Lecture 11 – Boundary Layers and Separation

Applied Computational Fluid Dynamics

Instructor: André Bakker

Overview

- Drag.
- The boundary-layer concept.
- Laminar boundary-layers.
- Turbulent boundary-layers.
- Flow separation.

The drag force

- The surrounding fluid exerts pressure forces and viscous forces on an object.


- The components of the resultant force acting on the object immersed in the fluid are the drag force and the lift force.

- The drag force acts in the direction of the motion of the fluid relative to the object.

- The lift force acts normal to the flow direction.

- Both are influenced by the size and shape of the object and the Reynolds number of the flow.

Drag prediction

- The drag force is due to the pressure and shear forces acting on the surface of the object.

- The tangential shear stresses acting on the object produce friction drag (or viscous drag). Friction drag is dominant in flow past a flat plate and is given by the surface shear stress times the area: $F_{d,viscous} = A.\tau_w$

- Pressure or form drag results from variations in the the normal pressure around the object: $F_{d,pressure} = \oint_A p \, da_n$

- In order to predict the drag on an object correctly, we need to correctly predict the pressure field and the surface shear stress.

- This, in turn, requires correct treatment and prediction of boundary layers and flow separation.

- We will discuss both in this lecture.

Viscous boundary layer

- An originally laminar flow is affected by the presence of the walls.
- Flow over flat plate is visualized by introducing bubbles that follow the local fluid velocity.
- Most of the flow is unaffected by the presence of the plate.
- However, in the region closest to the wall, the velocity decreases to zero.
- The flow away from the walls can be treated as inviscid, and can sometimes be approximated as potential flow.
- The region near the wall where the viscous forces are of the same order as the inertial forces is termed the boundary layer.
- The distance over which the viscous forces have an effect is termed the boundary layer thickness.
- The thickness is a function of the ratio between the inertial forces and the viscous forces, i.e. the Reynolds number. As Re increases, the thickness decreases.
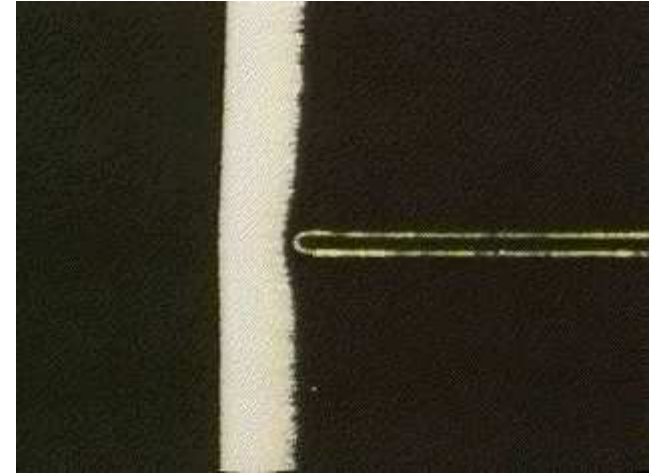
Effect of viscosity

- The layers closer to the wall start moving right away due to the no-slip boundary condition. The layers farther away from the wall start moving later.

- The distance from the wall that is affected by the motion is also called the viscous diffusion length. This distance increases as time goes on.

- The experiment shown on the left is performed with a higher viscosity fluid (100 mPa.s). On the right, a lower viscosity fluid (10 mPa.s) is shown.

Moving plate boundary layer

- An impulsively started plate in a stagnant fluid.

- When the wall in contact with the still fluid suddenly starts to move, the layers of fluid close to the wall are dragged along while the layers farther away from the wall move with a lower velocity.

- The viscous layer develops as a result of the no-slip boundary condition at the wall.

$\delta(t)$

Moving Plate
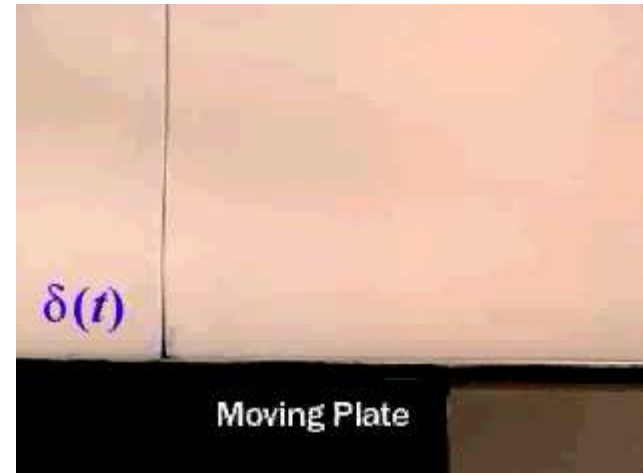
Viscous boundary layer thickness

- Exact equations for the velocity profile in the viscous boundary layer were derived by Stokes in 1881.
- Start with the Navier-Stokes equation:

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2}$$

- Derive exact solution for the velocity profile:

$$U = U_0 \left(1 - erf \left(\frac{y}{2\sqrt{\nu t}} \right) \right)$$

- *erf* is the error function: $erf(z) = \frac{2}{\sqrt{\pi}} \int_0^z e^{-t^2} dt$
- The boundary layer thickness can be approximated by:

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2} \implies \frac{U_0}{t} \approx \nu \frac{U_0}{\delta^2} \implies \delta \approx \sqrt{\nu t}$$
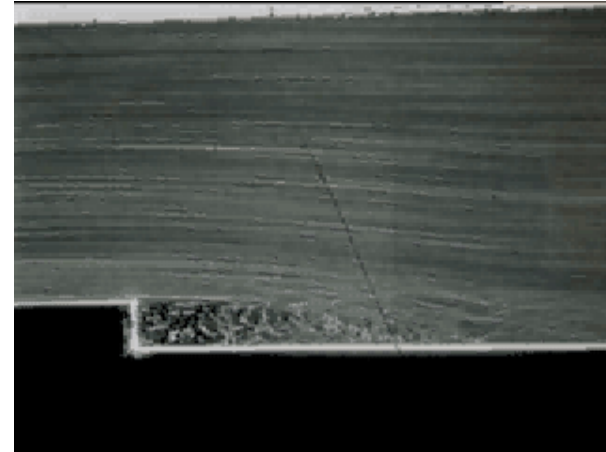
Flow separation

- 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.

separation point

wake
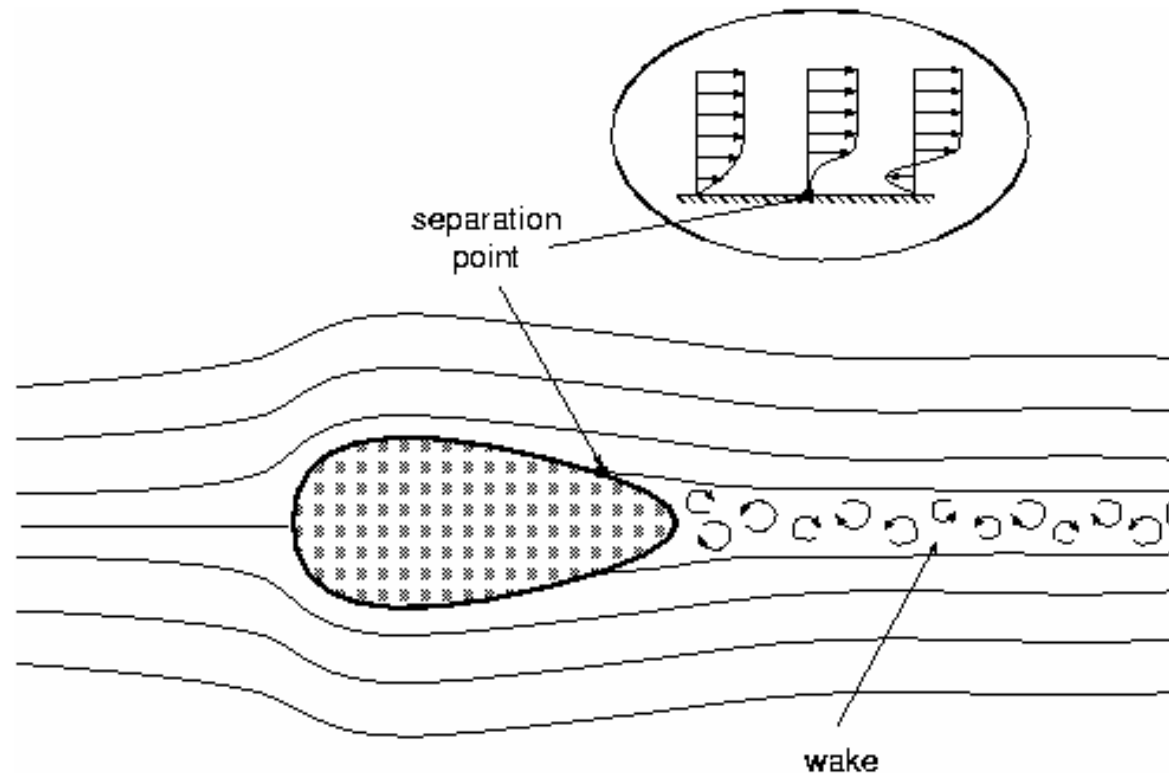
Separation at sharp corners

- Corners, sharp turns and high angles of attack all represent sharply decelerating flow situations where the loss in energy in the boundary layer ends up leading to separation.

- Here we see how the boundary layer flow is unable to follow the turn in the sharp corner (which would require a very rapid acceleration), causing separation at the edge and recirculation in the aft region of the backward facing step.
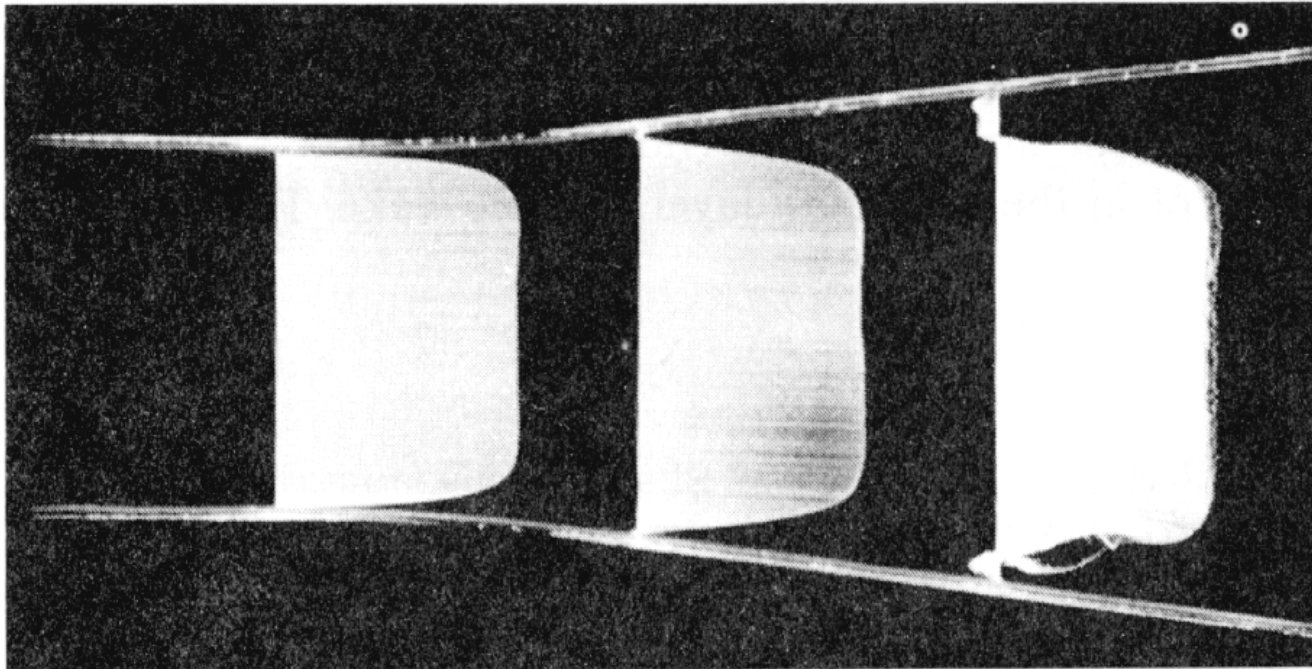
Flow around a truck

- Flow over non-streamlined bodies such as trucks leads to considerable drag due to recirculation and separation zones.

- A recirculation zone is clear on the back of the cab, and another one around the edge of the trailer box.

- The addition of air shields to the cab roof ahead of the trailer helps organize the flow around the trailer and minimize losses, reducing drag by up to 10-15%.

Flow separation in a diffuser with a large angle
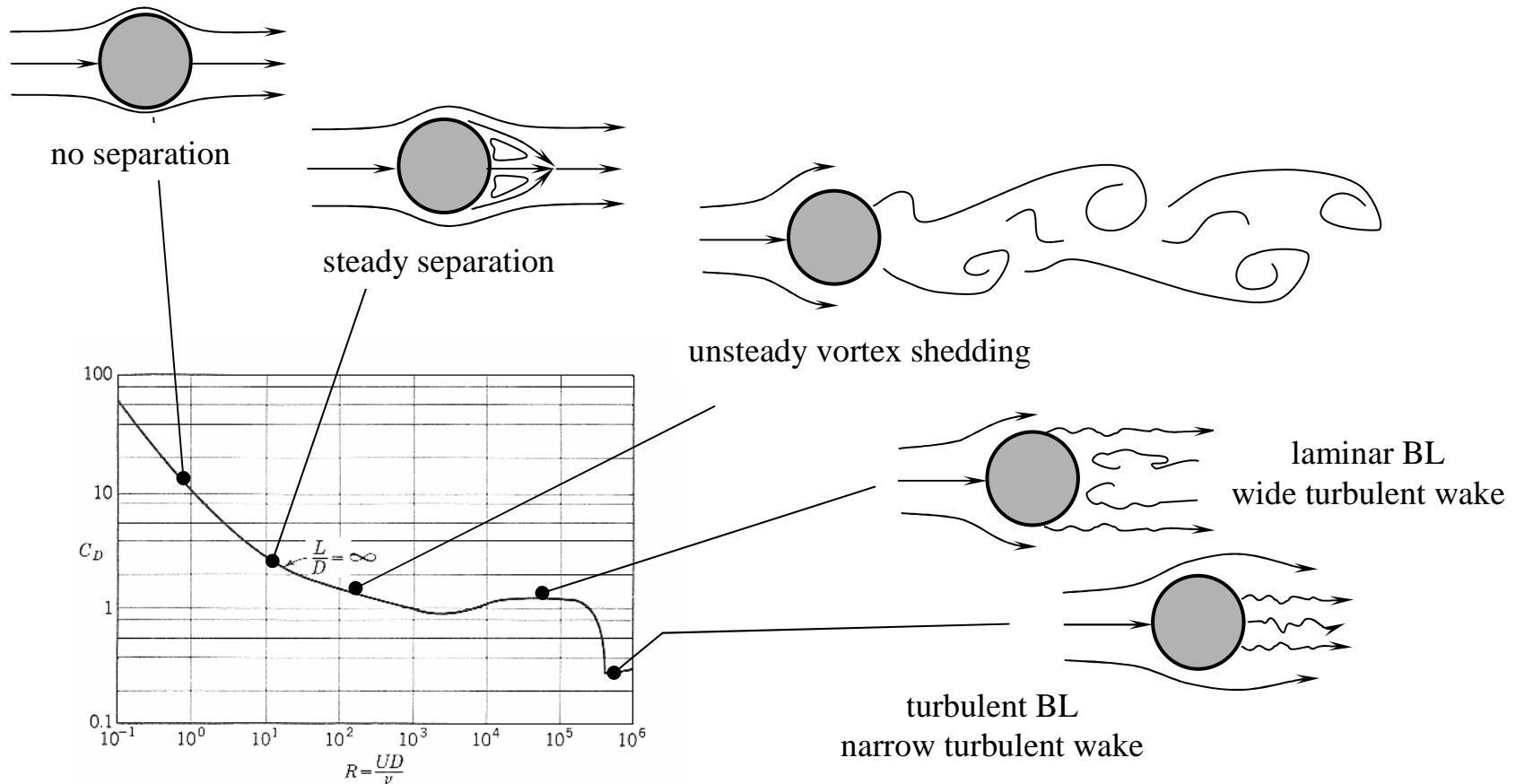
Inviscid flow around a cylinder

- The origins of the flow separation from a surface are associated with the pressure gradients impressed on the boundary layer by the external flow.

- The image shows the predictions of inviscid, irrotational flow around a cylinder, with the arrows representing velocity and the color map representing pressure.

- The flow decelerates and stagnates upstream of the cylinder (high pressure zone).

- It then accelerates to the top of the cylinder (lowest pressure).

$$\frac{p - p_\infty}{\frac{1}{2}\,\rho U^2}$$
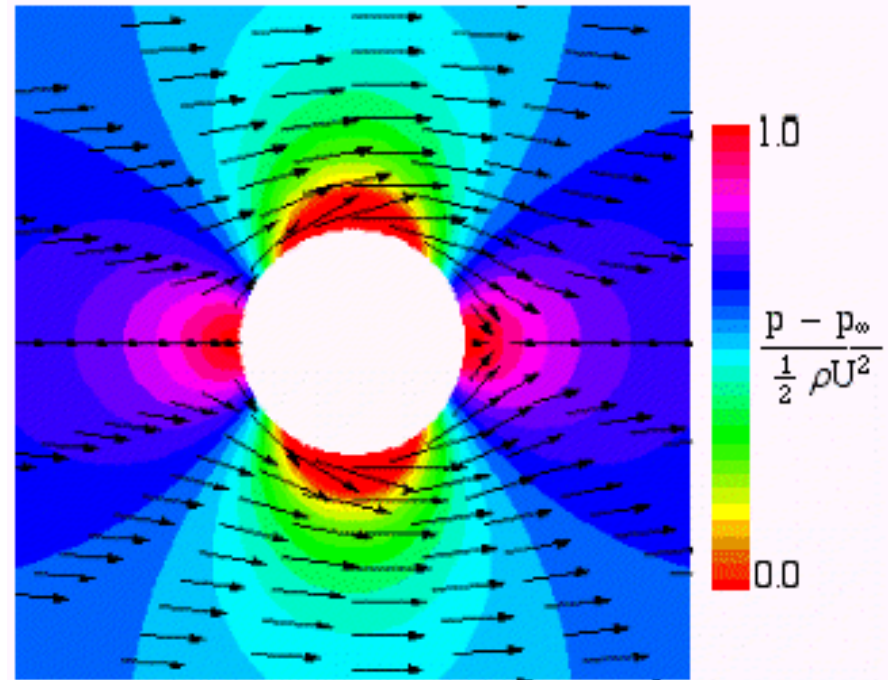
- Next it must decelerate against a high pressure at the rear stagnation point.

Drag on a smooth circular cylinder

- The drag coefficient is defined as follows: $\quad F_{drag} = C_D \frac{1}{2} \rho v^2 A_\perp$

no separation

steady separation

unsteady vortex shedding

laminar BL
wide turbulent wake

turbulent BL
narrow turbulent wake

Drag on a smooth circular cylinder

- At low Reynolds numbers (Re < 1), the inertia effects are small relative to the viscous and pressure forces. In this flow regime the drag coefficient varies inversely with the Reynolds number. For example, the drag coefficient CD for a sphere is equal to 24/Re.
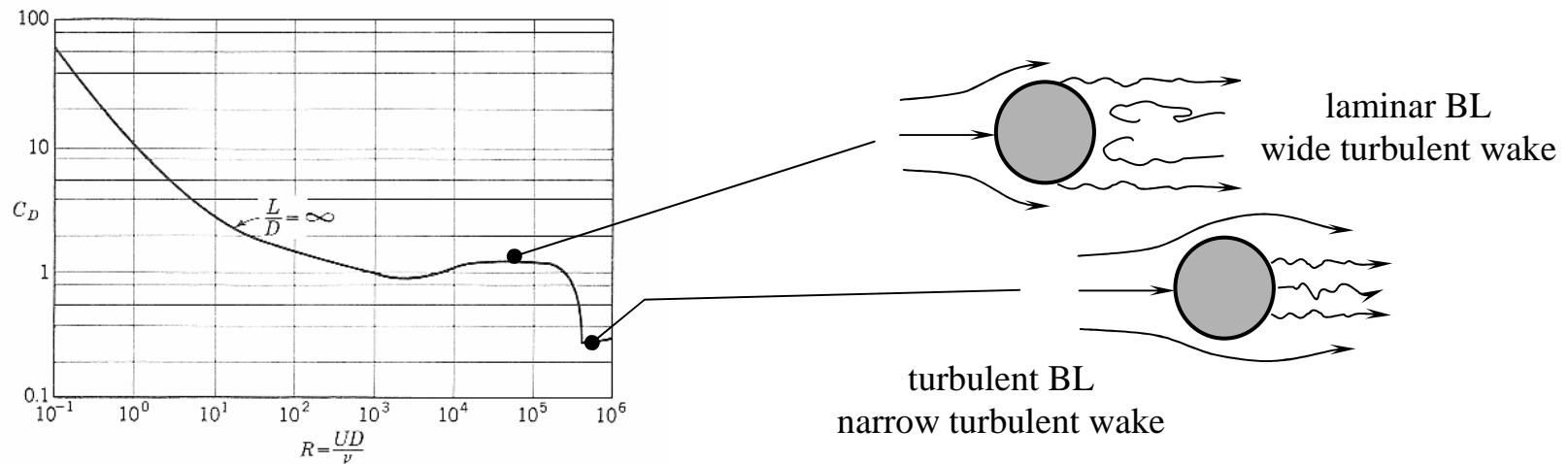
no separation

$R = \dfrac{UD}{\nu}$
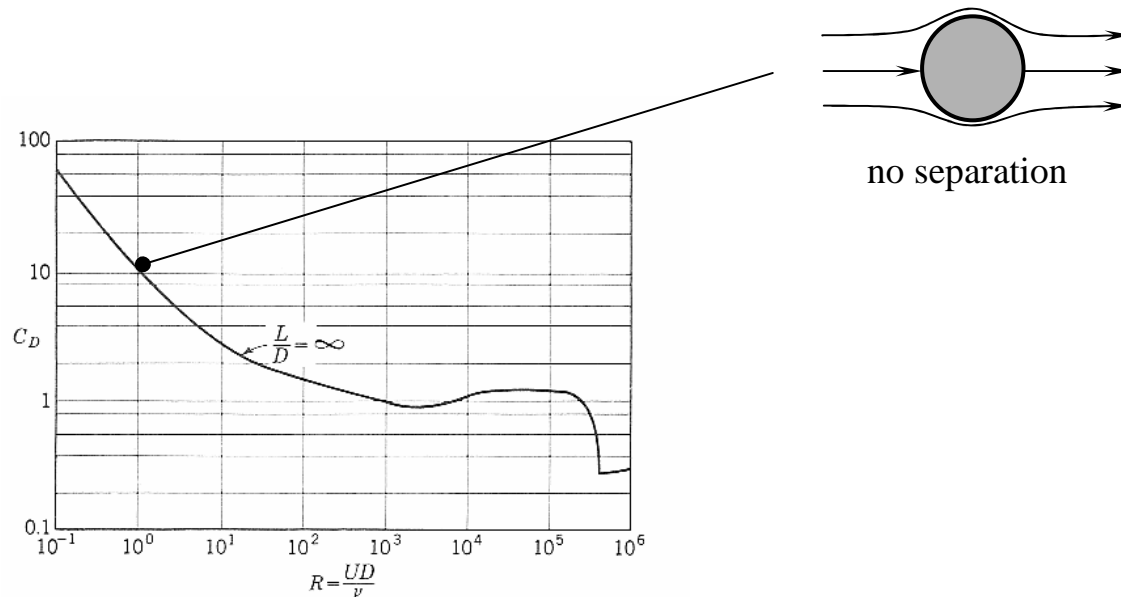
Drag on a smooth circular cylinder

- At moderate Reynolds numbers (1<Re<10^3), the flow begins to separate in a periodic fashion in the form of Karman vortices

steady separation

unsteady vortex shedding

$$C_D$$

$$\frac{L}{D} = \infty$$

$$R = \frac{UD}{\nu}$$
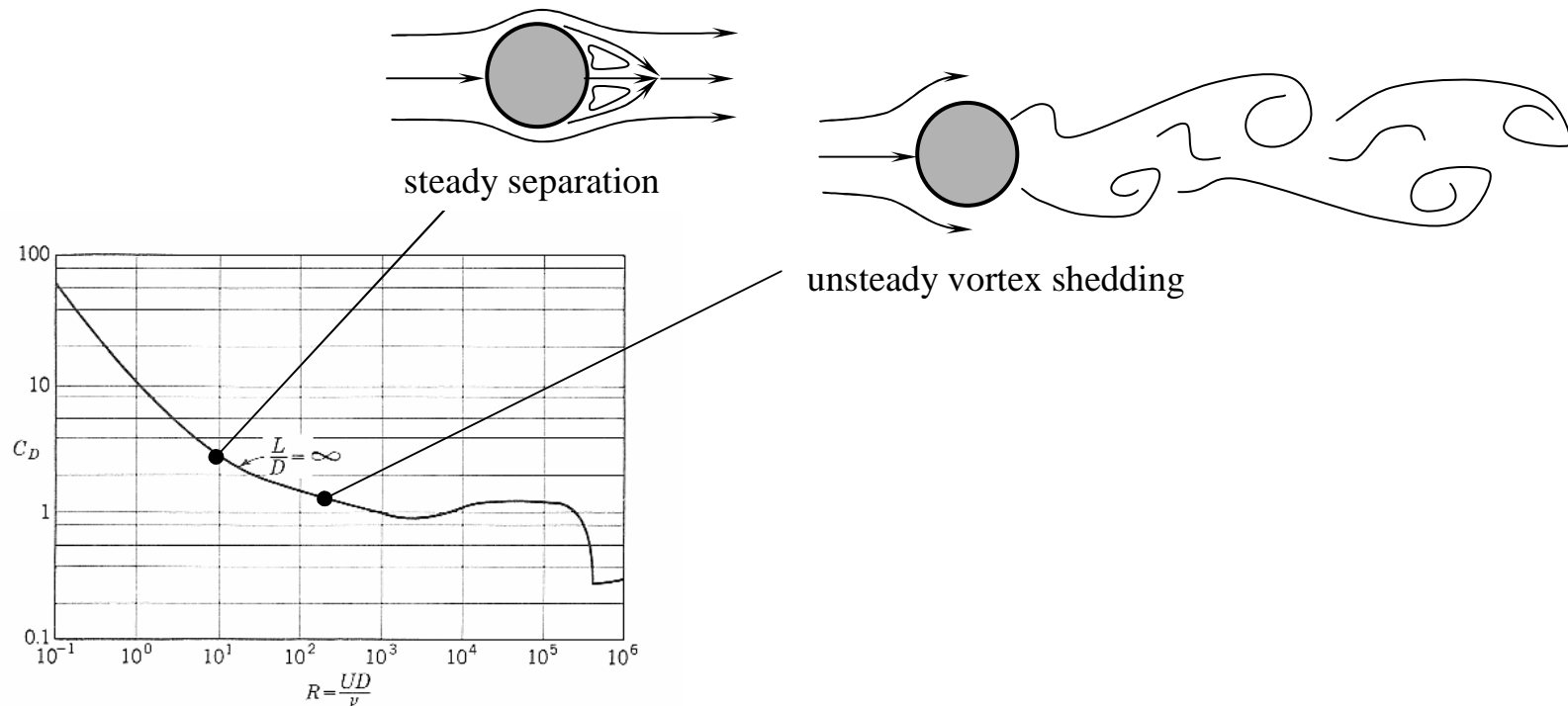
Drag on a smooth circular cylinder

- At higher Reynolds numbers ($10^3 <$ Re $< 10^5$), the flow becomes fully separated. An adverse pressure gradient exists over the rear portion of the cylinder resulting in a rapid growth of the laminar boundary layer and separation

- As the Reynolds number increases, the boundary layer transitions to turbulent, delaying separation and resulting in a sudden decrease in the drag coefficient.

laminar BL
wide turbulent wake

turbulent BL
narrow turbulent wake

Separation - adverse pressure gradients

- Separation of the boundary layers occurs whenever the flow tries to decelerate quickly, that is whenever the outer pressure gradient is negative, or the pressure gradient is positive, sometimes referred to as an adverse pressure gradient.

- In the case of the tennis ball, the flow initially decelerates on the upstream side of the ball, while the local pressure increases in accord with Bernoulli's equation.

- Near the top of the ball the local external pressure decreases and the flow should accelerate as the potential energy of the pressure field is converted to kinetic energy.


- However, because of viscous losses, not all kinetic energy is recovered and the flow reverses around the separation point.

18

Turbulent boundary layer

- Increased momentum transport due to turbulence from the free stream flow to the flow near the wall makes turbulent boundary layers more resistant to flow separation.

- The photographs depict the flow over a strongly curved surface, where there exists a strong adverse (positive) pressure gradient.

- The boundary layer has a high momentum deficit.

- In the case where the boundary layer is laminar, insufficient momentum exchange takes, the flow is unable to adjust to the increasing pressure and separates from the surface.

- In case where the flow is turbulent, the increased transport of momentum (due to the Reynolds stresses) from the free-stream to the wall increases the streamwise momentum in the boundary layer. This allows the flow to overcome the adverse pressure gradient. It eventually does separate nevertheless, but much further downstream.
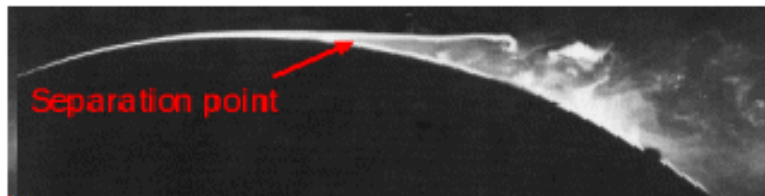

Laminar Separation
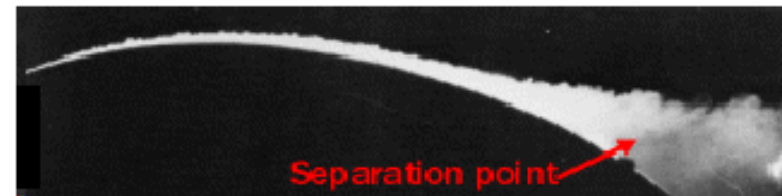
Turbulent Separation

19

Tripping the boundary layer

- Here we see how the addition of a trip wire to induce transition to turbulence changes the separation line further to the rear of the sphere, reducing the size of the wake and thus drastically diminishing overall drag.

- This well-known fact can be taken advantage of in a number of applications, such as dimples in golf balls and turbulence generation devices on airfoils.
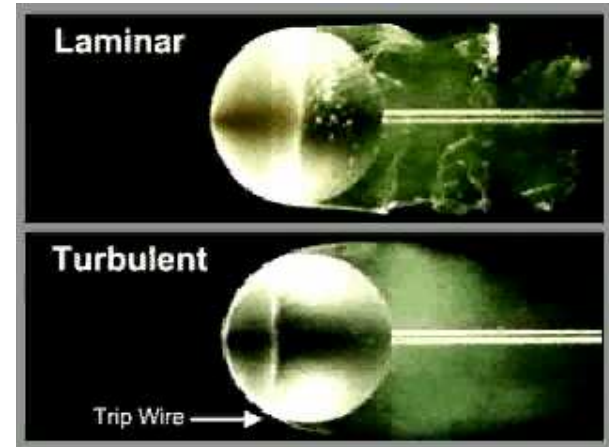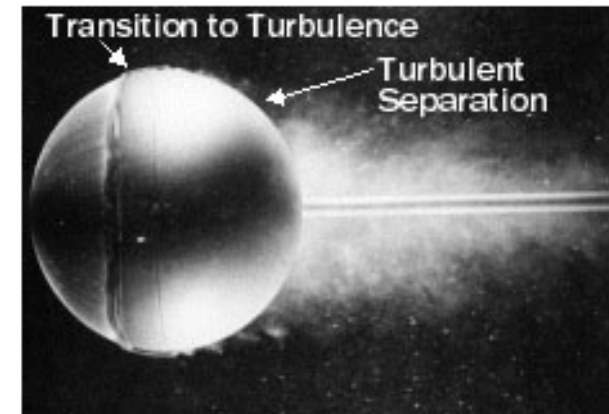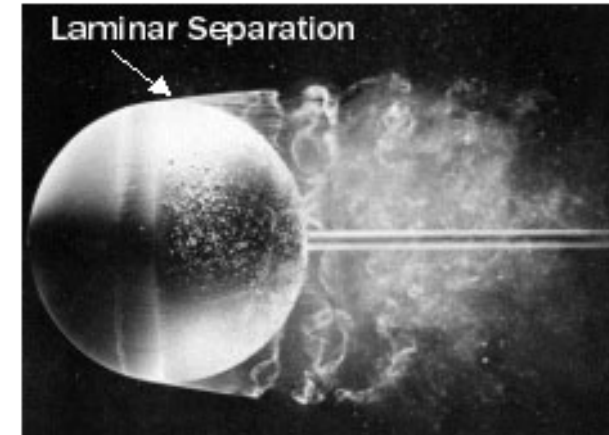
Sports balls

- Many games involve balls designed to use drag reduction brought about by surface roughness.

- Many sports balls have some type of surface roughness, such as the seams on baseballs or cricket balls and the fuzz on tennis balls.

- It is the Reynolds number (not the speed, per se) that determines whether the boundary layer is laminar or turbulent. Thus, the larger the ball, the lower the speed at which a rough surface can be of help in reducing the drag.

- Typically sports ball games that use surface roughness to promote an early transition of the boundary layer from a laminar to a turbulent flow are played over a Reynolds number range that is near the "trough" of the Cd versus Re curve, where drag is lowest.
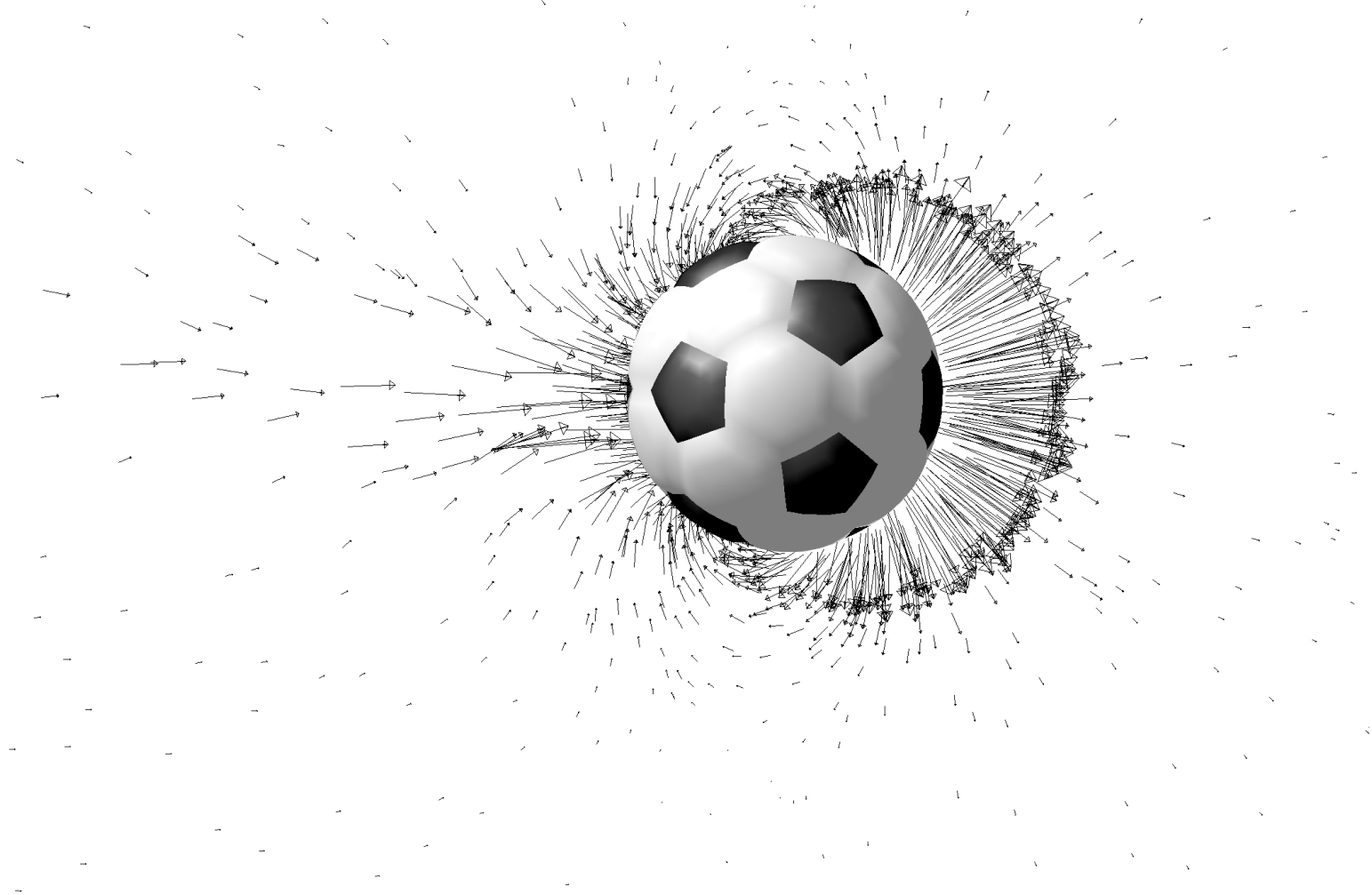
Flow in reference frame relative to the ball

- Note that we have been showing flow fields in the reference frame of the object, similar to the flow around the soccer ball shown here.
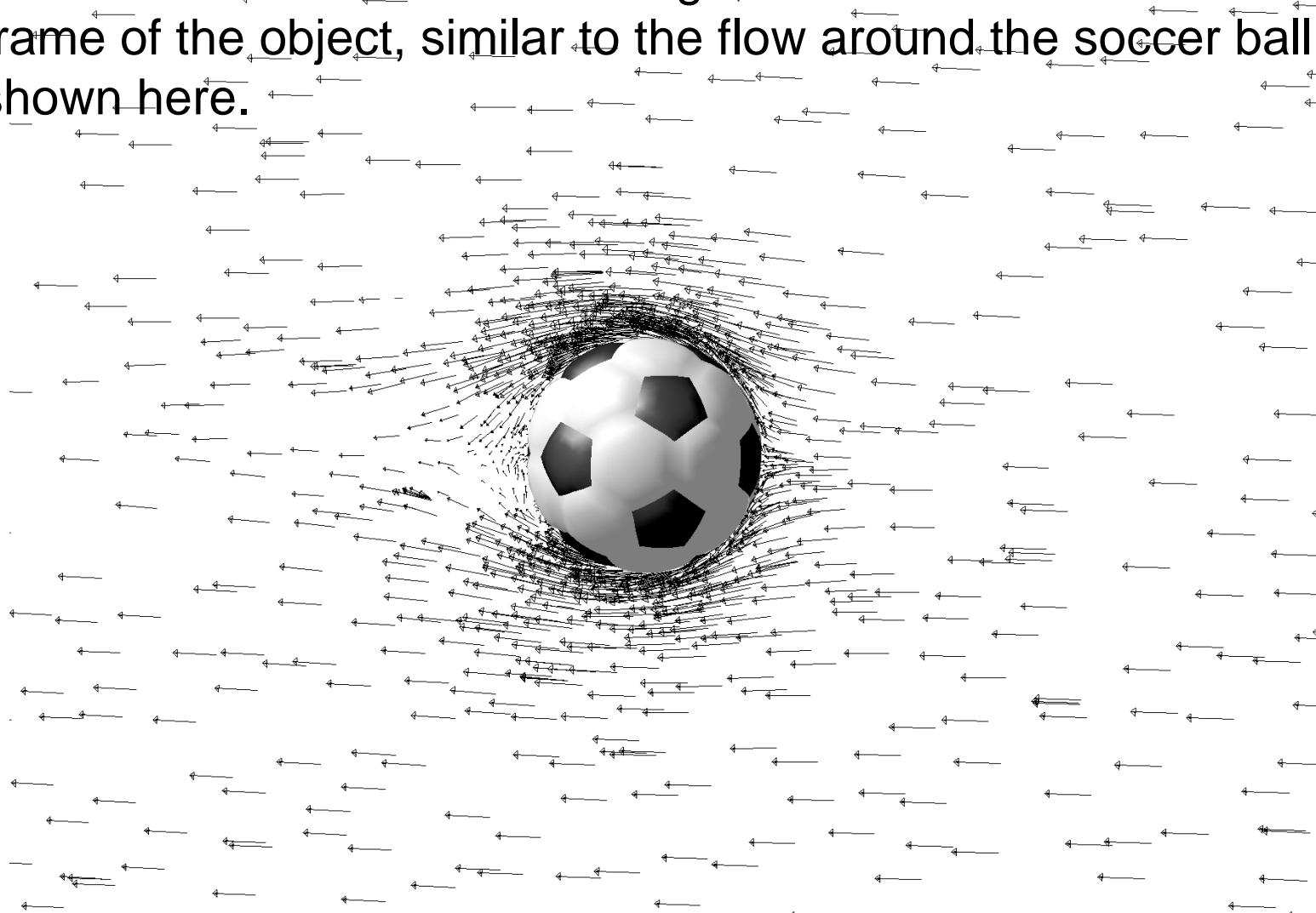
Flow in absolute reference frame

- However, one should keep in mind that the flow in the absolute reference frame may look quite different, as shown here.

Airfoil - effect of angle of attack

- The loss in pressure in the separated flow region behind solid bodies causes an imbalance between the upstream and downstream forces, contributing greatly to an increased net drag force.

- In the case of streamlined airfoils at low angle of attack, separation occurs only at the tip, with minimal losses. As the angle of attack increases, the separation point moves upstream, leading to increased drag.

Airfoil - effect of shape

- The pressure field is changed by changing the thickness of a streamlined body placed in the flow. The acceleration and deceleration caused by a finite body width creates favorable and unfavorable pressure gradients.

- When the body is thin, there are only weak pressure gradients and the flow remains attached. As the body is made thicker, the adverse pressure gradient resulting from the deceleration near the rear leads to flow separation, recirculation, and vortex shedding.

- Focusing in on the rear region of the flow, it is seen that as the body is again reduced in thickness, the separated region disappears as the strengths of the adverse pressure gradient is diminished.

25
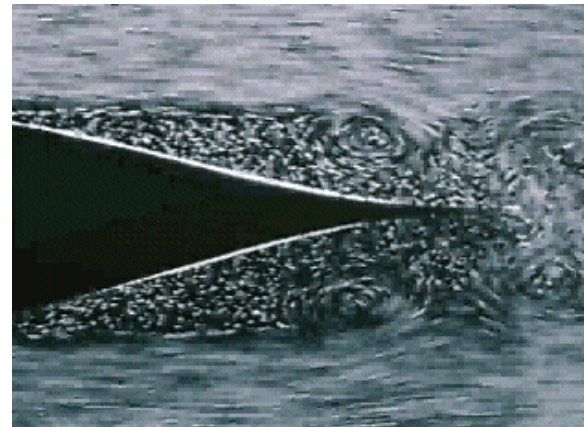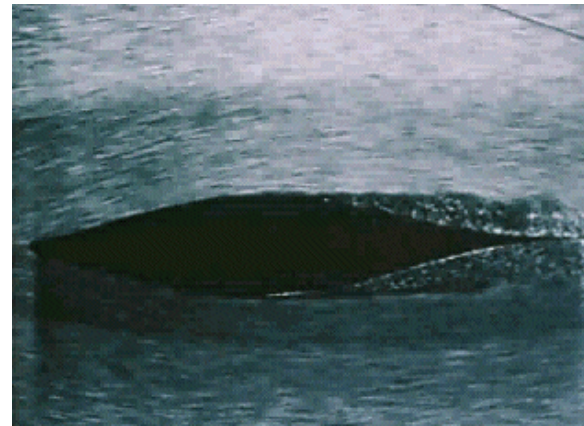
Suction

- Just as flow separation can be understood in terms of the combined effects of viscosity and adverse pressure gradients, separated flows can be reattached by the application of a suitable modification to the boundary conditions.

- In this example, suction is applied to the leading edge of the airfoil at a sharp angle of attack, removing the early separation zone, and moving the separation point much farther downstream.

No Suction - Separated Flow

Effect of Suction on Streamlines

Suction Applied
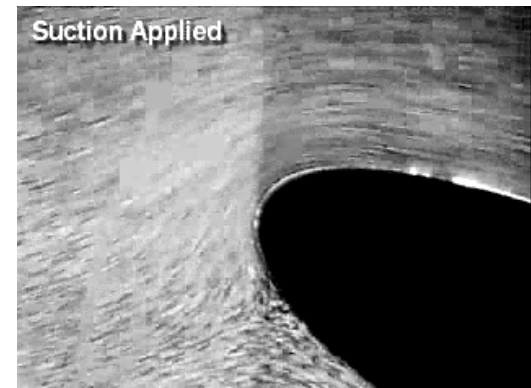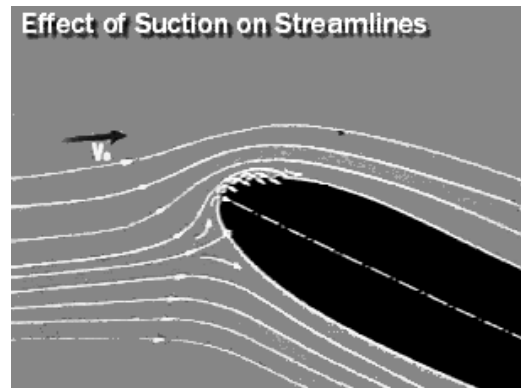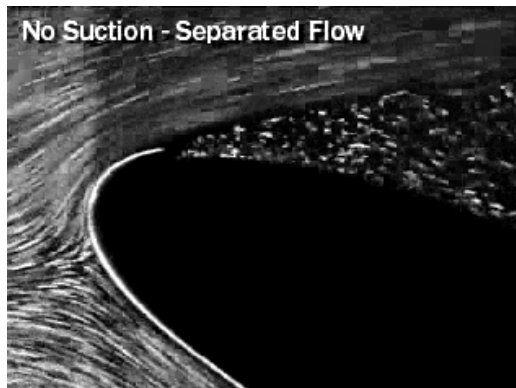
Blowing

- Separation in external flows, such as the flow past a sudden expansion can be controlled not only by suction but also by blowing.

- In this video, the region of separated flow is eliminated by the introduction of high momentum fluid at a point near the separation point.

- This acts to eliminate the adverse pressure gradient by accelerating the fluid close to the boundary, leading to re-attachment of the flow.

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 from the surface where the velocity reaches 99% of the free-stream velocity.
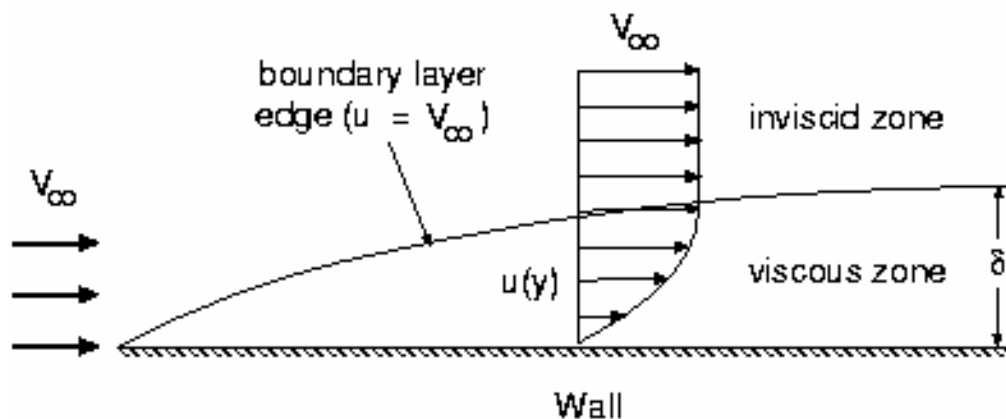
$$\delta = y, \; where \; u/U = 0.99$$

The turbulent boundary layer
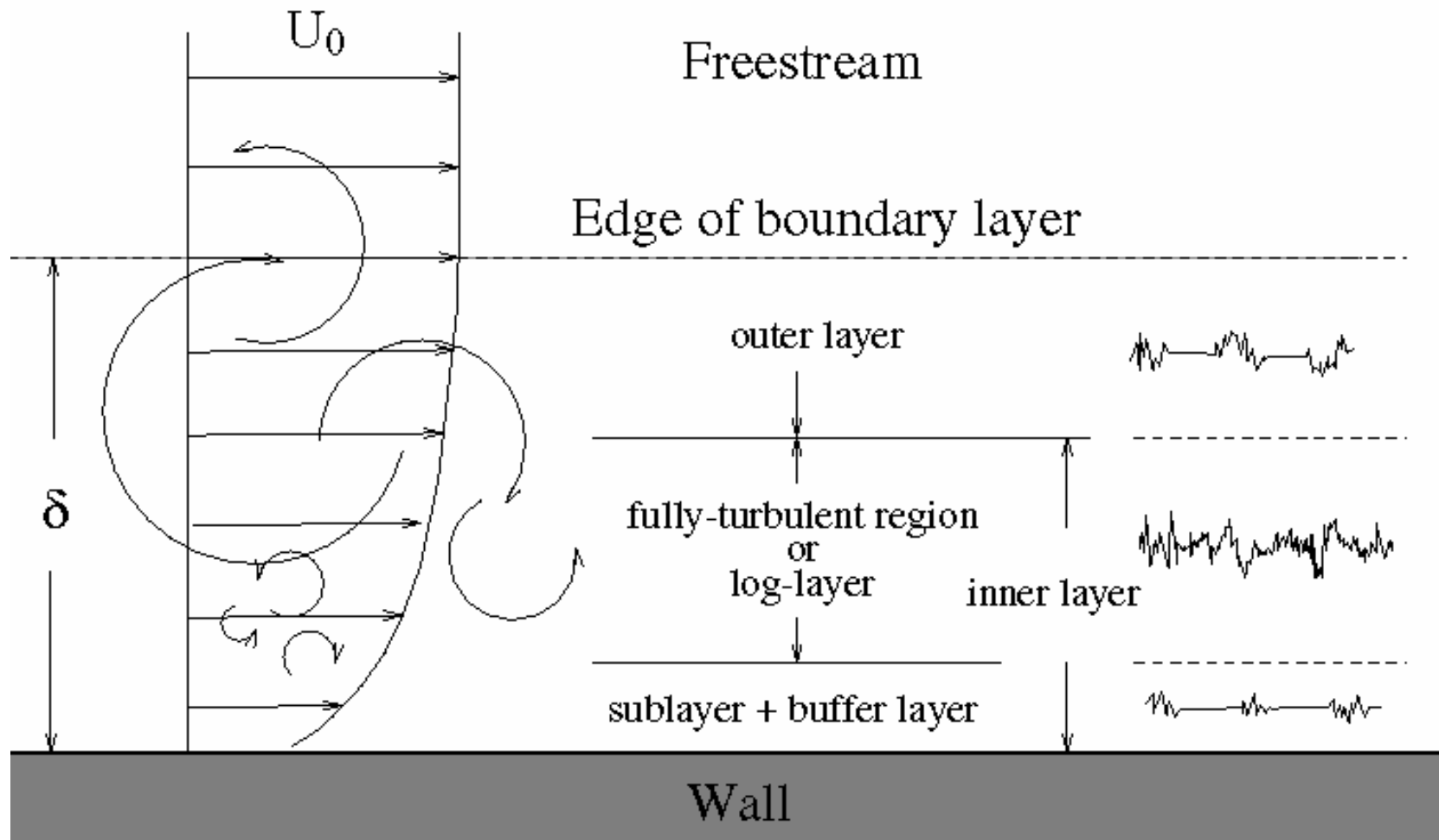
The turbulent boundary layer

- Important variables:
  - Distance from the wall: y.
  - Wall shear stress: τ_w. The force exerted on a flat plate is the area times the wall shear stress.
  - Density: ρ.
  - Dynamic viscosity: μ.
  - Kinematic viscosity: ν.
  - Velocity at y: U.
  - The friction velocity: $u_\tau = (\tau_w/\rho)^{1/2}$.
- We can define a Reynolds number based on the distance to the wall using the friction velocity: $y^+ = yu_\tau/\nu$.
- We can also make the velocity at y dimensionless using the friction velocity: $u^+ = U/u_\tau$.

Boundary layer structure

$$U/U_\tau = 2.5 \ln(U_\tau y/\nu) + 5.45$$

inner layer

outer layer

$u^+ = y^+$

viscous sublayer

buffer layer
or
blending
region

fully turbulent region
or
log-law region

Upper limit
depends on
Reynolds no.

$y^+ = 1$

$y^+ \simeq 5$

$y^+ \simeq 60$

U/U_τ

$\ln\ U_\tau\, y/\nu$
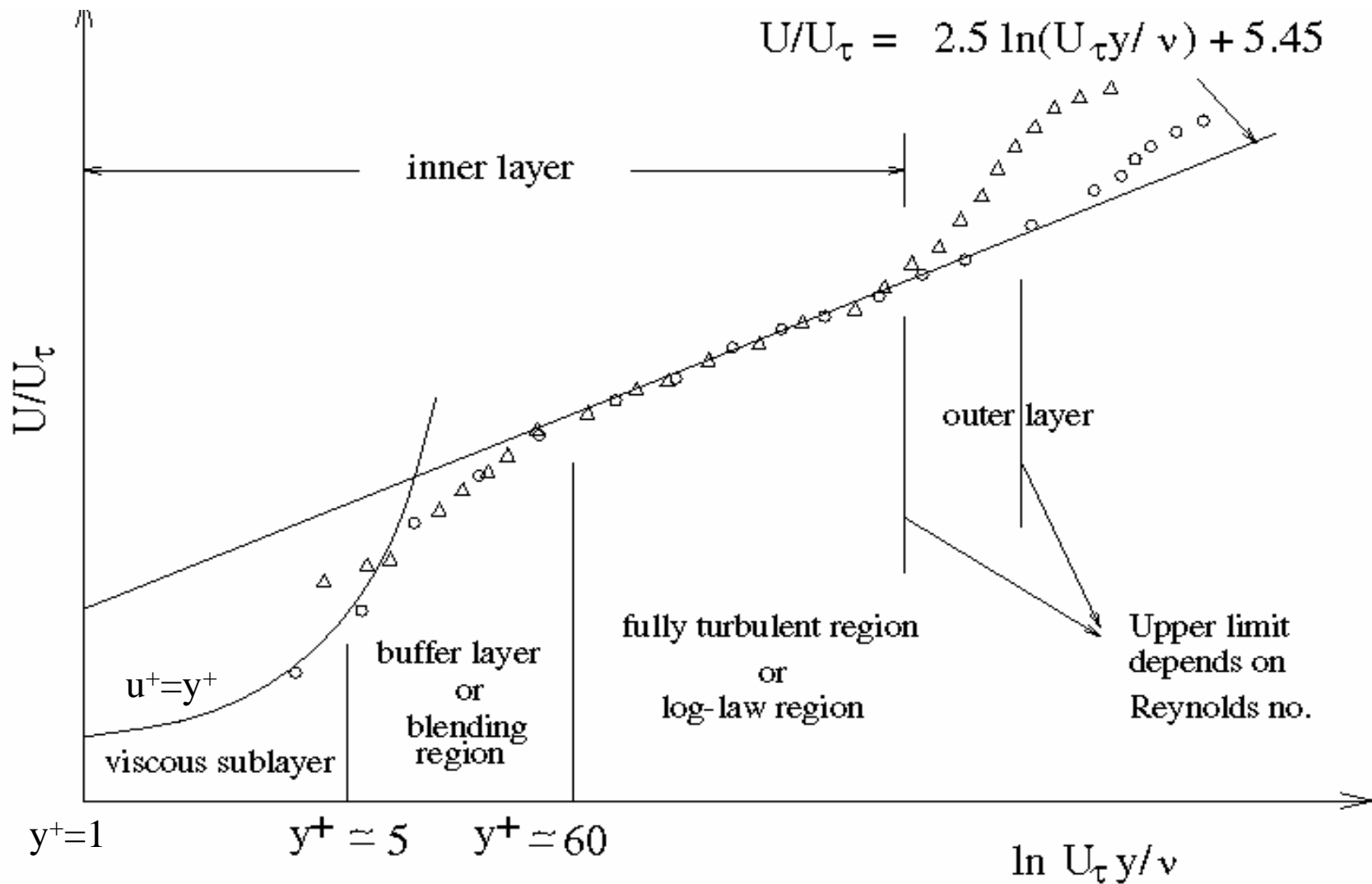
Standard wall functions

- The experimental boundary layer profile can be used to calculate τ_w. However, this requires y^+ for the cell adjacent to the wall to be calculated iteratively.

- In order to save calculation time, the following explicit set of correlations is usually solved instead:

$$y^* = \frac{\rho C_\mu^{1/4} k_P^{1/2} y_P}{\mu} \Rightarrow \left\{ \begin{array}{l} y^* \geq 11.225 \Rightarrow U^* = \frac{1}{\kappa}\ln\left(Ey^*\right) \\ \\ y^* < 11.225 \Rightarrow U^* = y^* \end{array} \right\} \Rightarrow \tau_w = \frac{U_P C_\mu^{1/4} k_P^{1/2}}{U^*/\rho}$$

- Here:
  - U_p is the velocity in the center of the cell adjacent to the wall.
  - y_p is the distance between the wall and the cell center.
  - k_p is the turbulent kinetic energy in the cell center.
  - κ is the von Karman constant (0.42).
  - E is an empirical constant that depends on the roughness of the walls (9.8 for smooth surfaces).

Near-wall treatment - momentum equations

- The objective is to take the effects of the boundary layer correctly into account without having to use a mesh that is so fine that the flow pattern in the layer can be calculated explicitly.

- Using the no-slip boundary condition at wall, velocities at the nodes at the wall equal those of the wall.

- The shear stress in the cell adjacent to the wall is calculated using the correlations shown in the previous slide.

- This allows the first grid point to be placed away from the wall, typically at $50 < y^+ < 500$, and the flow in the viscous sublayer and buffer layer does not have to be resolved.

- This approach is called the "standard wall function" approach.

- The correlations shown in the previous slide are for steady state ("equilibrium") flow conditions. Improvements, "non-equilibrium wall functions," are available that can give improved predictions for flows with strong separation and large adverse pressure gradients.

Two-layer zonal model

- A disadvantage of the wall-function approach is that it relies on empirical correlations.

- The two-layer zonal model does not. It is used for low-Re flows or flows with complex near-wall phenomena.

- Zones distinguished by a wall-distance-based turbulent Reynolds number: $Re_y \equiv \dfrac{\rho \sqrt{k}\, y}{\mu}$

- The flow pattern in the boundary layer is calculated explicitly.

- Regular turbulence models are used in the turbulent core region.

- Only k equation is solved in the viscosity-affected region.

- ε is computed using a correlation for the turbulent length scale.

- Zoning is dynamic and solution adaptive.
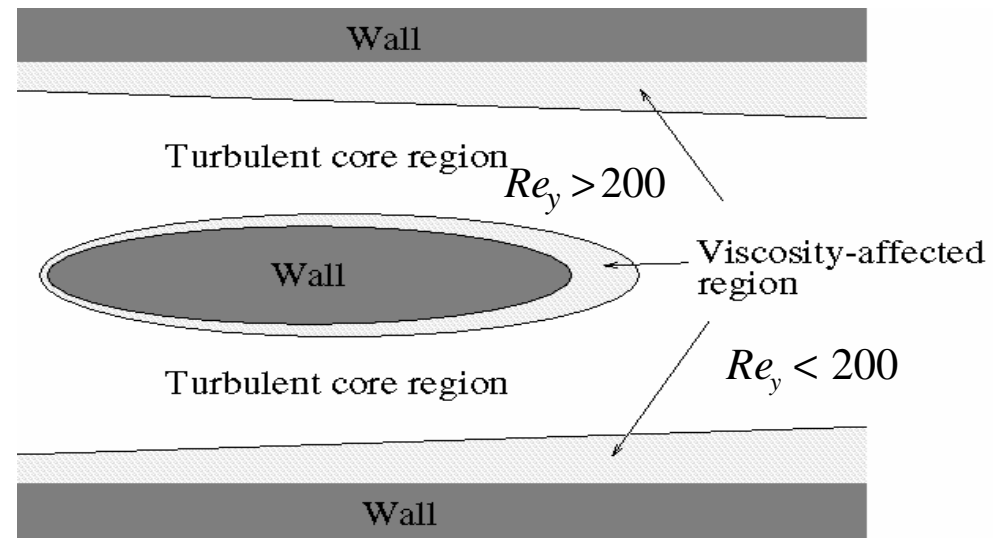
Near-wall treatment - turbulence

- The turbulence structure in the boundary layer is highly anisotropic.

- ε and k require special treatment at the walls.

- Furthermore, special turbulence models are available for the low Reynolds number region in the boundary layer.

- These are aptly called "low Reynolds number" models.

- This is still a very active area of research, and we will not discuss those here in detail.

Comparison of near-wall treatments

Approach	Strengths	Weaknesses
Standard wall-functions	Robust, economical, reasonably accurate	Empirically based on simple high- Re flows; poor for low-Re effects, massive transpiration, PGs, strong body forces, highly 3D flows
Non-equilibrium wall-functions	Accounts for pressure gradient (PG) effects. Improved predictions for separation, reattachment, impingement	Poor for low-Re effects, massive transpiration (blowing, suction), severe PGs, strong body forces, highly 3D flows
Two-layer zonal model	Does not rely on empirical law-of-the-wall relations, good for complex flows, applicable to low-Re flows	Requires finer mesh resolution and therefore larger cpu and memory resources

Computational grid guidelines
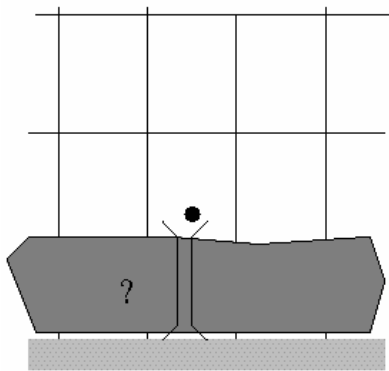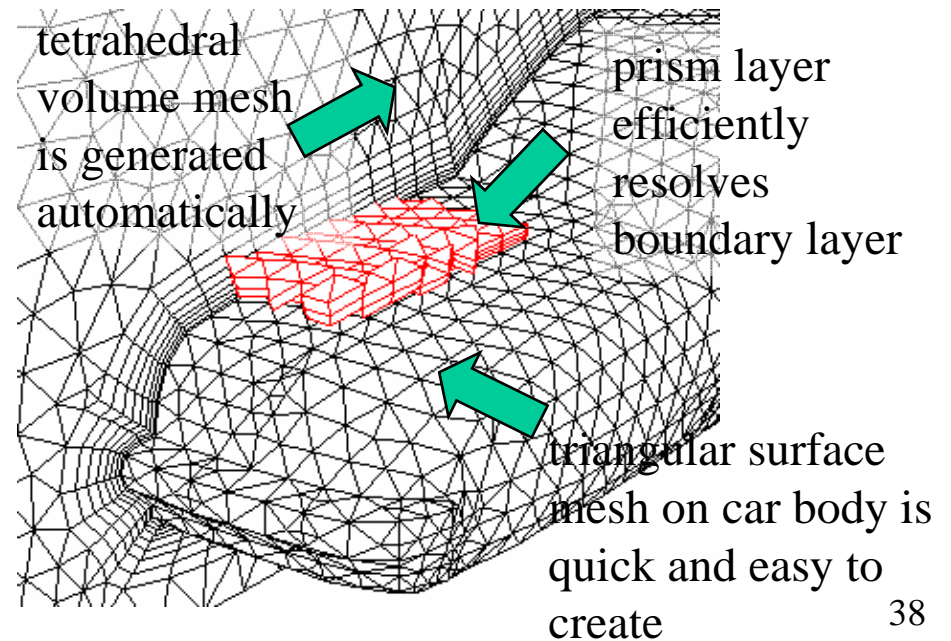

Wall Function Approach	Two-Layer Zonal Model Approach
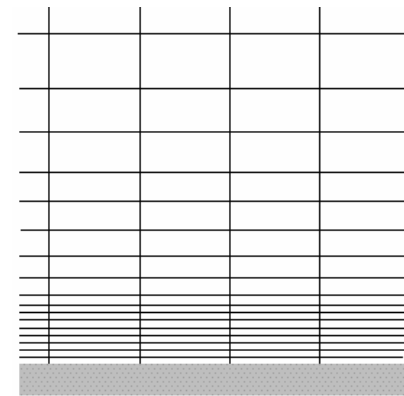
- First grid point in log-law region:

$$50 \leq y^+ \leq 500$$

- Gradual expansion in cell size away from the wall.

- Better to use stretched quad/hex cells for economy.

- First grid point at $y^+ \approx 1$.

- At least ten grid points within buffer and sublayers.

- Better to use stretched quad/hex cells for economy.

Obtaining accurate solutions

- When very accurate (say 2%) drag, lift, or torque predictions are required, the boundary layer and flow separation require accurate modeling.
- The following practices will improve prediction accuracy:
  - Use boundary layer meshes consisting of quads, hexes, or prisms. Avoid using pyramid or tetrahedral cells immediately adjacent to the wall.
  - After converging the solution, use the surface integral reporting option to check if y$^+$ is in the right range, and if not refine the grid using adaption.
  - For best predictions use the two-layer zonal model and completely resolve the flow in the whole boundary layer.

tetrahedral volume mesh is generated automatically

prism layer efficiently resolves boundary layer

triangular surface mesh on car body is quick and easy to create

38

Summary

- The concept of the boundary layer was introduced.
- Boundary layers require special treatment in the CFD model.
- The influence of pressure gradient on boundary layer attachment showed that an adverse pressure gradient gives rise to flow separation.
- For accurate drag, lift, and torque predictions, the boundary layer and flow separation need to be modeled accurately.
- This requires the use of:
    - A suitable grid.
    - A suitable turbulence model.
    - Higher order discretization.
    - Deep convergence using the force to be predicted as a convergence monitor.