LIFT AND DRAG

INTRODUCTION

An object immersed in a flow, or traveling through a flow (e.g. ball, airplane), will be subject to forces and moments from the flow, as shown in Figure 1.

When considering flow past two- and three-dimensional (2D or 3D) bodies, as engineers we are primarily interested in the lift and drag forces. These are a function of the speed of the object moving through still air, for instance, the flight speed of an airplane (U). But in aerodynamics, it is much easier to adopt the reference frame of the airplane. So, we think that the plane is still, and the air is moving towards the plane with speed U. The speed at which the air *far away* approaches the flying object is called *free-stream speed*.

Let us denote the air flow towards the body (free stream) by U. The drag (D) is the force parallel to the axis of the free stream (U) – it is a flow loss and must be overcome if the body is to move forward. The lift (L) is perpendicular to the free stream and is usually associated with airfoils and wings (Lift=Weight is required for airplanes flying at constant speed). All moments and forces **except drag** may cancel if the body has the right symmetry. Cylinders and bodies of revolution have no lift, side force or moments if they are placed with the axis in a straight angle with the free stream (as illustrated in Figure 2). They still have a significant drag.

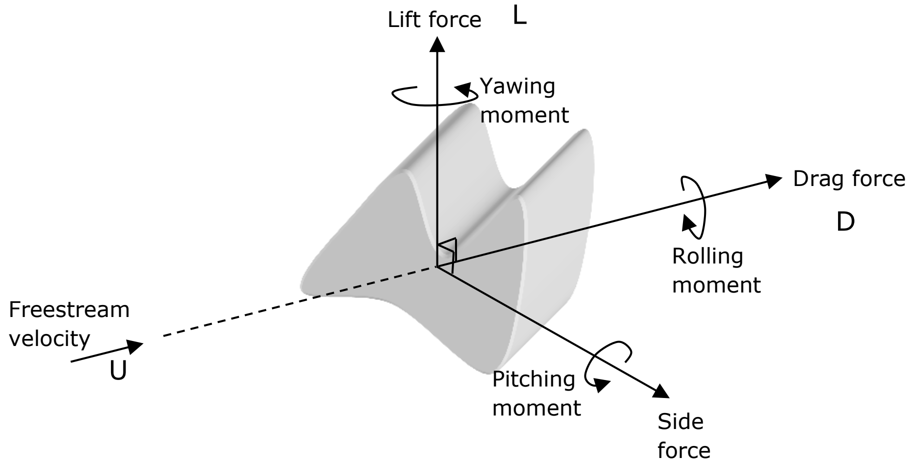
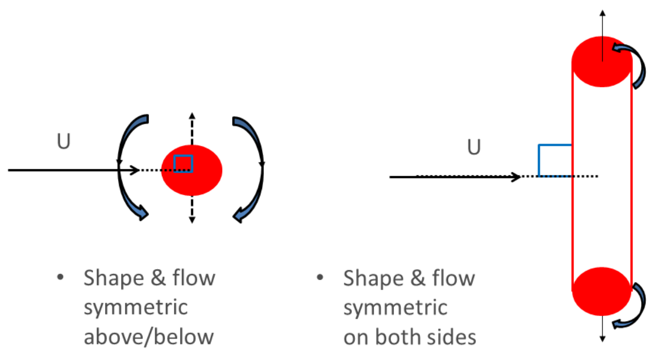


Figure 1 Definition of forces and moments on an arbitrary body immersed in a uniform flow

Example: Forces on a circular cylinder with free stream in 90° angle



- Shape & flow symmetric above/below -> forces above/below equal -> **No lift** ! - > No pitching moment
- Shape & flow symmetric on both sides -> forces equal on both sides/ends -> **No sideforce** ! -> No yawing moment

Figure 2 Lift, side force and moments which cancel for cylinder flow depending on free-stream angle

LIFT & DRAG: EXAMPLE


Figure 3 Success in Formula 1 is largely about lift&drag

Figure 3 gives an example where aerodynamics is very important. Aerodynamics means two things: maximizing lift (up or down) and minimizing drag.

For a car, the **negative lift (downforce)** needs to be maximised, so that the car can stay on the road in sharp turns etc. This is done using inverted airfoils (front and rear wings). The principle is the opposite of that of an airplane wing.

We also need to minimize the drag, because drag is a force opposing forward motion. If there is a lot of drag, a lot of engine power (and fuel!) is spent to overcome it and drive at a constant speed. Drag is minimized by giving the car a smooth *aerodynamic* shape, which avoids **flow separation**.

CFD AND EXPERIMENTATION PLAYS A KEY ROLE FOR DRAG (AND LIFT)

Flow separation plays a key role for drag, and influences lift as well[1]. There is no analytical formula to calculate flow separation in the boundary layer, so no analytical formula to calculate the drag. Drag can generally only be found by numerical simulations (CFD) or wind tunnel experiments. For example, the Formula 1 teams spend a significant budget on CFD and wind tunnel tests to optimize aerodynamics of every new car (i.e. minimize drag and maximize downforce).

We can determine the drag and lift of an object by placing it in a wind tunnel. One can measure the total drag and lift forces directly (by a force balance equipment, as you will do in Lab 1). Lift and drag are results of pressure and friction forces from the fluid, which act all around the surface of the immersed object. Local force values can also be measured, for instance, from pressure holes on the surface. Such information can be useful when we want to know which regions the drag or lift is strongest, and how to improve them. (In this topic, we will also relate drag to the wake behind the body. To visualize the wake, the velocity field can be measured by ex. Particle-image velocimetry, PIV). Most engineering knowledge about drag (and much about lift) is based on experimental wind tunnel data, which was used to construct empirical formulas and curves when there were no computers. We still use the empirical curves to determine drag coefficients.

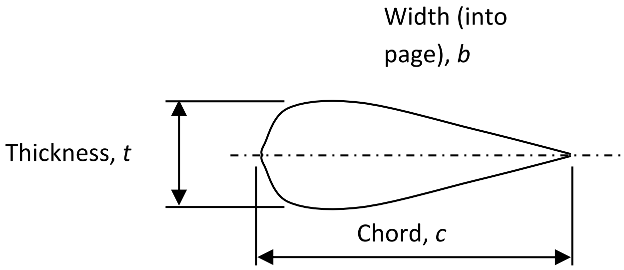
Computational Fluid Dynamics (CFD) uses computers to simulate the flow equations, and is a very useful investigative tool, where often more data can be extracted than in experiments. The CFD methods used in the industry are fast and efficient. However, they cannot always cope with laminar-turbulent transition and flow separation. Both phenomena can influence the drag. Academic research CFD codes capture those two phenomena better, but require a lot of computational power or focus on simple configurations. Hence, CFD can provide extremely useful ideas and guidelines for aerodynamic design, but the designs still must be verified experimentally.

[1] Flow separation happens in the boundary layer around the body, so it was also mentioned in Topic 1: Boundary Layers.

DRAG

First of all, the figure at the right introduces the notation we will use in this Topic, when discussing lift and drag of different bodies:

- Thickness t (perpendicular to the flow direction)
- Chord c (parallel to the flow direction)
- Width b (the third dimension "into the page", infinite for 2D bodies)



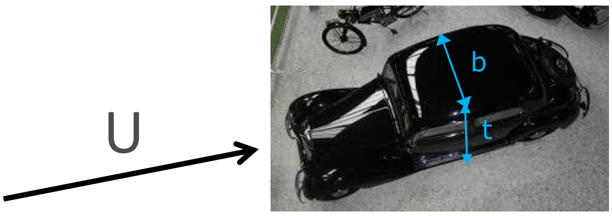
The drag is determined by how the body interacts with the flow around the body. Hence, we need to consider both the flow speed, and the dimensions of the body. When discussing drag, a very important coefficient is the drag coefficient.

Drag coefficient:

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

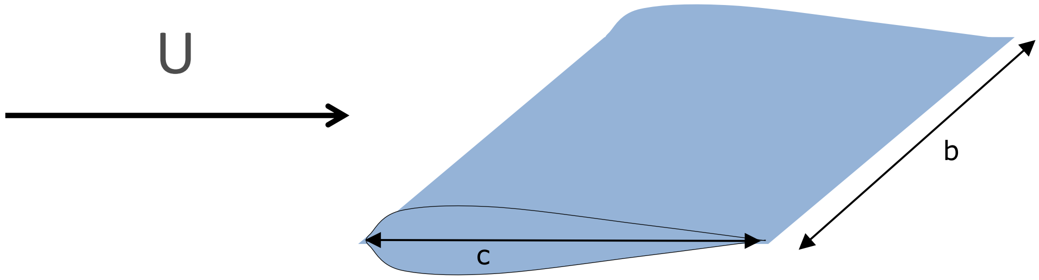
Here, U is the free stream velocity, D is the drag, and **the area A is one of**:

- **1. Frontal area** (*tb*) – usually for spheres, cylinders, cars, missiles etc (thick, stubby bodies)



or

- **2. Planform area** (*cb*) – usually for wings and hydrofoils (wide, flat bodies)



Wondering why two different areas for the two different type of bodies?
This is because there are two different types of drag[2], as explained below:

Pressure drag – often the largest part!

- Increases with the **frontal area**
- Has to do with flow separation:

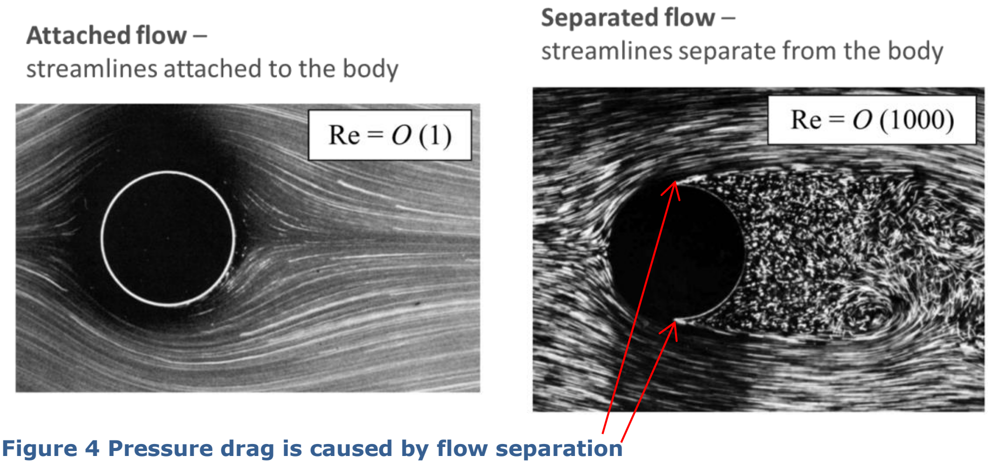


Figure 4 Pressure drag is caused by flow separation which creates a low pressure area (wake) behind the body

1. Bodies with a large frontal area (i.e. the thick, stubby bodies such as a car) experience mainly **pressure drag**. Pressure drag can be explained as follows: In the **free stream** far from the body, we can often consider the flow velocity U to be a constant. In Topic 1, you looked at **boundary layers**, which are regions of low velocity created near the body surface. In many normal engineering flows there will be a **separation** in the boundary layer. This means that the flow streamlines are no longer attached to the body, but they separate from the body, leaving a **wake region** behind the body. In the wake region, the pressure is very low. The difference between the high pressure region at the front of the body and the low pressure wake region creates a drag force on the body. This component of the total drag is called the pressure drag or form drag.

   Pressure drag is almost directly proportional to the width of the wake. The width of the wake is proportional to the frontal area. Therefore, to determine the drag for thick stubby bodies the frontal area is important.

[2] A third type of drag is vortex-induced drag, which will be considered later under "Lift: finite-span wings" (because induced drag increases with lift).

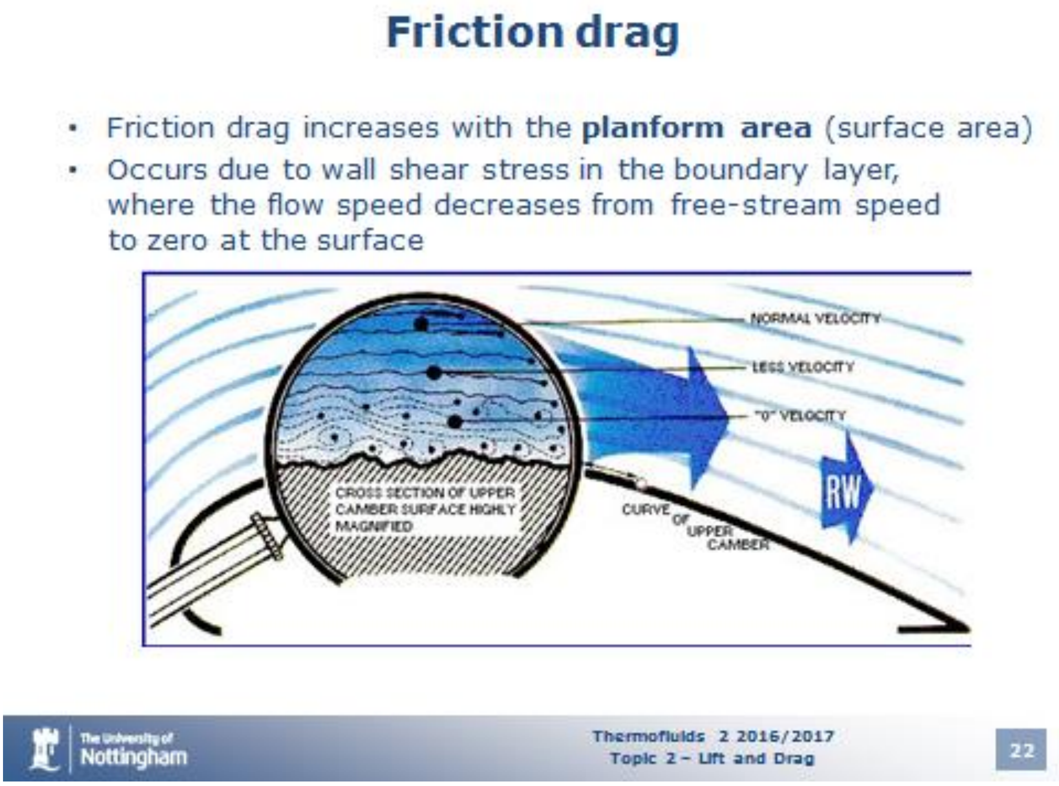


Figure 5 Friction drag is caused by surface friction.

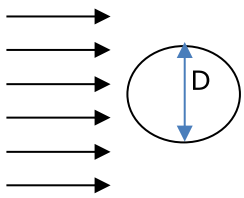
2. Bodies with a large parallel area to the flow (i.e. an airplane wing) experience mainly **friction drag**. Friction drag is similar to the friction forces in pipes you studied in MM2TF2. The cause of friction drag is the wall shear stress on the body surface, which is there because the you studied in Topic 1 – Boundary layers. Friction drag increases with area parallel to the flow. The friction drag is higher for **turbulent** flow than for laminar flow (observe that pressure drag is the opposite). Rough surface also increases friction drag. The friction drag was considered in detail in Fluids Topic 1 Boundary Layers.

   Friction drag is proportional to the sheared area of the body, which is the planform area (the area parallel to the flow). Therefore, to determine the drag for flat streamlined bodies the planform area is important.

It is important to remember that even if one type of drag dominates, a body never has only pressure drag, or only friction drag, but some of both. In calculating the total drag on a body we therefore have to add the pressure drag and the friction drag:

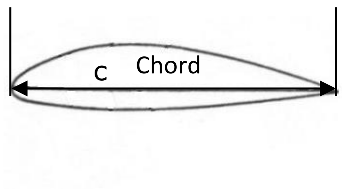
$$C_D = C_{D,pressure} + C_{D,friction}$$

Drag coefficient C_D depends on the Reynolds number Re of the body.
At low and moderate Reynolds numbers C_D is a function of Reynolds number; at higher Reynolds numbers C_D is independent of Re. Again we need to distinguish between the two types of bodies:
(1) For **stubby** bodies such as cylinders, spheres and disks the characteristic length in Re is the diameter D:



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

(2) For **streamlined** bodies the characteristic length in Re is the body length parallel to the free stream, such as the chord c for a wing:



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

It is most common for the pressure drag to be larger than the friction drag.

Knowing the body shape, and the Reynolds number, we can find the drag coefficient from a C_D - Re curve (and then rearrange the definition of C_D to find the drag force in m/s if needed).
Figure 6 shows how the drag coefficient for the illustrated streamlined, long cylinder (almost infinite depth into page) varies with thickness ratio at a fixed Reynolds number (10^6). Figure 6b has two curves, one for C_D non-dimensionalised with planform area and the other for frontal area. It is the same drag data in both cases. Figure 6a shows how the friction drag varies as a percentage of the total drag with increasing thickness, *t*. For a thin plate parallel to the flow t is very small and in the limit the pressure drag is zero and there is only skin friction. At the other end of the scale, for *t/c*=1 the friction drag makes a negligible contribution to the total drag.

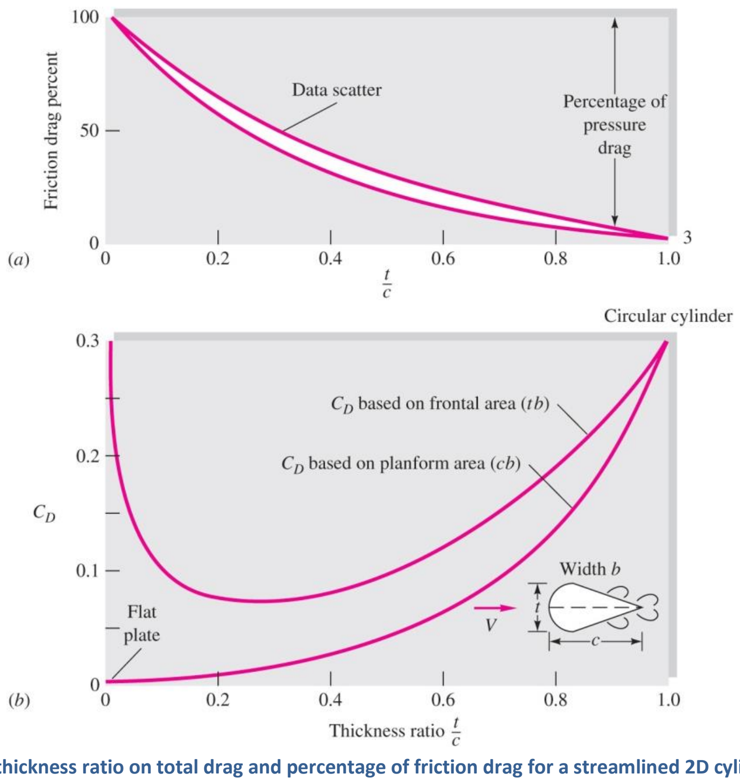


Figure 6: Effect of thickness ratio on total drag and percentage of friction drag for a streamlined 2D cylinder at Re=10^6 [1]

FLOW PAST A CIRCULAR CYLINDER

Consider flow past a circular cylinder.

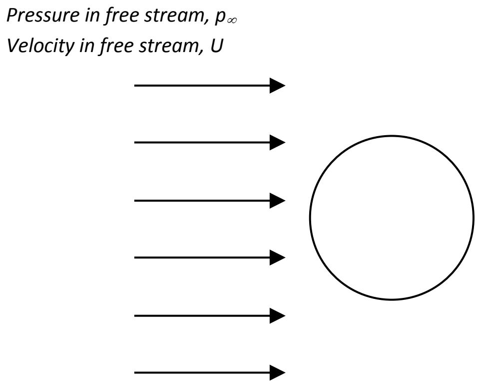


Figure 7: Flow past a circular cylinder

The pressure distribution around the cylinder for inviscid flow (no viscosity, no boundary layer, no separation) is given by:

$$\frac{p - p_\infty}{\frac{1}{2}\rho U^2} = 1 - 4sin^2\theta \qquad \text{Eq 1}$$

and is illustrated in Figure 8. (The quantitiy $\frac{p-p_\infty}{\frac{1}{2}\rho U^2}$ is termed the pressure coefficient, C_p.)

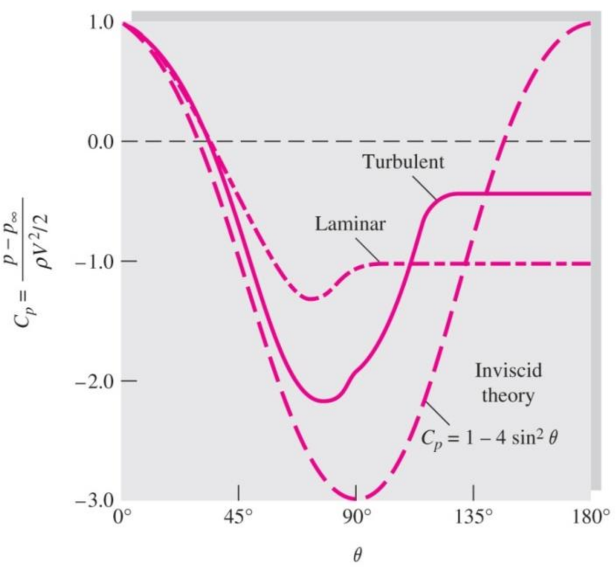


Figure 8: Pressure distribution around a circular cylinder for inviscid, laminar and turbulent flow [1]

The pressure distributions for laminar and turbulent flow are also shown. There is an adverse pressure gradient on the rear of the cylinder (velocity decreasing, pressure increasing) that leads to separation. In the laminar case this occurs at 82° whereas in the turbulent case separation does not occur until 120°.

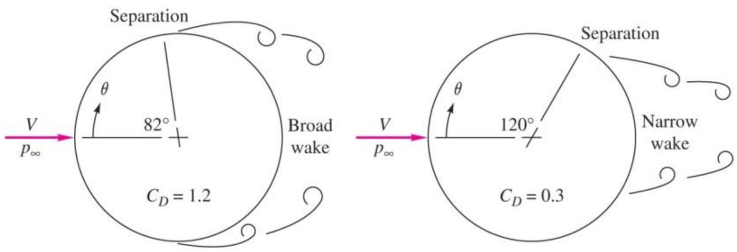


Figure 9: Laminar and turbulent separation and wake for a circular cylinder [1]

Because the flow separates earlier in the laminar case the wake is wider and consequently the pressure drag is higher than in the turbulent case.

A similar difference in separation behavior and drag is found for a sphere. The drag coefficient for a sphere with a turbulent boundary layer is significantly lower than for the laminar case. This is one of the reasons that golf balls are dimpled (rough) as this causes the boundary layer to become turbulent at lower flight speeds thereby reducing drag and enabling the ball to go further.

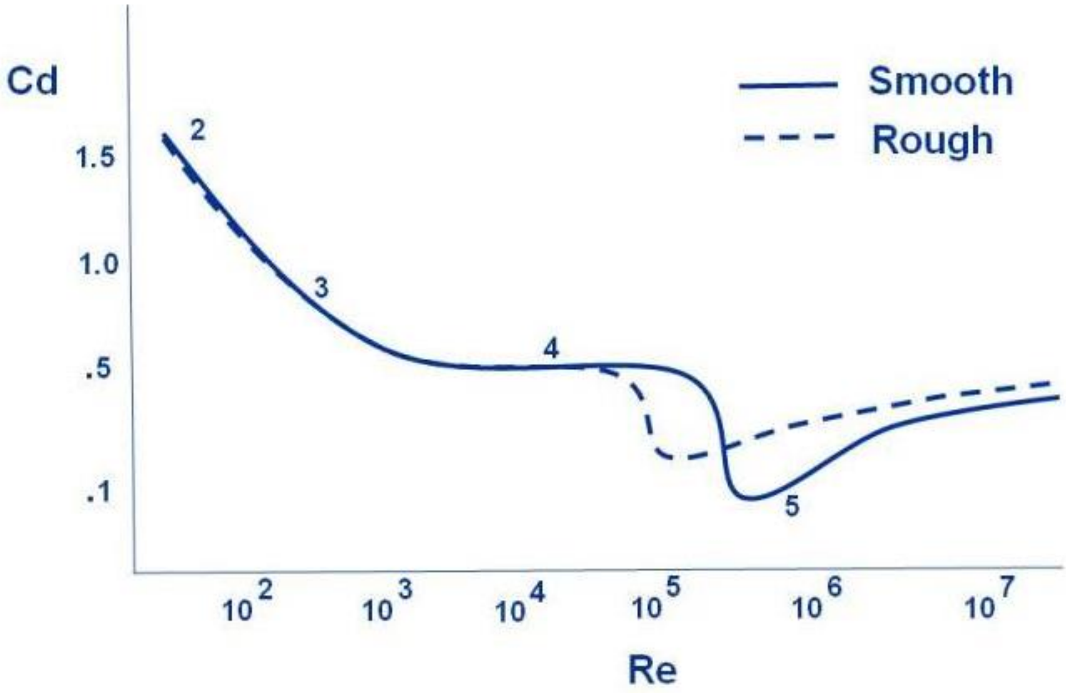


Figure 10: Drag coefficients for smooth and rough spheres [2]

Worked example 1

A golf ball has a diameter of 1.68” [3].
(a) What is the minimum ball speed at which the drag coefficient will be significantly affected by the dimples?
(b) Tiger Woods can launch the ball at 150 mph [4]. Where does this value sit at the Re-axis? Could Tiger Woods use a golf ball without dimples?

Answers: (a) 10.5 m/s, (b) 1.9×10^{5}

Your solution:

DRAG COEFFICIENTS FOR STANDARD 2-D BODIES

Drag coefficients for a few standard 2-D cylinders are given in Figure 11. The boundary layer on the sharp-edged bodies will tend to separate whether the boundary layer is laminar or turbulent because of the sharp edges. The elliptical cylinders behave in the same way as the circular cylinder and so the drag coefficient is sensitive to whether the boundary layer is laminar or turbulent.

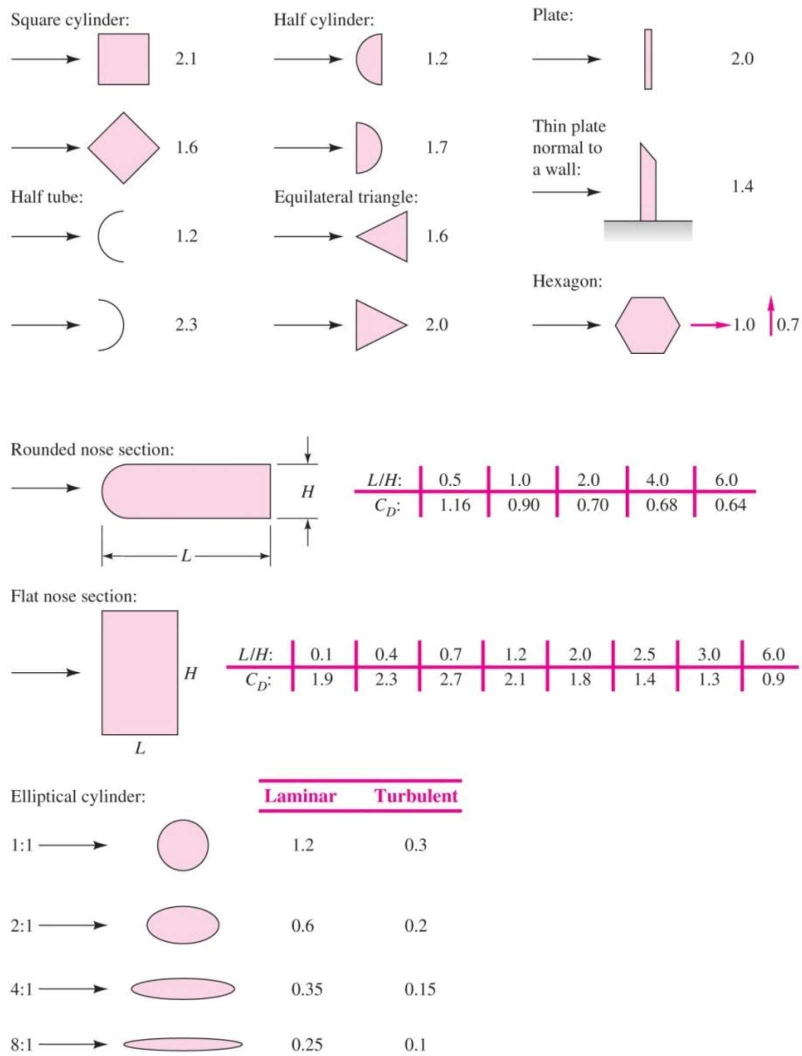


Figure 11: Drag coefficients for 2D bodies at $Re \geq 10^4$ [1]

In practice, each of these shapes will have its own Reynolds number-Drag coefficient characteristic. For example the one for a circular cylinder is shown in Figure 12, where the effect of surface roughness is also included. Consequently the data of Figure 11 should only really be regarded as an approximation. If highly accurate drag data is required then a highly accurate C_D-Re characteristic is necessary.

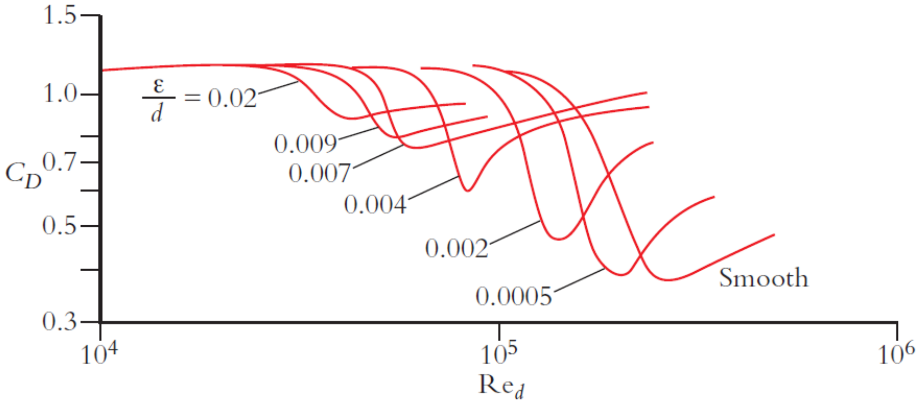


Figure 12: Drag coefficient for a circular cylinder as a function of Reynolds number and roughness [5]

In addition, the C_D data of Figure 11 is only for Reynolds numbers above 10^4. Below 10^4 the drag coefficient will be significantly higher, for example the full characteristic for infinite cylinders and spheres is shown in Figure 13.

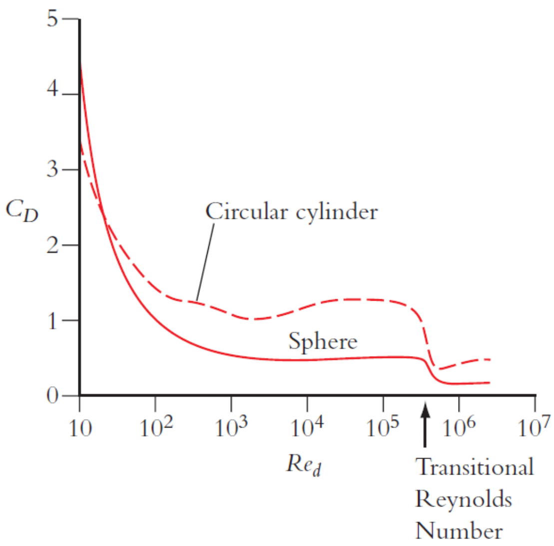
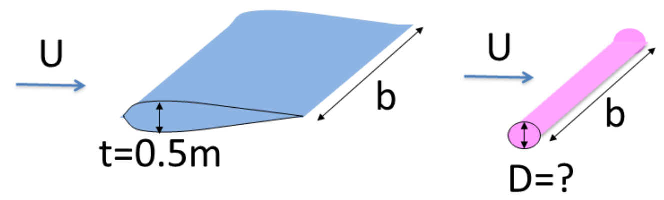


Figure 13: Full C_D-Re characteristic for infinite cylinders and spheres [5]

Worked Example 2

The streamlined cylindrical body (shown at left) is placed in an airflow (density 1.2 kg/m^3, viscosity $1.8x10^{-5}$kg/ms) has a drag coefficient of 0.12 based on frontal area when the Reynolds number **based on thickness t** is $4x10^5$. What diameter D is the circular cylinder (at right) of the same width b that has the same drag when placed in the same flow? Assume both cylinders are infinitely long.



Answer: D= 0.05m

Your solution:

Your solution to Worked example 2 continued:

The data of Figure 11 is for 2D (infinite length) cylinders. In practice, most engineering components will be too short to be regarded as 2D. For example for a circular cylinder in laminar flow at (10^4<) Re<$5\text{x}10^5$ there is an effect of L/D ratio way beyond L/D=40, as illustrated in Figure 14. The drag coefficient for an infinitely circular long cylinder in laminar flow is 1.2, as shown in Figure 11. (Please observe that the drag does not always increase with L/D ratio for other bodies. However, in general the drag is a function of the aspect ratio for all bodies.)

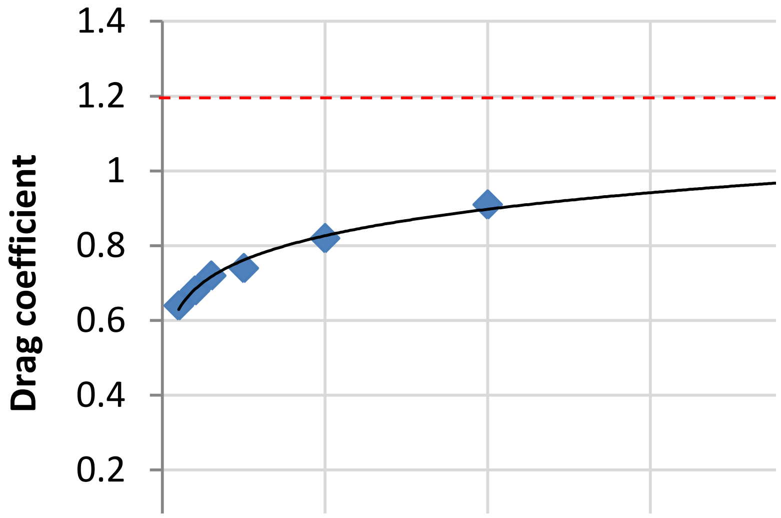
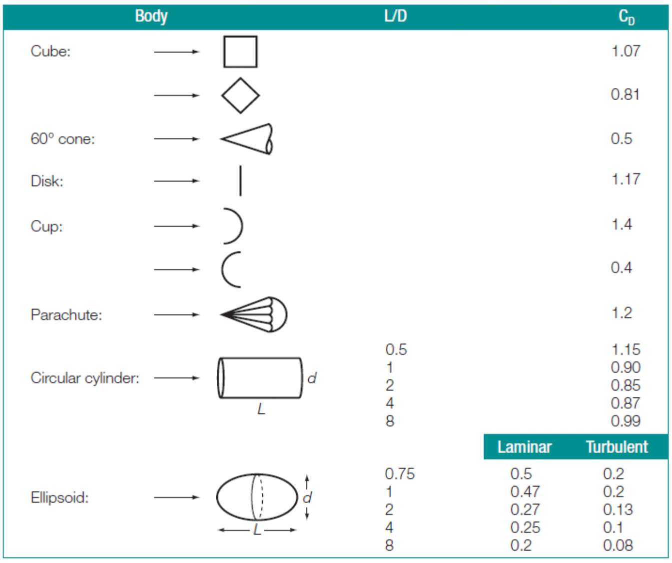


Figure 14: Effect of L/D ratio on C_D for circular cylinders in laminar flow (data from [1])

DRAG COEFFICIENTS FOR STANDARD 3-D BODIES

Drag coefficients for a few standard 3-D shapes are given in Figure 15. As with 2-D cylinders, sharp edges lead to flow separation and thus high drag that is largely insensitive to Reynolds number. For rounded bodies the point of flow separation depends on Reynolds number and so the drag coefficient does too. For example see the ellipsoid data in Figure 15.



Body	L/D	C_D	
Cube:		1.07	
(cube, diamond orientation)		0.81	
60° cone:		0.5	
Disk:		1.17	
Cup:		1.4	
(cup, reversed)		0.4	
Parachute:		1.2	
Circular cylinder:	0.5	1.15	
	1	0.90	
	2	0.85	
	4	0.87	
	8	0.99	
		Laminar	Turbulent
Ellipsoid:	0.75	0.5	0.2
	1	0.47	0.2
	2	0.27	0.13
	4	0.25	0.1
	8	0.2	0.08

Figure 15: Drag coefficients for a few 3-D bodies, $Re \geq 10^4$ [5]

It should be noted that the drag coefficients of Figure 15 are obtained from windtunnel (model fixed) tests. The sphere data (Figure 13) is obtained using either falling sphere tests or windtunnel tests. For a buoyant (rising) sphere such as a bubble, at lower Reynolds numbers there may be wake instability that will lead to the body spiraling upwards and C_D will be increased. In addition, a bubble formed from a less dense liquid rising in a more dense liquid may deform to a less resistive (lower C_D) shape.

Worked Example 3

Estimate the maximum bending moment at the base of a single spruce tree in a wide open field at a wind speed of 60 mph. The tree is 3m tall and can be treated as a cone (triangular cross-section) of base diameter 2m and height 3m. Take the density of air to be 1.2 kg/m^3 and the viscosity as 1.8×10^{-5} kg/ms.

U (m/s)	10	20	30	40
C_D ($\pm$0.2)	1.2	1.0	0.7	0.5

Data from [1]

Ans: 1029 Nm

Your solution:

Your solution to Worked Example 3 continued:

DRAG REDUCTION

In many cases drag reduction is highly desirable. For transport (road, rail, air) increased drag leads directly to increased fuel costs. Remembering that total drag is made up from friction drag and pressure drag, an important strategy is to reduce the pressure drag of immersed bodies by streamlining them ie shaping the bodies in such a way as to move the flow separation point further downstream. This will effectively reduce the width of wake (the area in the downstream of flow separation), leading to a reduction of the low pressure region in the rear of the immersed bodies [5].

White [1] has a chart showing how car drag coefficients have decreased with time up to the year 2000. For examples of more recent data the Toyota Prius V has a quoted C_D of 0.25 [7] and Nuna, the Solar World challenger from 2007 has a quoted C_D of 0.07 [9] – which is less that White's theoretical minimum of 015.

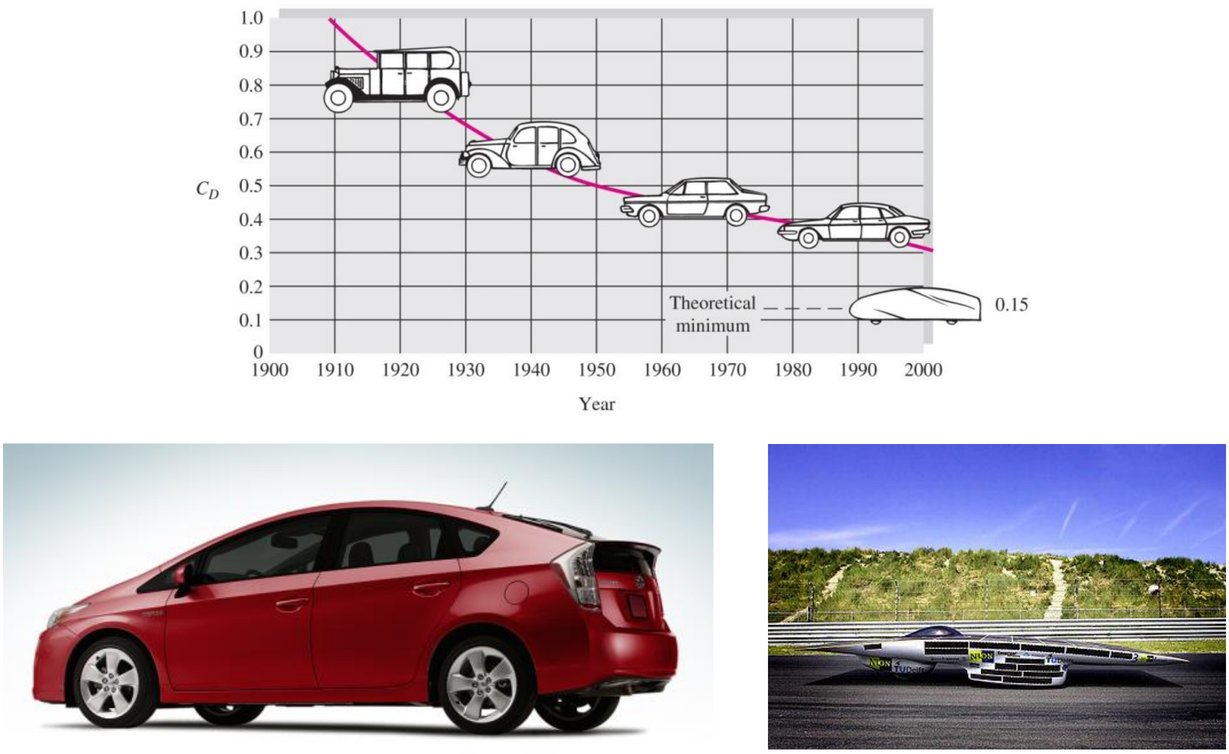


Figure 16: Car drag coefficients. a) Trend to 2000 [1]; b) Toyota Prius V [6], C_D = 0.25; c) Nuna [8], C_D = 0.07

Other fields where there are substantial benefits in drag reduction include:

- Oil flow in pipelines
- Racing yachts (vee-groove microriblets for example)
- Performance sports clothing (for swimming mainly)

There are numerous research projects and substantial funding in these and other areas.

Group activity

- With ~3 people around you, try identify 3 features in this figure which reduce drag
- Try to specify also which is reduced – is it pressure drag, friction drag (or both)?

Your notes from the discussion:

LIFT

Lifting bodies are primarily aerofoils (airfoil), hydrofoils (liquid based equivalent of aerofoil) and vanes. Usually the intention is to provide as much lift as possible while minimizing drag (lift is what you want, drag is what you pay for!). Aerodynamic lift and aerofoil performance is a huge area for study and only the basic concepts are introduced here.

A generic aerofoil is illustrated in Figure 17. The straight line joining the leading edge and the trailing edge is the chord. If the shape is not symmetrical then the aerofoil is *cambered* and the concept of the mean camber line is introduced. The mean camber line is the locus of midpoints between the upper and lower camber lines.

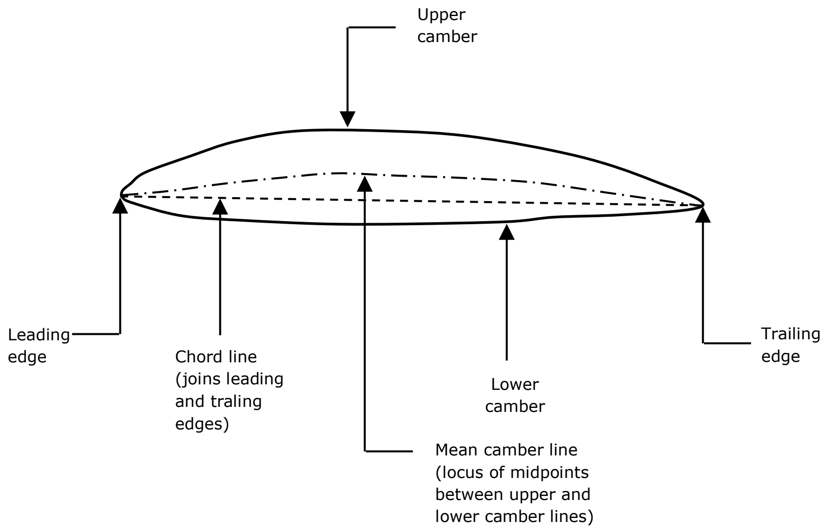


Figure 17: Definitions for aerofoils

Figure 18 illustrates the angle of attack, the angle between the free stream velocity vector and the chord line. Lift and drag coefficients for aerofoils are based on the planform area (*bc* in Figure 18). With reference to Figure 18, the Lift force, *L* and Drag force, *D* are non-dimensionalised into lift and drag coefficients as:

$$C_D = \frac{D/A_p}{\frac{1}{2}\rho V^2}, \qquad C_L = \frac{L/A_p}{\frac{1}{2}\rho V^2}$$

Eq 2

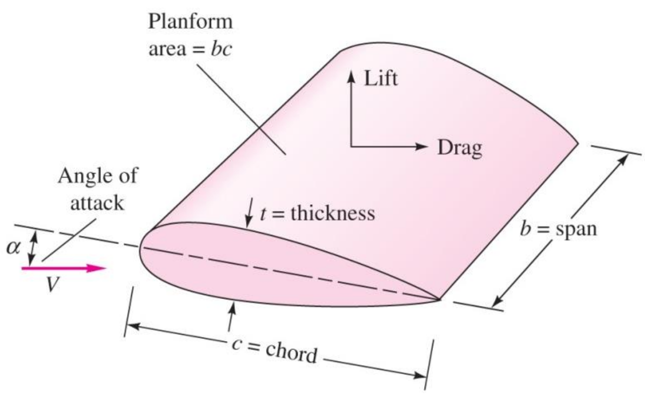


Figure 18: Further definitions for aerofoils [1]

If the chord length is not constant (eg tapered wing) then A_p is obtained by integration: $A_p = \int c\, db$

HOW DOES AN AEROFOIL GENERATE LIFT?

Airfoil creates lift by turning the flow around it. It creates a downwards force on the flow, and by Newton's 3rd law, the air creates an equal but opposite (=upwards) force on the airfoil.

You can observe this in all the airfoil figures below. At free flight, the free-stream flow approaching the airfoil leading edge is nearly horizontal. However, the flow leaving the airfoil at the trailing edge has been deflected down. The flow has changed direction, which means that it has accelerated, so a force down must have acted on it. By Newton's 3rd law, a force up must then have acted on the airfoil. The principle is illustrated in Figure 19 below which is from NASA web site. (For those who are interested, the same website also considers some common but incorrect explanations of lift and explains why they are wrong).

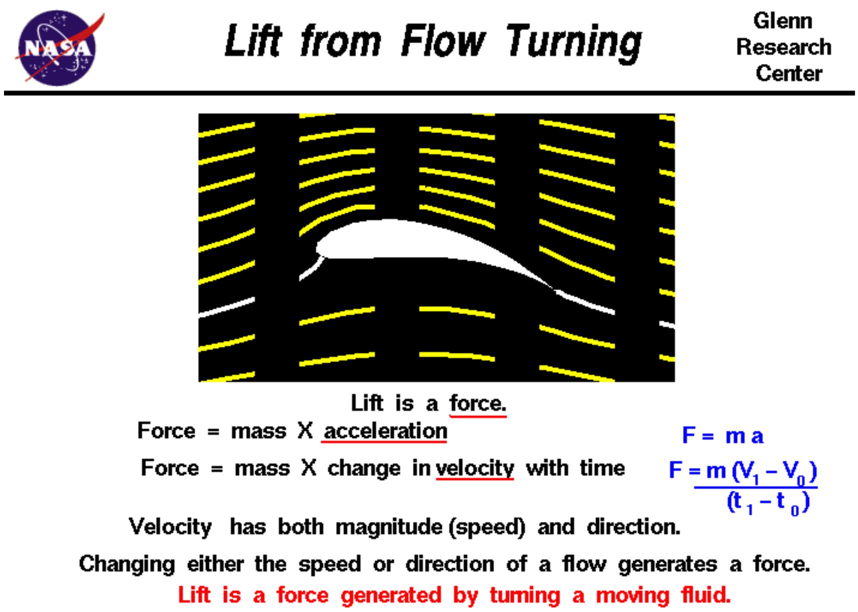


Figure 19 Principle of lift generation [20].

You might wonder actually how the airfoil makes the flow turn. This has to do with the shape of the airfoil, in particular the sharp trailing edge. From previous sections, you might remember that for shapes with corners, the flow separates in the corners (triangular prism, cube). The trailing edge is a sharp corner. The flow around the airfoil wants to separate at the trailing edge (this is called Kutta condition in aerodynamics) (. rather than on the smooth surfaces. Hence, the flow must follow the airfoil and this means turning down along with the airfoil surface.

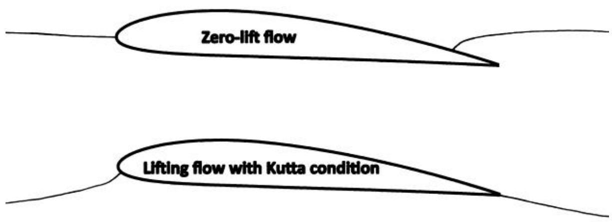
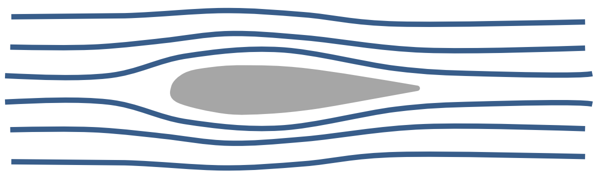
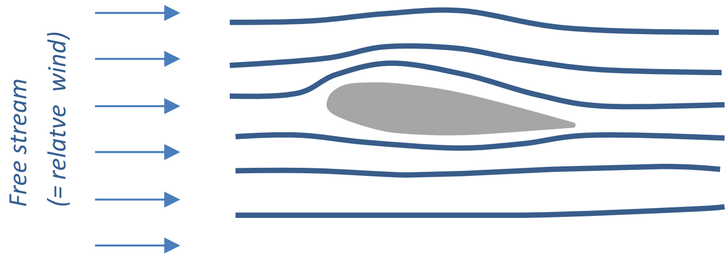


Figure 20
Bottom: Flow around a lifting airfoil separates at the sharp trailing edge (Kutta condition).
Top: An unrealistic zero-lift flow which would violate the Kutta condition.

For a symmetric aerofoil shape that is parallel to the flow, the flow is horizontal in the front and back of the aerofoil and no lift is generated. However, if the aerofoil is angled to the free-stream velocity vector (relative wind), then the flow leaving the trailing edge is turned down, creating lift. This is illustrated in Figure 21



a) Symmetric aerofoil parallel to free stream – no lift



b) Symmetric aerofoil at a small angle to free stream – lift is generated

Figure 21: Illustrating how lift is generated from a symmetric aerofoil at a small angle to the relative wind

The angle between the chord of the aerofoil and the relative wind is called the angle of attack, α. As illustrated in Figure 22, a symmetric aerofoil generates no lift when parallel to the relative wind (zero angle of attack) whereas a cambered aerofoil will generate a little bit of lift even at zero angle of attack. As long as the flow remains attached (low angle of attack) more lift is generated at a higher angle of attack.

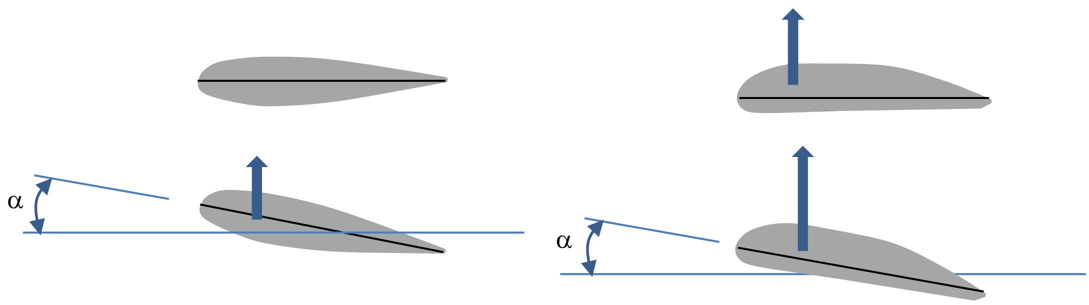


Figure 22: Symmetric and cambered aerofoils. A symmetric aerofoil only generates lift when it has a positive angle of attack. A cambered aerofoil will generate lift even at zero angle of attack.

In general lift and drag data is obtained by experimentation. For low Mach number flow C_L and C_D depend on roughness, angle of attack, α and chord Reynolds number.

There are many tested aerofoil designs and a huge number of them were developed by NACA[3]. A NACA aerofoil is described using a series of digits following the word "NACA." The parameters in the numerical code can be entered into equations to precisely generate the cross-section of the airfoil and calculate its properties.

The NACA four-digit wing sections define the profile by:

1. One digit describing maximum camber as percentage of the chord.
2. One digit describing the distance of maximum camber from the airfoil leading edge in tens of percents of the chord.
3. Two digits describing maximum thickness of the airfoil as percent of the chord.

For example, the NACA 2412 airfoil has a maximum camber of 2% located 40% (0.4 chords) from the leading edge with a maximum thickness of 12% of the chord. Four-digit series airfoils by default have maximum thickness at 30% of the chord (0.3 chords) from the leading edge [11]. There are also 5-digit, 6-digit, 7-digit and 8-digit series!

Although the boundary layer on an aerofoil is most commonly turbulent, laminar flow can be maintained up to chord-based Reynolds numbers of 30×10^6 by appropriate profile shaping. Laminar-flow aerofoils have favourable drag characteristics and are used in the design of many modern subsonic aircraft [12].

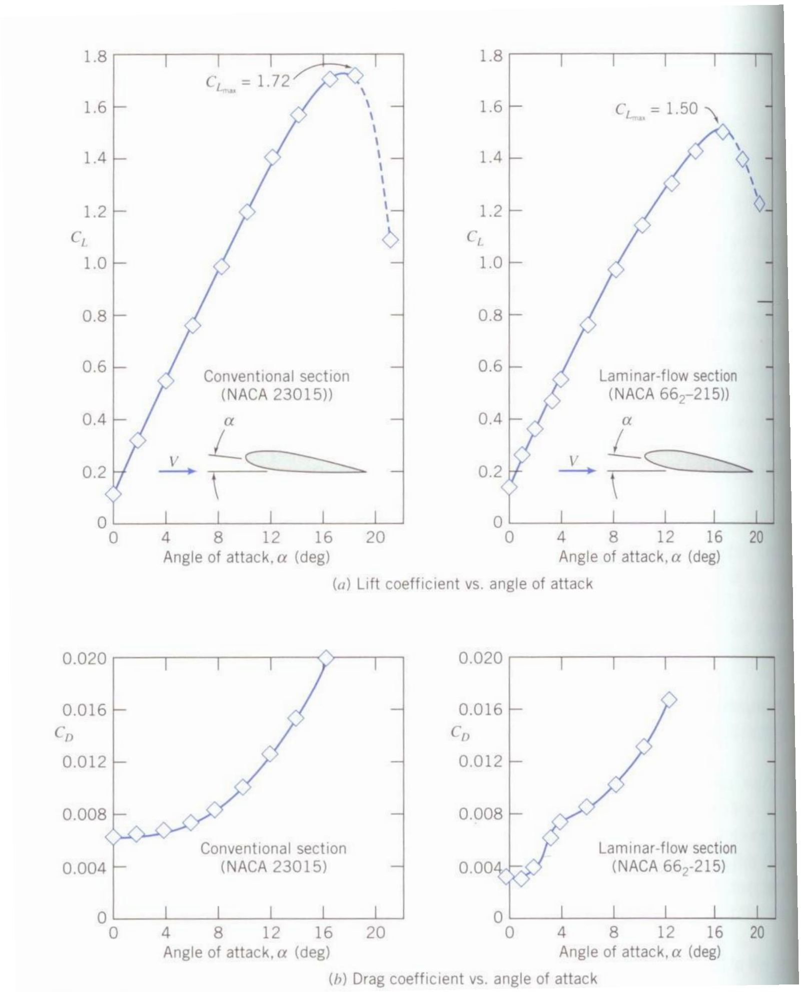


Figure 23: Lift and drag coefficients as a function of angle of attack for two aerofoil sections at Re=9×10^6. From [12]

[3] National Advisory Committee for Aeronautics (in USA) now replaced by NASA (National Aeronautics and Space Administration

STEADY HORIZONTAL FLIGHT

For an aircraft in steady horizontal flight, there are no accelerations and all the forces acting on the aircraft balance. In the vertical direction the total lift force (from all the lifting elements, wings, fuselage etc) must balance the total weight of the aircraft. By far the greater proportion of the lift comes from the wings and for initial calculations it is often a satisfactory approximation to assume all the lift comes from the wings, in which case C_L in Eq 3 can be taken as the C_L of the wings only.

$$W = L = C_L \frac{1}{2} \rho U^2 A \qquad \text{Eq 3}$$

In the horizontal direction in steady flight the thrust force generated by the engines or propellers must be equal to the total drag of the aircraft.

$$T = D = C_D \frac{1}{2} \rho U^2 A \qquad \text{Eq 4}$$

In Eq 4 the drag coefficient is that of the entire aircraft but again, as a first approximation this can be equated to the drag of the wings (for subsonic flight and relatively large aspect ratio wings) and under these circumstances the drag coefficient is that of the wings. Note that if T is the total thrust for the aircraft then A must be the total wing area (ie for both wings) if the equation is to balance.

Thrust force can be related to thrust power using the standard equation power = force x velocity. Thrust force is in Newtons and thrust power in Watts.

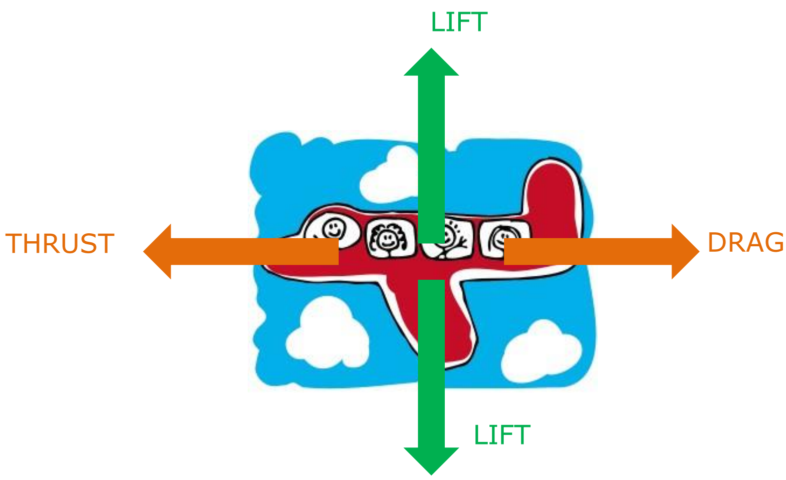


Figure 24: In steady, horizontal flight: Lift = Weight and Thrust = Drag

Worked Example 4

An Embraer E190 has a mass of 47,000 kg.

a) What is the lift force when the jet cruises in steady horizontal flight at 500 kmh at an altitude of 5000m?
b) Assuming all the lift comes from the wings calculate the lift coefficient given that the wing area is $92.5m^2$?
c) If the drag coefficient is 3% of the lift coefficient, what is the thrust required?
d) What thrust power are the engines generating at this flight condition?

Ans: 461.07 kN, 0.701, 13832N, 1921 kW

Your solution:

STALL

At a low angle of attack there is an adverse pressure gradient on the rear surfaces and either no separation or only a small amount (depending on the exact shape of the aerofoil). As the angle of attack is increased the separation point moves forward until there comes a point where there is a sudden decrease in lift. This is called stall and the angle at which it occurs is the stall angle. Typically lift at the stall condition is insufficient to support the weight of the aircraft and it will fall.

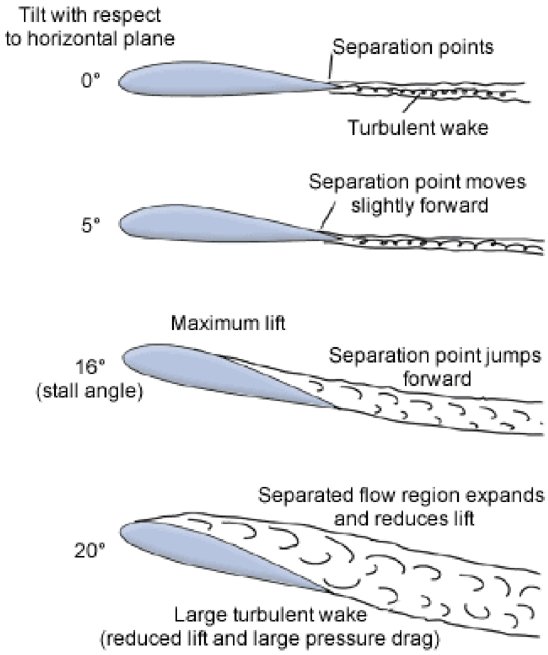


Figure 25: Showing effect of angle of attack on flow around an aerofoil [10]

As air flows over an airfoil, lift is generated as a consequence of the differential pressure between the lower (pressure) surface and the upper (suction) surface. Under normal operating conditions, air flows faster over the upper surface than the lower surface, thus pressure is lower on the upper surface and there is a resultant upwards force (lift). If the angle of attack of the airfoil is increased a little from zero then the lift is increased. However, there comes a point where further increase of the angle of attack does not cause further increase in lift. The flow is unable to remain attached to the upper surface and instead becomes detached. When this happens there is no longer higher speed flow over the top of the airfoil and there is a sudden, dramatic, loss of lift. This is known as stall and it occurs just after the maximum C_L condition.

Examination of Eq 3 shows that for a given aircraft weight, at the maximum lift condition there is a minimum velocity. If the velocity is any lower than this minimum velocity then the lift force for the aircraft is less than the weight. This minimum speed is often referred to as the stall speed because if the aircraft flies slower than the stall speed it will start to fall and any attempt to increase lift through increasing the angle of attack reduce in stall of the flow over the wings (because at the limiting condition with maximum lift the angle of attack is already at its maximum). Rearranging Eq 3 with $C_{L,max}$ and U_{stall} yields:

$$U_{\text{stall}} = \sqrt{\frac{2W}{\rho C_{L_{max}} A}}$$

Eq 5

It is generally accepted [1] a pilot needs a speed greater than about $1.2U_{stall}$ to avoid instability and stall and it is an FAA (Federal Aviation Authority) requirement for commercial aircraft to always operate above $1.2U_{stall}$.

This minimum safe speed is particularly important for landing because it is the time when a) the aircraft needs to be travelling slowly for safe touchdown and b) any sudden uncontrolled loss of lift is dangerous because of the proximity of the ground. The minimum safe landing speed is taken as 1.2 times the stall speed associated with the maximum lift condition for the wings when configured for landing.

Worked Example 5

The Geier 2 glider illustrated has a total wing area of $14m^2$ and a weight of 370 kg. What is its minimum safe landing speed if $C_{L\ max}$ is 1.28? Take air density at ground level to be 1.2 kg/m^3.

Ans:22 m/s

Your solution:

At landing a low speed is desirable and lowering U_{stall} can be achieved either by increasing the wing area A or C_{Lmax}. Flaps are hinged surfaces on the trailing edge of a wing. When the flaps are extended the effective wing area is increased and so the minimum speed of the aircraft is reduced. Figure 26 shows the wing flaps on a Boeing 737 as they go from retracted to fully extended ready for landing.

Figure 26: wing flaps on a Boeing 737 shown going from retracted to fully extended for landing [19]

Figure 27 shows shows lift and drag data for a NACA 23012 aerofoil, illustrating the significant increase in C_L even for a single slotted flap.

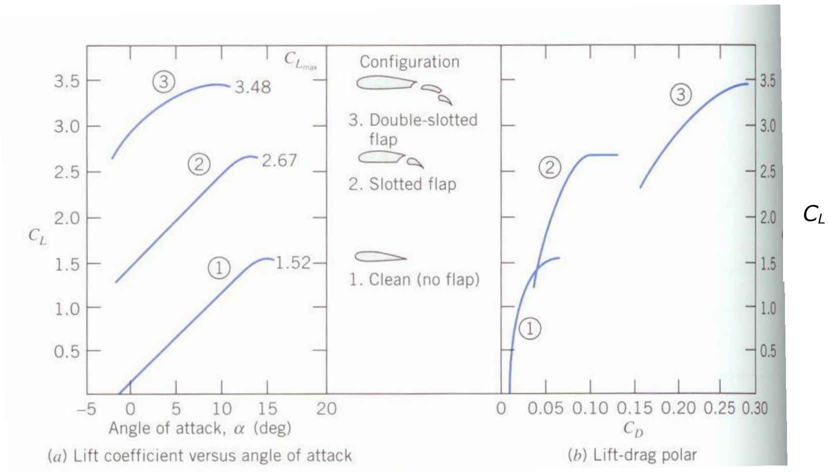


Figure 27: Effect of flaps on lift and drag of NACA 23012 aerofoil

FINITE SPAN WINGS

Although aerofoil data is typically presented for infinite length (2D) sections, in all real applications aerofoils are of finite span (aircraft wings are not infinitely long). These real-world aerofoils have less lift and more drag than the 2D data shows. One of the reasons for this is trailing vortices. Near the wing tip there is higher pressure on the lower surface than on the upper and this causes flow around the wing tip, as illustrated in Figure 28. Another consequence of trailing vortices is that drag is increased with the additional drag termed *induced* drag.

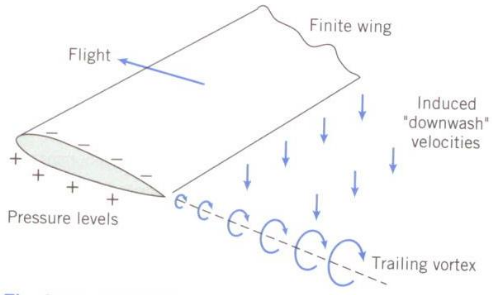


Figure 28: Schematic representation of wingtip trailing vortices [12]

The trailing vortices and consequent increase in drag and reduction in lift are concentrated near the tip of the wing and consequently are for more significant for short, stubby wings than for long ones. The effect of finite span is usually correlated with wing aspect ratio:

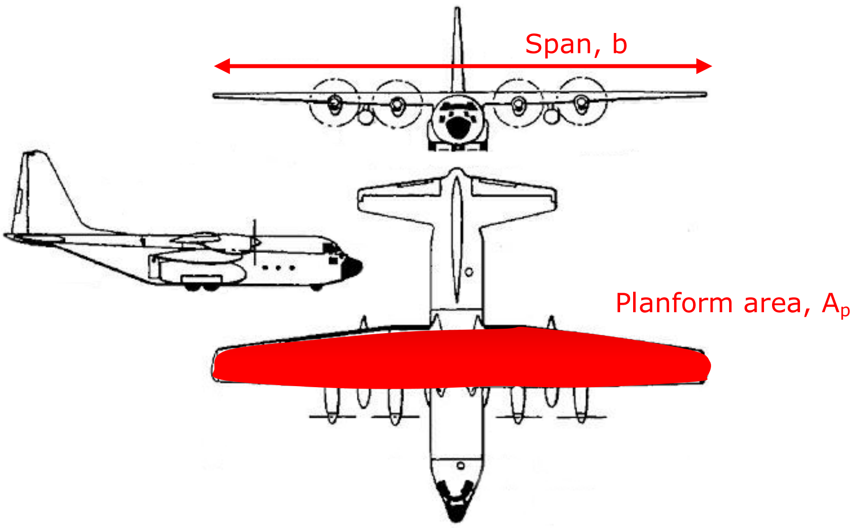


Figure 29: Illustrating wingspan and planform area (base illustration from [14])

Span is the tip to tip length of the wing, as illustrated in Figure 29. The platform area is the entire area of the aircraft wing (or wings if there are two separate halves). By definition [15]:

$$A_{Ratio} = \frac{b^2}{A_p} = \frac{b}{c} \; or \; \frac{b}{\bar{c}} \text{ for rectangular section and average chord respectively} \qquad \textbf{Eq 6}$$

For a rectangular wing the planform area = width x chord: $A_p = bc$

The effect of finite aspect ratio is illustrated in Figure 30 and Figure 31. Compared to the infinite aerofoil, the angle of attack has to increase by $\Delta\alpha$ to get the same lift. Theory predicts that this increase in effective angle of attack (in radians) is given by:

$$\Delta\alpha = \frac{C_L}{\pi A_{Ratio}}$$

Eq 7

Also, instead of being perpendicular to the motion the lift force line of action leans backwards from perpendicular by $\Delta\alpha$.

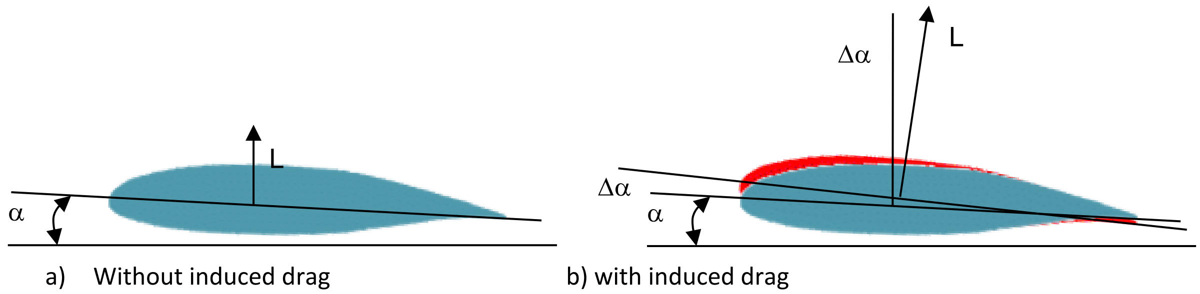


Figure 30: With induced drag a larger angle of attack is required for the same lift

There is therefore an induced component of drag such that:

$$\Delta C_D \approx C_L \Delta\alpha = \frac{C_L^2}{\pi A_{Ratio}}$$

Eq 8

Thus:

$$C_D = C_{D\infty} + \frac{C_L^2}{\pi A_{Ratio}}$$

Eq 9

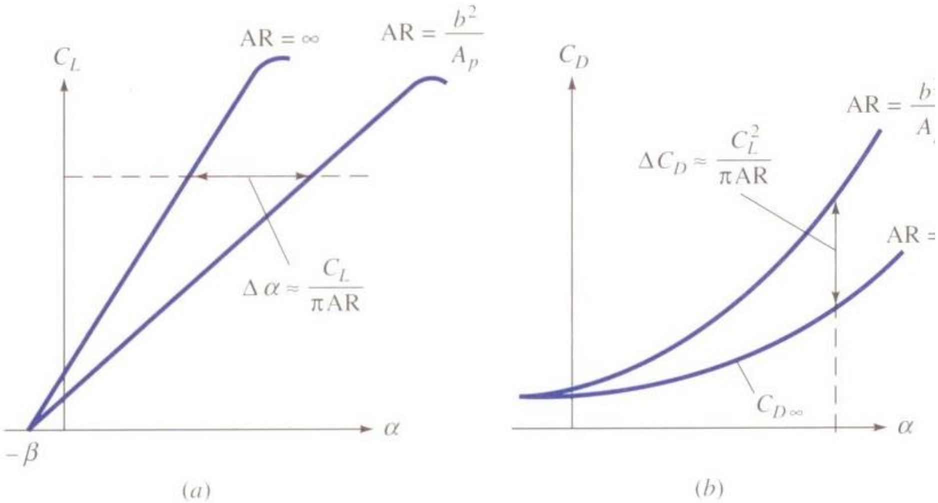


Figure 31: Effect of finite aspect ratio on lift and drag. a) shows how effective angle increases; b) shows how drag increases [1]

Worked Example 6

The Geier 2 glider of worked example 9 has a wingspan of 17.76m and a wing area of 14m^2. The wing aspect ratio is 22.53. The aerofoil is NACA 633-618 [13]. By how much must the angle of attack change from the infinite case for a C_L of 1.1? (use the symbols in the figure on the next page). What is the drag coefficient of this wing at this condition?

Ans 0.0155rad or 0.9°, 0.036

Your solution:

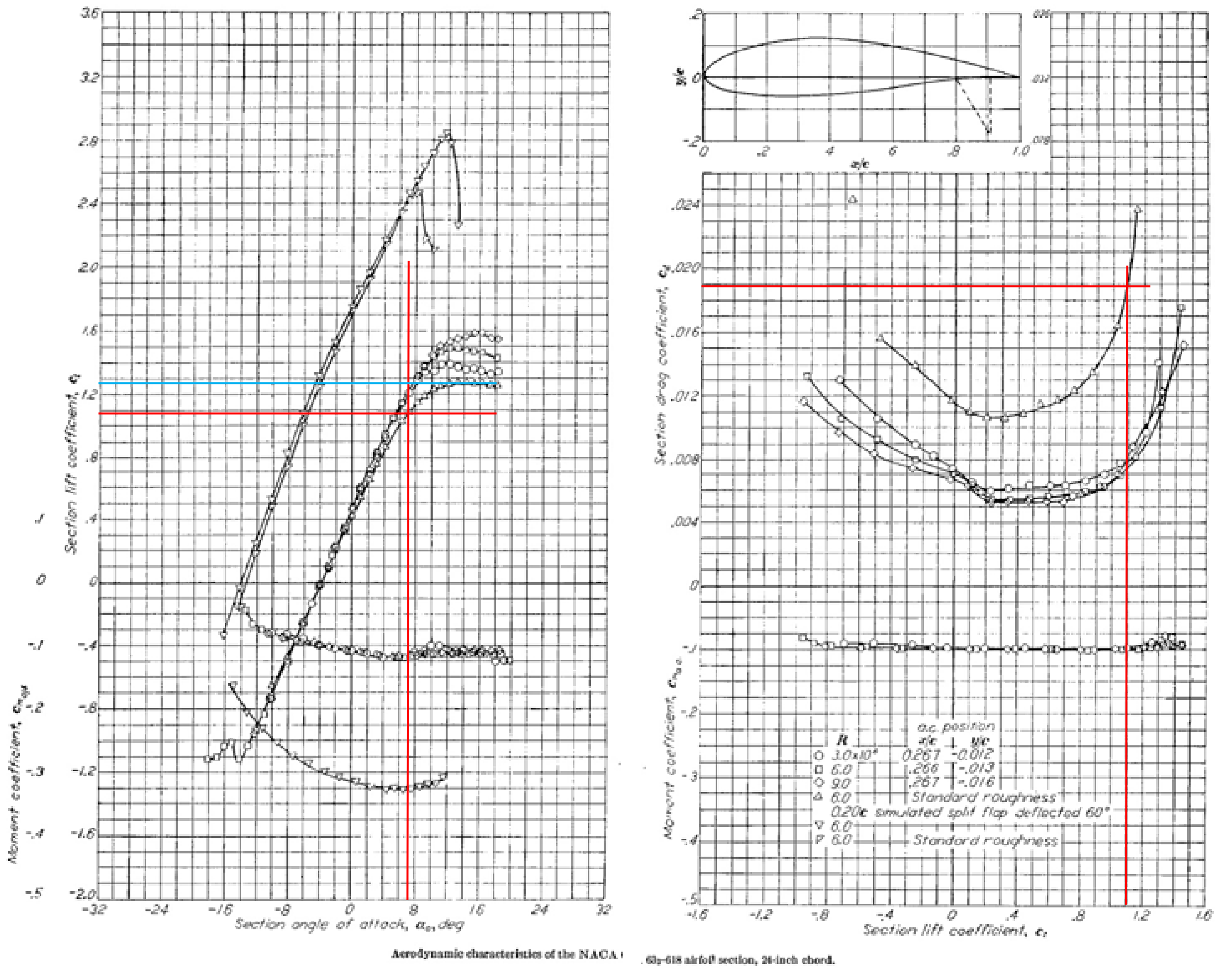


Figure 32: Lift and drag data for NACA 633-618 aerofoil [14]
(lines for worked example 10)

Aerodynamic lift is an important consideration in the design of high speed land vehicles. At high speeds lift can unload the tyres and a direct consequence of this reduction in steering control and stability. An inverted aerofoil creates a downforce (opposite to lift) on the vehicle and this improves traction.

An endplate or winglet added to the tip of the wing will reduce the strength of the trailing vortex and thereby reduce the induced drag. In addition a winglet will produce a small component of force in the flight direction – which also reduces overall drag.

Figure 33: Endplates (a) and winglets (b) reduce induced drag.

SPIN AND LIFT

Another area where aerodynamic lift plays a significant role is in sport. A spinning sphere behaves differently to a non-spinning one and in golf for example ball spin creates aerodynamic lift that keeps the ball in the air for longer. The effect of the spin is to modify the pressure distribution around the ball and thus the location of the boundary layer separation point. Figure 34a) shows the modified flow pattern for a smooth sphere; as the wake is directed downwards there is a lift force upwards, perpendicular to the spin axis and to V. This lift force is known as the Magnus effect (after Heinrich Magnus). If the spin were in the opposite direction there would be a down force and if the spin were about the vertical axis the ball would curve to the side (can lead to drift of artillery shells for example). Spin ratio, the ratio of sphere surface speed to free stream flow, makes a big difference to the lift coefficient as illustrated in Figure 34b).

$$Spin\ ratio = \frac{\omega R}{V} = \frac{\omega D}{2V} \qquad \textbf{Eq 10}$$

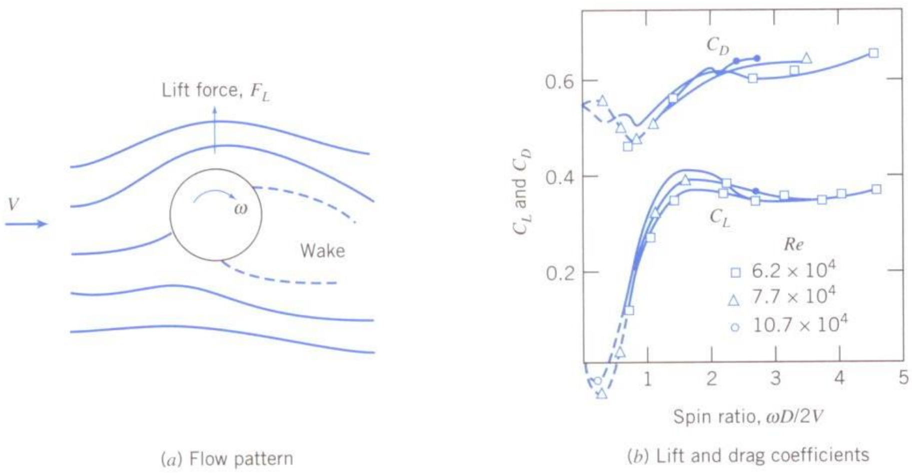


Figure 34: Spinning sphere. Modified flow patter and effect on lift and drag coefficients [12]

A similar effect is found for golf balls, as shown in Figure 35, although the values of C_D and C_L are somewhat different because of the rough surface of a golf ball.

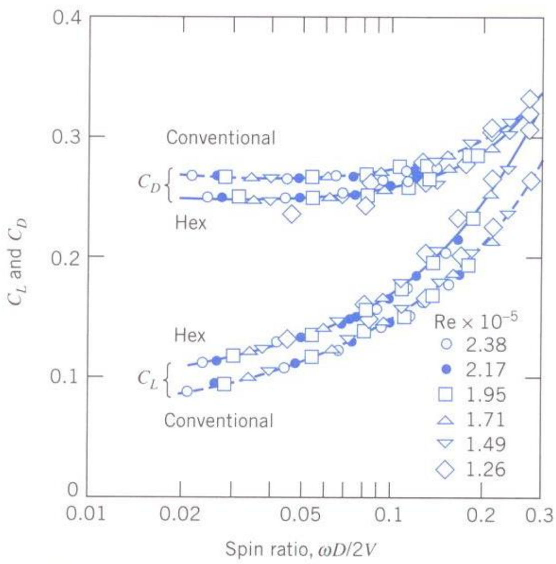


Figure 35: Effect of spin on lift for golf balls [12]

Worked Example 6

A smooth tennis ball of mass 57g and diameter 64 mm is hit at 25 m/s with top-spin of 7500 rpm. Calculate the aerodynamic lift on the ball. How does this compare to the weight of the ball? Discuss qualitatively how this will affect its trajectory. Take air properties to be ρ=1.2kg/m^3 and μ=1.8x10^{-5} kg/ms.

Ans: 0,31N, 55% of weight

Your solution:

Space for additional notes:

REFERENCES

1. Frank White, Fluid Mechanics 5th Edition, 2003, ISBN: 0-07-119911-X, Chapter 7.
2. http://www.grc.nasa.gov/WWW/K-12/airplane/dragsphere.html, accessed Nov 2010
3. http://en.wikipedia.org/wiki/Golf_ball#Regulations, accessed Nov 2010
4. http://answers.yahoo.com/question/index?qid=20070503055442AADo5OZ, accessed Nov 2010
5. An introduction to Mechanical Engineering Pt 2, pub: Hodder, ISBN-10: 0340939966
6. http://www.toyota.com/prius-hybrid/photos/2010-Prius-Hybrid-11.html , accessed Nov 2010
7. http://www.toyota.com/prius-hybrid/specs.html accessed Nov 2010
8. http://en.wikipedia.org/wiki/Nuna accessed Nov 2010
9. http://en.wikipedia.org/wiki/Automobile_drag_coefficient , accessed Nov 2010
10. http://quest.arc.nasa.gov/aero/virtual/demo/aeronautics/tutorial/wings.html accessed Nov 2010
11. http://en.wikipedia.org/wiki/NACA_airfoil , accessed Nov 2010
12. RW Fox, AT McDonald & PJ Pritchard, Introduction to Fluid Mechanics, 6th edition, 2004, USBN: 0-471-20231-2
13. http://www.sailplanedirectory.com/zwf2.htm , accessed Nov 2010
14. https://www.ffa.int/mcs/node/1571 , accessed Oct 2014
15. http://www.grc.nasa.gov/WWW/k-12/airplane/geom.html , accessed Oct 2014
16. http://www.scribd.com/doc/37212290/ABOTT-Summary-of-Airfoil-Data , accessed Nov 2010
17. http://www.f1technical.net/forum/viewtopic.php?f=6&t=7745&start=0 , accessed Nov 2010
18. http://www.aspireaviation.com/category/airbus/ , accessed Nov 2010
19. http://commons.wikimedia.org/wiki/Category:Cabin_window_views_of_aircraft_wings , accessed Nov 2010
20. https://www.grc.nasa.gov/www/k-12/airplane/right2.html , accessed Oct 2016