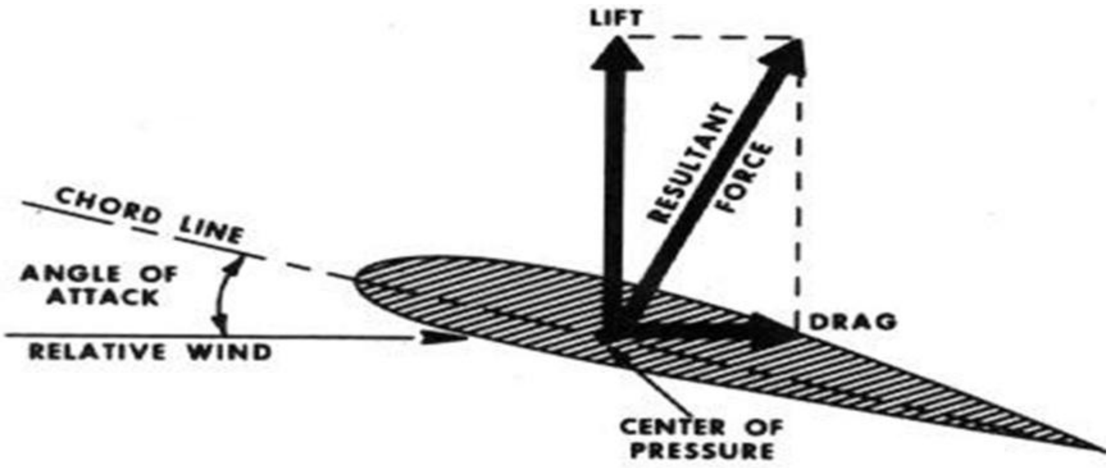
- Airfoil has maximum thickness of 15% of chord (0.15c)
- Camber of 2% (0.02c) located 40% back from airfoil leading edge (0.4c)

AERODYNAMIC CHARETERISTICS



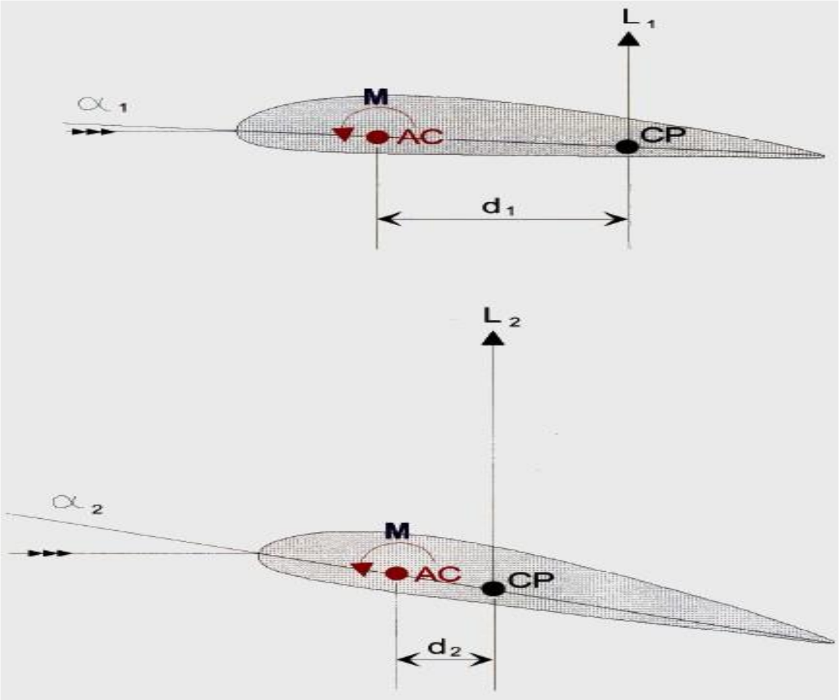
- **Relative Wind:** Direction of V_∞
  - We used subscript ∞ to indicate far upstream conditions
- **Angle of Attack, a:** Angle between relative wind (V_∞) and chord line

- Total aerodynamic force, R, can be resolved into two force components
- Lift, L: Component of aerodynamic force perpendicular to relative wind
- Drag, D: Component of aerodynamic force parallel to relative wind



Center of Pressure The center of pressure is the point where the total sum of a pressure field acts on a body. In aerospace, this is the point on the airfoil (or wing) where the resultant vector (of lift and drag) acts.

As the airfoil angle of attack changes, the pressure field changes. Due to this, the *center of pressure changes with variation in the angle of attack*. In the airplane's normal range of flight attitudes, if the angle of attack is increased, the center of pressure moves forward; and if decreased, it moves rearward



The resultant (or the pressure forces) also cause a moment on the airfoil. As the angle of attack increases, the pitching moment at a point (for example, the center of gravity) also changes. However, the pitching moment remains constant at a particular point, which is called the aerodynamic center.

For symmetric airfoils in subsonic flight the aerodynamic center is located approximately 25% of the chord from the leading edge of the airfoil. This point is described as the quarter-chord point.

Thus the *aerodynamic center does not change with variation in angle of attack*. Due to this, the aerodynamic center, rather than the center of pressure is used in the analysis of longitudinal stability.

Wing of infinite aspect ratio

In aeronautics, the aspect ratio of a wing is the ratio of its span to its mean chord. It is equal to the square of the wingspan divided by the wing area. Thus, a long, narrow wing has a high aspect ratio, whereas a short, wide wing has a low aspect ratio.

Aspect ratio and other features of the planform are often used to predict the aerodynamic efficiency of a wing because the lift-to-drag ratio increases with aspect ratio, improving fuel economy in aircraft.

Flow velocity over top of airfoil is faster than over bottom surface

Streamtube A

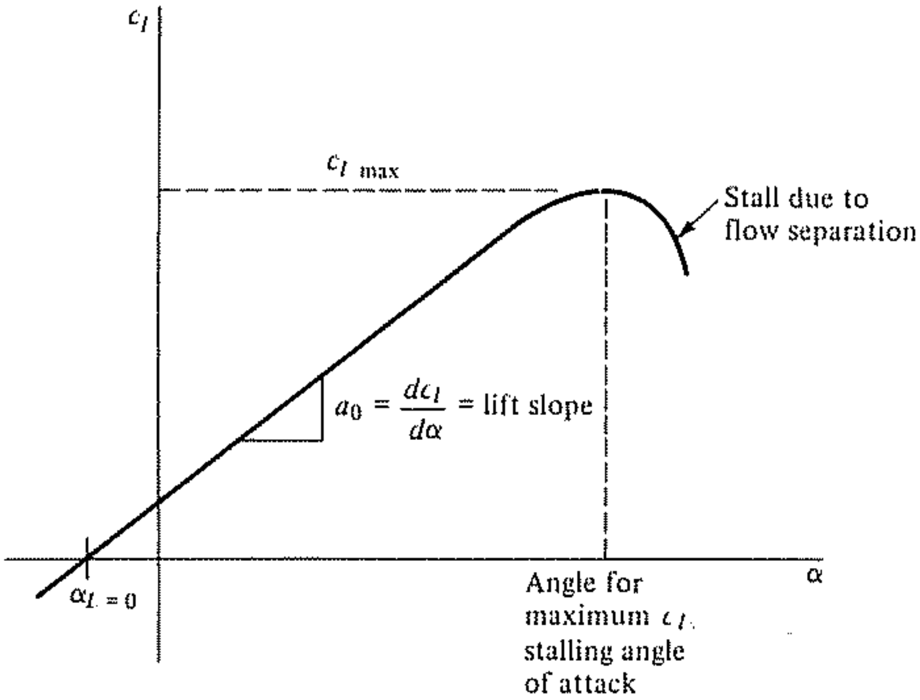
senses upper portion of airfoil as an obstruction. Streamtube A is

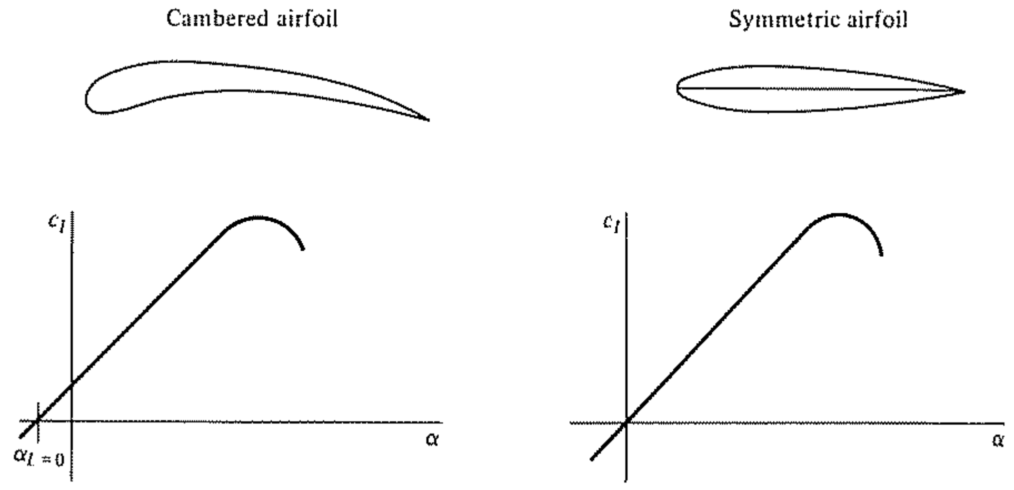
squashed to

smaller cross-sectional area

Mass c

V↑





Lift coefficient (or lift) linear variation with angle of attack,

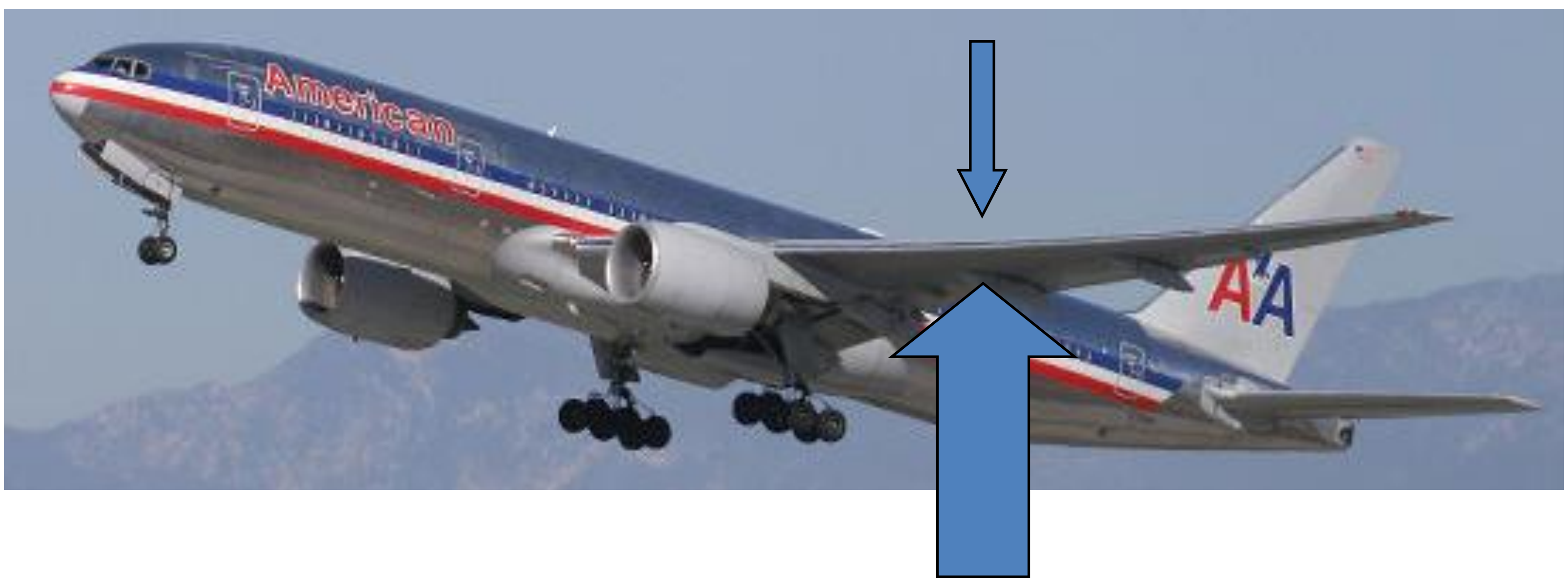
Cambered airfoils have positive lift when a=0

Symmetric airfoils have zero lift when a=0

At high enough angle of attack, the performance of the airfoil rapidly degrades → stall

The aspect ratio is ratio of the square of the wingspan b to the projected[wing area s which is equal to the ratio of the wingspan b to the mean aerodynamic chord

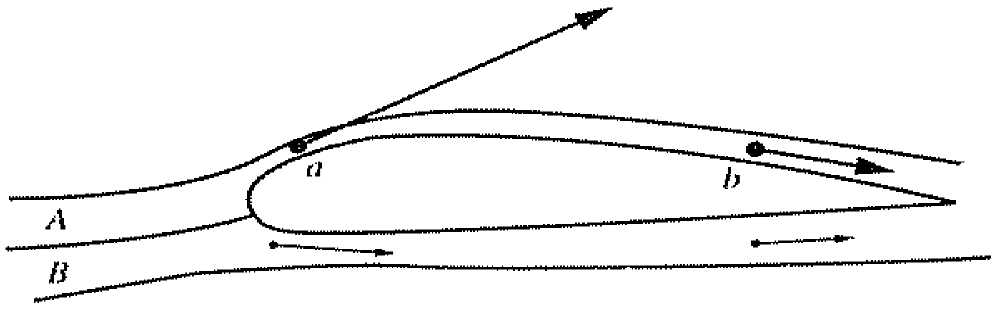
$$AR=b^2/s$$



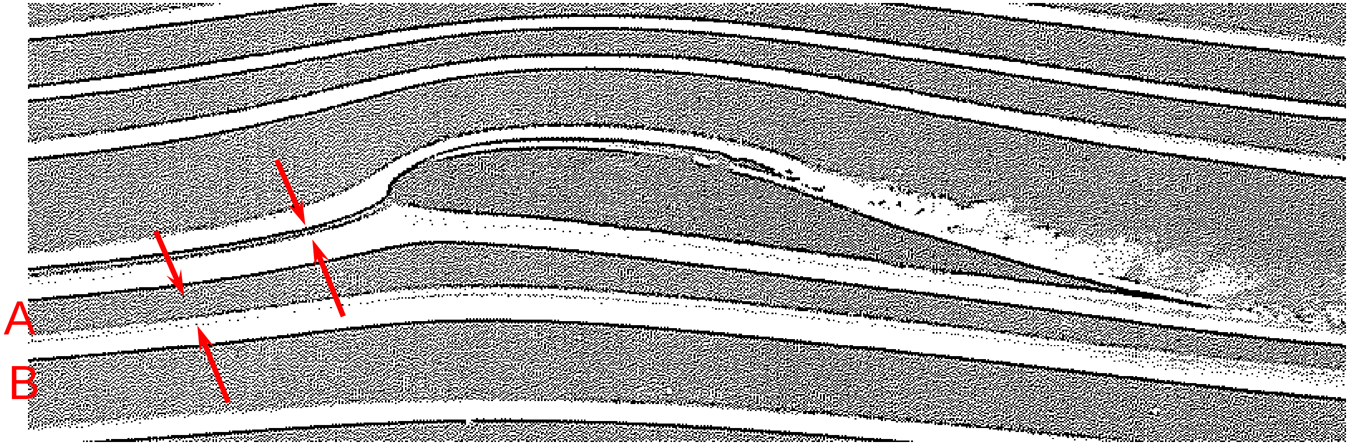
HOW DOES AN AIRFOIL GENERATE LIFT?

- Lift due to imbalance of pressure distribution over top and bottom surfaces of airfoil (or wing)
  - If pressure on top is lower than pressure on bottom surface, lift is generated
  - *Why* is pressure lower on top surface?
- We can understand answer from basic physics:
  - Continuity (Mass Conservation)
  - Newton's 2nd law (Euler or Bernoulli Equation)

HOW DOES AN AIRFOIL GENERATE LIFT?



Streamtube A is squashed most in nose region (ahead of maximum thickness)



Kutta condition

The Kutta condition is a principle in steady-flow fluid dynamics especially aerodynamics that is applicable to solid bodies with sharp corners, such as the trailing edges of airfoils. It is named for German mathematician and aerodynamicist Martin Wilhelm Kutta.

In fluid flow around a body with a sharp corner, the Kutta condition refers to the flow pattern in which fluid approaches the corner from both directions, meets at the corner, and then flows away from the body. None of the fluid flows around the corner, remaining attached to the body.

The Kutta condition is significant when using the Kutta–Joukowski theorem to calculate the lift created by an airfoil with a cusped trailing edge. The value of circulation of the flow around the airfoil must be that value which would cause the Kutta condition to exist.

Thin airfoil theory

Thin airfoil theory is a simple theory of airfoils that relates angle of attack to lift for incompressible, inviscid flows. It was devised by German-American mathematician Max Munk and further refined by British aerodynamicist Hermann Glauert and others in the 1920s. The theory idealizes the flow around an airfoil as two-dimensional flow around a thin airfoil. It can be imagined as addressing an airfoil of zero thickness and infinite wingspan.

Thin airfoil theory was particularly notable in its day because it provided a sound theoretical basis for the following important properties of airfoils in two-dimensional flow
(1) on a symmetric airfoil, the center of pressure and aerodynamic center lies exactly one quarter of the chord behind the leading edge

(2) on a cambered airfoil, the aerodynamic center lies exactly one quarter of the chord behind the leading edge

(3) the slope of the *lift coefficient versus angle of attack* line is units per radian

As a consequence of the section lift coefficient of a symmetric airfoil of infinite wingspan is

$$c_l = 2\Pi\alpha$$

Panel Methods

Linear Strength Vortex Panel Method for a Two Element Airfoil

Panel methods break up an airfoil geometry into "panels" and then solve for the flow around the panels. There are many different panel method variations and each variation has its own strengths and weaknesses.

Panel methods have two key features that distinguish themselves from each other: the formulation of the boundary conditions and the type of singularity element used to describe the flow field around the airfoil.
element since it can model both lift and pressure.

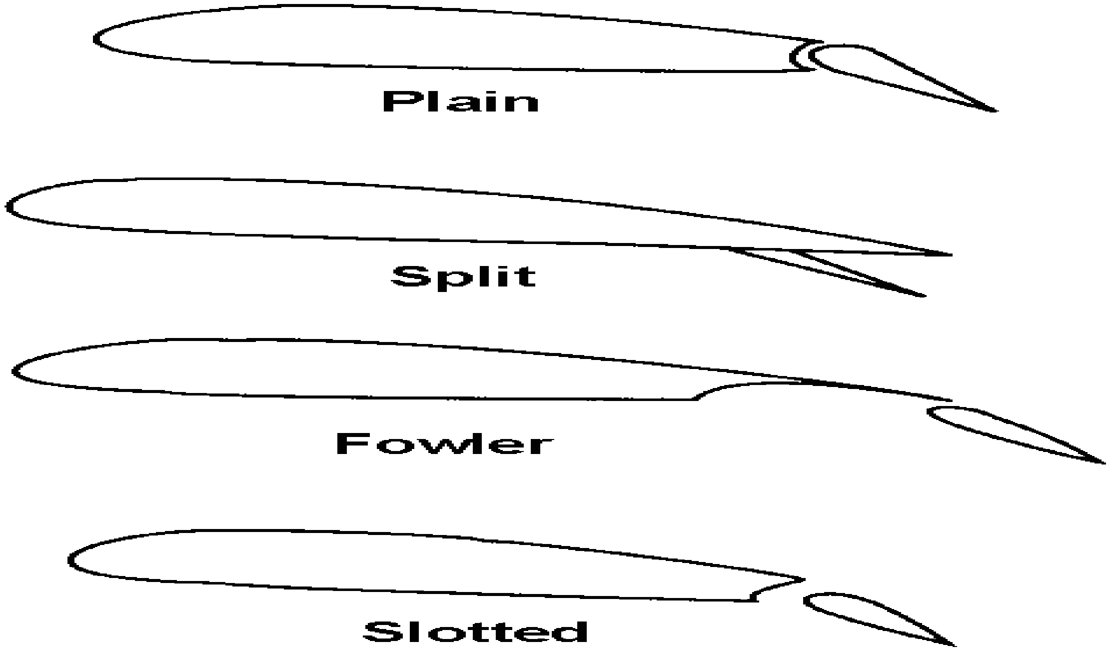
These are the various assumptions that go into developing potential flow panel methods:

- Inviscid
- Incompressible $\nabla \cdot V = 0$
- Irrotational $\nabla \times V = 0$
- Steady

However, the incompressible flow assumption may be removed from the potential flow derivation leaving:

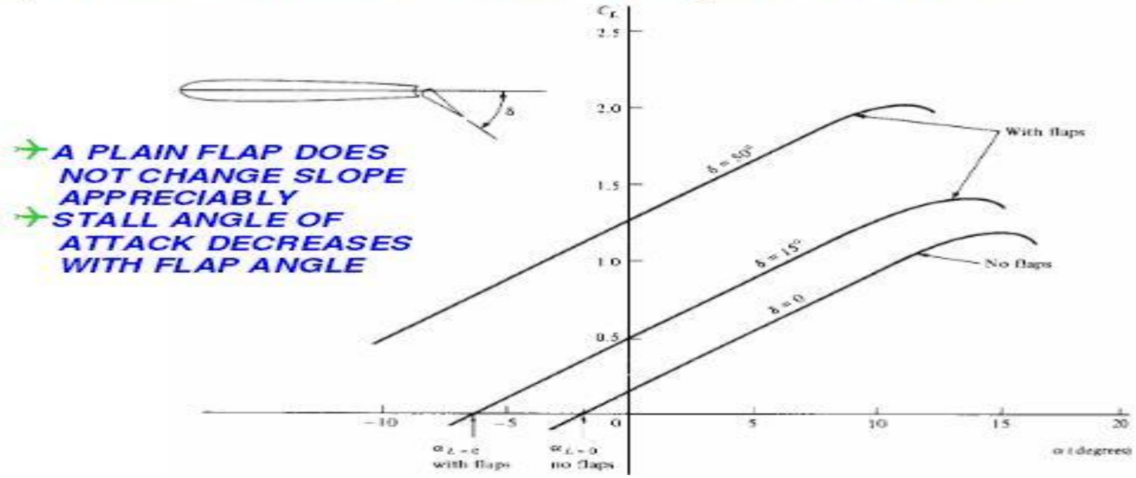
- Potential Flow (inviscid, irrotational, steady) $\nabla^2 \varphi = 0$.

High lift airfoils



High lift devices.

- Flaps are the most common high lift device



The most common high-lift device is the flap, a movable portion of the wing that can be lowered to produce extra lift. When a flap is lowered this re-shapes the wing section to give it more camber. Flaps are usually located on the trailing edge of a wing, while leading edge flaps are used occasionally.

UNIT-III
Finite Wing Theory

A *vortex line* is a line whose tangent is everywhere parallel to the local vorticity vector. The vortex lines drawn through each point of a closed curve constitute the surface of a *vortex tube*. Finally, a *vortex filament* is a vortex tube whose cross-section is of infinitesimal dimensions.

In fluid dynamics, circulation is the line integral around a closed curve of the velocity field. Circulation is normally denoted Γ

$$\Gamma_C = \oint_C \mathbf{v} \cdot d\mathbf{r},$$

Kelvin and Helmhotz theorem

According to the *Kelvin circulation theorem*, which is named after Lord Kelvin (1824-1907), the circulation around any co-moving loop in an inviscid fluid is independent of time. The proof is as follows. The circulation around a given loop

$$\Gamma_C = \oint_C \mathbf{v} \cdot d\mathbf{r}.$$

$$d\mathbf{v} = d(d\mathbf{r}/dt) = d(d\mathbf{r})/dt$$

However, for a loop that is co-moving with the fluid, we have . Thus,

$$\frac{d\Gamma_C}{dt} = \oint_C \frac{d\mathbf{v}}{dt} \cdot d\mathbf{r} + \oint_C \mathbf{v} \cdot d\mathbf{v}.$$

$$d\mathbf{v}/dt = D\mathbf{v}/Dt$$

$$\frac{D\mathbf{v}}{Dt} = -\nabla\left(\frac{p}{\rho} + \Psi\right),$$

$$\frac{d\Gamma_C}{dt} = -\oint_C \nabla\left(\frac{p}{\rho} - \frac{1}{2}v^2 + \Psi\right) \cdot d\mathbf{r} = 0,$$

$$\mathbf{v} \cdot d\mathbf{v} = d(v^2/2) = \nabla(v^2/2) \cdot d\mathbf{r}$$

$$p/\rho - v^2/2 + \Psi$$

One corollary of the Kelvin circulation theorem is that the fluid particles that form the walls of a vortex tube at a given instance in time continue to form the walls of a vortex tube at all subsequent times.

Helmholtz’s third theorem:

In the absence of rotational external forces, a fluid that is initially irrotational remains irrotational.

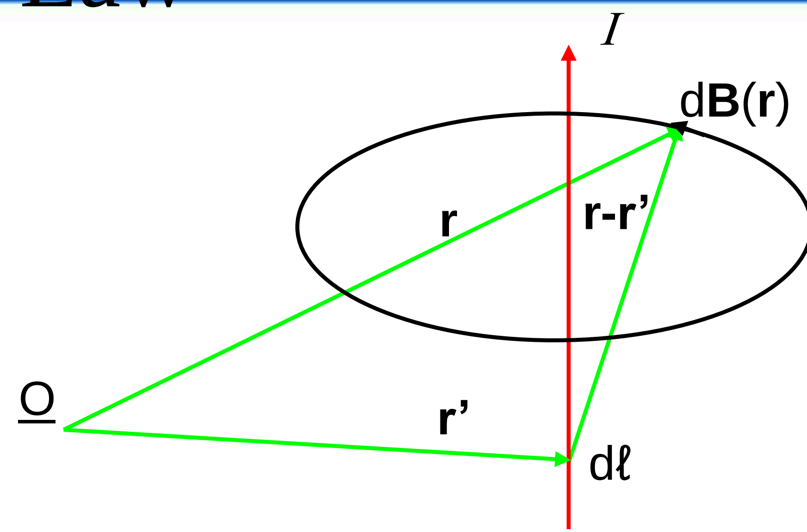
Helmholtz’s theorems apply to inviscid flows. In observations of vortices in real fluids the strength of the vortices always decays gradually due to the dissipative effect of viscous forces.

Alternative expressions of the three theorems are as follows:

1. The strength of a vortex tube does not vary with time.
2. Fluid elements lying on a vortex line at some instant continue to lie on that vortex line. More simply, vortex lines move with the fluid. Also vortex lines and tubes must appear as a closed loop, extend to infinity or start/end at solid boundaries.
3. Fluid elements initially free of vorticity remain free of vorticity.

Biot-Savart Law

- The analogue of Coulomb’s Law is the Biot-Savart Law
- Consider a current loop (*I*)
- For element d**ℓ** there is an associated element field d**B**



$$d\mathbf{B}(\mathbf{r}) = \frac{\mu_o I}{4\pi}\frac{d\ell \times (\mathbf{r}-\mathbf{r}')}{|\mathbf{r}-\mathbf{r}'|^3}$$

d**B** perpendicular to both d**ℓ** and **r** - **r’**

Inverse square dependence on distance

$\mu_o/4p = 10^{-7}$ Hm^{-1}

Integrate to get Biot-Savart Law

$$\mathbf{B}(\mathbf{r}) = \frac{\mu_o I}{4\pi}\oint_{\ell}\frac{d\ell \times (\mathbf{r}-\mathbf{r}')}{|\mathbf{r}-\mathbf{r}'|^3}$$

Biot-Savart Law examples

(1) Infinite straight conductor

$$\mathbf{B}(\mathbf{r}) = \frac{\mu_o I}{4\pi} \int_{-\infty}^{\infty} \frac{d\ell \times (\mathbf{r} - \mathbf{r'})}{|\mathbf{r} - \mathbf{r'}|^3}$$

$$d\ell \times (\mathbf{r} - \mathbf{r'}) = |d\ell||\mathbf{r} - \mathbf{r'}| \sin\theta \, \hat{n}$$

$$|\mathbf{r} - \mathbf{r'}|^2 = r^2 + z^2$$

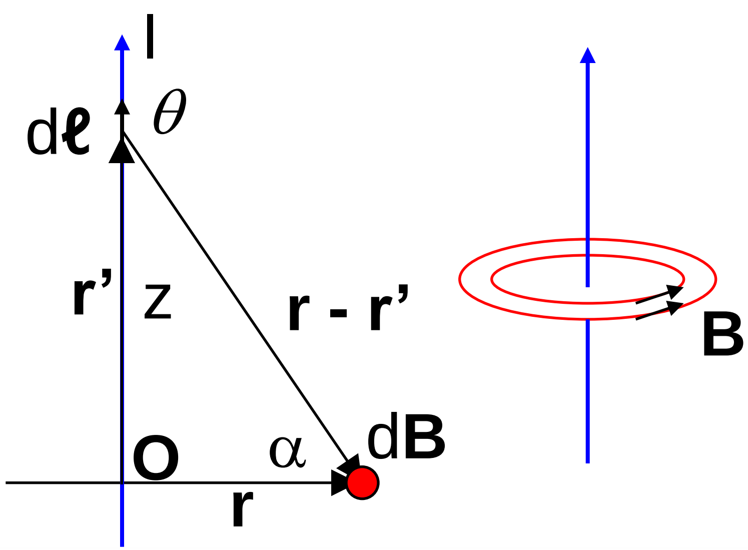
d**ℓ** and **r, r'** in the page

d**B** is into the page

B forms concentric circles about the conductor

$$\theta = \pi/2 + \alpha \qquad \sin\theta = \cos\alpha = \frac{r}{(r^2 + z^2)^{1/2}}$$

$$\frac{d\ell \times (\mathbf{r} - \mathbf{r'})}{|\mathbf{r} - \mathbf{r'}|^3} = \frac{|d\ell||\mathbf{r} - \mathbf{r'}| \sin\theta}{|\mathbf{r} - \mathbf{r'}|^3} \hat{n} = \frac{r \, dz}{(r^2 + z^2)^{3/2}} \hat{n}$$



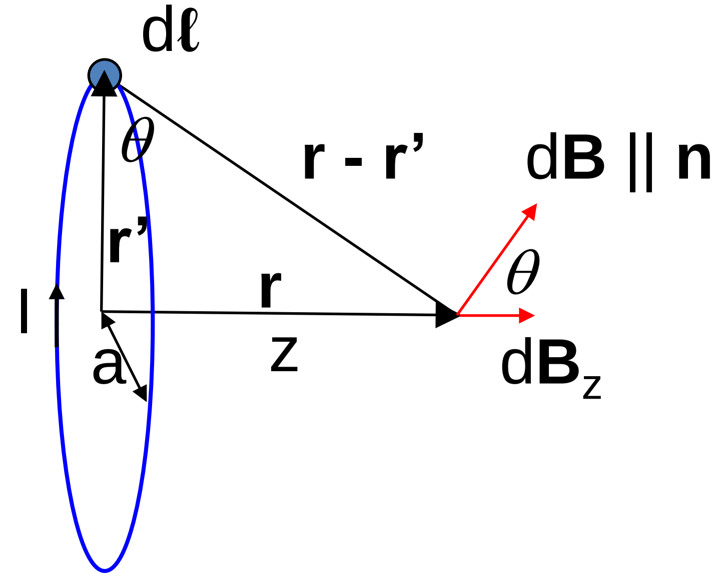
$$\mathbf{B} = \frac{\mu_o I}{4\pi} \int_{-\infty}^{\infty} \frac{r \, dz}{(r^2 + z^2)^{3/2}} \hat{n}$$

$$\int_{-\infty}^{\infty} \frac{r \, dz}{(r^2 + z^2)^{3/2}} = \frac{2}{r}$$

$$\mathbf{B} = \frac{\mu_o I}{2\pi r} \hat{n}$$

Biot-Savart Law examples

(2) Axial field of circular loop



Loop perpendicular to page, radius *a*

d**ℓ** out of page at top and **r, r'** in the page

On-axis element d**B** is in the page,

perpendicular to **r** - **r'**, at *q* to axis.

Magnitude of element d**B**

$$dB = \frac{\mu_o I}{4\pi} \frac{d\ell \times (\mathbf{r}-\mathbf{r}')}{|\mathbf{r}-\mathbf{r}'|^3} = \frac{\mu_o I}{4\pi} \frac{|d\ell|}{|\mathbf{r}-\mathbf{r}'|^2}\hat{n} \Rightarrow dB_z = \frac{\mu_o I}{4\pi} \frac{d\ell}{|\mathbf{r}-\mathbf{r}'|^2}\cos\theta$$

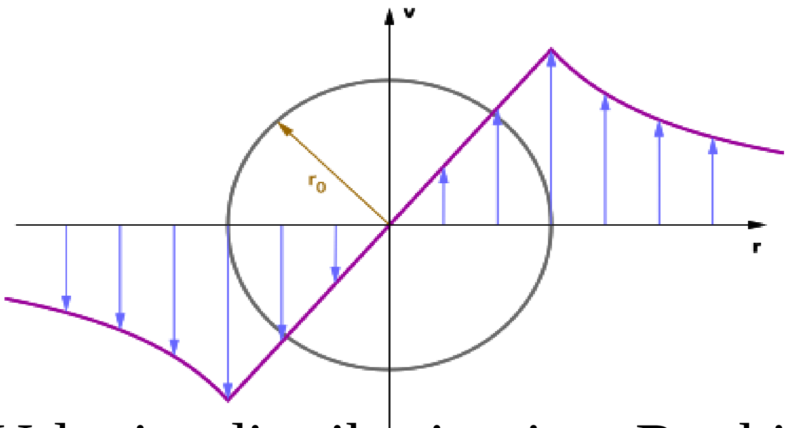
$$\cos\theta = \frac{a}{|\mathbf{r}-\mathbf{r}'|} = \frac{a}{(a^2+z^2)^{1/2}}$$

Integrating around loop, only z-component of d**B** contributes net result

Rankine vortex

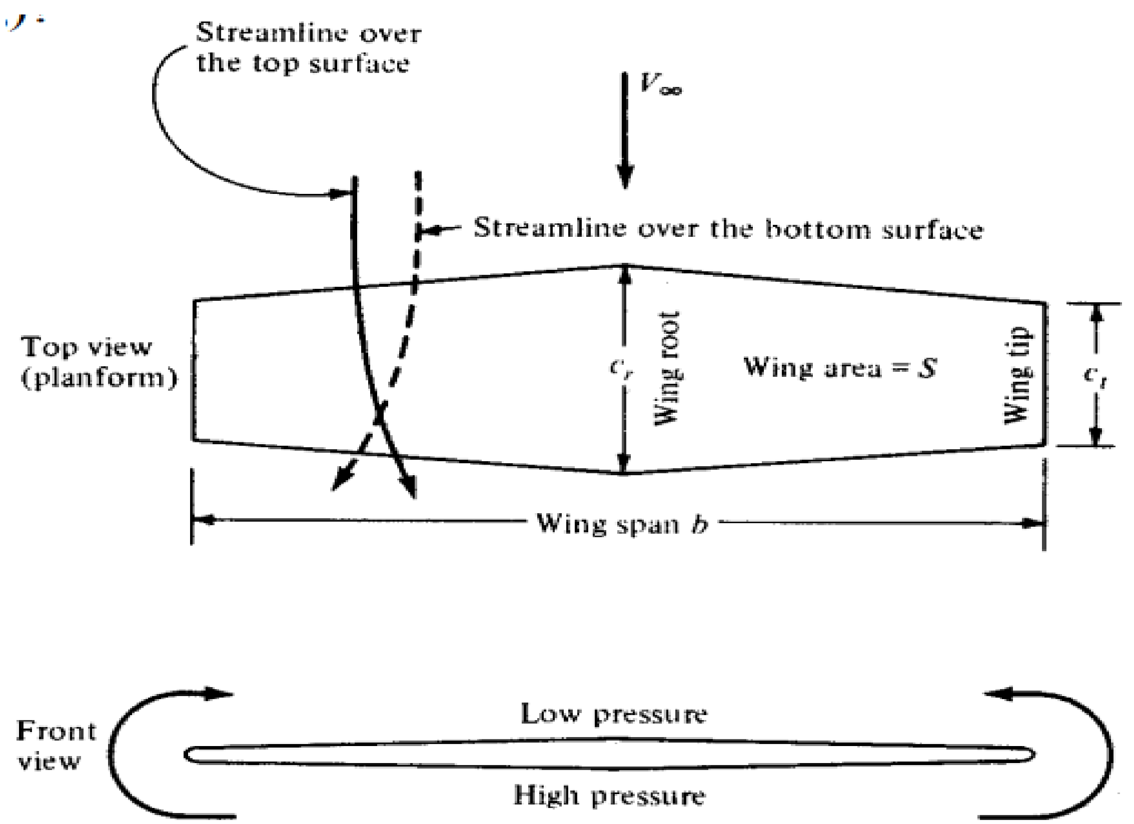
The Rankine vortex is a type of vortex in a viscous fluid. It is named after its discoverer, William John Macquorn Rankine.

A swirling flow in a viscous fluid can be characterized by a forced vortex in its central core, surrounded by a free vortex. In an inviscid fluid, on the other hand, a swirling flow consists entirely of the free vortex with a singularity at its center point instead of the forced vortex core. The tangential velocity of a Rankine vortex.



Velocity distribution in a Rankine vortex.

Flow past finite wings

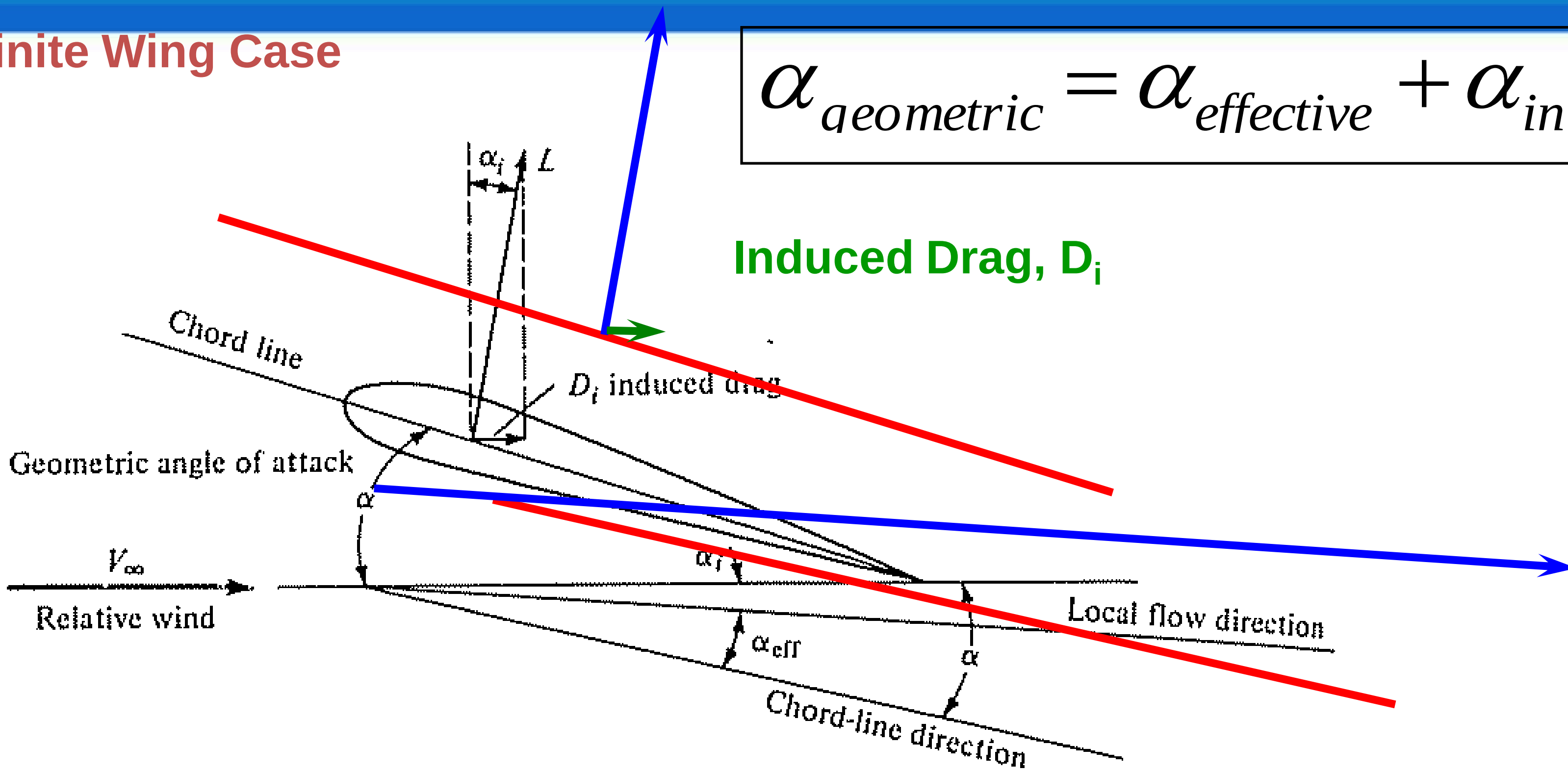


FINITE WING DESCRIPTION

Finite Wing Case

$$\alpha_{geometric} = \alpha_{effective} + \alpha_{induced}$$

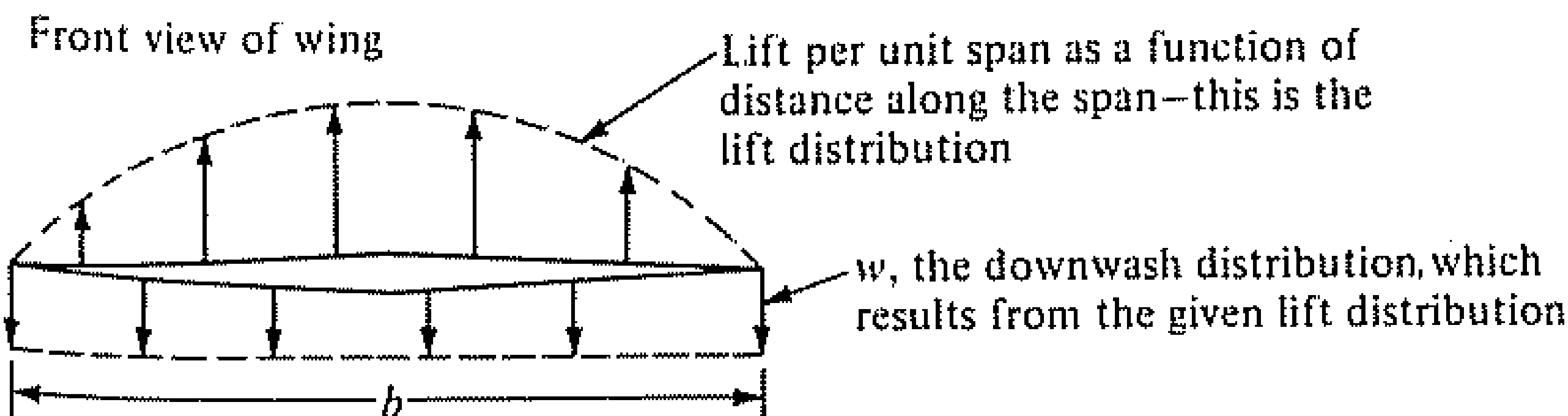
Induced Drag, D_i



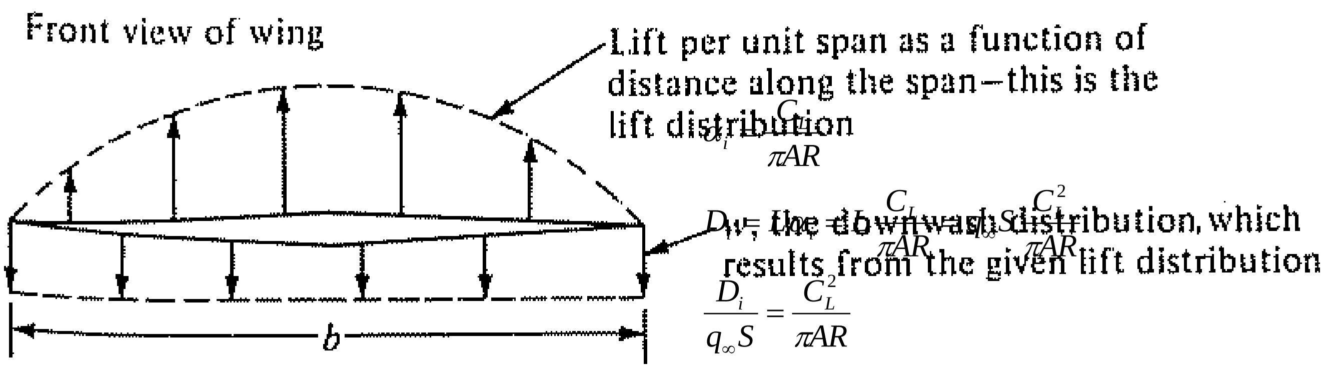
- Drag is measured in direction of ***incoming*** relative wind (that is the direction that the airplane is flying)
- Lift vector is tilted back
- Component of L acts in direction parallel to ***incoming*** relative wind → results in a new type of drag

INDUCED DRAG

- Calculation of angle a_i is not trivial (MAE 3241)
- Value of a_i depends on *distribution of downwash* along span of wing
- Downwash is governed by *distribution of lift* over span of wing



- Special Case: Elliptical Lift Distribution (produced by elliptical wing)
- Lift/unit span varies elliptically along span
- This special case produces a uniform downwash



$$\alpha_i = \frac{C_L}{\pi AR}$$

$$D_i = q_\infty S \frac{C_L^2}{\pi AR}$$

$$\frac{D_i}{q_\infty S} = \frac{C_L^2}{\pi AR}$$

$$C_{D,i} = \frac{C_L^2}{\pi AR}$$

Key Results:
Elliptical Lift Distribution

The Elliptical Lift Distribution

Solving the Fundamental Equation of Finite Wing Theory requires us to guess at a Γ (y) distribution and show it's a correct guess. (The same approach we used for the γ(x) distribution for thin airfoils.) As a first guess we consider an elliptic distribution:

$$\Gamma(y) = \Gamma_0 \left[1 - \left(\frac{2y}{b}\right)^2\right]^{1/2}$$

This distribution has circulation Γ0 at the root (y = 0) and Γ = 0 at the wingtips

$$w = -\frac{\Gamma_0}{2b}$$

$$c(y) = c_0 \left[1 - \left(\frac{2y}{b}\right)^2\right]^{1/2}$$

$$S = \frac{\pi c_0 b}{4} \quad \text{and} \quad AR = \frac{4b}{\pi c_0}$$

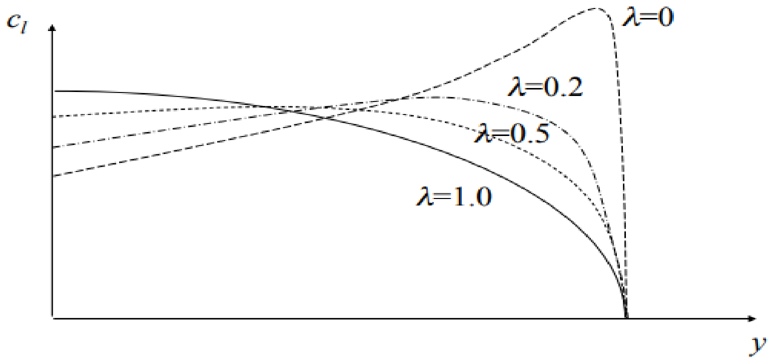
Tapered Wings

• Taper ratio is defined as à Reduction of the amount of lift near the wing-tip. à Tip vortex is weaker à Induced drag is smaller

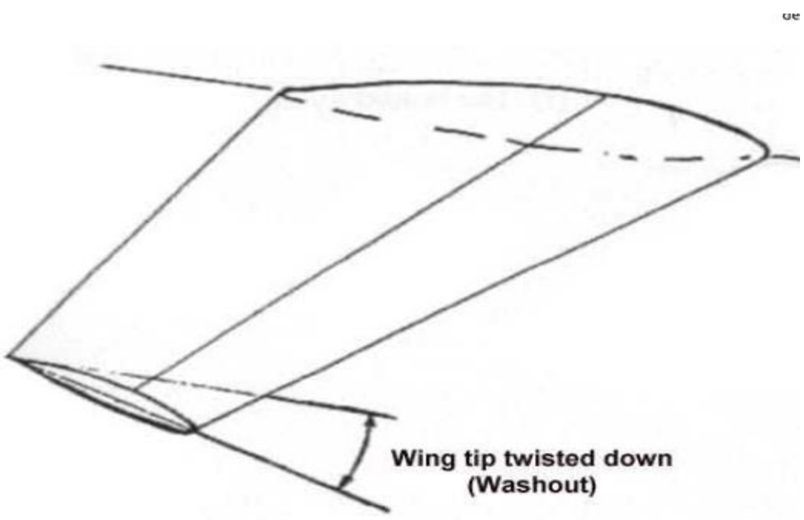
• Taper also reduces structural weight

• As the chord at the root is unchanged the maximum lift is not severely affected by taper

• If the taper is not too high, the stalling characteristics are acceptable, even without twist



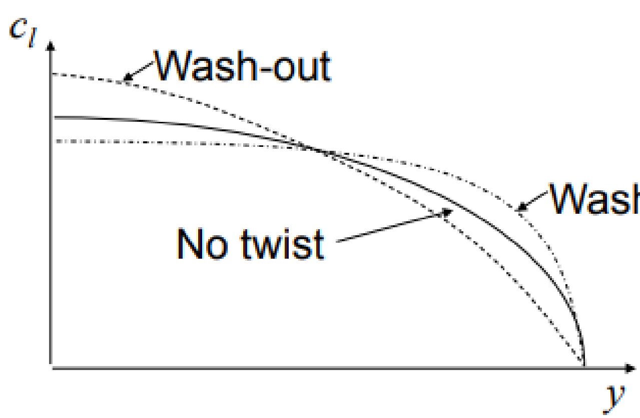
Effect of wing taper on lift distribution

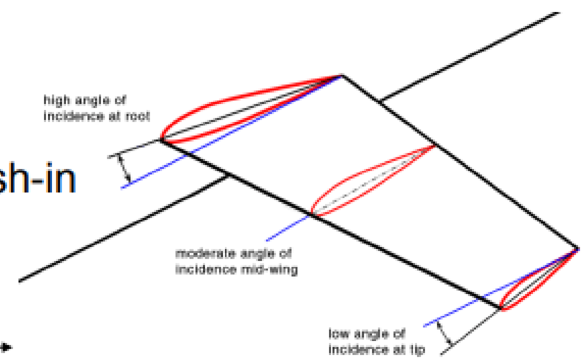


Twisted wings

Wing twist (1)

- Wash-in: αtip > αroot
- Wash-out: αtip < αroot



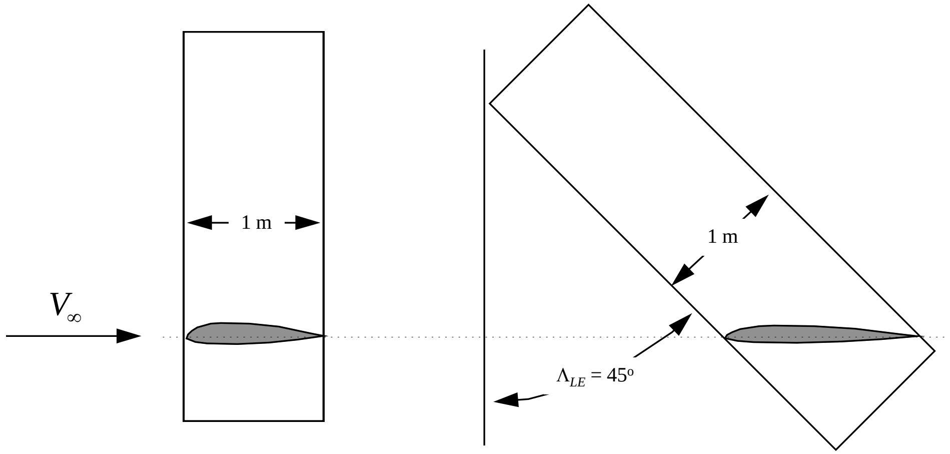


Wings often have wash-out to reduce structural weight and improve stall characteristics.

- The point of initial stalling should be sufficiently inboard, around 0.4s from the wing root.
- This can be achieved with suitable twist. If the stall point is too far outboard, a little washout will bring it inboard.
- However, a washout of more than 5o results in an unacceptable increase in induced drag.

Effect of Wing Sweep

In addition to reducing airfoil thickness, aircraft designers can also raise a wing's M_{crit} by sweeping it either forward or aft. To understand how this works, consider the untapered, swept wing in Figure 4.34. Sweeping the wing without changing its shape increases the effective chord length. Figure 4.34 shows why this is true.



The Effect of Wing Sweep on Streamwise Thickness-to-Chord Ratio

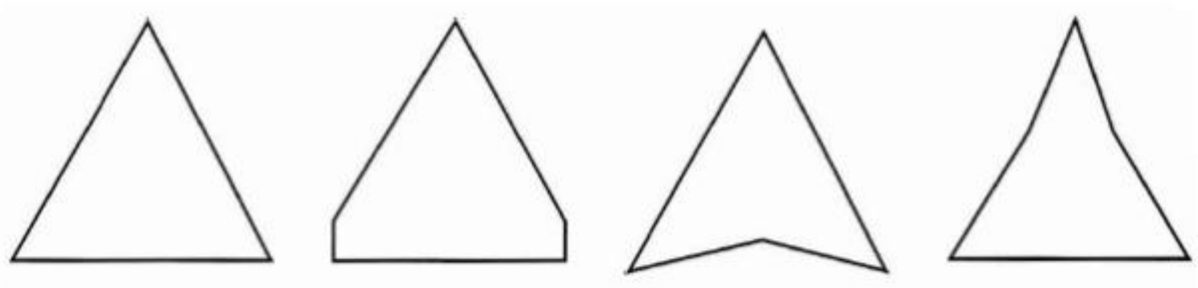
Delta wing

The delta wing is a wing shaped in the form of a triangle. It is named for its similarity in shape to the Greek uppercase letter delta (Δ).

Canard delta – Many modern fighter aircraft, such as the JAS 39 Gripen, the Eurofighter Typhoon and the Dassault Rafale use a combination of canards and a delta wing.
Tailed delta – adds a conventional tailplane (with horizontal tail surfaces), to improve handling. Common on Soviet types such as the Mikoyan-Gurevich MiG-21.
Cropped delta – tip is cut off. This helps avoid tip drag at high angles of attack. Used for example in all three Eurocanards (cropped, tailless delta combined with a canard).

Primary and secondary vortex

Swept wings that have platforms such as shown in Fig are called delta wings. dominant aspect of this flow is the two vortices that are formed along the highly swept leading edges, and that trail downstream over the top of the wing. This vortex pattern is created by the following mechanism. The pressure on the bottom surface of the wing is higher than the pressure on the top surface



(a) Simple delta (b) Cropped delta (c) Notched delta (d) Double delta

Thus, the flow on the bottom surface in the vicinity of the leading edge tries to curl around the leading edge from the bottom to the top. If the leading edge is relatively sharp, the flow will separate along its entire length. This separated flow curls into a primary vortex above the wing just inboard of each leading edge. The stream surface which has separated at the leading edge loops above the wing and then reattaches along the primary attachment line.

The primary vortex is contained within this loop. A secondary vortex is formed underneath the primary vortex, with its own separation line, and its own reattachment line.

Source Panel Method

Steps to determine the solution:

1. Write down the velocities, ui , vi , in terms of contributions from all the singularities. This includes qi , g from each panel and the influence coefficients which are a function of the geometry only.
2. Find the algebraic equations defining the "influence" coefficients. To generate the system of algebraic equations:
3. Write down flow tangency conditions in terms of the velocities (N eqn's., N+1 unknowns).
4. Write down the Kutta condition equation to get the N+1 equation.
5. Solve the resulting linear algebraic system of equations for the qi , g .

.

6. Given qi , g , write down the equations for uti, the tangential velocity at each panel control point.

7. Determine the pressure distribution from Bernoulli's equation using the tangential velocity on each panel. We now carry out each step in detail. The algebra gets tedious, but there's no problem in carrying it out.

8. As we carry out the analysis for two dimensions, consider the additional algebra required for the general three dimensional case

Vortex panel Method

PANEL is an exact implementation of the analysis given and is essentially the program given by Moran.6 Other panel method programs are available in the textbooks by Houghton and Carpenter,10 and Kuethe and Chow.11 Moran's program includes a subroutine to generate the ordinates for the NACA 4-digit and 5-digit airfoils (see Appendix A for a description of these airfoil sections).

The main drawback is the requirement for a trailing edge thickness that's exactly zero. To accommodate this restriction, the ordinates generated internally have been altered slightly from the official ordinates.

The extension of the program to handle arbitrary airfoils is an exercise. The freestream velocity in PANEL is assumed to be unity, since the inviscid solution in coefficient form is independent of scale. PANEL's node points are distributed employing the widely used cosine spacing function

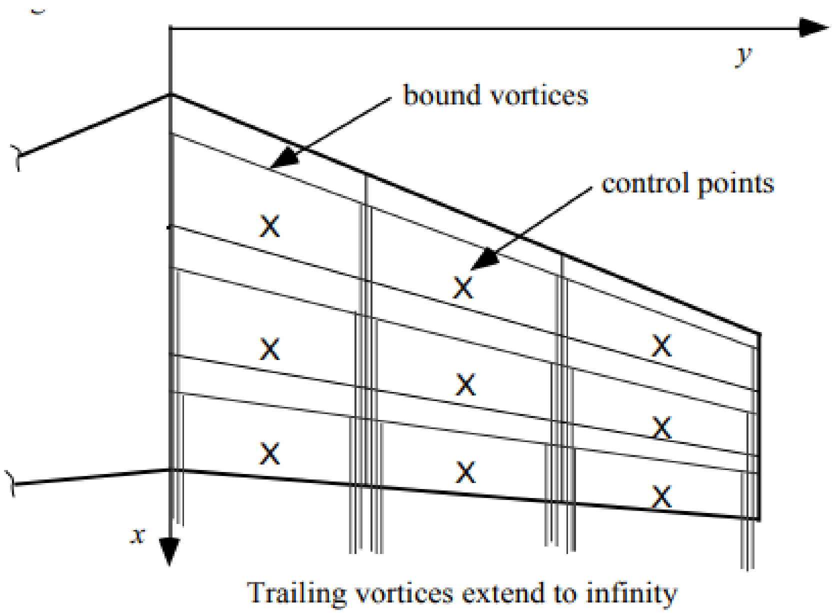
The Classical Vortex Lattice Method

There are many different vortex lattice schemes. In this section we describe the “classical” implementation. Knowing that vortices can represent lift from our airfoil analysis, and that one approach is to place the vortex and then satisfy the boundary condition using the “1/4 - 3/4 rule,” we proceed as follows:

1. Divide the planform up into a lattice of quadrilateral panels, and put a horseshoe vortex on each panel.
2. Place the bound vortex of the horseshoe vortex on the 1/4 chord element line of each panel.
3. Place the control point on the 3/4 chord point of each panel at the midpoint in the spanwise direction (sometimes the lateral panel centroid location is used) .

4. Assume a flat wake in the usual classical method.

5. Determine the strengths of each Gn required to satisfy the boundary conditions by solving a system of linear equations. The implementation is shown schematically



Note that the lift is on the bound vortices. To understand why, consider the vector statement of the Kutta-Joukowski Theorem, F = rV ´G . Assuming the freestream velocity is the primary contributor to the velocity, the trailing vortices are parallel to the velocity vector and hence the force on the trailing vortices are zero.

More accurate methods find the wake deformation required to eliminate the force in the presence of the complete induced flowfield.

UNIT-IV

Flow past non-lifting bodies and interference effects

INTRODUCTION

- Main function of the wing is to provide lift
  - Modeled using lifting-line theory
  - Uses method of singularities involving vortices
- Main function of the fuselage is to provide space for payload
  - Design of a slender body which offers low drag
  - Lift component of fuselage is relatively small
  - Can be considered as a non-lifting body
  - Modeled using method of singularities involving sources and doublets

ROAD MAP

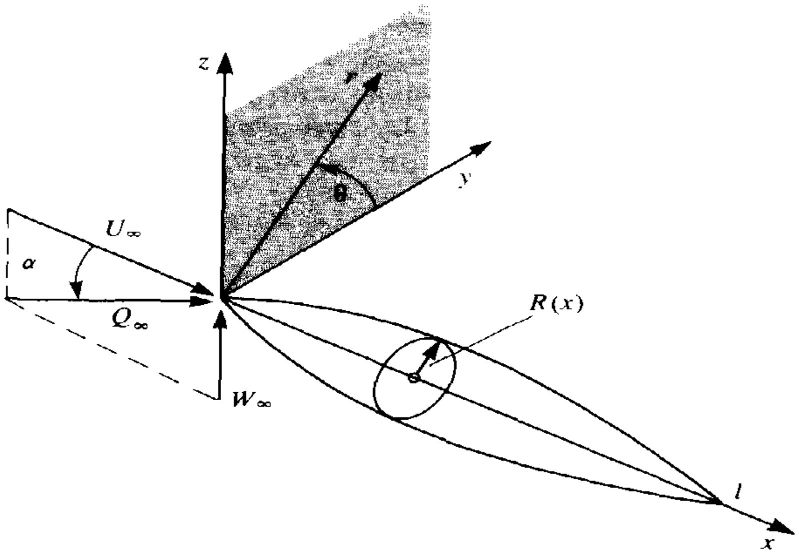
- Study the aerodynamic characteristics of non-lifting bodies like fuselage, using the slender body theory
- Study the interference effects between wing and fuselage
- Briefly note the effect of propeller slipstream on wing/tail
- Briefly note a few aspects of flow over the whole airplane

FLOW PAST NON-LIFTING BODIES

- Main function of the fuselage is to provide space for passengers/cargo
  - Design of a slender body which offers low drag
  - Lift component of fuselage is relatively small
  - Can be considered as a non-lifting body
- Analyze the flow over fuselage using the slender body theory
  - Uses method of singularities involving sources and doublets

SLENDER BODY THEORY

- Flow past a slender body



- Assumptions for modeling as a slender body
  - Low slenderness ratio, R(x)/l << 1
  - Small angle of attack, α << 1
  - Small ratio of body radius to length, |dR(x)/dx| << 1

SLENDER BODY THEORY

- Solve Laplace equation governing the flow past the slender body, represented in cylindrical (

$$\nabla^2\Phi = \frac{\partial^2\Phi}{\partial x^2} + \frac{\partial^2\Phi}{\partial r^2} + \frac{1}{r}\frac{\partial\Phi}{\partial r} + \frac{1}{r^2}\frac{\partial^2\Phi}{\partial\theta^2} = 0$$

- In this coordinate ... eestream velocity is

$$\mathbf{Q}_\infty = U_\infty \mathbf{e}_x + W_\infty \mathbf{e}_z$$

$$\approx Q_\infty[\mathbf{e}_x + \alpha(\sin\theta\, \mathbf{e}_r + \cos\theta\, \mathbf{e}_\theta)]$$

- Solve Laplace equation governing the flow past the slender body, represented in cylindrical coordinate system

$$\nabla^2\Phi = \frac{\partial^2\Phi}{\partial x^2} + \frac{\partial^2\Phi}{\partial r^2} + \frac{1}{r}\frac{\partial\Phi}{\partial r} + \frac{1}{r^2}\frac{\partial^2\Phi}{\partial\theta^2} = 0$$

- In this coordinate system, freestream velocity is

$$\mathbf{Q}_\infty = U_\infty \mathbf{e}_x + W_\infty \mathbf{e}_z$$

$$\approx Q_\infty[\mathbf{e}_x + \alpha(\sin\theta\,\mathbf{e}_r + \cos\theta\,\mathbf{e}_\theta)]$$

- Boundary condition for Laplace equation
  - Solid wall condition
  - Surface of the body: $F \equiv r - R(x) = 0$
  - **Q.n** = 0 => $\nabla\Phi^* \cdot \nabla F = 0$
- Application of boundary condition gives:

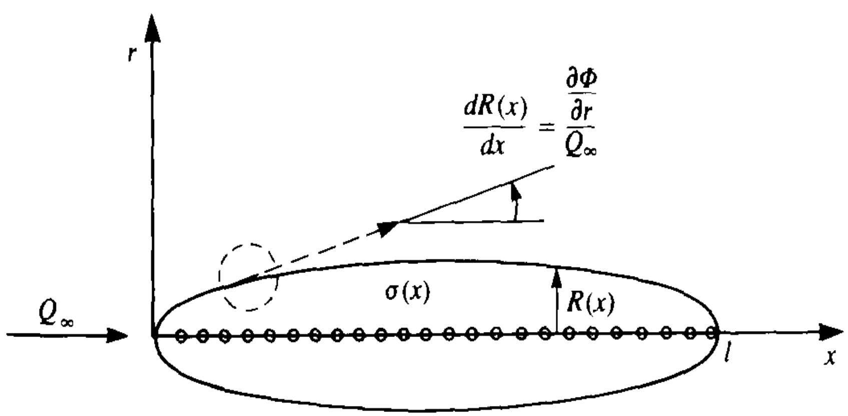
$$\frac{\partial \Phi}{\partial r}(x, R, \theta) = Q_\infty R'(x) - Q_\infty \alpha \sin\theta$$

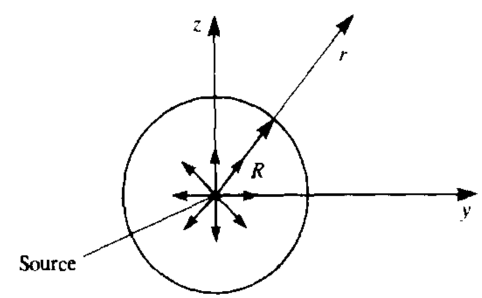
$$\frac{\partial \phi_a(x,R,\theta)}{\partial r} + \frac{\partial \phi_t(x,R,\theta)}{\partial r}$$

- The above condition is superposition of:
  - Longitudinal axisymmetric flow and
  - Transverse flow

AXISYMMETRIC LONGITUDINAL FLOW

- Laplace equation: $$\frac{\partial^2\Phi}{\partial x^2}+\frac{\partial^2\Phi}{\partial r^2}+\frac{1}{r}\frac{\partial\Phi}{\partial r}=0$$
- Corresponding boundary condition: $$\frac{\partial\Phi}{\partial r}(x, R, \theta)=Q_\infty R'(x)$$
- Modeled using method of singularities involving sources:





AXISYMMETRIC LONGITUDINAL FLOW

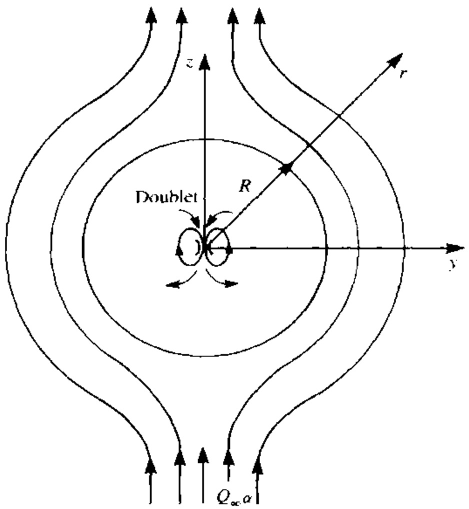
- Using potential theory, solution for axisymmetric longitudinal flow:

$$\Phi(r, x) = \frac{-Q_\infty}{4\pi} \int_0^l \frac{S'(x_0)\, dx_0}{\sqrt{(x - x_0)^2 + r^2}}$$

$$q_r(r, x) = \frac{\partial \Phi}{\partial r} = \frac{Q_\infty}{4\pi} \int_0^l \frac{S'(x_0) r\, dx_0}{[(x - x_0)^2 + r^2]^{3/2}}$$

$$q_x(r, x) = \frac{\partial \Phi}{\partial x} = \frac{Q_\infty}{4\pi} \int_0^l \frac{S'(x_0)(x - x_0)\, dx_0}{[(x - x_0)^2 + r^2]^{3/2}}$$

- Laplace equation: $\nabla^2\Phi = \frac{\partial^2\Phi}{\partial x^2} + \frac{\partial^2\Phi}{\partial r^2} + \frac{1}{r}\frac{\partial\Phi}{\partial r} + \frac{1}{r^2}\frac{\partial^2\Phi}{\partial\theta^2} = 0$
- Corresponding boundary condition: $\frac{\partial\Phi}{\partial r}(x, R, \theta) = -Q_\infty\alpha\sin\theta$
- Modeled using method of singularities involving doublets:



- Using potential theory, solution for transverse flow:

$$\Phi(r, \theta, x) = Q_\infty \alpha R^2 \frac{\sin\theta}{r}$$

$$q_r(r, \theta, x) = -Q_\infty \alpha R^2 \frac{\sin\theta}{r^2}$$

$$q_\theta(r, \theta, x) = Q_\infty \alpha R^2 \frac{\cos\theta}{r^2}$$

$$q_x(r, \theta, x) = \frac{\partial \Phi}{\partial x} = 2 Q_\infty \alpha R R' \frac{\sin\theta}{r}$$

COMPLETE SOLUTION FOR FLOW PAST SLENDER BODY (LIKE FUSELAGE)

- Adding up the potentials for longitudinal and transverse flows, we get:

$$\frac{dF_x}{dx} = \left\{ p_\infty - \frac{1}{2}\rho Q_\infty^2 \left[\alpha^2 + \frac{q_{xA}}{Q_\infty} + (R')^2 \right] \right\} S'$$

$$\frac{dF_y}{dx} = 0$$

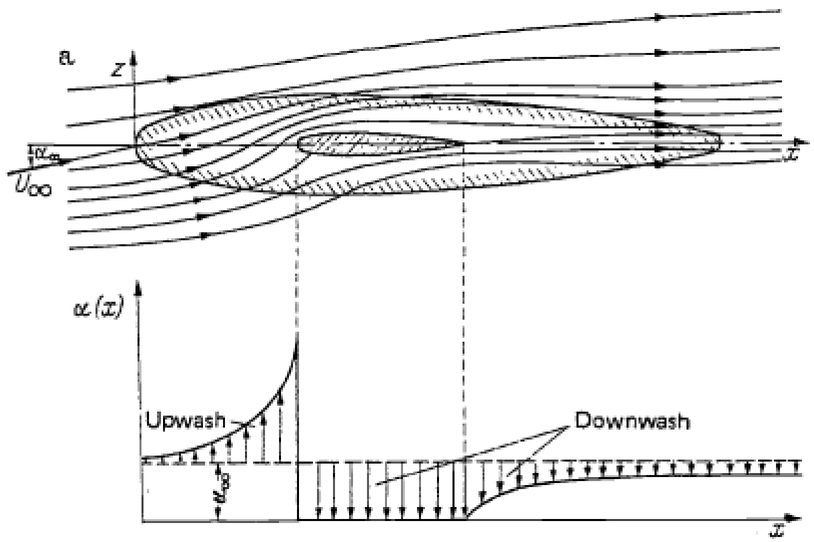
$$\frac{dF_z}{dx} = \rho Q_\infty^2 \alpha S'$$

- Conclusions from the above expressions:
  - The side force distribution is zero and therefore the side force is also zero.
  - The normal force distribution is proportional to the AOA and the rate of change of cross-sectional area.
  - The normal force distribution can become zero if the body's ends are pointed.
  - The axial force can also become zero if the body's ends are pointed
  - For pointed slender bodies, there is no lift and pressure drag, but there is an aerodynamic pitching moment.

WING-BODY INTERFERENCE

- Interference effects can be of the same order of magnitude as the aerodynamic effects of the individual parts
- Wing-fuselage interference
  - Wing affects flow field around fuselage
  - Fuselage affects flow field around wing
  - Two cases to be discussed:
    - Symmetric flow around the wing-fuselage system
    - Asymmetric flow around the wing-fuselage system

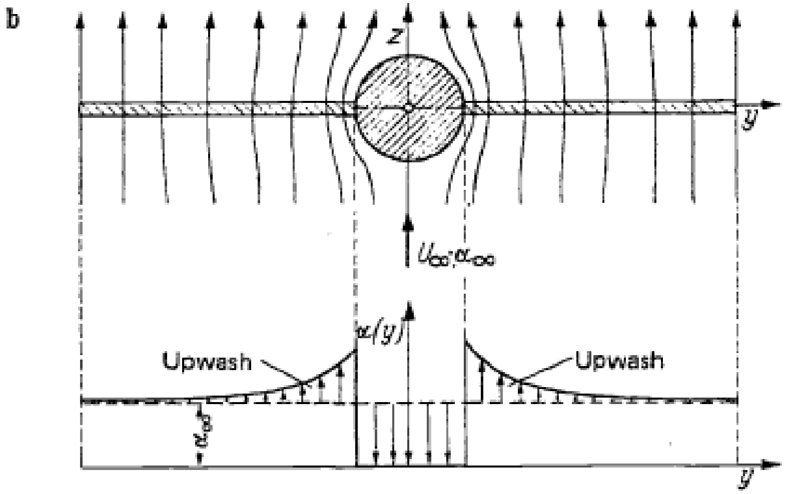
- Along the fuselage axis:
  - Additive velocities normal to the fuselage axis are induced by the wing
  - Near wing-fuselage penetration, the flow is parallel to the wing chord



SYMMETRIC FLOW: EFFECT OF WING ON FUSELAGE

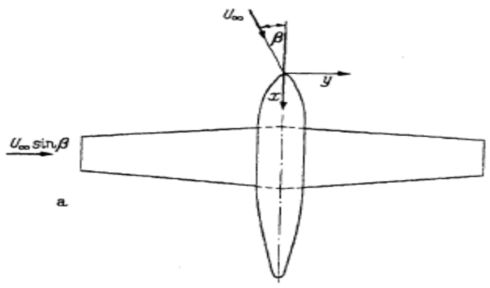
- Fuselage is therefore in a curved flow
- Angle-of-attack distribution $\alpha(x)$ varies along the fuselage axis
- This induced AOA distribution, shows that fuselage is subjected to an additive nose-up pitching moment

- Component of the incident flow velocity normal to the fuselage axis U.*sin*α ≈ Uα, generates additive upwash velocities in the vicinity of the fuselage
- Induced velocities normal to the plane of the wing with an additive symmetric angle-of-attack distribution over the wing span (twist angle)



ASYMMETRIC FLOW: EFFECT OF WING ON FUSELAGE

- Flow field of a wing-fuselage system at subsonic velocity in asymmetric, incident flow (angle of sideslip ≠ 0) :

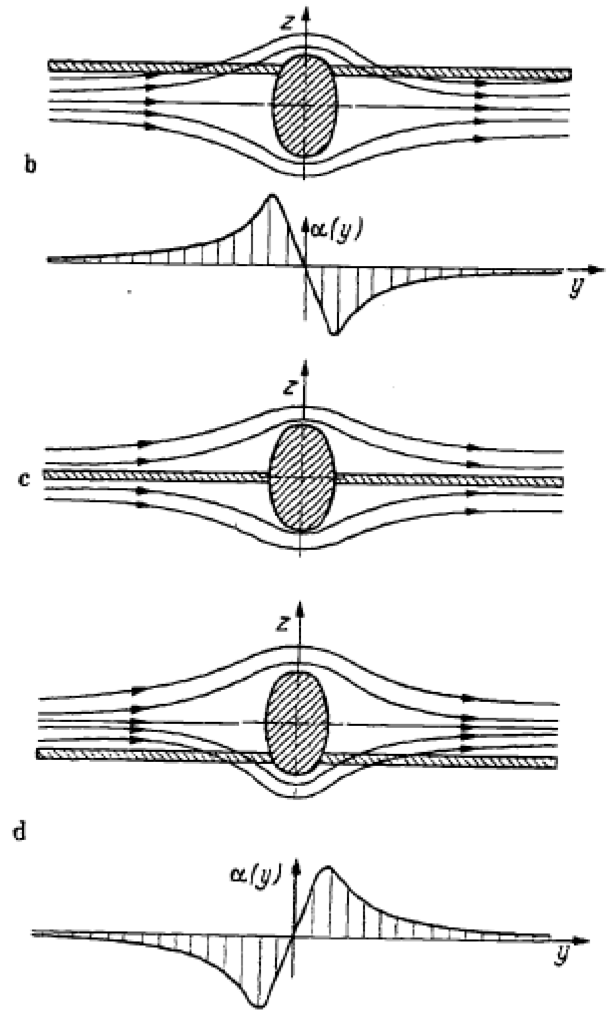


- Flow fields divided into:
  - Incident flow parallel to the plane of symmetry, of velocity $U.cos\beta \approx U$
  - Cross flow normal to the plane of symmetry, of velocity $U.sin\beta \approx U\beta$

CROSS FLOW ON WING-FUSELAGE SYSTEM

- Lift distributions over the wing span generated by the cross flow have reversed signs for high-wing and low-wing airplanes.
- The rolling moment due to sideslip:
  - Positive for the high-wing airplane
  - Zero for the mid-wing airplane
  - Negative for the low-wing airplane

CROSS FLOW ON WING-FUSELAGE SYSTEM



EFFECTS OF PROPELLER STREAM ON THE WING/TAIL DOWNSTREAM

- Experimental studies showed that when the wing/tail was under the effect of the slipstream (jet) from a propeller whose axis was fixed in the direction of the undisturbed wind, the rotation and the dynamic pressure changes in the jet resulted in a nonsymmetrical variation in the lift. Study of the downwash relations led to the result that the two portions into which the jet is divided by the wing/tail did not again reunite behind the wing/tail but that each portion experienced a lateral deviation in the direction of the jet rotation.

FLOW OVER WHOLE AIRPLANE

- Aircraft lift coefficient
  - Complete aircraft usually generates more lift than its wing alone

$$C_{L\alpha\ (whole\ aircraft)} = C_{L\alpha\ (wing)} + C_{L\alpha\ (horizontal\ tail)}$$

- Aircraft drag coefficient

$$C_D = C_{D_{0L}} + \frac{C_L^2}{\pi AR e}$$

where:

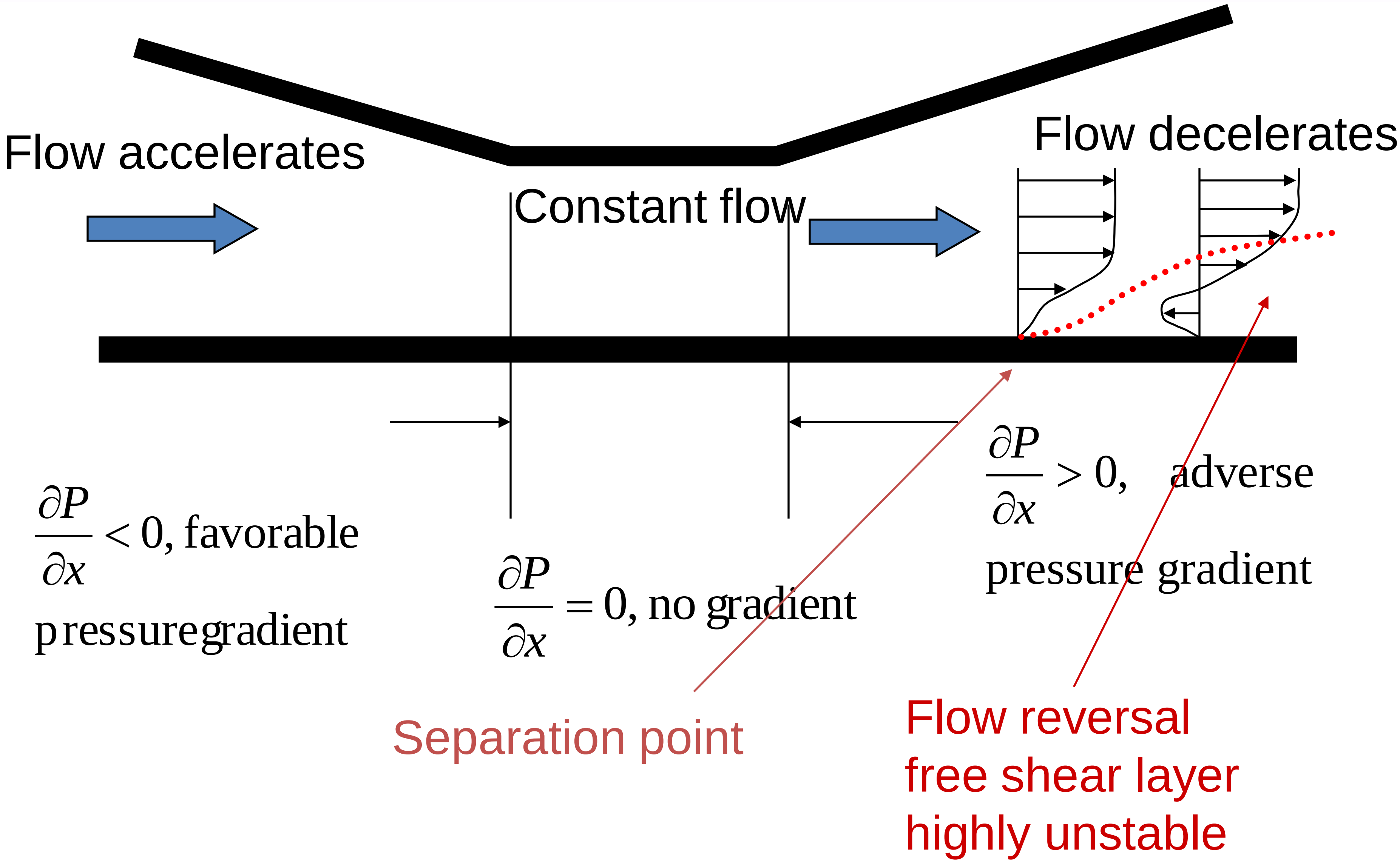
- C_{DOL} = zero-lift drag coefficient, parasite drag coefficient
- e = Oswald (aircraft) efficiency factor
- *AR* = Aspect ratio of the wing

UNIT-V
Boundary Layer Theory

Boundary layer

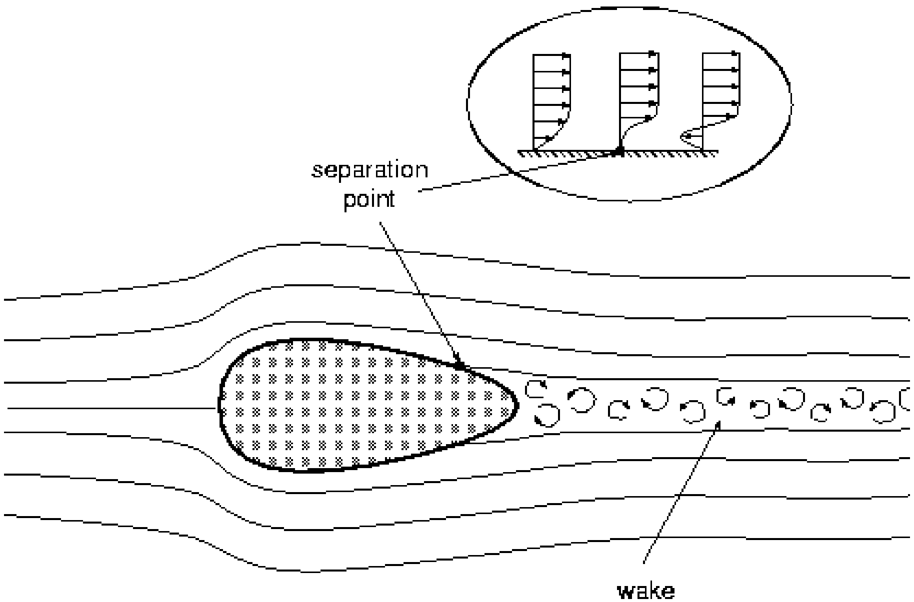
- In physics and fluid mechanics, a boundary layer is an important concept and refers to the layer of fluid in the immediate vicinity of a bounding surface where the effects of viscosity are significant.
- In the Earth's atmosphere, the atmospheric boundary layer is the air layer near the ground affected by diurnal heat, moisture or momentum transfer to or from the surface. On
  an aircraft wing the boundary layer is the part of the flow close to the wing, where viscous forces distort the surrounding non-viscous flow.

Boundary Layer and separation



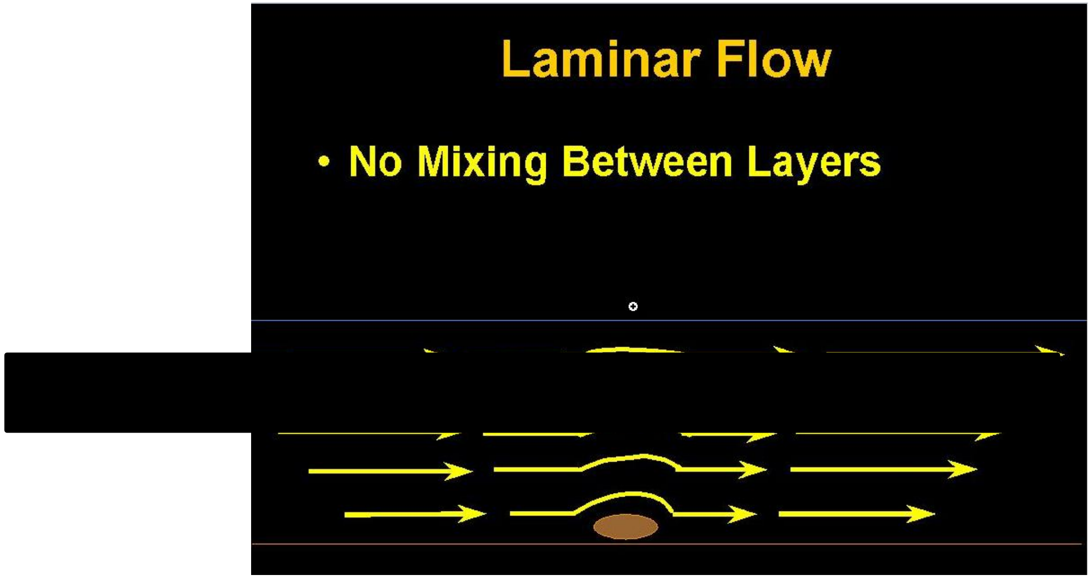
Flow separation

- Flow separation occurs when:
  - the velocity at the wall is zero or negative and an inflection point exists in the velocity profile,
  - and a positive or adverse pressure gradient occurs in the direction of flow.



Laminar flow

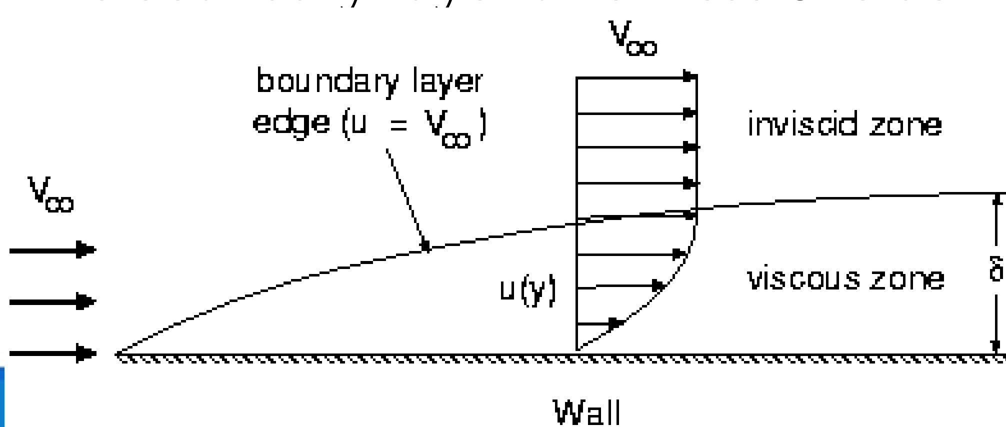
- Also known as streamline flow
- Occurs when the fluid flows in parallel layers, with no disruption between the layers
- The opposite of turbulent flow (rough)



The turbulent boundary layer

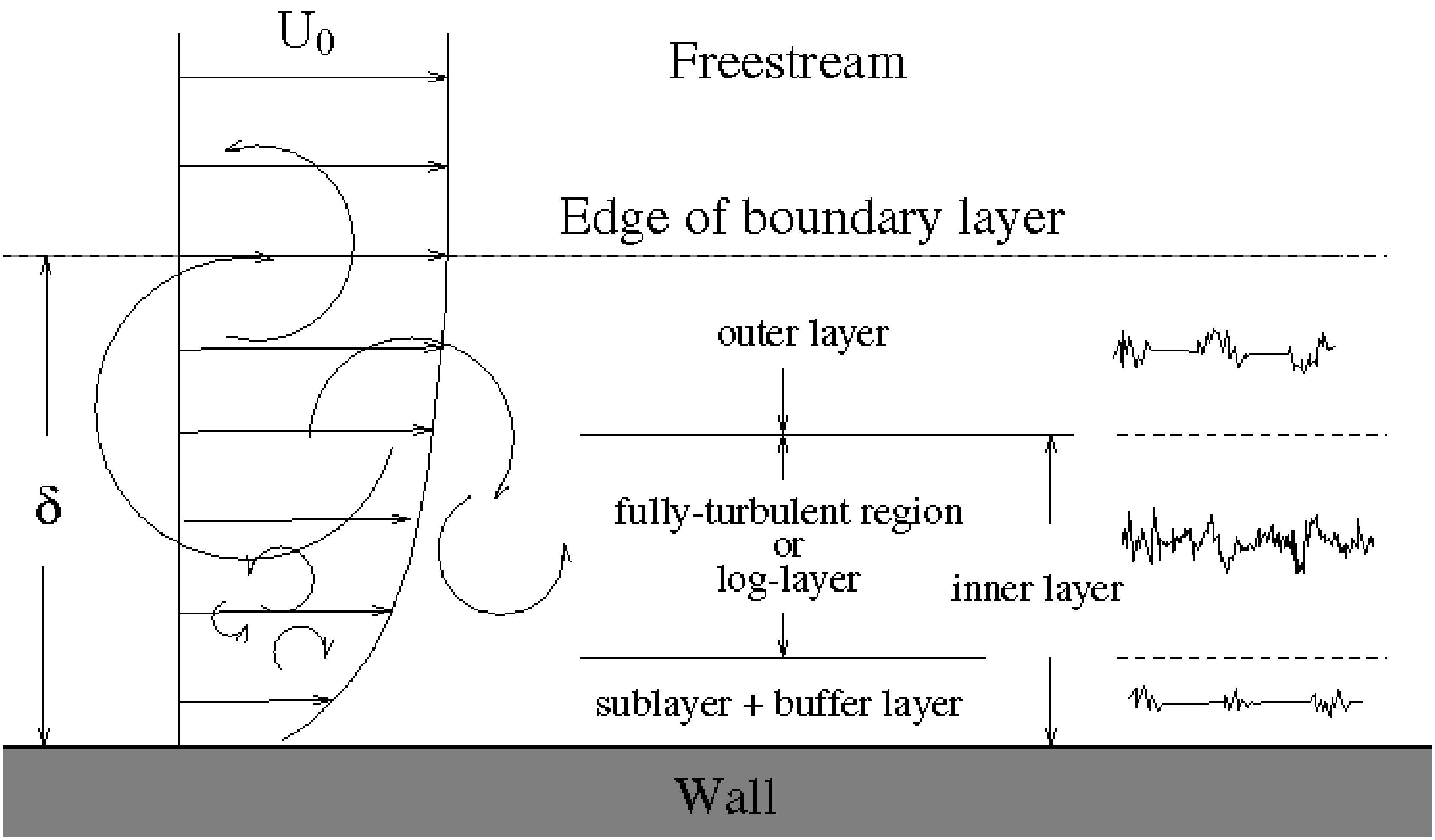
- In turbulent flow, the boundary layer is defined as the thin region on the surface of a body in which viscous effects are important.
- The boundary layer allows the fluid to transition from the free stream velocity U_τ to a velocity of zero at the wall.
- The velocity component normal to the surface is much smaller than the velocity parallel to the surface: $v << u$.
- The gradients of the flow across the layer are much greater than the gradients in the flow direction.
- The boundary layer thickness δ is defined as the distance away

ches 99% of the free-



$$\delta = y,\ where\ {}^{u}\!/_{U} = 0.99$$

The turbulent boundary layer



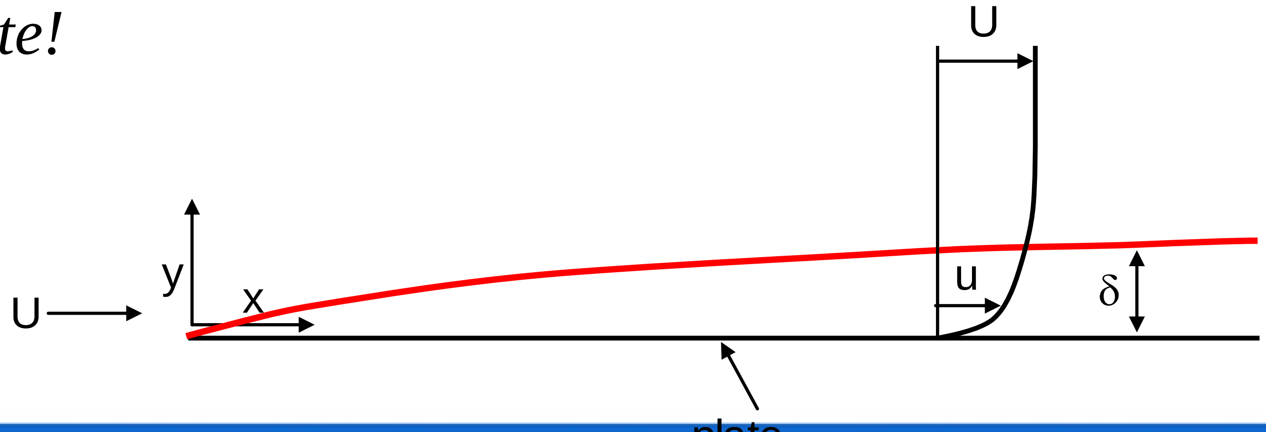
BOUNDARY LAYER ON A FLAT PLATE

Consider the following scenario.

1. A *steady* potential flow has constant velocity U in the x direction.
2. An infinitely thin flat plate is placed into this flow so that the plate is parallel to the potential flow (0 *angle of incidence*).

Viscosity should retard the flow, thus creating a boundary layer on either side of the plate. Here only the boundary layer on one side of the plate is considered. The flow is assumed to be laminar.

Boundary layer theory allows us to calculate the *drag on the plate!*



A steady, rectilinear potential flow in the x direction is described by the relations

$$\phi = Ux \quad , \quad u = \frac{\partial \phi}{\partial x} = U \quad , \quad v = \frac{\partial \phi}{\partial y} = 0$$

According to Bernoulli's equation for potential flows, the dynamic pressure of the potential flow p_{pd} is related to the velocity field as

$$p_{pd} + \frac{1}{2}\rho(u^2 + v^2) = const$$

Between the above two equations, then, for this flow

$$\frac{\partial p_{pd}}{\partial x} = \frac{\partial p_{pd}}{\partial y} = 0$$

BOUNDARY LAYER EQUATIONS FOR A FLAT PLATE

For the case of a steady, laminar boundary layer on a flat plate at 0 angle of incidence, with vanishing imposed pressure gradient, the boundary layer equations and boundary conditions become (see Slide 15 of BoundaryLayerApprox.ppt with $dp_{pds}/dx = 0$)

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{dp_{pds}}{dx} + \nu\frac{\partial^2 u}{\partial y^2}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

→

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = \nu\frac{\partial^2 u}{\partial y^2}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

$$u|_{y=0} = 0 \quad , \quad v|_{y=0} = 0 \quad , \quad u|_{y=\infty} = U$$

Tangential and normal velocities vanish at boundary: tangential velocity = free stream velocity far from plate

The definition of the displacement thickness for compressible flow is based on mass flow rate.

$$U_\infty \delta^* = \int_0^\infty (U_\infty - u)\,dy$$

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

$$dy = \delta d\eta = \sqrt{\frac{\nu x}{U_\infty}}\, d\eta$$

$$\rho U_\infty^2 \delta^{**} = \int_0^\infty \rho u (U_\infty - u)\,dy$$

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

$$or \quad \delta^{**} = 0.664\sqrt{\frac{\nu x}{U_\infty}} = \frac{0.664x}{\sqrt{\mathrm{Re}_x}}$$

Effect of curvature on boundary layer

- The effect of transverse surface curvature on the turbulent boundary layer is reviewed by recourse to experiments on axial flow along a circular cylinder. Three flow regimes are identified depending on values of the two controlling parameters, namely, the Reynolds number and the ratio of the boundary layer thickness to cylinder radius.
- The boundary layer flow resembles a wake when both parameters are large. As expected, the effect of curvature is small when the Reynolds number is large and the boundary layer is thin. When the boundary layer is thick and the Reynolds number is small, which is typical of laboratory investigations, the effect of transverse curvature is felt throughout the boundary layer with evidence for relaminarization at the low Reynolds numbers. This review describes the experimental evidence and points out gaps that remain

Temperature boundary layer

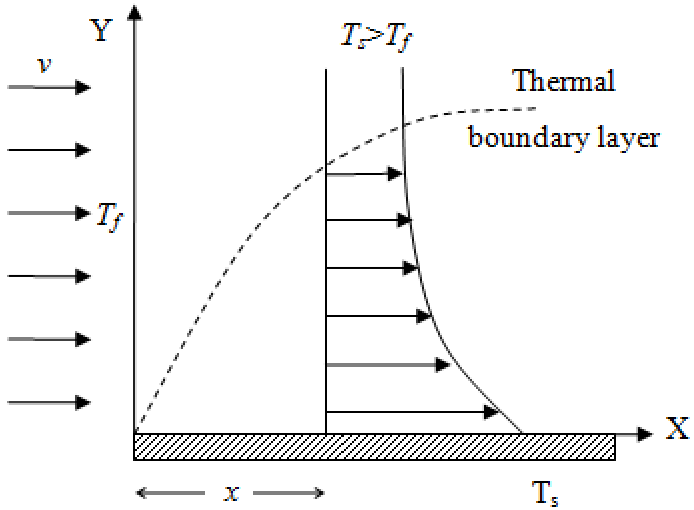


Fig.4.3: Thermal boundary layer flow past a flat surface

If $T_s > T_f$, the fluid temperature approaches asymptotically and the temperature profile at a distance x However, a thermal boundary may be defined (similar to velocity boundary) as the distance from the surface to the point where the temperature is within 1% of the free stream fluid temperature (T_f).

- Outside the thermal boundary layer the fluid is assumed to be a heat sink at a uniform temperature of T_f. The thermal boundary layer is generally not coincident with the velocity boundary layer, although it is certainly dependant on it.
- That is, the velocity, boundary layer thickness, the variation of velocity, whether the flow is laminar or turbulent etc are all the factors which determine the temperature variation in the thermal boundary layer.