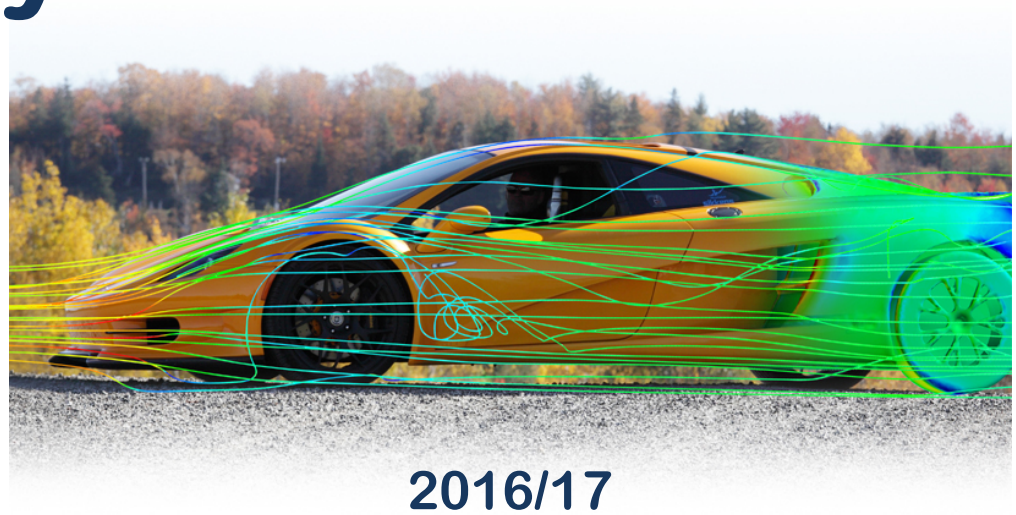
Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

Vehicle Dynamics: Aerodynamics



2016/17

Tim Tudor
tim.tudor@uwtsd.ac.uk

Aerodynamics
Introduction

The course

- During this course we will look at the science of aerodynamics (Fluid mechanics) and how it may be applied to vehicles to achieve the design goals just outlined.

Topics Include

- Fluid Physics
- Coefficients of pressure
- Boundary Layers
- Separation
- Turbulence
- Wing Theory
- Wing Design
- Computational Fluid Dynamics
- Wind tunnel testing
- External Aerodynamics
- Common aerodynamic devices
- Internal devices

Aerodynamics
Introduction

Assessment

- Assessment will take place by means of a single exam question in the Vehicle Dynamics examination at the end of semester 2

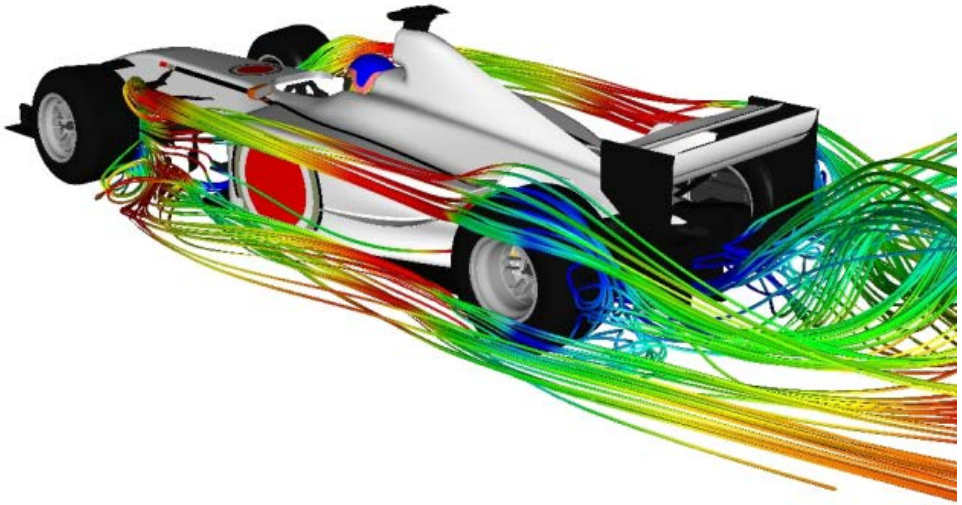
- This will take the form of answer 3 from 4 questions, with Roger and Mark contributing three questions and Aerodynamics the fourth question

Aerodynamics
Introduction

Aerodynamics Definition

– Simple definition of aerodynamics is;

"The study of the flow of air around and through a vehicle, primarily if it is in motion."

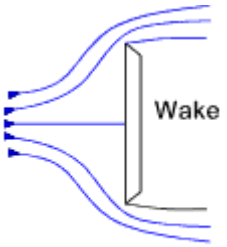


Aerodynamics
Introduction

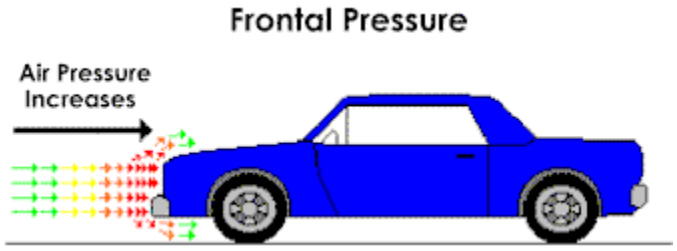
The role of aerodynamics in vehicles



- **Drag** – Resistance of air acting in rearward direction. Slows cars down on a given straight
  - Frontal Pressure –
    - As air molecules approach vehicle they slow and compress increasing pressure on front surface of car
    - As pressure is lower round sides and top/bottom air molecules 'spill' into lower pressure speeding up flow in these areas.



Bluff Body

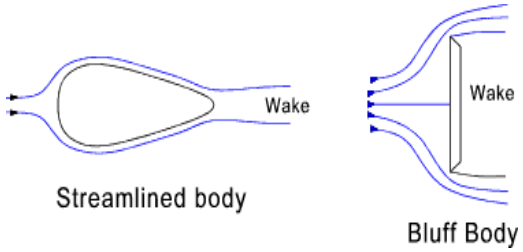


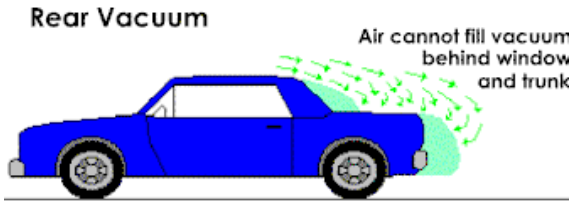
Aerodynamics
Introduction

The role of aerodynamics in vehicles



- Rear Vacuum–
  - As vehicle moves through the air it 'punches a hole in the air' leaving a vacuum behind the vehicle. This is because the air molecules travelling around the vehicle can't fill the vacuum as fast as it is being created. (Flow detachment)
  - The vacuum is an area of low pressure and acts on the rear of the car
    → Rearward force
    → Drag





The role of aerodynamics in vehicles

- Frictional Drag–
  - As air moves over vehicle surface, friction between the air and surface looses energy as heat.
  - This friction effect results in a contribution to drag.
  - Depends on surface finish
  - Important on planes but only a small contribution to overall drag on vehicles.
  - Total drag is made up of pressure and friction components.

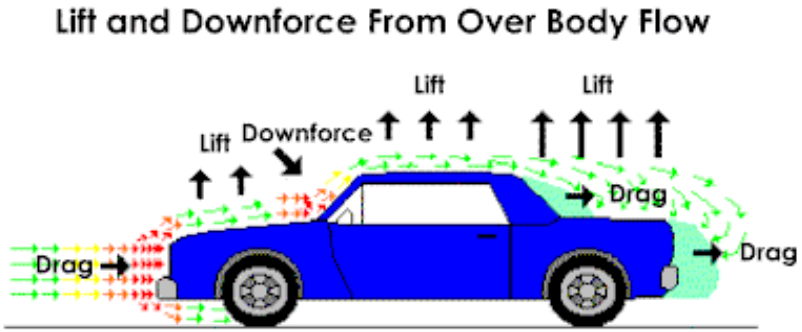
Aerodynamics
Introduction

The role of aerodynamics in vehicles

- All vehicles also experience lift/downforce when they move through the air.

- Bernoulli –

  For a given volume of air, the higher the speed the air molecules are travelling, the lower the pressure.



- Air speeds up over bonnet → Lift
- Air slows down on windscreen → Lift + Frontal Drag as inclined
- Air Speeds back up over roof and trunk→ Lift

Aerodynamics
Introduction

The role of aerodynamics in vehicles

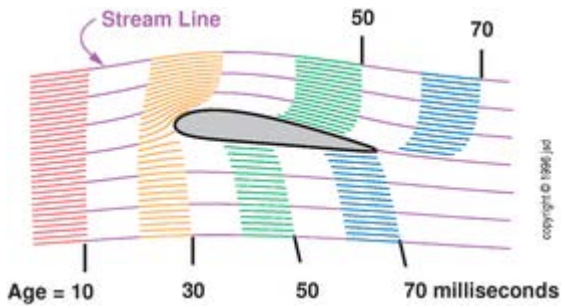


Race cars :

- **Downforce –** (Lift in downwards direction) Increases load on contact patches, increasing available grip, enabling cars to negotiate corners at higher velocities

- Downforce is often described as negative lift.

  - Aeroplane wings produce ‘lift’ which acts against gravity to keep the plane in the sky. Race cars use inverted wings.



Aerodynamics
Introduction

The role of aerodynamics in vehicles

Race cars :

- Drag and Downforce are the main parts of the aerodynamic equation and it should be noted that cars with high Downforce usually have high drag.
- The best aerodynamic solution depends on the track – e.g. Monza V Monaco
- What is aerodynamic efficiency?
  - Aero Efficiency = Downforce / Drag
- Do we want the same from aero on road cars?





Aerodynamics
Introduction

The role of aerodynamics in vehicles



Road cars :

- For road cars the emphasis is usually more on drag than Downforce.
- Lower drag yields higher fuel economy – Smaller engine.
- The trade-off is generally with passenger space in the car and wheelbase.

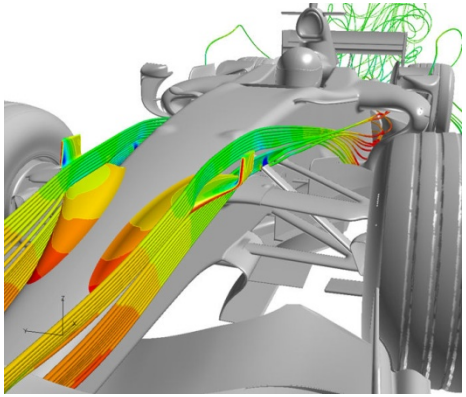
i.e. You could have a very low drag car if you only needed to carry one person and didn't need to park!





Aerodynamics
Introduction

The role of aerodynamics in vehicles



Additional considerations:

In addition to Downforce and Drag an aerodynamicists must consider the following :

- **Engine cooling –**

  The engine needs to be sufficiently cooled while minimising the resulting drag.



- **Engine intake –**

  In race cars, the engine intake should be designed to maximise pressure in the plenum chamber while minimising vehicle drag.

Aerodynamics
Introduction

The role of aerodynamics in vehicles





Additional considerations:

- **Occupant conditions**

  If the occupant is plagued by heat or noise, the race car will not perform and the road car will not sell. In racing this is particularly important in endurance racing.

- **Stability**
- It would be undesirable to have high Downforce in a corner if the Downforce suddenly disappears if the vehicle hits a bump r kerb. (eg ground effect cars, Ayrton Senna's death??, Mercedes sports prototype)



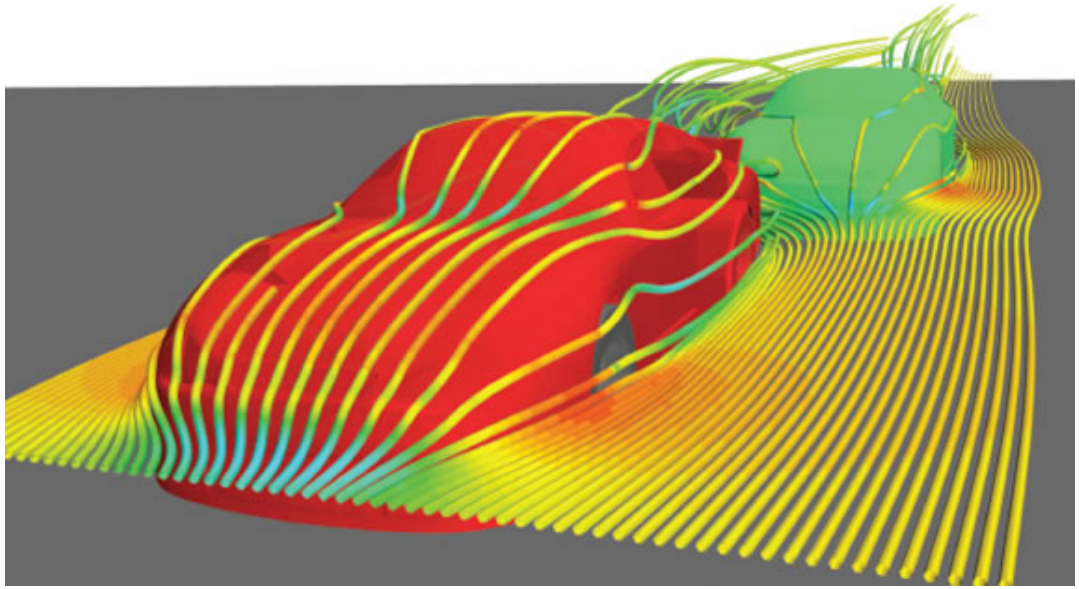
Aerodynamics
Introduction

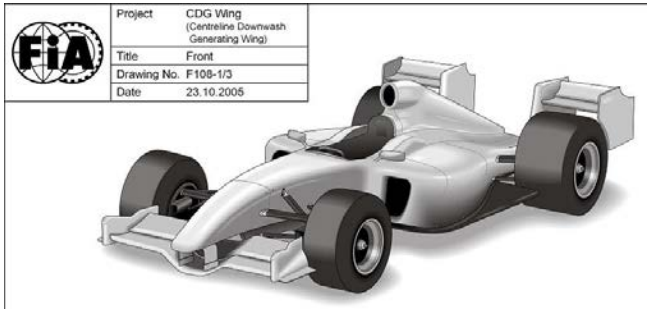
The role of aerodynamics in vehicles

Additional considerations:

- **Drafting and following vehicles**

  It is obvious that during racing vehicles will be in close proximity with each other. Aerodynamics of vehicle may be optimised to reduce the effect of following in 'dirty air'





The role of aerodynamics in vehicles

Additional considerations:

- Stability is obviously an important subject. An aerodynamicist should be striving for;
  - Consistent downforce/drag
  - Consistent balance front/rear
  - Minimising effects of side winds
  - Minimising effects of ‘dirty’ air – creating dirty air?

Aerodynamics
The fundamentals

Fluid Mechanics

- The area of physics with governs the topic of aerodynamics is called Fluid Mechanics
- Before we can look at aerodynamics we need to understand the basics
- We will only be looking at basic fluid mechanics during this course
- Concentrate on providing knowledge applicable to vehicles

Aerodynamics
The fundamentals

Pressure

- Pressure is a scalar quantity (No direction)
- It is a property of a fluid at a particular point.
- However the force exerted on a boundary, by a pressure, does have direction.
- Pressure is measured in terms of this force.

Pressure = Force/Area (N/m^2)

Aerodynamics
The fundamentals

Pressure

- Now there is another way of thinking about pressure

Pressure = Energy/Volume

Pressure = Energy per unit volume

Units show that the above statements hold

Energy/Volume = Jm^{-3}
= $N\ m\ \ m^{-3}$
= $N\ m^{-2}$
= Force/Area
= Pressure

Aerodynamics
The fundamentals

Bernoulli's Equation - FUNDEMENTAL

- Named after the Dutch-Swiss mathematician Daniel Bernoulli
- Published in his book *Hydrodynamica* in 1738!

3 forms of energy in a fluid–

- Kinetic energy – ½ m v^2 → ½ ρ v^2
- Potential Energy – mgh → ρgh
- Pressure energy

(Convert per unit volume changes masses to densities)

Aerodynamics
The fundamentals

Bernoulli's Equation - FUNDEMENTAL

- 1st law of thermodynamics states that energy is constant.

  Therefore, the sum of these three forms of energy must be constant.

  $P_{Static} + \frac{1}{2} rV^2 + \rho gh$ **is constant**

  OR

  $P_{static\ 0} + \frac{1}{2} \rho V_0^2 + \rho g h_0 = P_{Static\ 1} + \frac{1}{2} \rho V_1^2 + \rho g h_1$

Aerodynamics
The fundamentals

Bernoulli's Equation - FUNDEMENTAL

- In normal vehicle aerodynamics there is no significant change of height, therefore potential energy term is negligible.

- For vehicle aerodynamics we can approximate Benoullis equation as :

$$P_{Static} + \tfrac{1}{2} \rho V^2 = \text{Constant}$$

$$P_{Static} + \tfrac{1}{2} \rho V^2 = P_{Total}$$

P_{Static} = The pressure measured by a gauge on an aerodynamic surface

P_{Total} = The pressure measured by a gauge where the air is stationary

$\tfrac{1}{2} \rho V^2$ is also known as **dynamic pressure**

Aerodynamics
The fundamentals

Bernoulli's Equation - FUNDEMENTAL

- Basically states

If you increase the velocity of a fluid you decrease its pressure

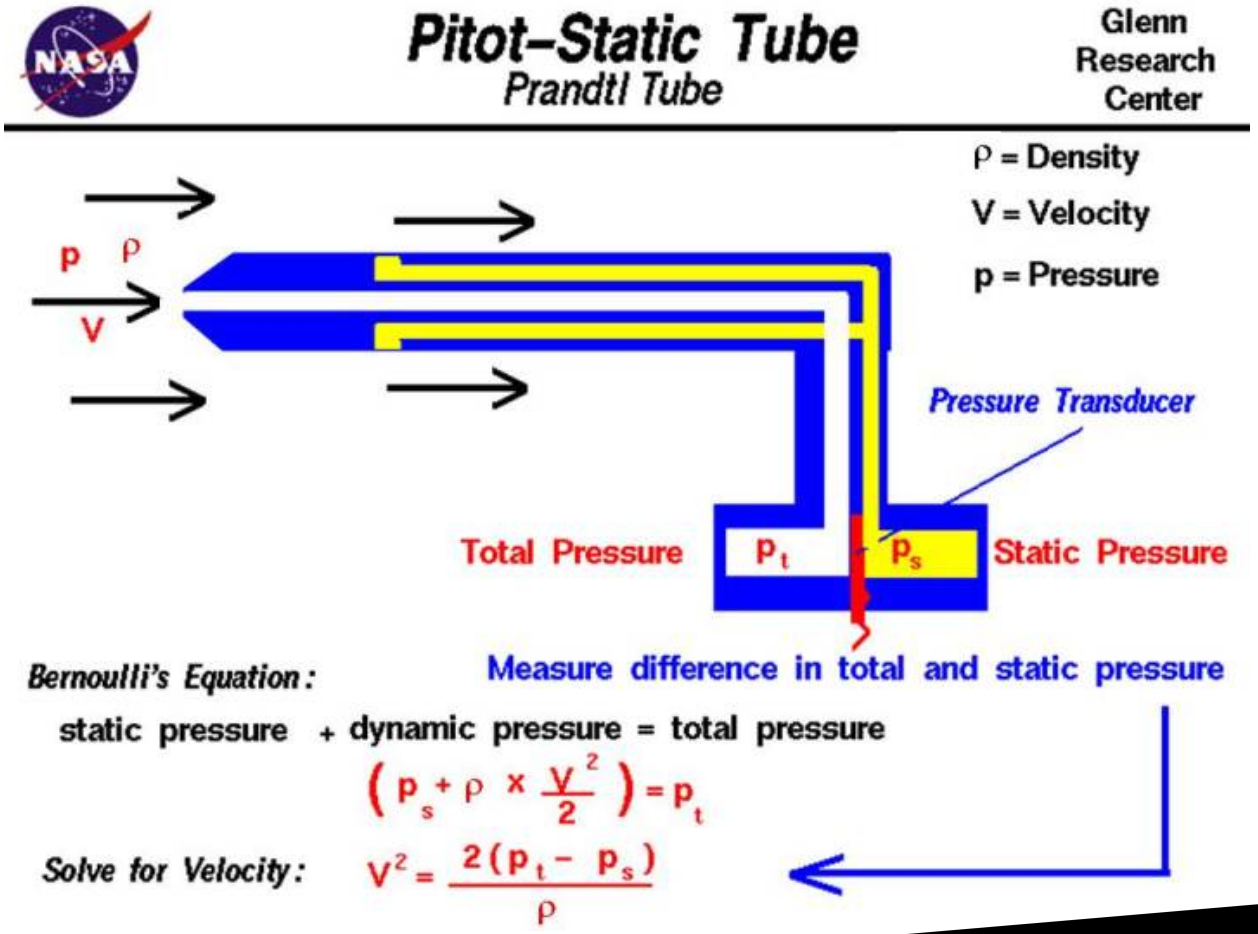
AND

If you decrease the velocity of a fluid you increase its pressure

Aerodynamics

The fundamentals

Bernoulli's Equation - FUNDEMENTAL



Mass Flow Rate

- Mass of fluid flowing past a point per second
- E.g. Tap fills an empty bucket weighing 2kg for 7 seconds

Calculate mass flow rate if after 7 s bucket weighs 8Kg

Mass Flow rate = Fluid Mass / time

= (8-2)/7 = 6/7 = 0.857 Kg/s

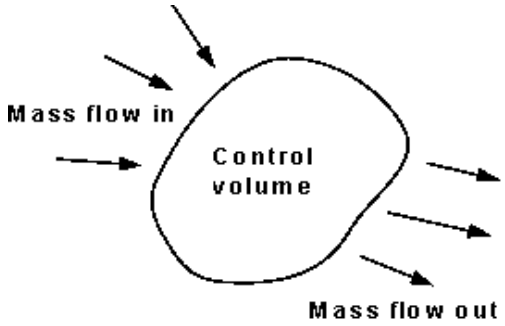
Volumetric Flow Rate

- Volume = mass / density
- Volumetric flow rate = mass flow rate / density

Aerodynamics
The fundamentals

Continuity

- Matter cannot be created or destroyed - *conservation of mass*
- The principle is applied to fixed volumes → **control volumes**



Mass entering per unit time = Mass leaving per unit time + Increase of mass in the control volume per unit time

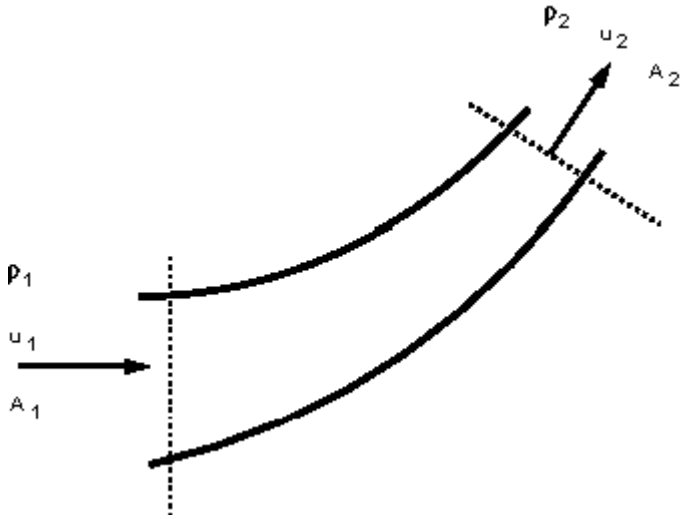
For **steady** flow there is no increase in the mass within the control volume, so

Mass entering per unit time = Mass leaving per unit time

Aerodynamics
The fundamentals

Continuity

- If Fluid flows through shaped pipe on RHS
- No fluid flows across the boundary



$$\rho_1 \delta A_1 u_1 = \rho_2 \delta A_2 u_2 = \text{Constant} = \dot{m}$$

- Mass entering per unit time at 1 = Mass leaving per unit time at 2

Aerodynamics
The fundamentals

Continuity

- The flow of fluid through a real pipe (or any other vessel) will vary due to the presence of a wall
- So we use the *mean* velocity

$$\rho_1 A_1 u_{m1} = \rho_2 A_2 u_{m2} = \text{Constant} = \dot{m}$$

- Assuming fluid is incompressible
  - Density is constant so we can divide through

$$A_1 u_1 = A_2 u_2 = Q$$

Aerodynamics
The fundamentals

Continuity

Example

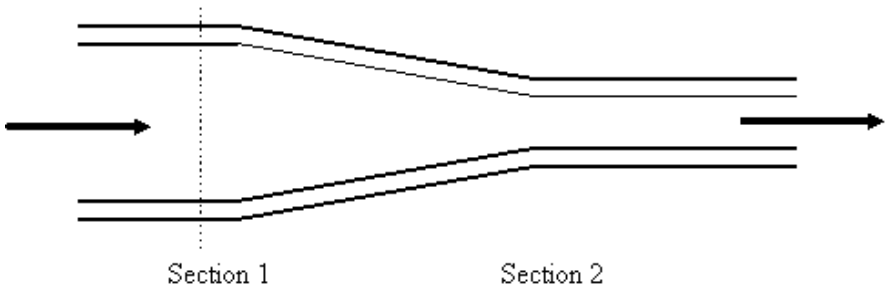
- If at a point 1 in a pipe the area is $A_1 = 10 \times 10^{-3} m^2$ and the pipe converges to an area at 2 of $A_2 = 3 \times 10^{-3} m^2$ calculate the velocity at 2 given the mean velocity at 1 is

$$u_1 = 2.1 m/s$$

$$A_1 u_1 = A_2 u_2 = Q$$

$$u_2 = \frac{A_1 u_1}{A_2}$$

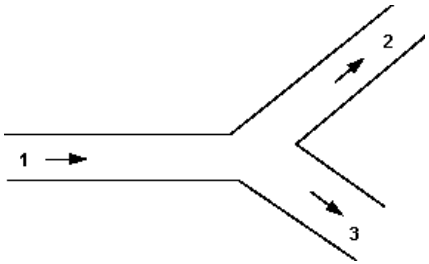
$$= 7.0 m/s$$



Aerodynamics
The fundamentals

Continuity

- If pipe 1 diameter = 50mm, mean velocity 2m/s, pipe 2 diameter 40mm takes 30% of total discharge and pipe 3 diameter 60mm. What are the values of discharge and mean velocity in each pipe?



$$Q_1 = A_1 u_1 = \left(\frac{\pi d^2}{4}\right) u$$

$$= 0.00392 m^3 / s$$

$$Q_2 = 0.3 Q_1 = 0.001178 m^3 / s$$

$$Q_1 = Q_2 + Q_3$$

$$Q_3 = Q_1 - 0.3 Q_1 = 0.7 Q_1$$

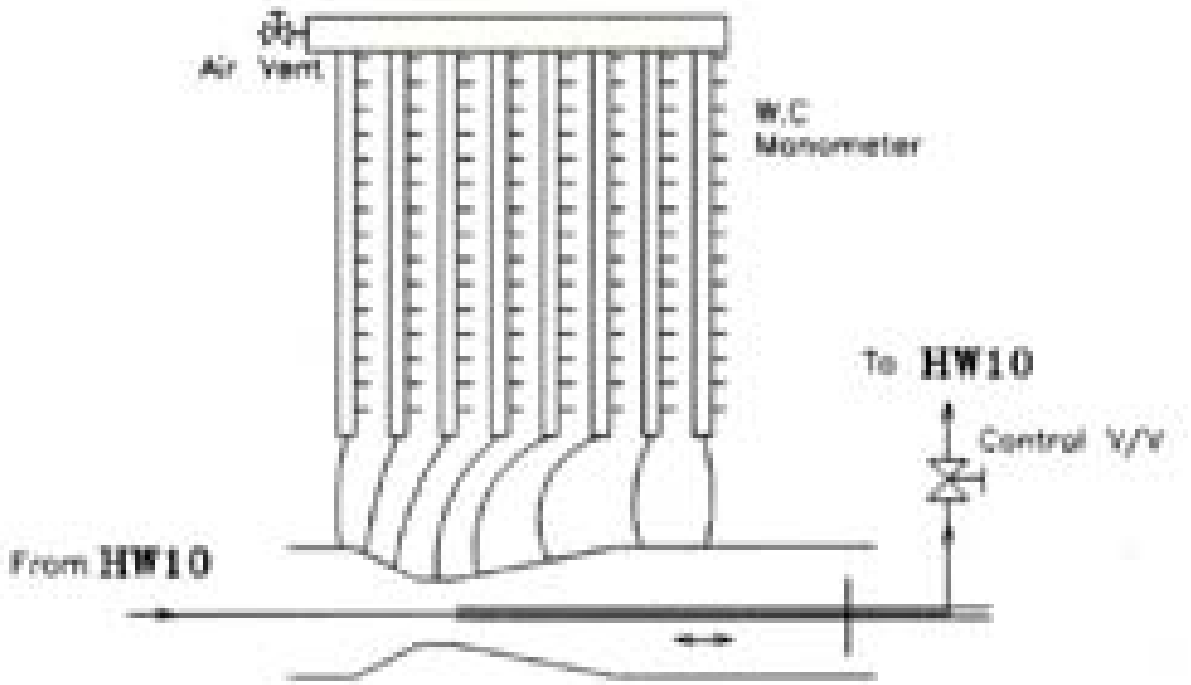
$$= 0.00275 m^3 / s$$

$$Q_2 = A_2 u_2 \qquad Q_3 = A_3 u_3$$

$$u_2 = 0.936 m / s \qquad u_3 = 0.972 m / s$$

Aerodynamics
The fundamentals

Continuity & Bernoulli's



Aerodynamics
The fundamentals

Continuity & Bernoulli's

– Pressure not totally recovered

Work is done on the fluid – Heat loss

$Q = mc\Delta T$
$Q = mc(T_o - T_i)$

m would actually be mass flow rate....

E.g. Pressure at outlet is x mm lower than inlet
$Q = mgh_i - mgh_O = mgh_i - mg(h_i - x)$

Equate 2 equations to give temp rise in fluid

Example Question

- A Pitot tube is connected to a water manometer and a height difference of 12 cm is recorded. What is the wind speed if the measurement is taken at standard temperature and pressure.

Density of air = 1.223 Kg/m^3

Density of water = 1000 Kg/m^3

$$Dynamic\ Pressure = \frac{1}{2}\rho v^2 = 12cm\ of\ \ water$$

$$\frac{1}{2}\rho_{air} v^2 = \rho_{water} gh$$

$$v = \sqrt{\frac{2gh\rho_{water}}{\rho_{air}}} = \sqrt{\frac{2\times 9.81\times 0.12\times 1000}{1.223}}$$

$$v = 43.87\ m/s$$

Example Question 2

- A 600cc formula student engine revs at 8000 rpm – Assuming no losses, what is the velocity of air in the 20mm round restrictor and hence , if the engine is operating in a pressure 1.02 E5 N/m^2 what would the absolute pressure in the restrictor be?

Density of air = 1.223 Kg/m^3

Volume of air drawn in per second :

$$V = \frac{RPM}{60} \times \frac{Capacity_{Total}}{2} = \frac{8000 \times 600}{60 \times 2}$$

$$V = 40{,}000 cc/s$$

$$V = 40{,}000 \times 1 \times 10^{-6} = 0.04 m^3/s$$

Area of restrictor :

$$A = \pi r^2 = \pi \times \left(\frac{20 \times 10^{-3}}{2} \right)^2 = 3.14 \times 10^{-4} m^2$$

Velocity of air:

$$V_{air} = v_{air} \times A$$

$$v_{air} = \frac{V_{air}}{A} = \frac{0.04}{3.14 \times 10^{-4}} = 127 m/s$$

Example Question 2 Continued

- A 600cc formula student engine revs at 8000 rpm – Assuming no losses, what is the velocity of air in the 20mm round restrictor and hence , if the engine is operating in a pressure 1.02 E5 N/m^2 what would the absolute pressure in the restrictor be?

Velocity of air : 127 m/s

Dynamic pressure :

$$dynamic\ \ pressure = \frac{1}{2}\rho v^2$$

$$= \frac{1}{2} \times 1.223 \times 127^2$$

Static Pressure :

$$= 9862.9\ Pa$$

$$P_{total} = P_s + P_d$$

$$P_s = P_{total} - P_d$$

$$P_s = 1.05 \times 10^5 - 9862.9Pa \approx 92100Pa$$

Aerodynamics
The fundamentals

Viscosity

- Viscosity, μ, is the property of a fluid which offers resistance to sheer deformation.
- Due to cohesion and interaction between molecules
- Different fluids deform at different rates under the same shear stress.
- Fluid with a high viscosity such as syrup, deforms more slowly than fluid with a low viscosity such as water.
- How sticky the fluid is!

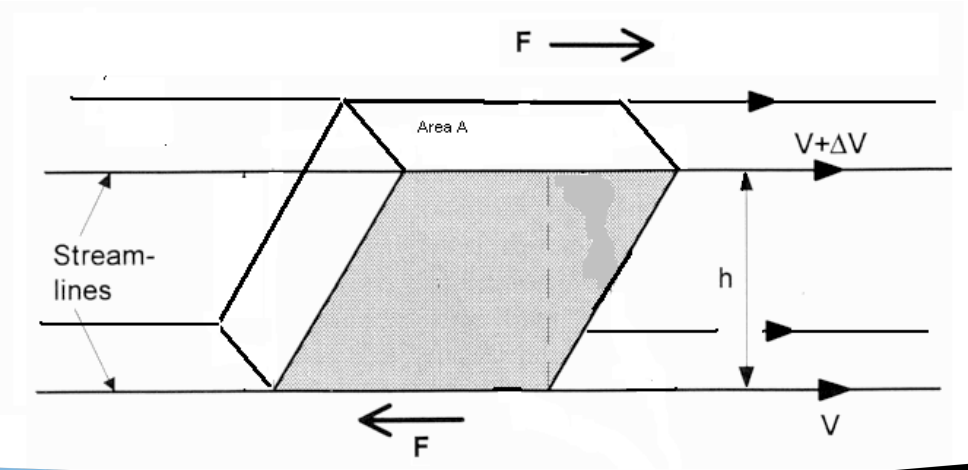
Aerodynamics
The fundamentals

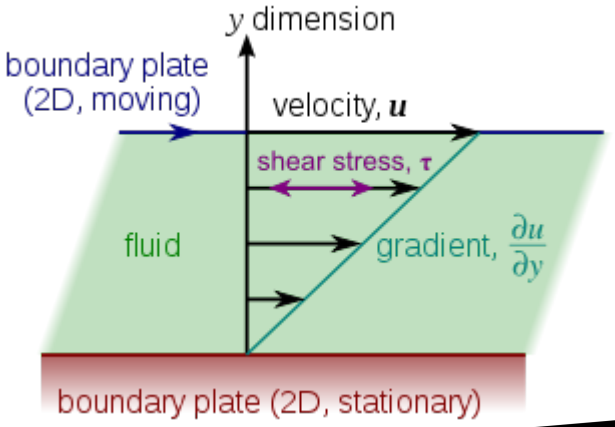
Dynamic Viscosity

- Dynamic Viscosity, is defined as the shear force, per unit area, required to drag one layer of fluid with unit velocity past another layer a unit distance away.

$$\mu = \frac{(F/A)}{(\Delta V/h)} = \frac{\text{Shear Stress}}{\text{Rate of shear strain}}$$

$$\tau = \mu \frac{du}{dy}$$





Dynamic Viscosity

- Units: Newton seconds per square metre, or Kilograms per meter per second.

  (Although often expressed in Poise, P, where 10 P = 1 $Kgm^{-1}s^{-1}$.)

- Typical values:

  Water =1.14 $Kgm^{-1}s^{-1}$
  Air =1.78 $Kgm^{-1}s^{-1}$
  Mercury =1.552 $Kgm^{-1}s^{-1}$
  Paraffin Oil =1.9 $Kgm^{-1}s^{-1}$

Aerodynamics
The fundamentals

Viscosity in different types of fluid

- Newton's law of viscosity,

$$\tau = \mu \frac{du}{dy}$$

is a constitutive equation.

- It is not a fundamental law of nature but an approximation that holds in some materials and fails in others.

Aerodynamics
The fundamentals

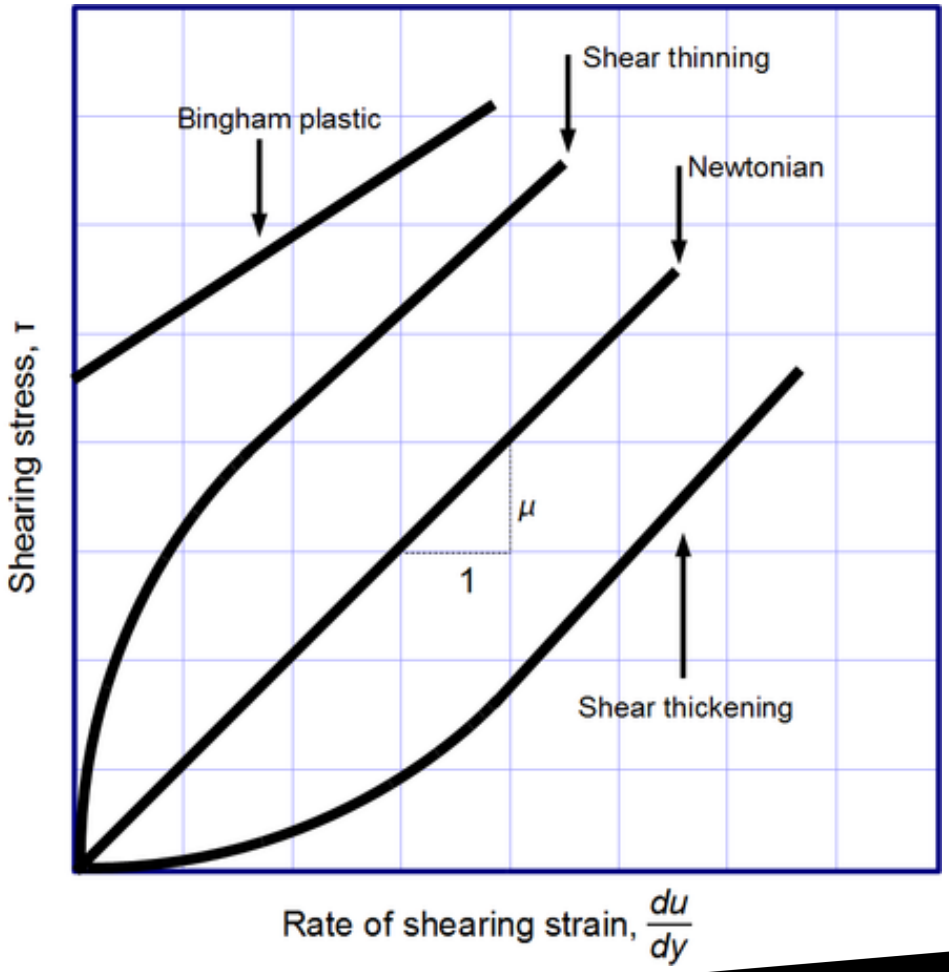
Viscosity in different types of fluid

- **Newtonian fluids** : such as water and most gases which have a constant viscosity.

- **Non-Newtonian Fluids :**
  - **Shear thickening**: viscosity *increases* with the rate of shear.
  - **Shear thinning**: viscosity *decreases* with the rate of shear.
  - **Thixotropic**: materials which become *less* viscous over time when shaken, agitated, or otherwise stressed.
  - **Rheopectic**: materials which become *more* viscous over time when shaken, agitated, or otherwise stressed.
  - A **Bingham plastic** is a material that behaves as a solid at low stresses but flows as a viscous fluid at high stresses.
  - A **magnetorheological fluid** viscosity changes with magnetic field

Aerodynamics
The fundamentals

Viscosity in different types of fluid



Aerodynamics
The fundamentals

Examples

A hydraulic piston of length 20mm and diameter 15mm has a clearance of 0.01 mm with the cylinder.

Assuming the lubrication is totally viscous and the oil viscosity is 1.6 E-2 Ns/m2 calculate the force required to push the piston at 0.1 m/s

$$\mu = \frac{F/A}{\Delta V/h} = \frac{Fh}{A\Delta V} \qquad Note: \quad h = \frac{clearance}{2} = \frac{0.01\times10^{-3}}{2} = 0.005\times10^{-3}m$$

$$F = \frac{\mu A\Delta V}{h} = \frac{1.6\times10^{-2}\times0.1\times\pi\times15\times10^{-3}\times20\times10^{-3}}{0.005\times10^{-3}}$$

$$= 3.02\times10^{-1}N$$

Aerodynamics
The fundamentals

Examples

If the piston had a mass of 0.1 Kg, what would it's velocity be if it fell freely through the cylinder.

$$F = mg = 9.81 \times 0.1 = 0.981N$$

$$F = \frac{\mu A \Delta V}{h} \qquad \Delta V = \frac{Fh}{\mu A}$$

$$= \frac{0.981 \times 0.005 \times 10^{-3}}{1.6 \times 10^{-2} \times \pi \times 15 \times 10^{-3} \times 20 \times 10^{-3}}$$

$$= 3.25 \times 10^{-1} m/s$$

Aerodynamics
The fundamentals

Kinematic Viscosity

This is another fluid property which becomes important in aerodynamics.

It is simply

$$\nu = \frac{\mu}{\rho}$$

Where

μ is the viscosity of the fluid

ρ is the density of the fluid

Water =1.14 , Air =1.46 , Mercury =1.145, Paraffin Oil =2.375 .

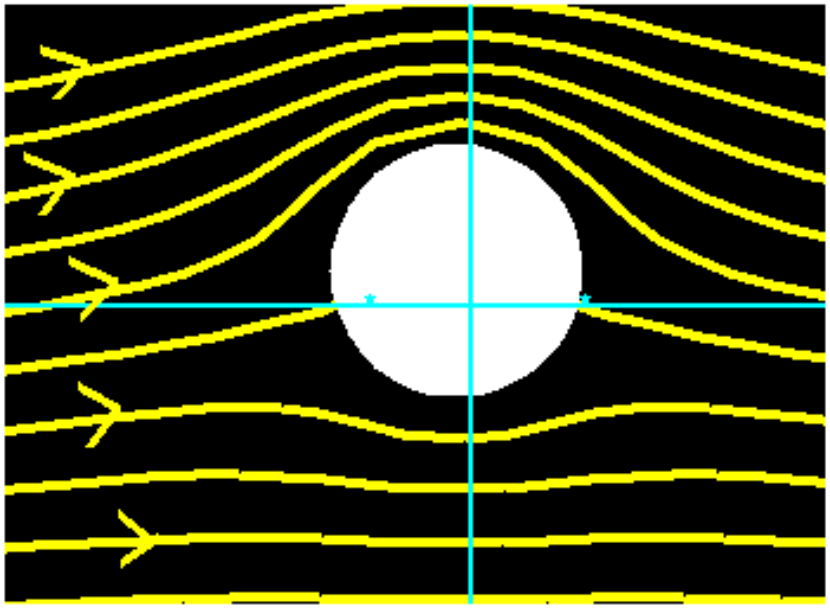
Aerodynamics
The fundamentals

Streamlines

Streamline is a path traced out by a massless particle moving with the flow.

Velocity is tangent to streamline at every point.

Mass does not cross streamlines.

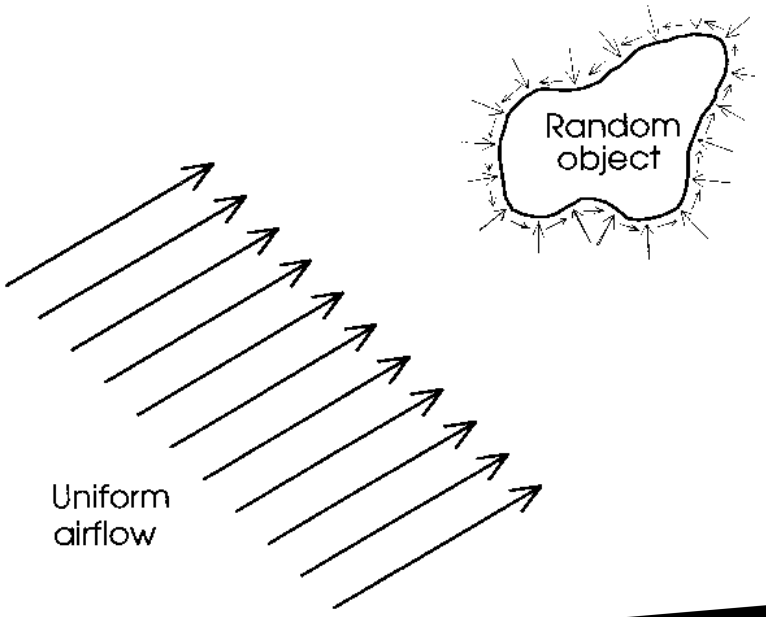


Aerodynamics
The fundamentals

Aerodynamic Forces

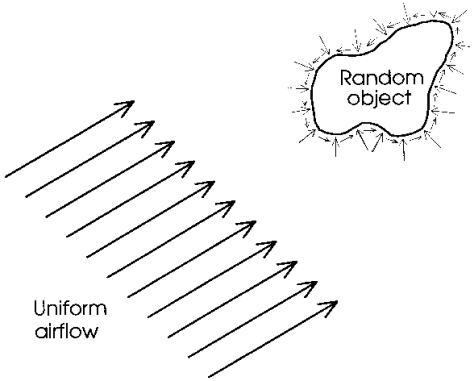
We have already looked briefly at aerodynamic force in terms of its effect on racing cars but the question is this :

“How exactly is aerodynamic force imposed on an object?”



Aerodynamics
The fundamentals

Aerodynamic Forces



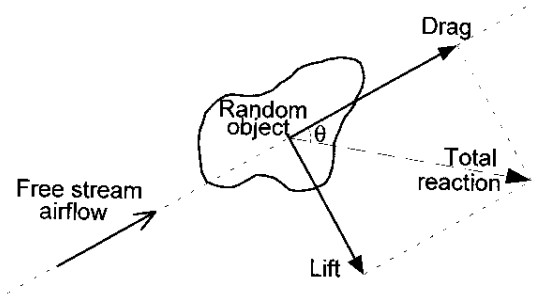
Consider the object and flow outlined in the diagram above. Air flows around the object in a complicated way leading to variations of pressure and velocity over the surface.

For each tiny element of surface there is a pressure and a rate of shear which give rise to a perpendicular and tangential force respectively.

If the pressure and viscous forces are added for all the elements of the surface, we are left with a single aerodynamic force (and moment) which act on the object.

Aerodynamics
The fundamentals

Aerodynamic Forces



This total aerodynamic force is then split into two components Drag and lift.

Technically, drag is the component of the overall aerodynamic force **parallel** to the flow. Lift is **perpendicular** to the flow.

Note : viscous forces or 'skin friction' play a very small part in generating the aerodynamic force compared to pressure forces.

Aerodynamics
The fundamentals

Cp – Coefficient of pressure

Up to a point the flow field around a body is similar if the body has the same shape, irrespective of the size of the body and the speed of flow.

It is therefore useful to analyse shapes in flow using non –dimensional coefficients. One such coefficient is the coefficient of pressure which acts on a point of an aerodynamic surface.

$$C_p = \frac{P - P_{static\text{-}pitot}}{P_{dynamic}}$$

Where P = static pressure in the flow near the body

$P_{dynamic}$ = dynamic pressure in the free stream flow

$P_{static\text{-}pitot}$ = Pressure in the free stream flow

Aerodynamics

Cp – Coefficient of pressure

$$\frac{P_S - P_{s\infty}}{\frac{1}{2}\rho V_\infty^2} = 1 - \frac{\frac{1}{2}\rho V^2}{\frac{1}{2}\rho V_\infty^2}$$

$$Cp = 1 - \frac{V^2}{V_\infty^2}$$

Aerodynamics
The fundamentals

Cp – Coefficient of pressure

At stagnation :

$$Cp = 1 - \frac{V^2}{V_\infty^2}$$

$$Cp = 1 - \frac{0^2}{V_\infty^2}$$

$$Cp = 1 - 0$$

$$Cp = 1$$

Aerodynamics
The fundamentals

Cp – Coefficient of pressure

Cps for various flow velocities

Local Velocity V	Pressure Coefficient Cp
Stagnation point, V = 0	1
Free stream, V = V_∞	0
Low local velocity V < V_∞	Between 0 and 1
High local velocity V > V_∞	Less than 0, i.e negative

Cp – Coefficient of pressure – Example

Let's consider a race car travelling at 45m/s (roughly 100 mph).

If the air under the vehicle is forced to accelerate to double the vehicle velocity (through clever underfloor shaping!) what is the Cp?

Air accelerates to 90 m/s and freestream is 45 m/s

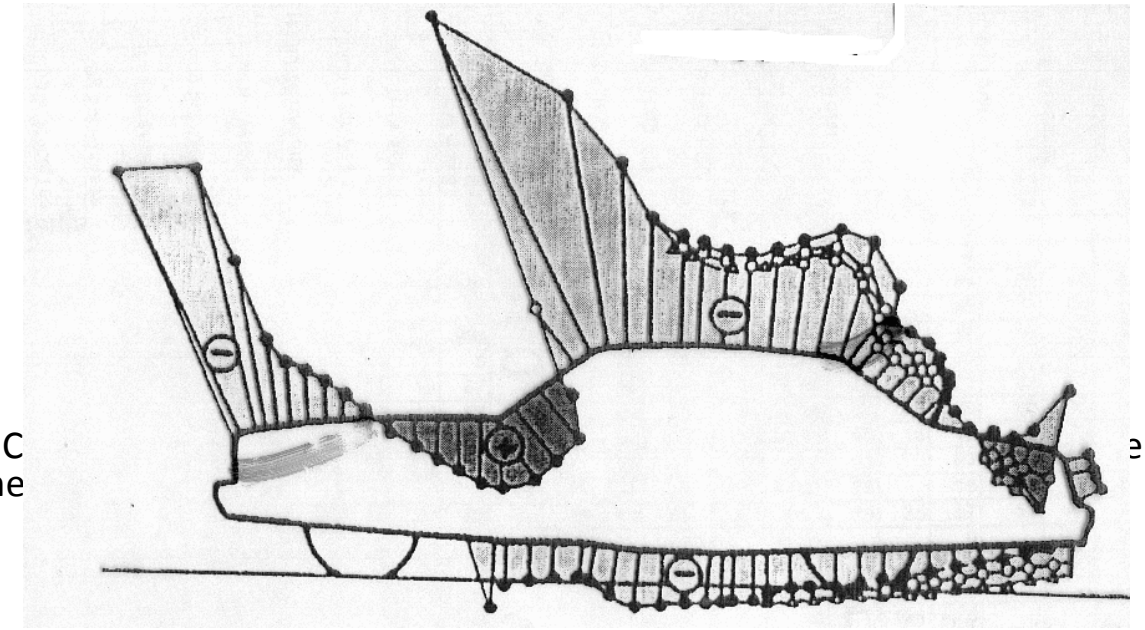
$$Cp = 1 - \frac{V^2}{V_\infty^2}$$

$$Cp = 1 - \frac{90^2}{45^2} = 1 - 4 = -3 \text{ !!}$$

Aerodynamics
The fundamentals

Cp – Coefficient of pressure

When analysing flow it is useful to draw a map of these coefficients of pressure around a body:

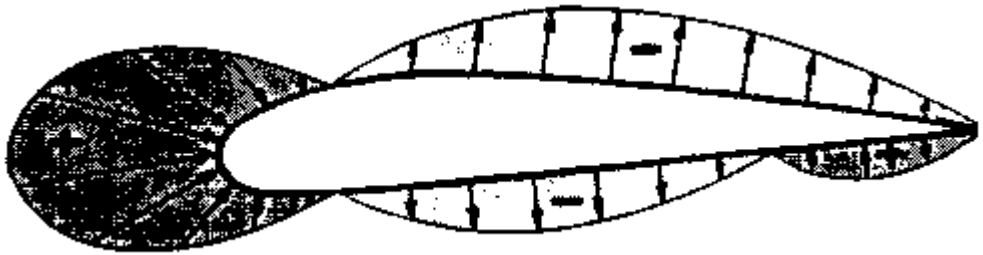


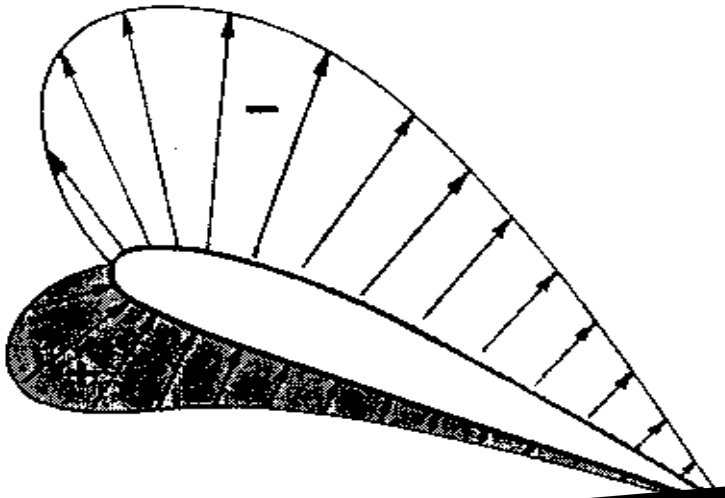
Because C e speed where the

Aerodynamics

The fundamentals

Cp – Coefficient of pressure

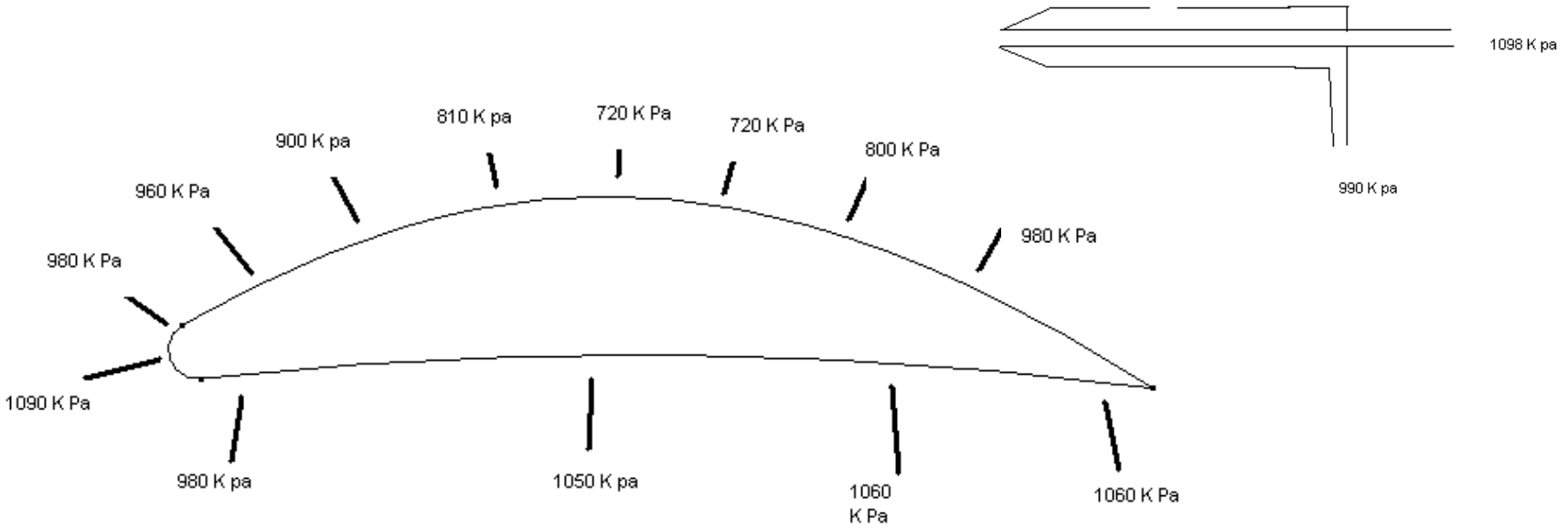




Aerodynamics
The fundamentals

Cp – Example

The diagram below shows a wing profile with the absolute pressure measurements taken at tapings on the wing.

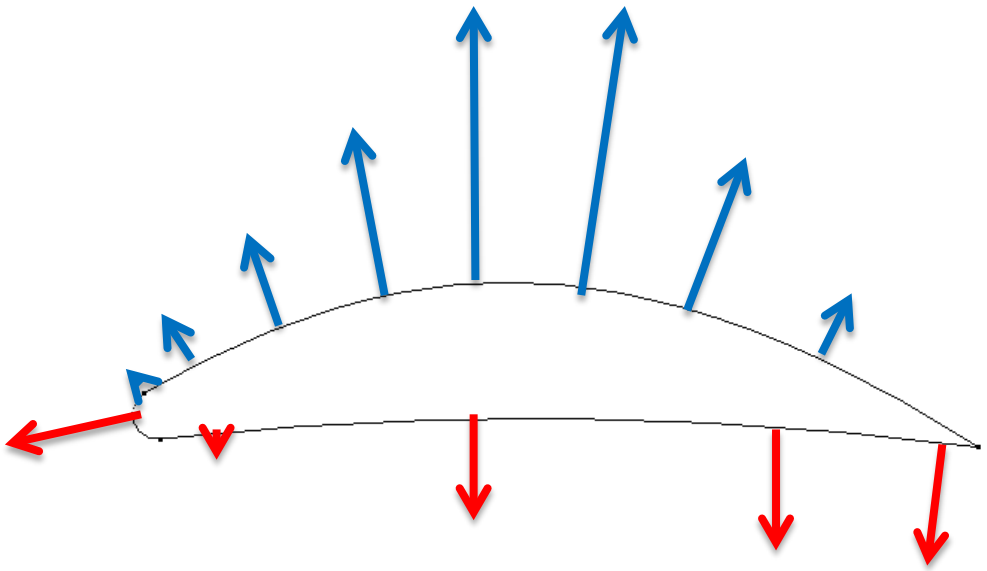


Aerodynamics
The fundamentals

Cp – Example

P_infi S	990	Pdy_infi	108
P_Total	1098		

	P_Static	Cp
1	1090	0.925926
2	980	-0.09259
3	960	-0.27778
4	900	-0.83333
5	810	-1.66667
6	720	-2.5
7	720	-2.5
8	800	-1.75926
9	980	-0.09259
10	1060	0.648148
11	1060	0.648148
12	1050	0.555556
13	980	-0.09259



Aerodynamics
The fundamentals

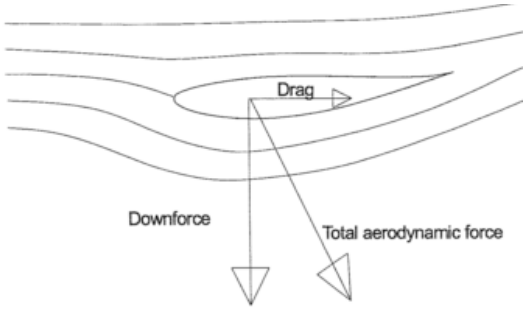
The forces involved

As we have seen, the total aerodynamic effect of a body interacting with a fluid flow is an aerodynamic force and moment.

The force can be thought of in terms of its lift and drag component.

What influences this overall force ?

- Velocity of the stream
- Density of the fluid
- Size of the object
- Exact shape of the object (including surface roughne



Aerodynamics
The fundamentals

The forces involved

The local pressure differences felt by a body such as a wing can be expressed mathematically.

$$Cp = \frac{\Delta P}{q} \qquad q = 1/2\,\rho V^2$$

$$\Delta P = Cp \times q = Cp \times 1/2\,\rho V^2$$

Aerodynamics
The fundamentals

The forces involved

To calculate the total force it is necessary to calculate the sum of all the local pressures acting on the body.

To calculate the resulting force me must then multiply by the area this pressure is acting on, as Pressure = Force / Area

$$F = Cp \times 1/2\, \rho V^2 \times A$$

Where A is the reference area upon which the sum of the pressure changes is deemed to act

The forces involved

For calculating the drag and lift of a body we use **appropriate coefficient**s which account for how the overall shape of that body affects the flow , and **distributions of velocity and pressures** over it.

The **magnitude** of these coefficients **depends** on the **shape and surface finish** of the object and does not depend on the density of air, the velocity of the flow or the size of the object.

The previous equation can be written in the forms

$$Drag\ Force = C_D \times 1/2\, \rho V^2 \times A$$

$$Lift\ Force = C_L \times 1/2\, \rho V^2 \times A$$

Aerodynamics
The fundamentals

The forces involved

And it follows we can do the same to find an equation for the aerodynamic moment

$$Moment = C_M \times 1/2\,\rho V^2 \times A$$

So we have

$$C_D = \frac{Drag\ Force}{1/2\,\rho V^2 \times A} \qquad C_L = \frac{Lift\ Force}{1/2\,\rho V^2 \times A} \qquad C_M = \frac{Moment}{1/2\,\rho V^2 \times A}$$

These equations define C_L, C_D , C_M - the coefficients of lift, drag and moment respectively

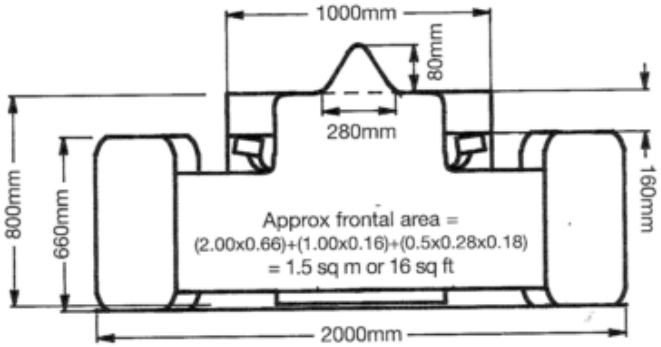
Aerodynamics
The fundamentals

The forces involved

The trick with these equations is selecting the correct area A

To calculate the drag force for a whole vehicle the area is taken to be the frontal area of the vehicle

(as drag is a result of pressures acting on surfaces which are normal to the direction of the vehicle)



$$Drag\ Force = C_D \times 1/2\,\rho V^2 \times A$$

Aerodynamics
The fundamentals

Example – Lift from a given wing

- A racing car rear wing has a plan width of 0.8m and a length of .9m If C_L is -3.1, calculate the downforce available at 220 km/h assuming normal atmospheric conditions.

- Density of air = 1.223 Kg/m^3

$$Area = 0.8 \times 0.9 = 0.72 m^2$$

$$Lift\ Force = C_L \times 1/2\, \rho V^2 \times A$$

$$Lift\ Force = -3.1 \times 1/2 \times 1.223 \times \left(\frac{220 \times 10^3}{60 \times 60}\right)^2 \times 0.72$$

$$Lift\ Force = -5097.19N$$

$$Downforce = 5097.19N$$

Aerodynamics
The fundamentals

Example – C_L for a given wing

- A racing car rear wing has a plan width of 0.15m and a length of 1m If the downforce available at 180 km/h is 500N calculate the C_L for the wing assuming normal atmospheric conditions.

- Density of air = 1.223 Kg/m^3

$$Area = 0.15 \times 1 \; = 0.15 m^2$$

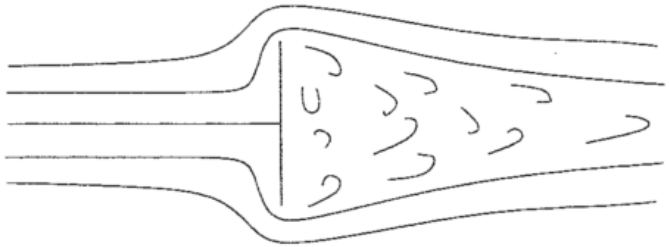
$$V = \frac{180 \times 10^3}{60 \times 60} \quad = 50 m/s$$

$$C_L = \frac{Lift\ Force}{1/2\,\rho V^2 \times A} = \frac{-500}{0.5 \times 1.223 \times 50^2 \times 0.15} \quad = -2.18$$

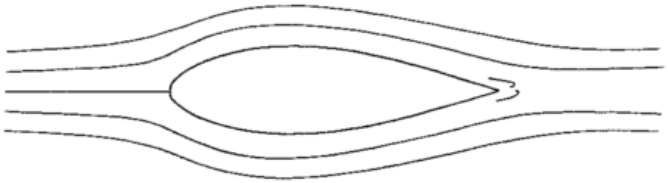
Aerodynamics
The fundamentals

Drag coefficients

- A flat plate will have a drag coefficient probably exceeding 1.1. Where as a tear drop shape will have a Cd of around 0.04



Flat plate, $C_D > 1.1$



Tear drop shape, $C_D \sim 0.04$

- Land speed record car ~ 0.1
- Passenger car ~ 0.3→0.4
- Competition car ~ 0.4(Road racer)→1.4 (Hillclimb car)!

Aerodynamics
The fundamentals

Drag Example

- An F1 car has an approximate frontal area of around 1.5m^2
- If it had a C_D of 0.75 and travelled at 200mph (89.4m/s) calculate the drag force
- Density of air = 1.223Kg/m^3

$$Drag\ Force = C_D \times 1/2\ \rho V^2 \times A$$

$$Drag\ Force = 0.75 \times 1/2 \times 1.223 \times (89.4)^2 \times 1.5$$

$$Drag\ Force = 5498N$$

Aerodynamics
The fundamentals

Drag and available power

- Aside – There is an equation that can be used to predict the theoretical maximum top speed of a vehicle based upon the drag equation and available power at the wheels

$$bhp = \frac{C_D \times V^3 \times A}{1225}$$

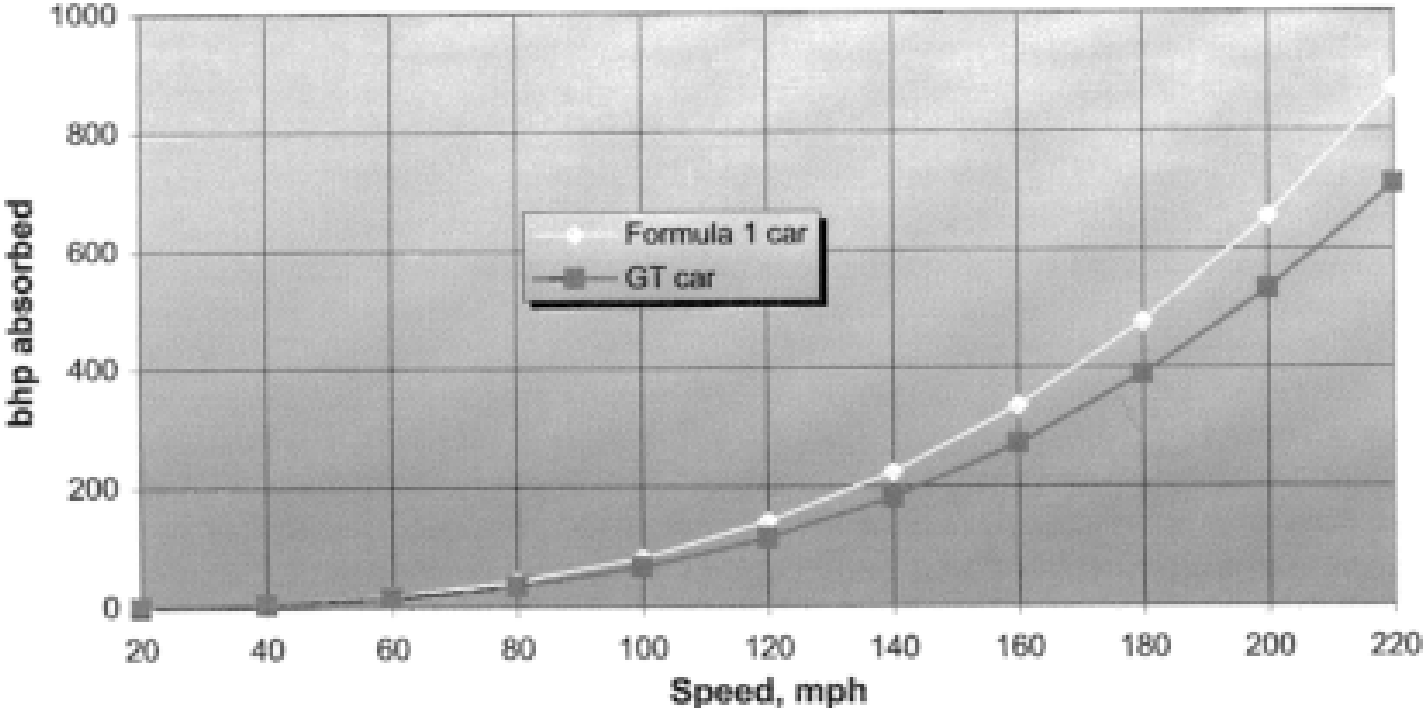
- Calculate the theoretical maximum speed for the two cars below

Car	C_D	A	Bhp	Max speed
F1	0.75	1.486	612.5	196.03 mph
Mclaren GTR	0.5	1.812	510	197.6 mph

Aerodynamics

The fundamentals

Absorbed power



Aerodynamics
The fundamentals

Analysis of the effect of downforce on cornering speed

- If a racing driver can get through a corner faster, the additional velocity will result in gains right through the subsequent straight.
- The velocity of a vehicle as it exits a corner is therefore absolutely vital to racing success.
- In this analysis we will look at the maximum velocity in a turn of radius R

Aerodynamics
The fundamentals

Analysis of the effect of downforce on cornering speed

Nomenclature :

R	=	Radius of turn
m	=	Mass of car
g	=	Acceleration due to gravity
F	=	Maximum lateral friction
μ	=	Coefficient of friction between tyre and road
C_L	=	Coefficient of lift (positive is upwards)
A	=	Plan area of vehicle
ρ	=	Density of air
V	=	Velocity of vehicle

Aerodynamics
The fundamentals

Analysis of the effect of downforce on cornering speed

Assumptions:

All of these are not true in the real world but serve to simplify the analysis.

R is constant – In the real world the driver would take a racing line
m is constant
The car is perfectly balance front/rear
The speed of the car in the corner is less than the maximum speed of the car

Aerodynamics
The fundamentals

Analysis of the effect of downforce on cornering speed

Max cornering speed **without** downforce

$$F = \mu mg \qquad Centripetal\ \ force = \frac{mV^2}{R}$$

At limit

$$Centripetal\ \ force = F$$

$$\frac{mV^2}{R} = \mu mg$$

$$V^2 = \frac{\mu mgR}{m} = \mu gR$$

Therefore

$$V = \sqrt{\mu gR}$$

Aerodynamics
The fundamentals

Analysis of the effect of downforce on cornering speed

Max cornering speed **with** downforce

$$Lift\ Force = C_L \times 1/2\,\rho V^2 \times A$$

$$\frac{mV^2}{R} = \mu(mg - C_L A \times \frac{1}{2}\rho V^2)$$

$$mV^2 = R\mu mg - R\mu C_L A \times \frac{1}{2}\rho V^2$$

$$mV^2 + R\mu C_L A \times \frac{1}{2}\rho V^2 = R\mu mg$$

$$V^2(m + R\mu C_L A \times \frac{1}{2}\rho) = R\mu mg$$

$$V^2 = \frac{R\mu mg}{(m + R\mu C_L A \times \frac{1}{2}\rho)} \qquad V = \sqrt{\frac{R\mu mg}{(m + R\mu C_L A \times \frac{1}{2}\rho)}}$$

Aerodynamics
The fundamentals

Analysis of the effect of downforce on cornering speed –

A car has the following characteristics :
Mass = 500Kg
C_LA = -0.8
mu = 1.5
Turning Radius = 35m
Air has a density of 1.223 Kg/m^3

What is its maximum velocity , as limited by aerodynamics through the corner if the car is perfectly balanced?

What would the corresponding velocity be with no aero package.

For the same car what are the equivalent velocities for a radius of 90m

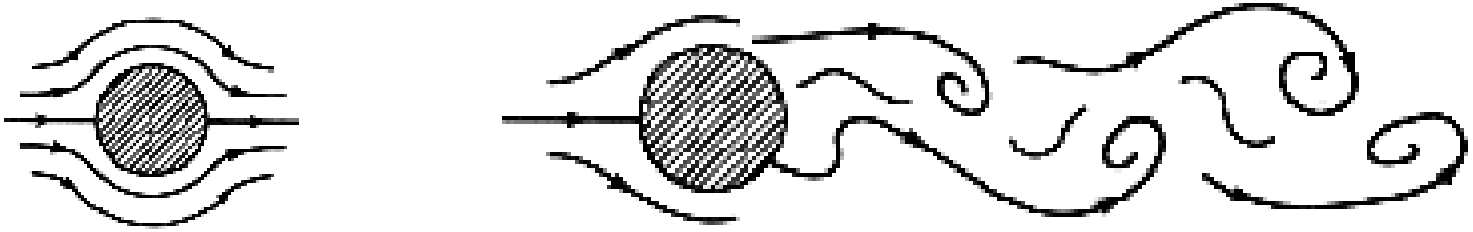
Aerodynamics

Turbulence + Separation

Is our flow field (streamlines) similar?

Last week I stated that you can assume Cp's are the same for any size of shape or speed of flow.

However, in practice flow fields can change as size of object, flow velocity, density and viscosity change

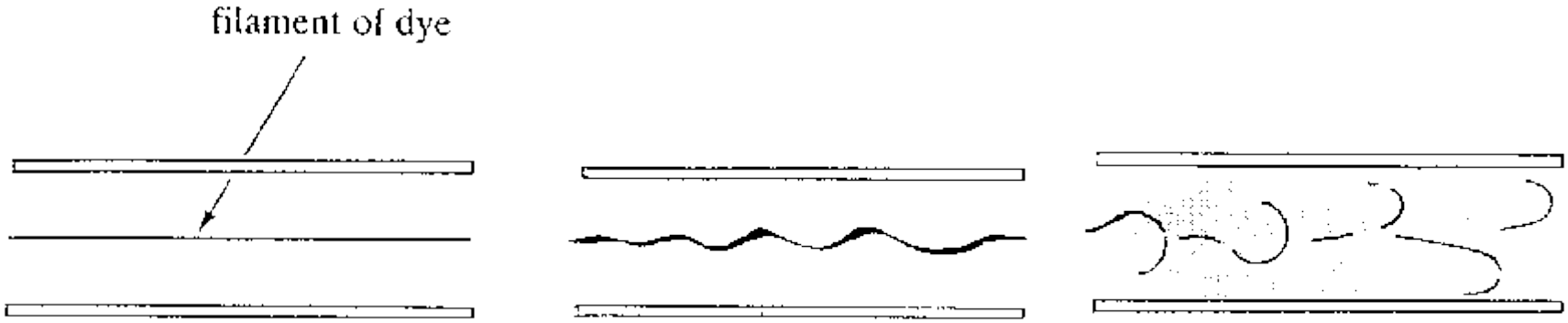


Aerodynamics

Turbulence + Separation

Laminar and turbulent flow

If we were to take a pipe of free flowing water and inject a dye into the middle of the stream, what would we expect to happen?



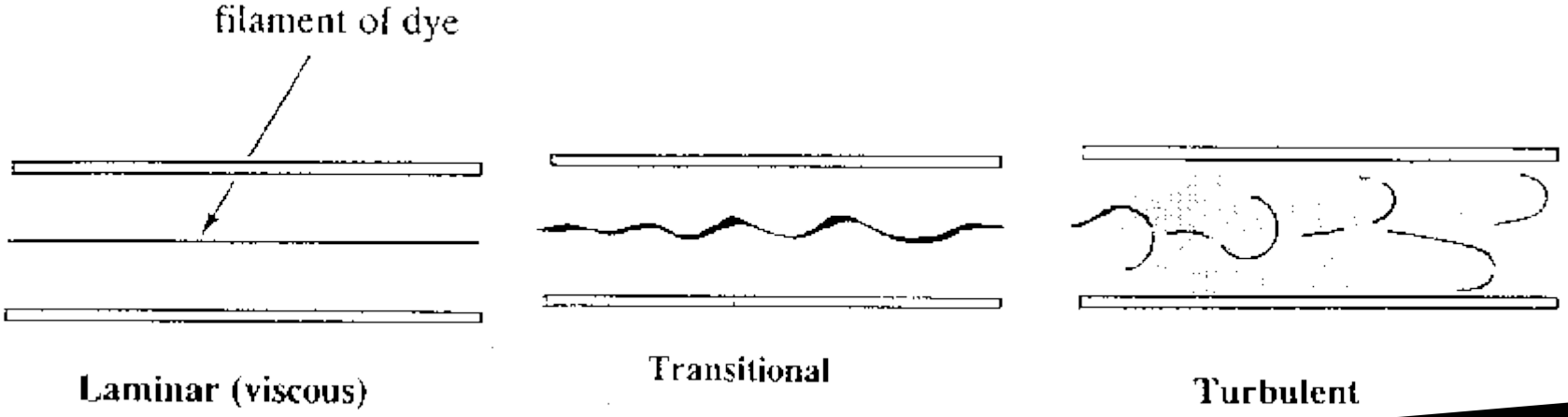
Aerodynamics

Turbulence + Separation

Laminar and turbulent flow

In reality any three of the possibilities could occur, it depends on the flow velocity!

Slow velocity velocity ***Fast***

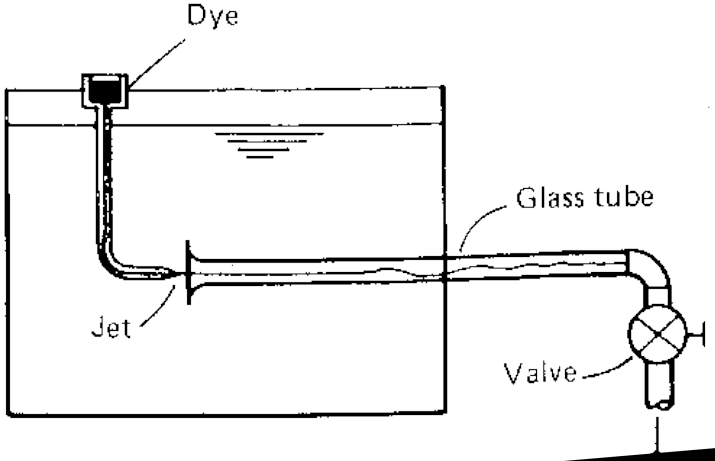


Aerodynamics

Turbulence + Separation

Laminar and turbulent flow

- But what is fast or slow? And at what speed does the flow pattern change?
- The phenomenon was first investigated in the 1880s by Osbourne Reynolds in an experiment which has become a classic in fluid mechanics.



Aerodynamics
Turbulence + Separation

Laminar and turbulent flow

- After many experiments he observed that the following expression

$$\frac{\rho u d}{\mu}$$

where ρ = density, u = mean velocity, d = diameter and μ = viscosity

could help predict the change in flow type.

Aerodynamics
Turbulence + Separation

Laminar and turbulent flow – Reynolds's Number

- This value is known as the Reynolds number, Re:

$$\mathrm{Re} = \frac{\rho u d}{\mu}$$

Laminar flow: Re < 2000
Transitional flow: 2000 < Re < 4000
Turbulent flow: Re > 4000

Re is dimensionless

Aerodynamics
Turbulence + Separation

Laminar and turbulent flow

- We can go through an example to discover at what velocity the flow in a pipe stops being laminar. If the pipe and the fluid have the following properties:

  water density r = 1000 kg/m^3
  pipe diameter d = 0.5m
  $\mu = 0.55 \times 10^{-3}$ Ns/m^2

- We want to know the maximum velocity when the Re is 2000.

$$\mathrm{Re} = \frac{\rho u d}{\mu} = 2000$$

$$u = \frac{2000\,\mu}{\rho d} \cdot = \frac{2000 \times 0.55 \times 10^{-3}}{1000 \times 0.5}$$

$$u = 0.0022\,m/s$$

Aerodynamics

Turbulence + Separation

Laminar and turbulent flow

- So what does Reynolds number mean?
- We can say that the number has a physical meaning, by doing so it helps to understand some of the reasons for the changes from laminar to turbulent flow.

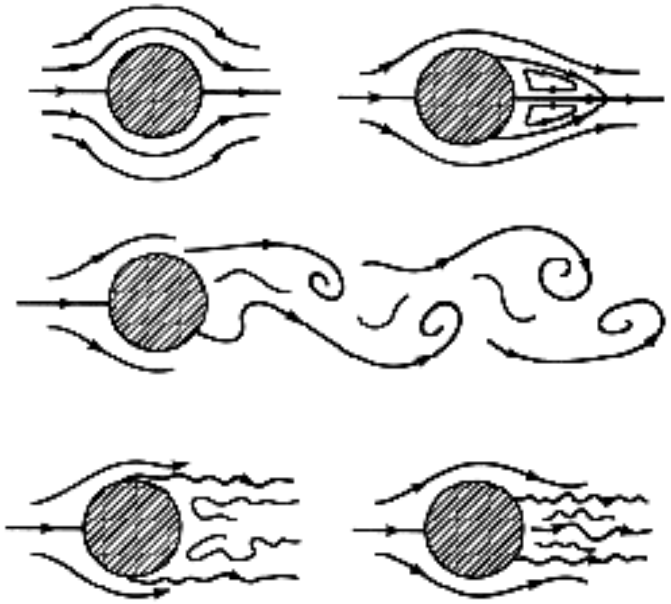
$$\mathrm{Re} = \frac{\rho u d}{\mu} = \frac{Inertial\ Forces}{Viscous\ Forces}$$

- It can be interpreted that when the inertial forces dominate over the viscous forces (when the fluid is flowing faster and Re is larger) then the flow is turbulent.
- When the viscous forces are dominant (slow flow, low Re) they are sufficient enough to keep all the fluid particles in line, then the flow is laminar.

Aerodynamics
Turbulence + Separation

Laminar and turbulent flow

- Flow fields around a cylinder at reynold's number = 0.2, 12,120,30000 and 500000



Aerodynamics
Turbulence + Separation

Derivation of Reynolds's number

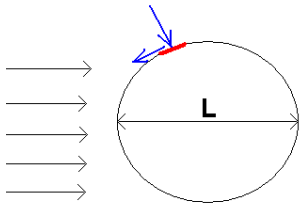
- We can also derive the equations for Reynolds's number by looking at the ratio of pressure forces to viscous forces.

- If this ratio is the same for two different cases, eg. Size of velocities, we can assume the flow pattern over the whole body will be the same, which will give a condition called dynamic similarity.

$$F_v = A\mu\frac{dV}{dy} \qquad F_p = C_1 \times A \times \frac{1}{2}\rho V^2 \qquad C_1 A = C_2 \times L^2 \qquad F_p = C_2 \times L^2 \times \frac{1}{2}\rho V^2$$

- Given $dV \propto V$ $y \propto L$ $A \propto L^2$

$$F_v = A\mu\frac{dV}{dy} \Rightarrow C_3 L^2 \mu\frac{V}{L} = C_3 L\mu V$$



Aerodynamics

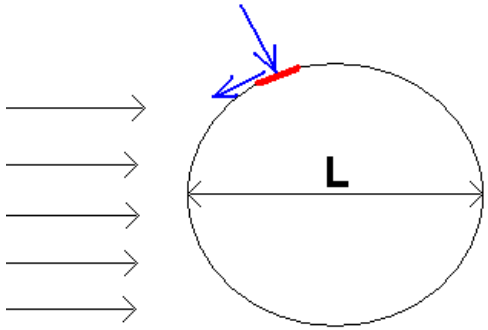
Turbulence + Separation

Derivation of Reynolds's number

$$F_p = C_2 \times L^2 \times \frac{1}{2}\rho V^2 \qquad F_v = C_2 L \mu V$$

$$\frac{F_p}{F_v} = \frac{C_2 \times L^2 \times 0.5\rho V^2}{C_3 \mu V L} = C_4 \times \frac{L\rho V}{\mu}$$

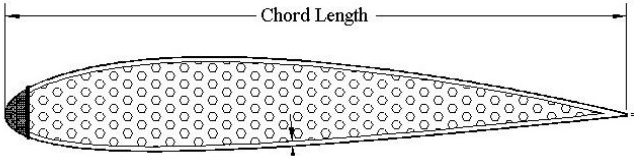
$$\mathrm{Re} = \frac{\rho V L}{\mu}$$



Aerodynamics
Turbulence + Separation

Characteristic length

- In the pipe example we used the diameter to calculate the Reynolds number.
- However we will not only apply the theory to pipes, so we must replace the diameter in the equation with a characteristic length L
- For whole car we use the vehicle length
- For wing we use the chord
- Flow over a plate we use the plate length.



$$\mathrm{Re} = \frac{\rho V L}{\mu}$$

Aerodynamics
Turbulence + Separation

Example

- A F1 car has a length of 4m. Given the density of air is 1.223Kg/m^3 and it's viscosity is 1.78 x10^{-5} kg/ms what is the Reynolds number of the car when travelling at 40m/s?

$$\mathrm{Re} = \frac{\rho V L}{\mu} = \frac{1.223 \times 40 \times 4}{1.78 \times 10^{-5}} = 1.1 \times 10^7$$

- If we wish to test a 45% scale model of the car and achieve the same flow pattern, what velocity must the required flow be?

$$\mathrm{Re} = \frac{\rho V L}{\mu}$$

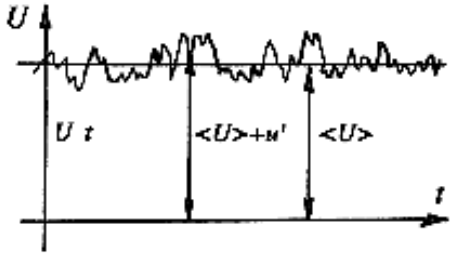
$$V = \frac{\mu \,\mathrm{Re}}{\rho L} = \frac{1.78 \times 10^{-5} \times 1.1 \times 10^7}{1.223 \times 0.45 \times 4} = 88.9 m/s$$

$$88.9 = \frac{40}{0.45}!!$$

Aerodynamics
Turbulence + Separation

Turbulence

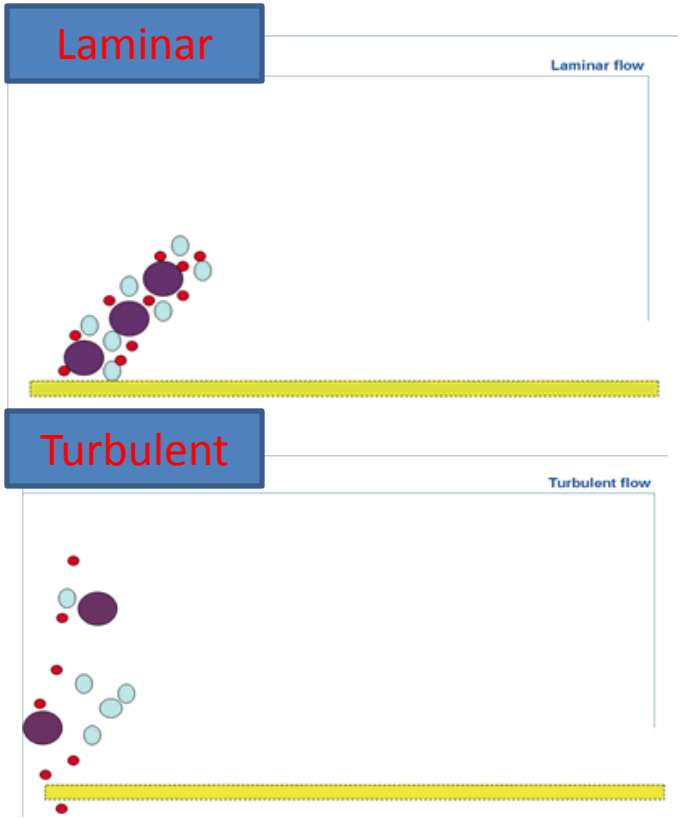
- So what is turbulent flow?
  - Occurs when **inertial forces overwhelm viscous forces**.
  - Can be described as a Stochastic (**random**) flow
  - If you monitored the flow velocity in the U direction at a point in a turbulent flow you would get a trace as below. It has a mean flow speed U, and a fluctuating component u' of the velocity.



Aerodynamics

Turbulence + Separation

Turbulence



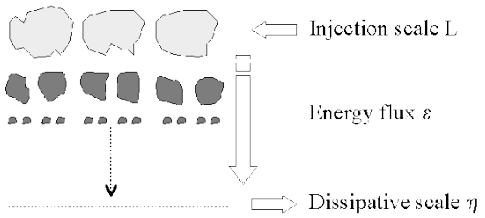
Aerodynamics
Turbulence + Separation

Turbulence

- Turbulence is **a three dimensional phenomena** even if the mean flow is 2D or 1D. So you would also measure a fluctuating component for velocities in both the v and w directions too.

$$u = U + u' \qquad v = V + v' \qquad w = W + w'$$

- Turbulent flow contains eddy motions (where the fluctuating component comes from) of all sizes and a large part of the mechanical energy in the flow goes into the formation of these eddys.
- Eddys also know as vortices
- The large vortices are continually produced by the mean flow field and form progressively smaller vortices as viscous action converts some energy to heat.
- When the energy has been transfered through vortice sizes to the smallest vortice Re ~ 1, viscous forces are equal to inertial and so the vortice is quickly destroyed dissipating the energy as heat.
- This is called the **energy cascade**!

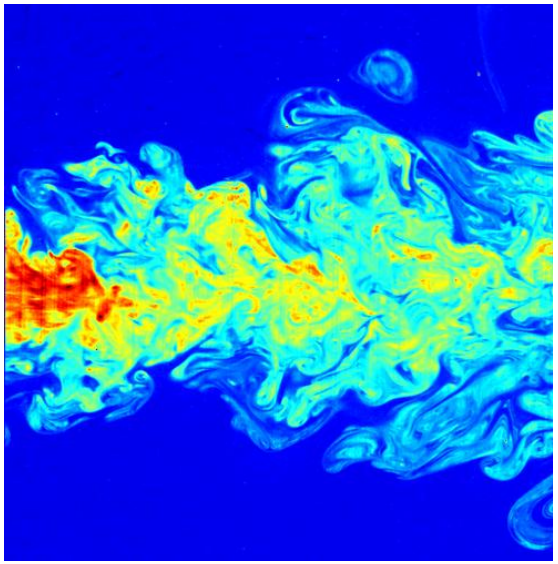


Aerodynamics

Turbulence + Separation

Turbulence



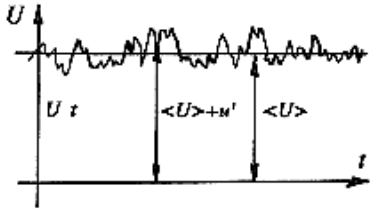


Aerodynamics
Turbulence + Separation

Turbulence

- A useful parameter for quantifying turbulence levels is the turbulence intensity parameter

$$Ti = \sqrt{\frac{\overline{u}'}{U^2}}$$

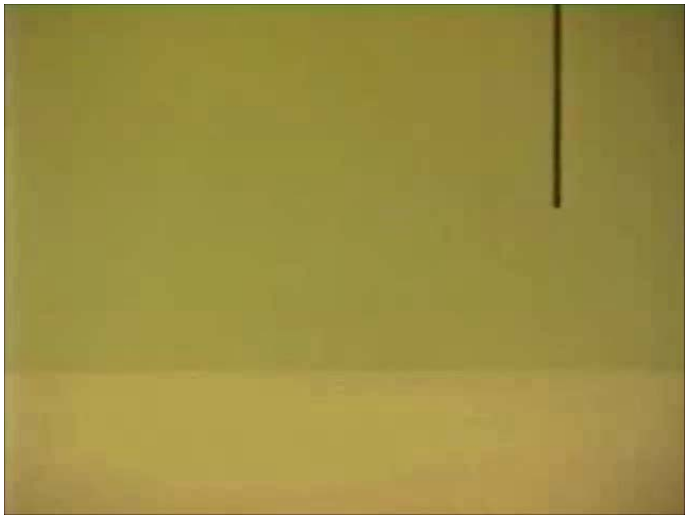


- For a well designed wind tunnel Ti is around 0.01
- For flow in the wake of a body Ti can be around 0.5
- Ti is normally expressed as a percentage figure. Ti= 0.5 = 50%

Aerodynamics
Turbulence + Separation

Boundary Layers

- When a fluid flows over a stationary surface, e.g. the bed of a river, or the wall of a pipe (surface of a wing), the fluid touching the surface is brought to rest by the shear stress t_o at the wall.
- Viscous effect slows down elements of the flow close to the surface as the molecules of the fluid 'stick' to the boundary.
- **NON SLIP CONDITION**

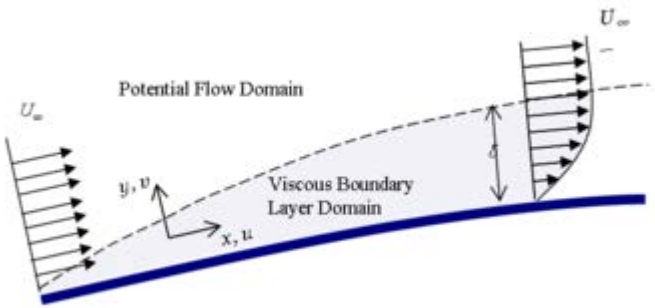


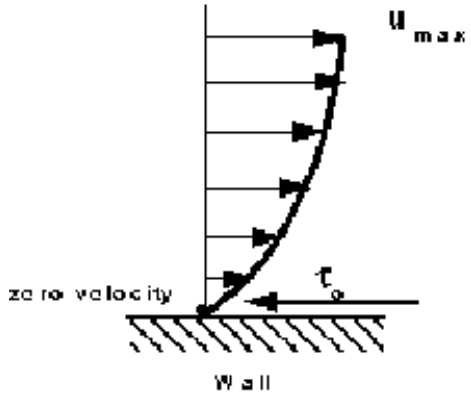
Aerodynamics
Turbulence + Separation

Boundary Layers

- The velocity increases from the wall to a maximum in the main stream of the flow.

- Looking at this two-dimensionally we get the velocity profile from the wall to the free stream flow.



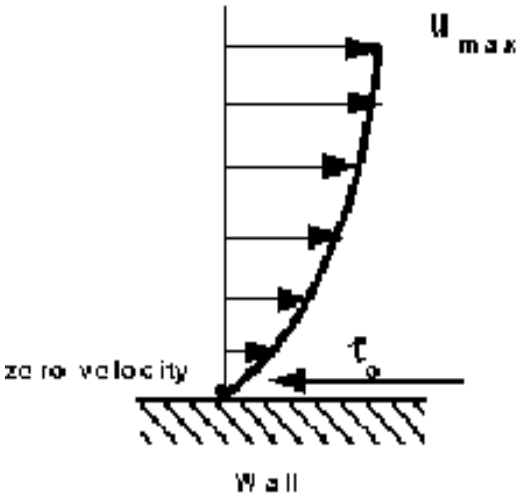


Aerodynamics

Turbulence + Separation

Boundary Layers

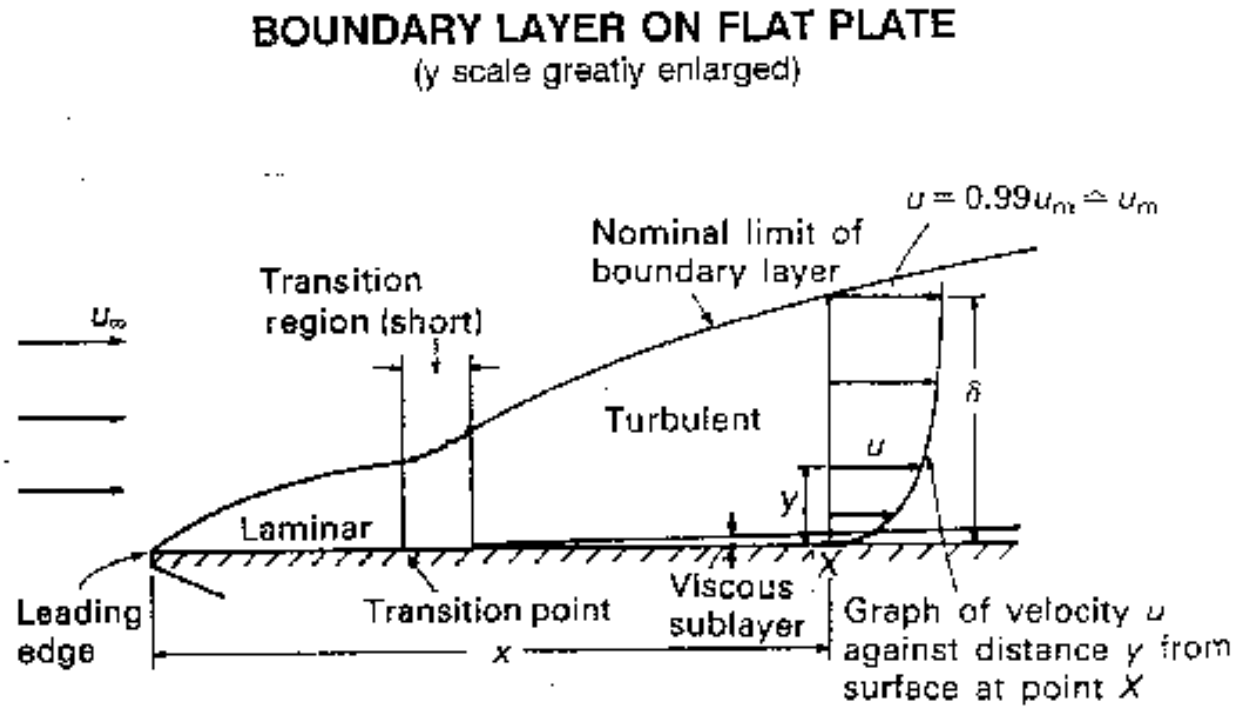
- If we consider a flat plate in the middle of a fluid, we will look at the build up of the velocity profile as the fluid moves over the plate.
- Upstream the velocity profile is uniform, (free stream flow) a long way downstream we have the velocity profile on the right. This is the known as **fully developed flow**. But how do we get to that state?
- This region, where there is a velocity profile in the flow due to the shear stress (viscous effects) at the wall, we call the **boundary layer**.
- The **boundary layer thickness** is taken to be the distance from the surface to the point where the local flow velocity is 99% of the free-stream velocity



Aerodynamics
Turbulence + Separation

Boundary Layers

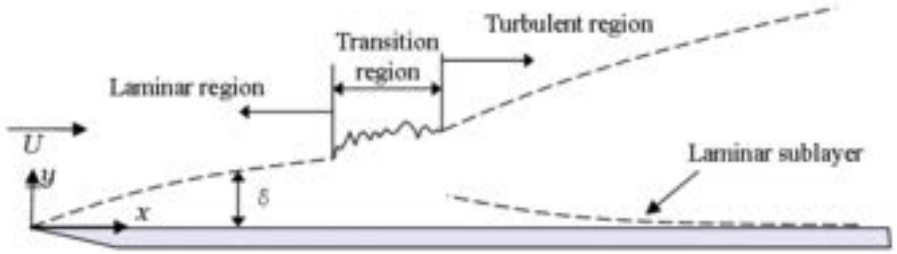
- The stages of the formation of the boundary layer are shown in the figure below:



Aerodynamics
Turbulence + Separation

Boundary Layers – Formation

- The boundary layer grows from zero when a fluid starts to flow over a solid surface.
- As is passes over a greater length more fluid is slowed by friction between the fluid layers close to the boundary.
- Hence the thickness of the slower layer increases.

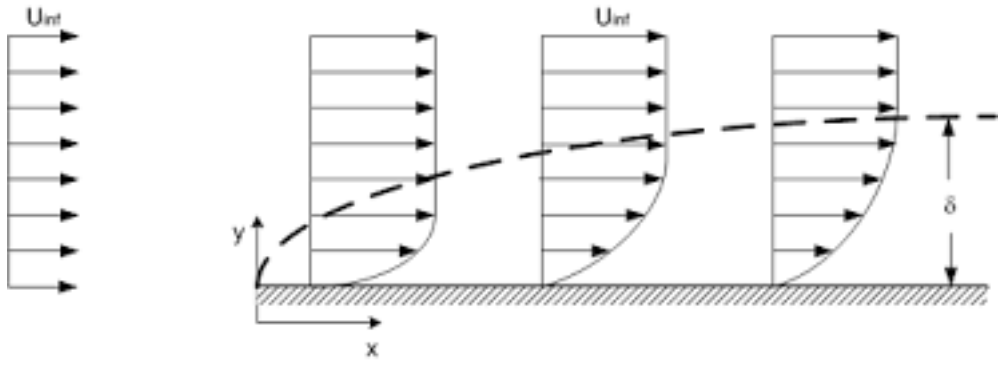


Aerodynamics
Turbulence + Separation

Boundary Layers – Formation

- The fluid near the top of the boundary layer is dragging the fluid nearer to the solid surface along. The mechanism for this dragging may be one of two types:

- The first type occurs when the normal viscous are large enough to exert drag effects on the slower moving fluid close to the solid boundary.



Aerodynamics
Turbulence + Separation

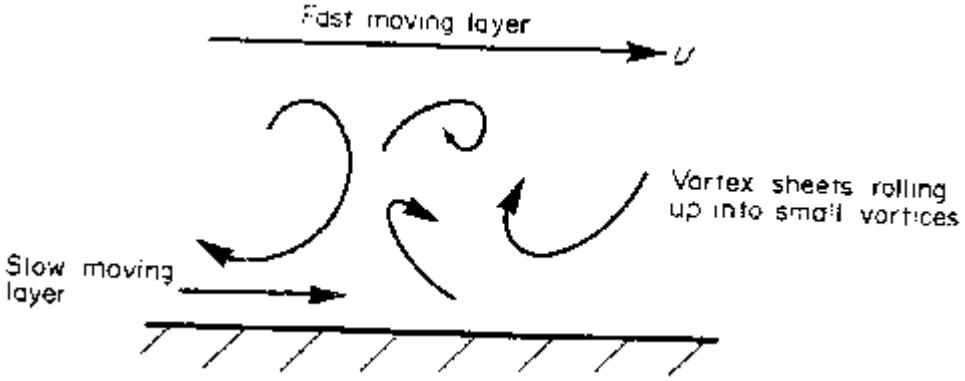
Boundary Layers – Formation

- As the boundary layer thickens , so the velocity gradient becomes smaller
- The shear stress decreases until it is no longer enough to drag the slow fluid near the surface along.
- If this viscous force was the only action then the fluid would come to a rest. It, of course, does not come to rest but the second mechanism comes into play.
- Up to this point the flow has been **laminar** and Newton's law of viscosity has applied. This part of the boundary layer is known as the **laminar boundary layer**

Aerodynamics
Turbulence + Separation

Boundary Layers – Formation

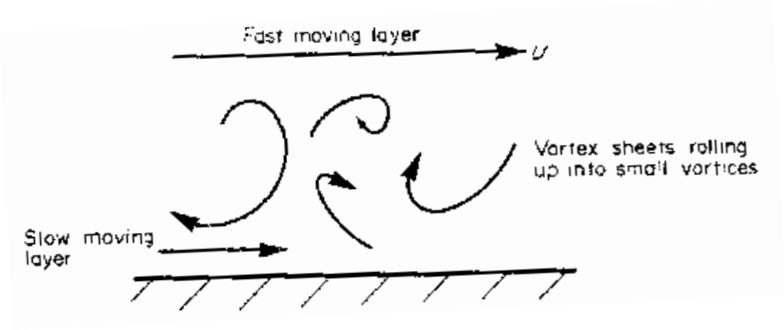
- Eventually when the boundary layer thickness is great enough, the viscous shear stresses are no longer able to hold the flow in layers and the fluid starts to rotate. Now in the transitional region.



Aerodynamics
Turbulence + Separation

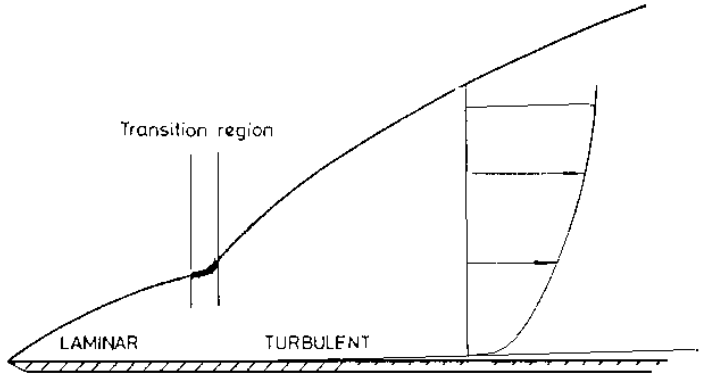
Boundary Layers – Formation

- This causes the fluid motion to rapidly become **turbulent**.



- Fluid from the fast moving region moves to the slower zone transferring momentum and thus maintaining the fluid by the wall in motion.

- Conversely, slow moving fluid moves to the faster moving region slowing it down. The net effect is an increase in momentum in the boundary layer



- We call the part of the boundary layer the **turbulent boundary layer**.

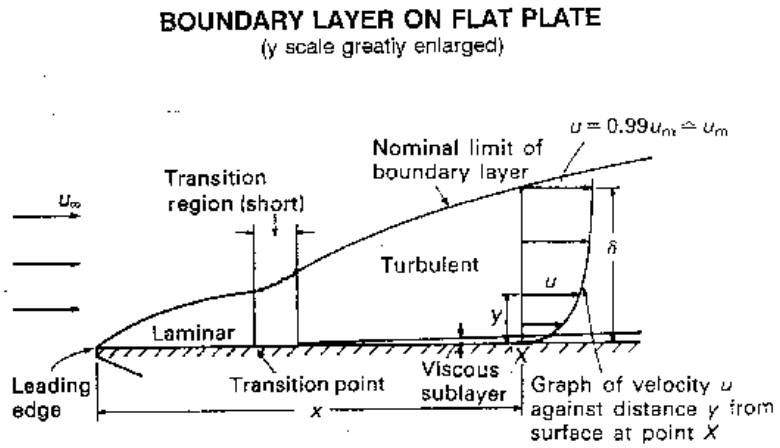
Aerodynamics
Turbulence + Separation

Boundary Layers – Formation

- At points very close to the boundary the velocity gradients become very large and the viscous shear forces again becoming large enough to maintain the fluid in laminar motion.

- This region is known as the **laminar sub-layer**.

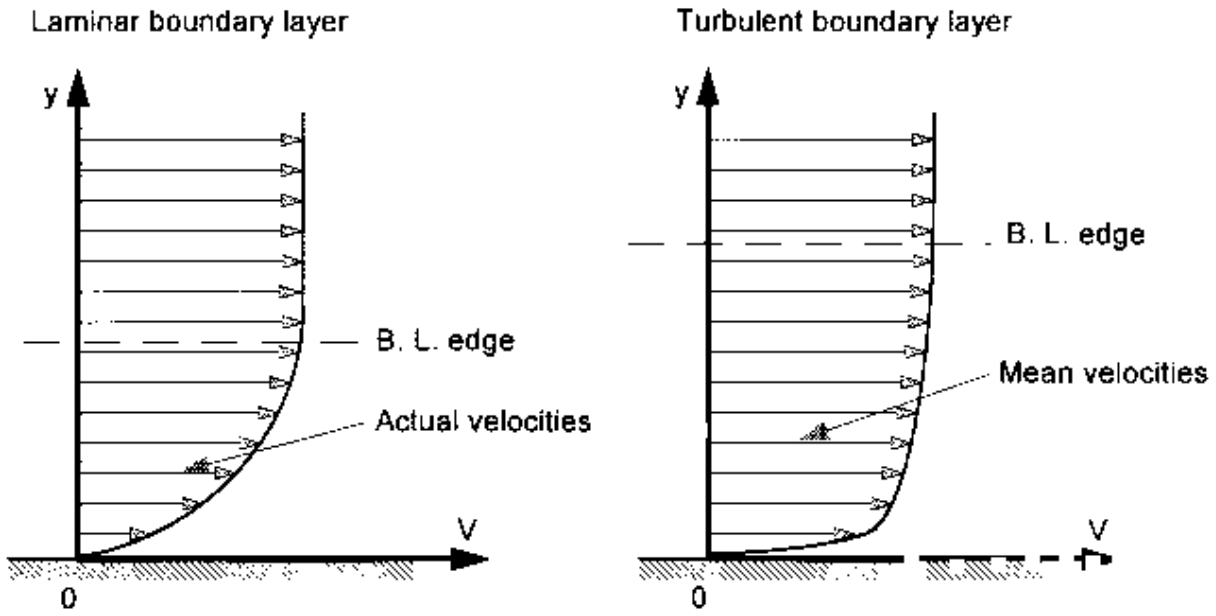
- This layer occurs within the turbulent zone and is next to the wall and very thin - a few hundredths of a mm.



Aerodynamics
Turbulence + Separation

Boundary Layers –

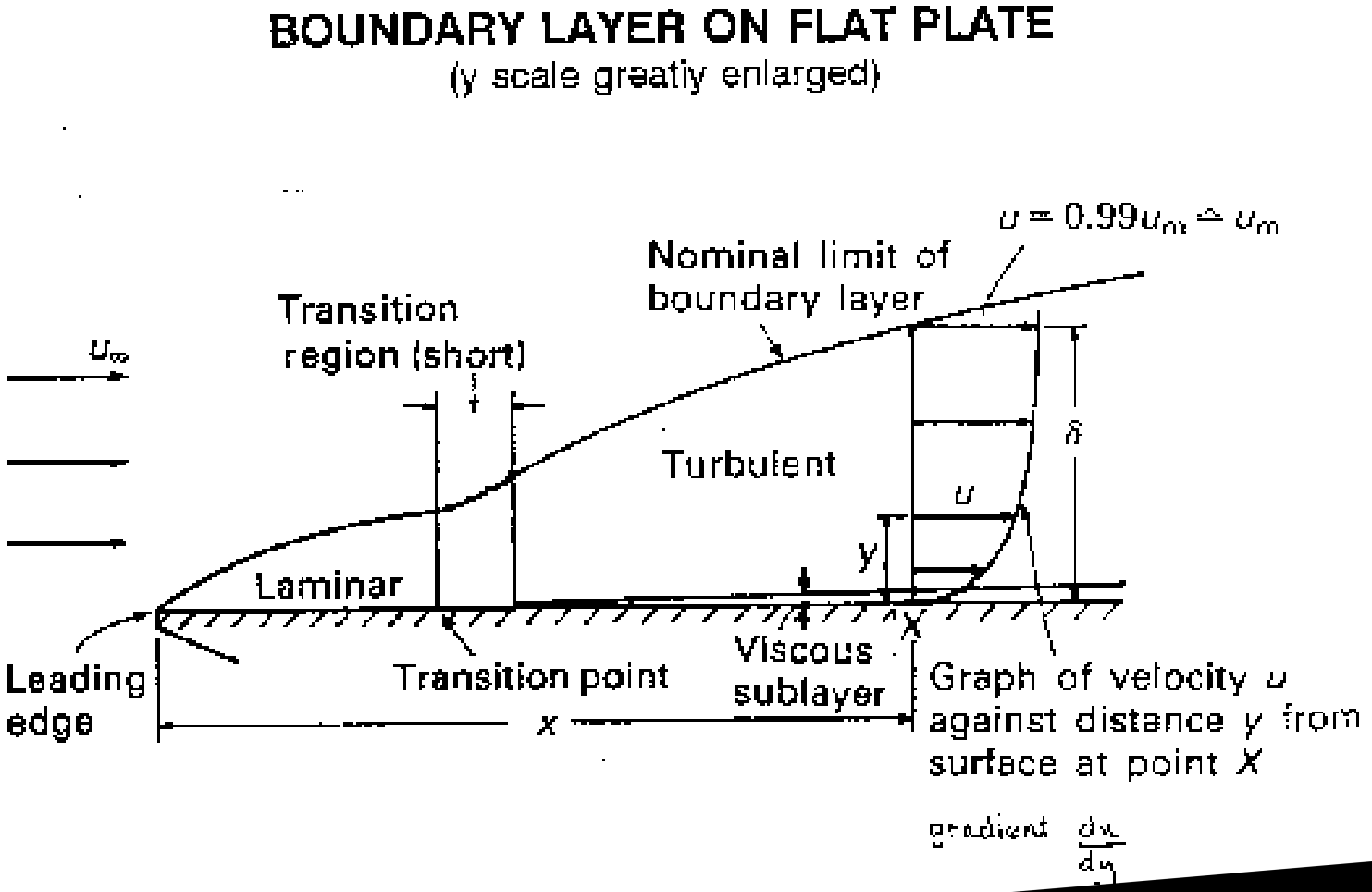
- Comparison of velocity profiles for laminar and turbulent boundary layers.



Aerodynamics

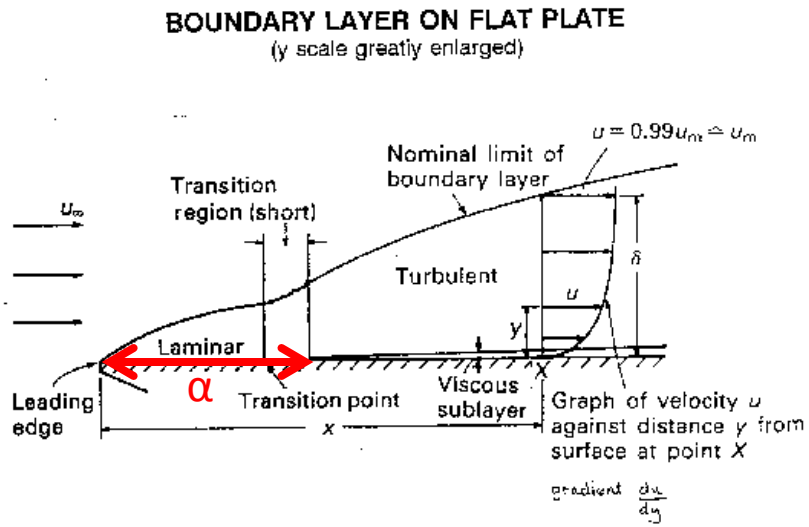
Turbulence + Separation

Boundary Layers



Aerodynamics
Turbulence + Separation

Boundary Layers



Given that the boundary layer flow transition between laminar and turbulent occurs at a Reynold's number of 500,000 for this case, calculate the distance, α, from the leading edge of the surface to the point at which transition occurs.

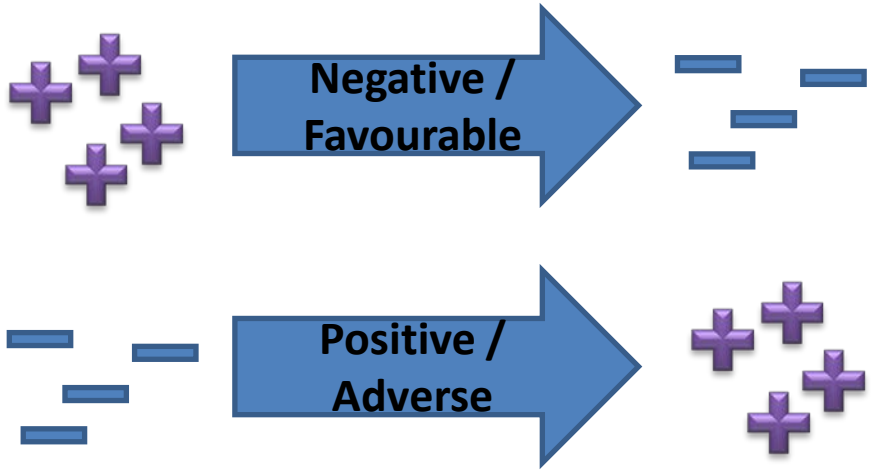
Assume ρ= 1.2047 Kg/m^3 and μ = 1.8205x 10^{-5} Ns/m^2

$$\mathrm{Re} = \frac{\rho u l}{\mu} \qquad l = \frac{\mathrm{Re}\,\mu}{\rho u} = \frac{500000 \times 1.8205 \times 10^{-5}}{1.2047 \times 26} = 0.291m$$

Aerodynamics
Turbulence + Separation

Pressure Gradient

- Where air flows from a region of high pressure to a region of low pressure, the pressure gradient is **negative**.
- Where the flow is from low to higher pressure, the gradient is **positive**.
- The greater the pressure gradient is, the faster the boundary layer growth.

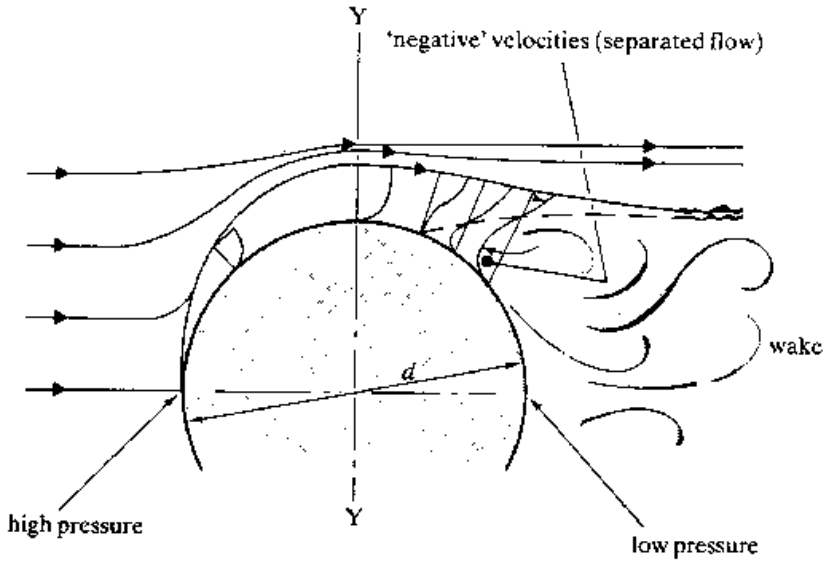


Aerodynamics
Turbulence + Separation

Separation

- Understanding the phenomenon of **separation** is a key to understanding aerodynamics as a whole.

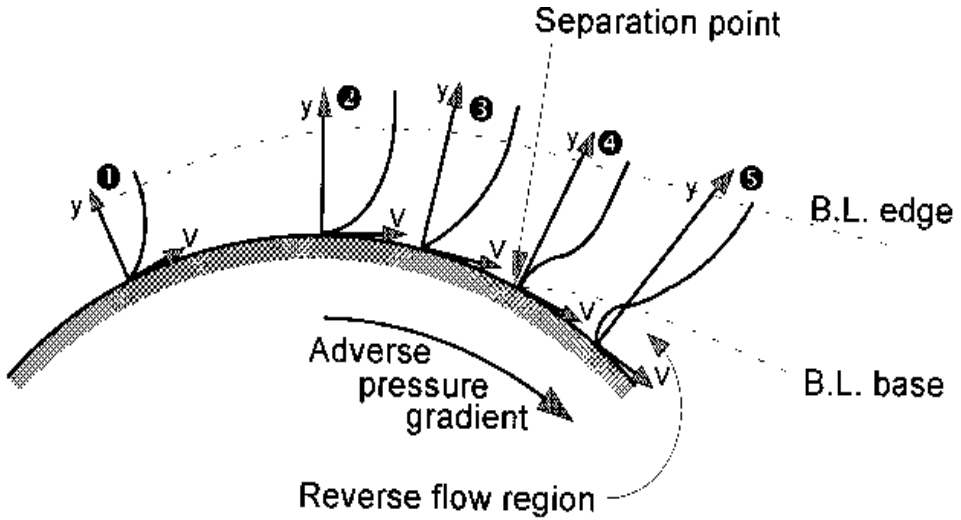
- Lets look at flow around a 2D cylinder



Aerodynamics

Turbulence + Separation

Separation



Stage 1-2

The flow has a negative pressure gradient. The local flow is speeding up as it goes round the cylinder as it has further to go than the surrounding fluid. The boundary layer is growing but not at a rapid pace.

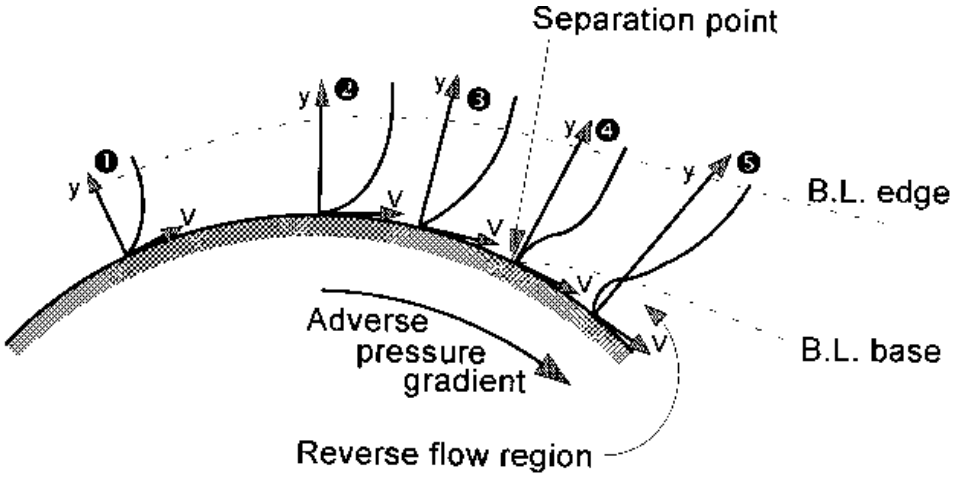
Stage 2-3.

It reaches a maximum at Y, where it also has also dropped in pressure. The pressure gradient is now positive. This means the height of the boundary layer starts to grow rapidly and the airflow slows down

Aerodynamics

Turbulence + Separation

Separation



Stage 3-4

If the flow was inviscid , the kinetic energy of the air would push the air into the higher pressure at the back of the cylinder until it reached the back end.

However, the air does have viscosity and some of the kinetic energy has been used up overcoming viscous forces.

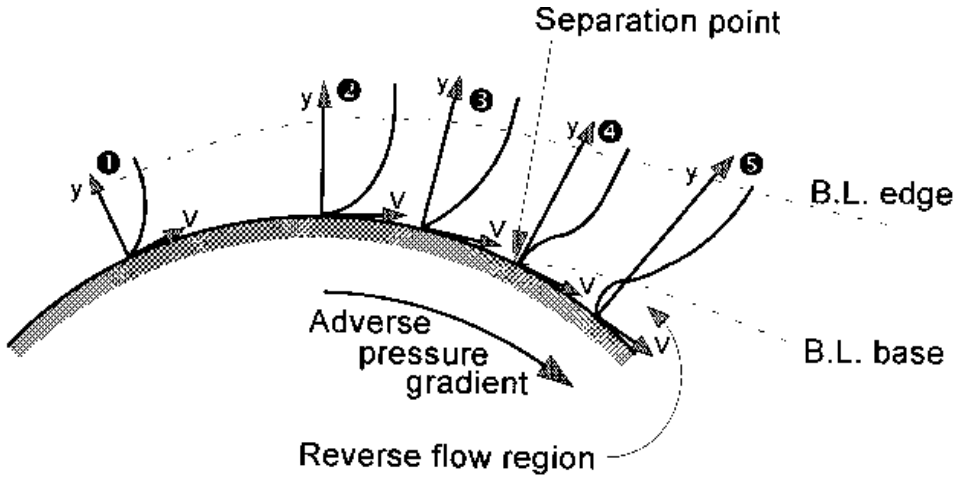
This means that the air does not have enough kinetic energy to overcome the net pressure force trying to slow it down. The air close to the surface comes to rest.

Aerodynamics
Turbulence + Separation

Separation

Stage 4-5

The continuing adverse pressure gradient causes back flow at stage 5.



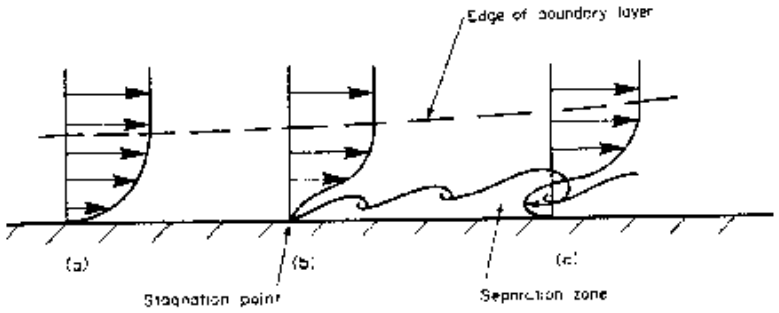
Note that above the line marked B.L. Base, the boundary layer looks similar to what we had before. The boundary layer is said to have separated. The region below B.L. base is said to be the wake. In the wake the air is turbulent.

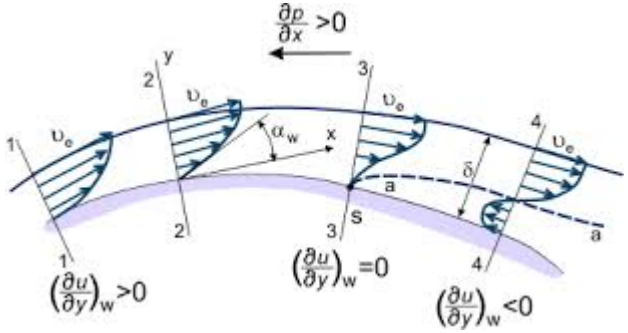
Aerodynamics

Turbulence + Separation

Separation

Although the positive pressure gradient is the same throughout the cross section of the BL it's most significant effect is on the fluid near the surface as it has less momentum than the fluid further out.





The net pressure force brings this fluid to rest. Back flow may then occur and hence the flow is no longer following the contour of the object and separation has occurred.

Aerodynamics

Turbulence + Separation

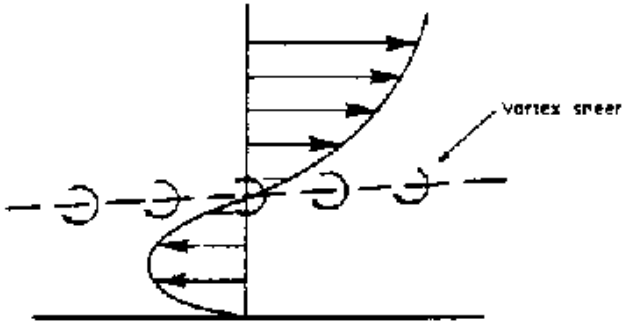
Separation



Aerodynamics
Turbulence + Separation

Separation

- Can only occur where an adverse pressure gradient exists
- Flow over a flat plate with zero or negative pressure gradient will never separate no matter how long the plate is.
- The line separating the forward and reverse flow is known as the separation streamline.



Aerodynamics
Turbulence + Separation

Separation

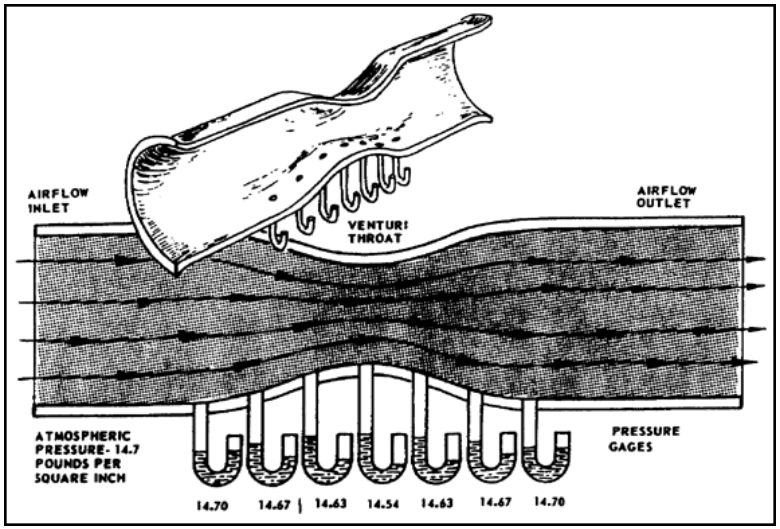
- As a result of reverse flow large eddies are formed which dissipate energy as heat
- Since the energy is dissipated as heat the pressure downstream is usually similar to that at the separation point.
- Laminar BLs are more prone to separation than turbulent ones. Turbulent BL can survive some adverse pressure gradient before separating.
- The greater the adverse pressure gradient the sooner separation occurs.

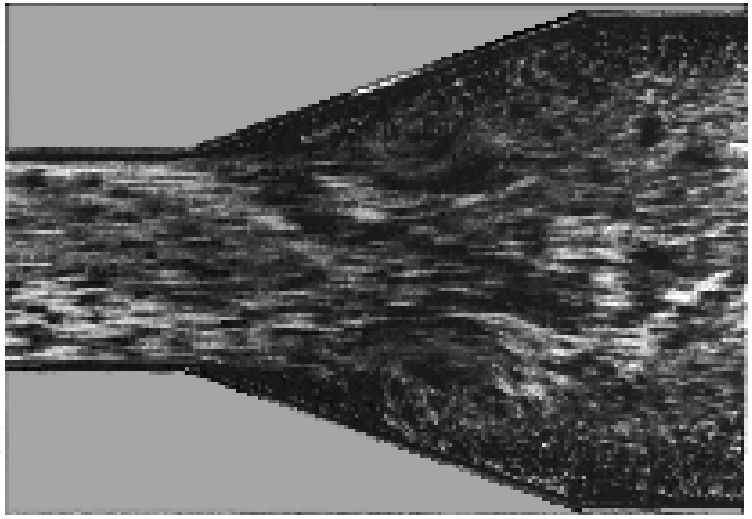


Aerodynamics
Turbulence + Separation

Separation

- Note! The surface need not be curved for separation to occur!
- An adverse pressure gradient will be found in a diffuser and hence separation will occur unless the angle is small or the boundary layer is energised (fed momentum to overcome the net pressure force)

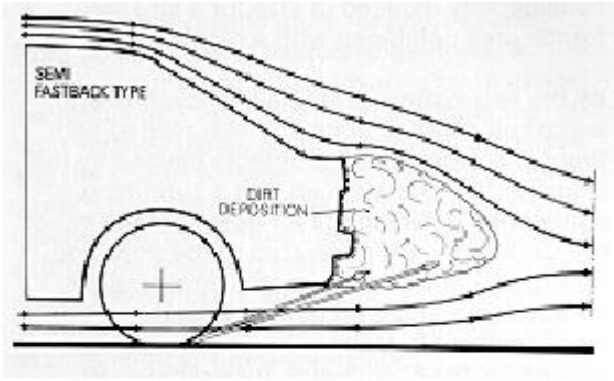


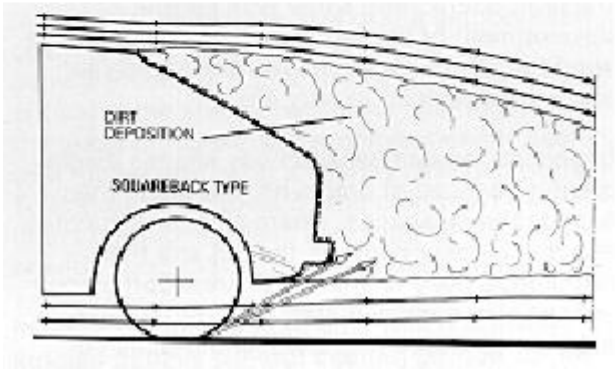


Aerodynamics
Turbulence + Separation

Separation in vehicles

- Vehicles all have zones where the boundary layer has separated.
- Much of the art of vehicle aerodynamics is about controlling boundary layer separation points – rather than trying to eliminate them.
- A controlled boundary layer separation point always leads to greater aerodynamic stability and often results in less drag.



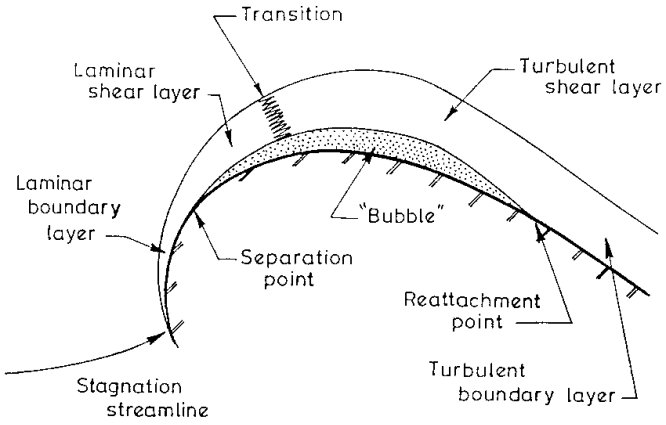


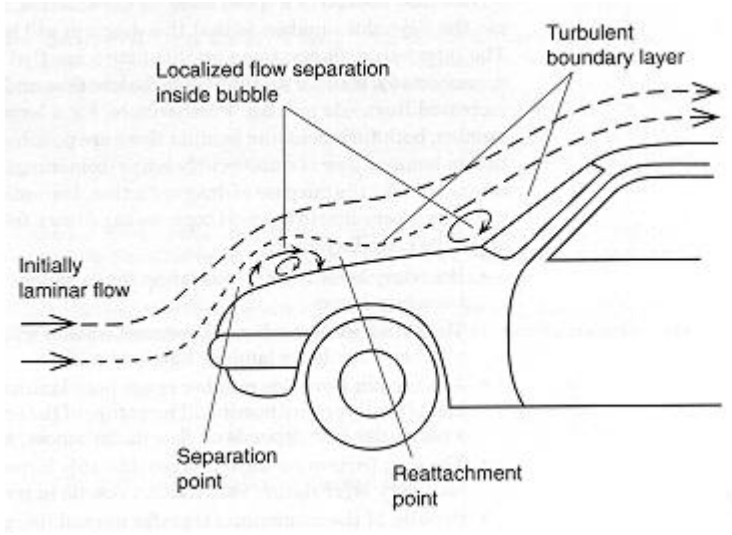
Aerodynamics
Turbulence + Separation

Separation bubbles

- This phenomenon occurs reasonably frequently in vehicles – particularly booted cars.

- The boundary layer separates but re-attaches itself further down the vehicle

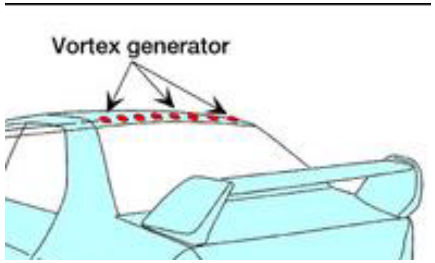


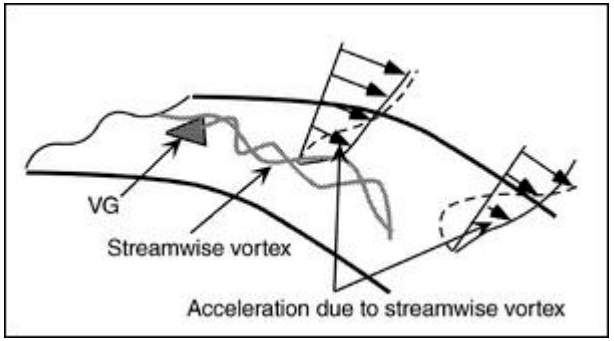


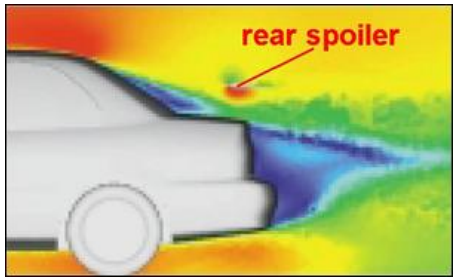
Aerodynamics
Turbulence + Separation

Vortex Generators

- Vortex generators can help delay separation
- Create vortices at edge of boundary layer which takes energy from free stream to increase kinetic energy in boundary layer
- This delays flow reversal









Aerodynamics

Mach number

- What is mach number?

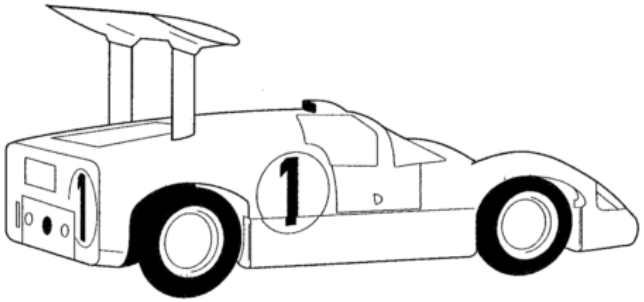
$$Mach\ number = \frac{flow\ velocity}{speed\ of\ sound}$$

- When you approach mach 1 you can no longer assume flow is incompressible. Flow is compressed and shock waves result. Energy goes into compressing the flow. Bernoulli's equation assumed incompressible flow!
- **There is no upstream influence in a supersonic flow**; disturbances are only transmitted downstream
- When an air intake chokes, flow velocity is mach 1. Air cannot travel faster than mach 1 under pressure difference.
- Subsonic – Flow velocities < mach 1
- Transonic – Free stream < mach 1 but some local velocities of flow around body > mach 1
- Supersonic – flow speed > mach 1

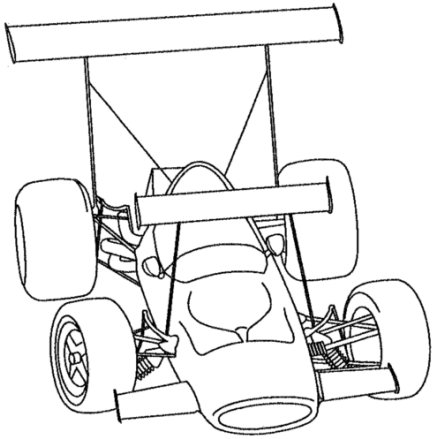
Aerodynamics
Wings

Wings on cars!

Started at the end of 1965 when the Chaparral 2C sports car appeared with large inverted wing to produce downforce. Sported controllable angle of incidence.



F1 cars didn't follow until 1968 with Ferrari v12 3000.



Wings were developed in F1, seeking clear air and feeding loads directly into suspension. These aspects were banned due to accidents!

Aerodynamics
Wings

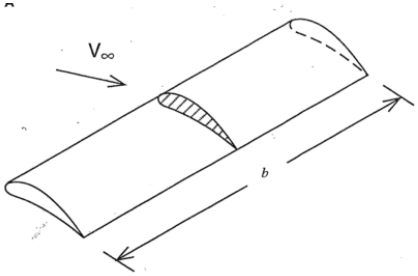
Wings terminology

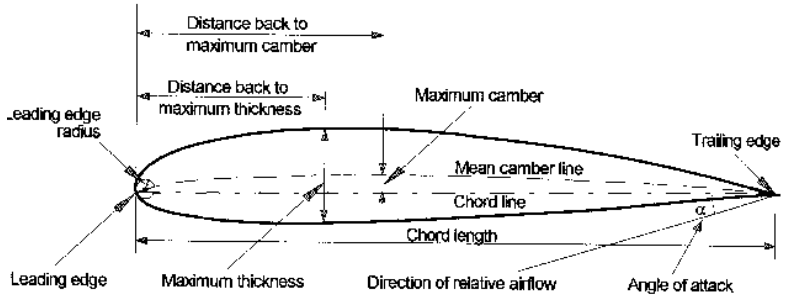
A wing is a three dimensional aerofoil.

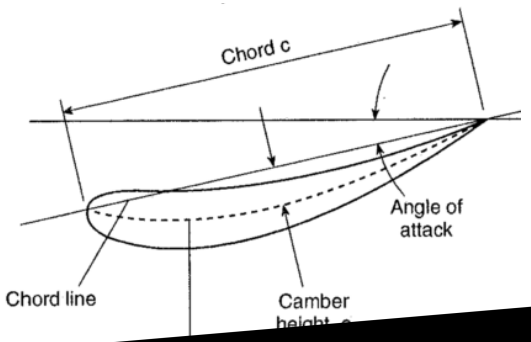
An aerofoil is the 2D cross section of a wing.

Note terminologies from diagrams.

- Chord
- Chord Line
- Camber
- Mean Camber Line
- Leading Edge
- Trailing Edge
- Angle of attack







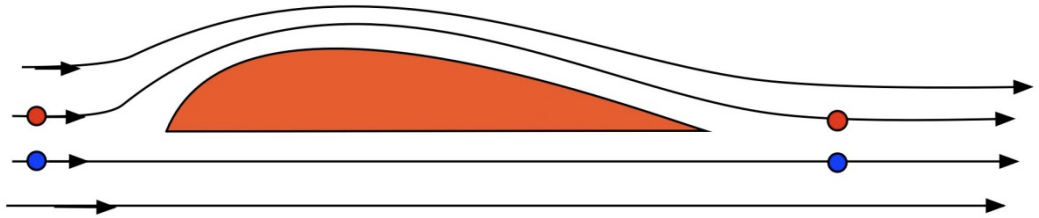
Aerodynamics

Wings

How wings produce lift - The equal time condition (Fallacy!)

The argument goes something like the vertically adjacent air molecules that get separated by the wing must rejoin aft of the wing after the wing passes. This forces the air over the top of the wing to go faster as it has further to travel.

Bernoulli's principle leads to lower air pressure above the wing as the air has to travel faster as it has more distance to cover than the air below and there's your lift.



The problem is that **THERE IS NO EQUAL TIME RULE**. No reason for the air molecules to meet back up

Aerodynamics
Wings

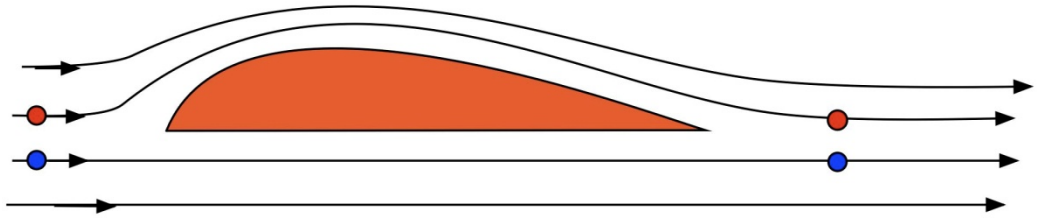
How wings produce lift – Bad flow diagrams!

The figure below also is an impossible air flow picture.

Notice that after the passage of the wing the air looks just as it did before the wing passed.

If there has been no pushing on the air, how can there have been any pushing on the wing. (Newton's conservation of momentum law).

No lift could be generated from such and air flow pattern.



How wings produce lift – Theories

How wings actually produce lift is a subject which has been debated among scientists for many years.

The truth is no one is 100% certain but there are a number of theories which seem likely.

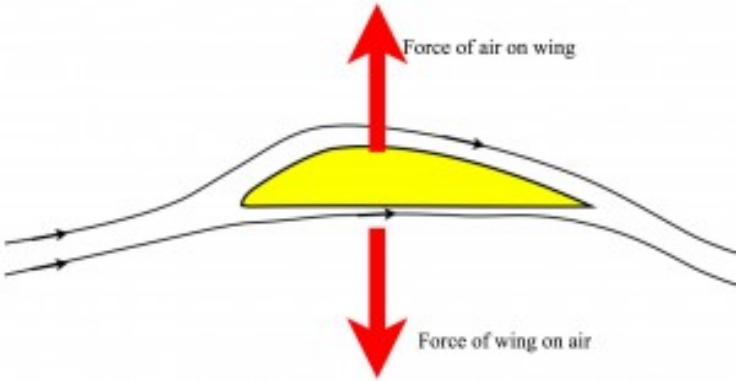
I'll present a couple of the widely accepted theories.

Aerodynamics

Wings

How wings produce lift – Newton.....

Below is a better air flow pattern.



Notice that overall there has been a **redirection** of the air flow and this can only be accomplished by a downward force on the air.

This in turn would cause an equal and opposite force upward on the wing via Newton's third law. In fact this seems to be a perfectly good way to generally explain and quantify lift but it doesn't explain why air would go faster over the top of the wing which it does in reality.

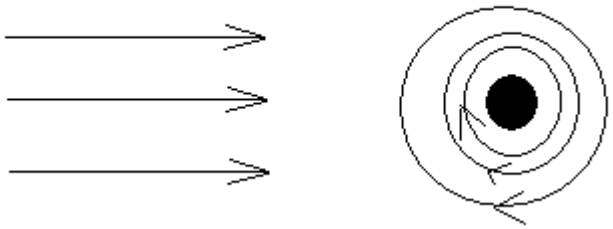
Aerodynamics

Wings

How wings produce lift – Circulation

Consider the bar shown in the diagram below. If the bar forces the air to circulate around it (e.g. rotating), consider the interaction of the local flow with the free stream flow.

Clearly air above the bar will be speeded up and air below the bar will be slowed down. The bar will therefore produce lift by Bernoulli's effect.



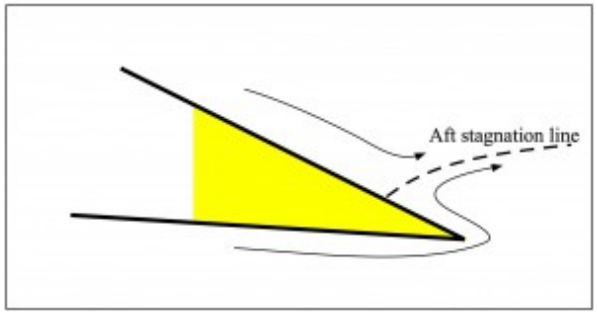
Aerodynamics

Wings

How wings produce lift – Circulation

One of the most accepted theories if the circulation theory (which predates the Wright Brothers by several years!)

It is a multi-step process.



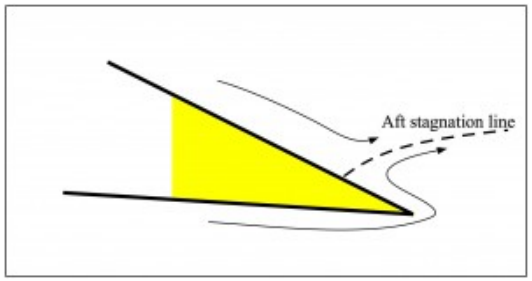
1. As the wing begins to move through the air the air under the wing has to wrap around the aft edge to meet the air moving along the top of the wing.

   This meeting point is called the aft stagnation line. There is one at the front of the wing as well where the air separates to go above the wing or below the wing. This point has to do with angle of attack.

Aerodynamics
Wings

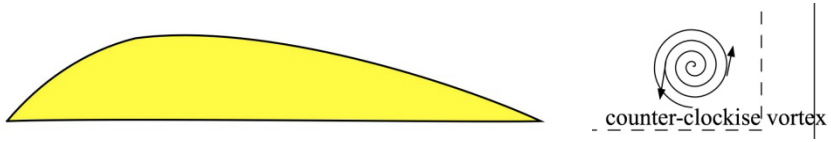
How wings produce lift – Circulation

2. The path the air has to take starts a counter-clockwise flow.



3. This flow is shed by the wing as it accelerates and is left behind.

This is when the stagnation line moves aft to the back edge of the wing. This is also called the Kutta condition.

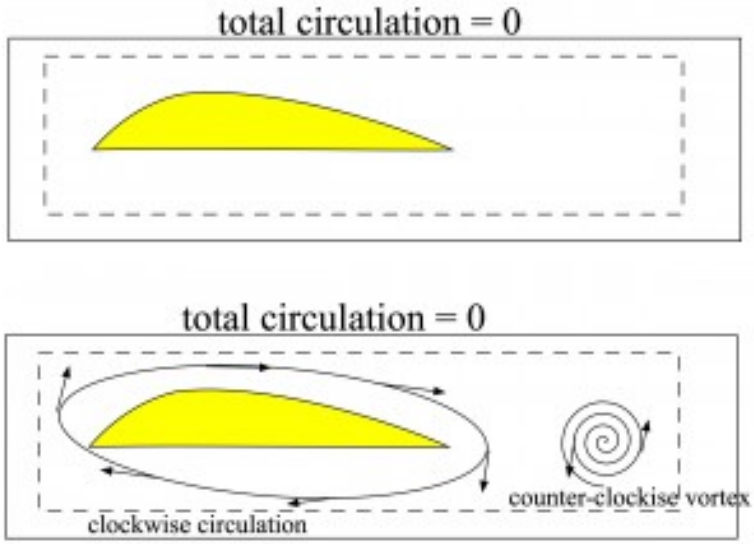


Aerodynamics
Wings

How wings produce lift – Circulation

4. Now before they wing was moving there was no net circulation.
   There is a theorem (Thomson's) that the net flow must remain zero.

5. So, a clockwise flow is induced around the wing so that the total circulation remains zero.



Aerodynamics
Wings

How wings produce lift – Circulation

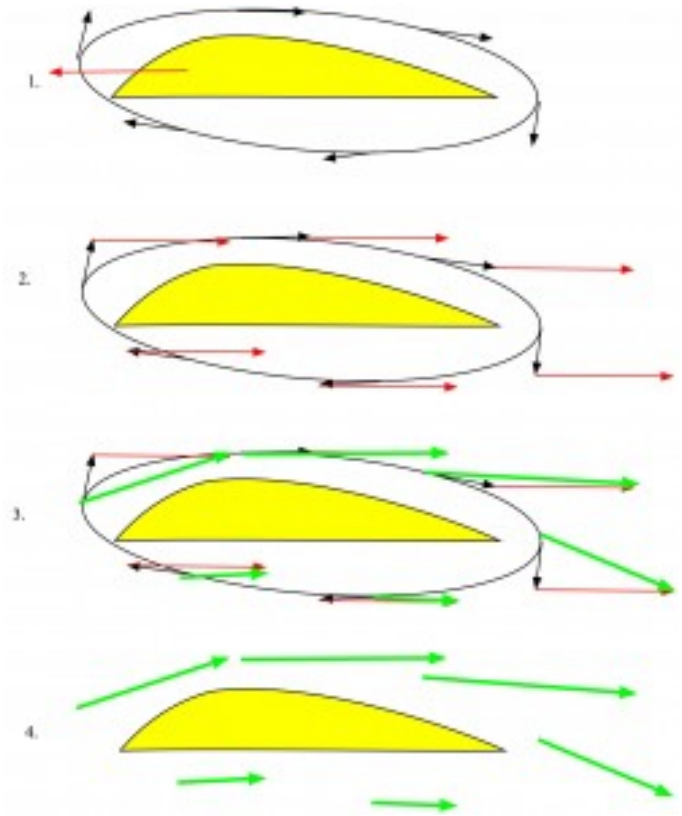
6. The wing now has air moving around it in a clockwise direction AND air flowing past it left to right due to the motion of the wing

Frame 1 I show just the circulation velocities as black.

Frame 2 shows the addition of the red horizontal velocity of the air

Frame 3 shows the sum of these two velocities in green.

Frame 4 shows just the green and we see that indeed we now have air going faster over the top than the bottom.



Aerodynamics

Wings

How wings produce lift – Circulation

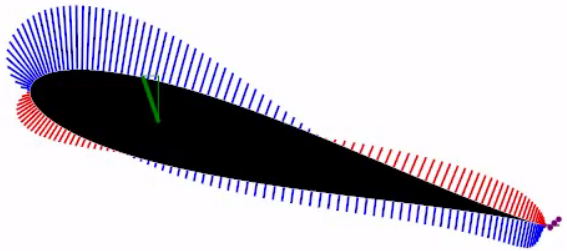


http://vimeo.com/10137777

Aerodynamics

Wings

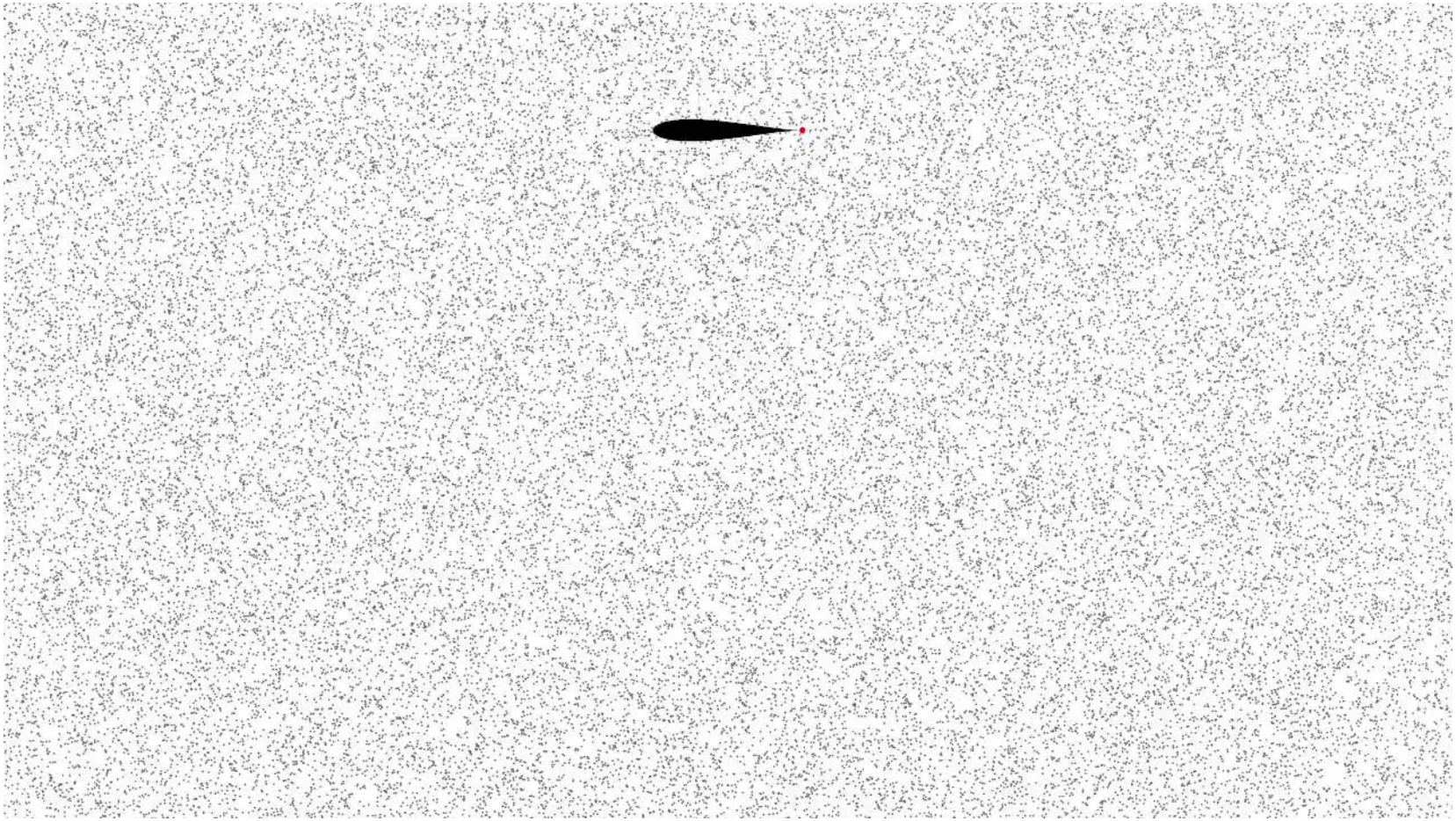
How wings produce lift – Circulation



Aerodynamics

Wings

How wings produce lift – Circulation



Aerodynamics
Wings

How wings produce lift

It is the element of circulation created by a wing which creates lift.

In fact lift is proportional to circulation and the wing can be seen as a circulation generator.

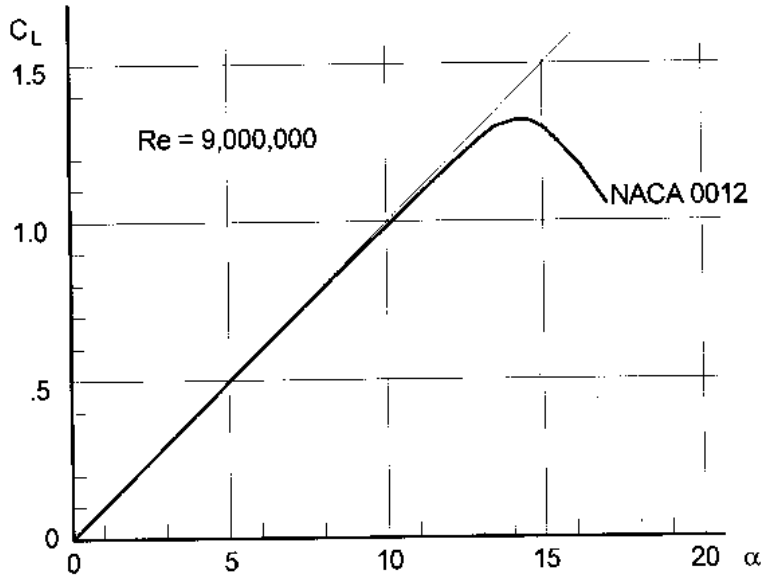
The amount of circulation generated by a wing is proportional the angle of attack α

Therefore we can assume that wing lift is proportional to α

We will now look at the 'lift curve' of a simple aerofoil shape

Aerodynamics
Wings

How wings produce lift



It can be seen that lift is proportional to a **until** the angle of attack reaches around13 degrees.

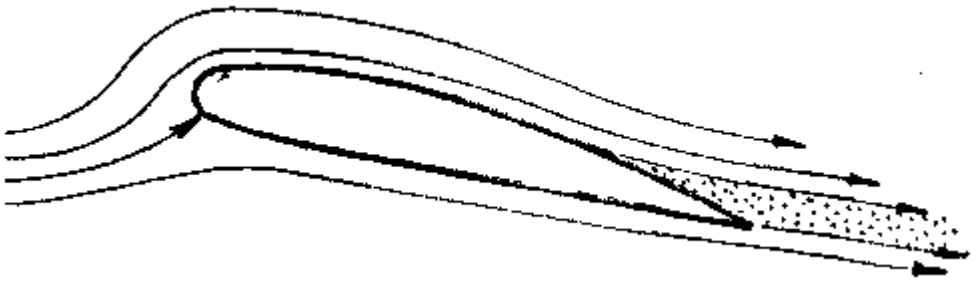
At this point the wing is said to **stall**.

Aerodynamics
Wings

Wings stalling

Stalling occurs when the boundary layer separates from the wing :

There are 2 types of stall –
a) **trailing edge**

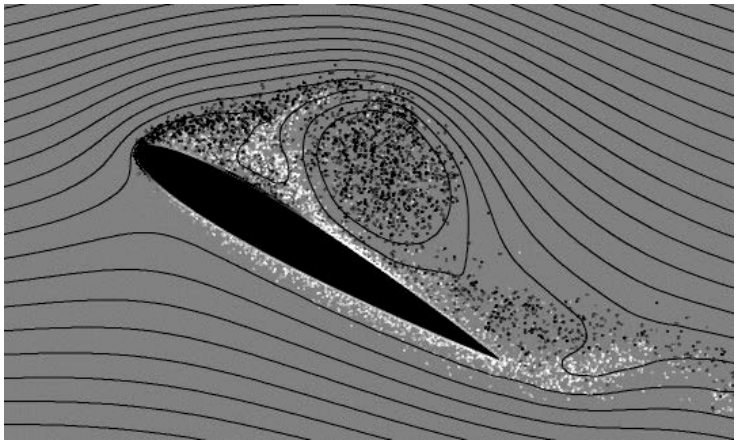


This stal f the lift curve is
roundec edge stall

Aerodynamics
Wings

Wings stalling

b) **Leading edge**



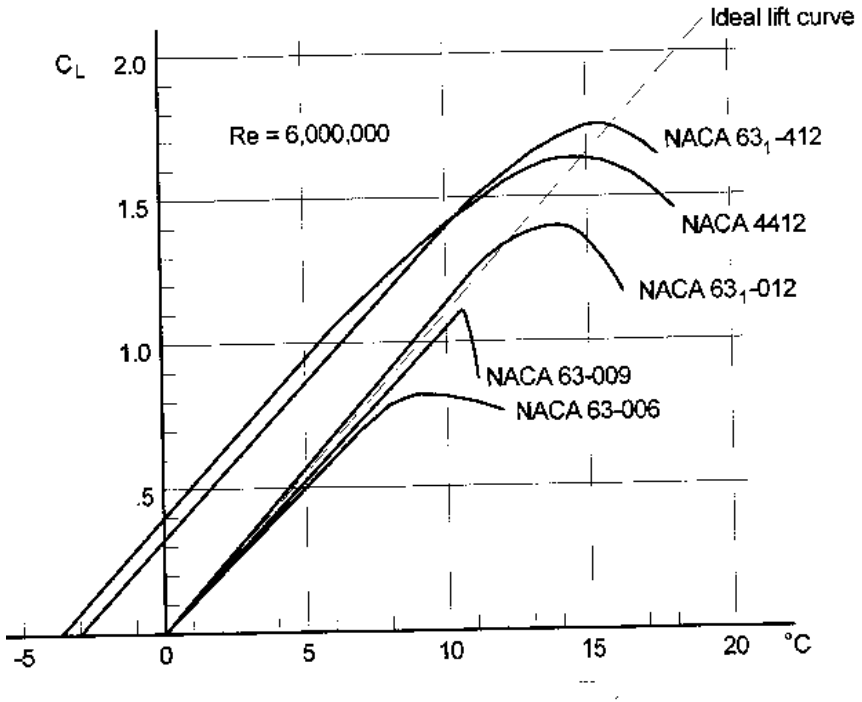
This stall is catastrophic in that it occurs very quickly and results in significant loss of lift. In motorsport, a stall of this nature while cornering causes sudden loss of traction which is difficult to adjust to.

Note wings with sharp front edges are much more likely to exhibit leading edge stalls.

Aerodynamics
Wings

Wings stalling

The graph below shows the lift curve for a number of different aerofoil sections.



Aerodynamics

Wings

Wings stalling - Prevention

All aerofoils are limited by stalling – i.e. separation over the curved top surface of the wing.

As already discussed, separation occurs because of a lack of energy in the boundary layer - If we can find a means to increase the energy in the boundary layer, we may be able to prevent separation. This diagram shows how a slotted wing achieves precisely that

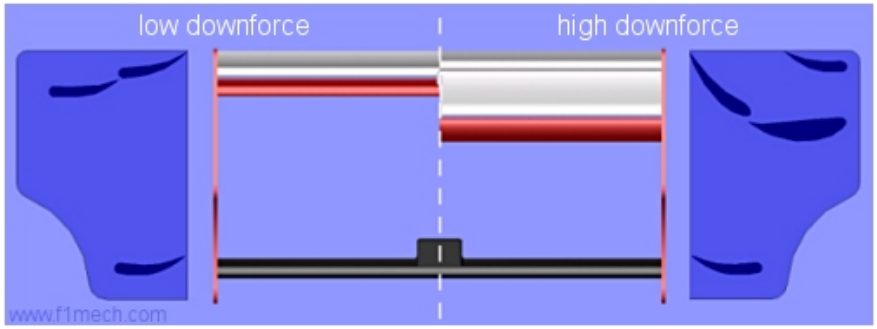


The underside of the wing is high pressure. Above the wing the pressure is low. Therefore the pressure difference will force air through the slot at high velocity.

This high speed air is directed parallel to the wing surface and therefore increases the velocity and energy of the boundary layer over the wing – thus preventing a stall.

Wings stalling - Prevention

This principle can be extended with the use of ‘compound wings’





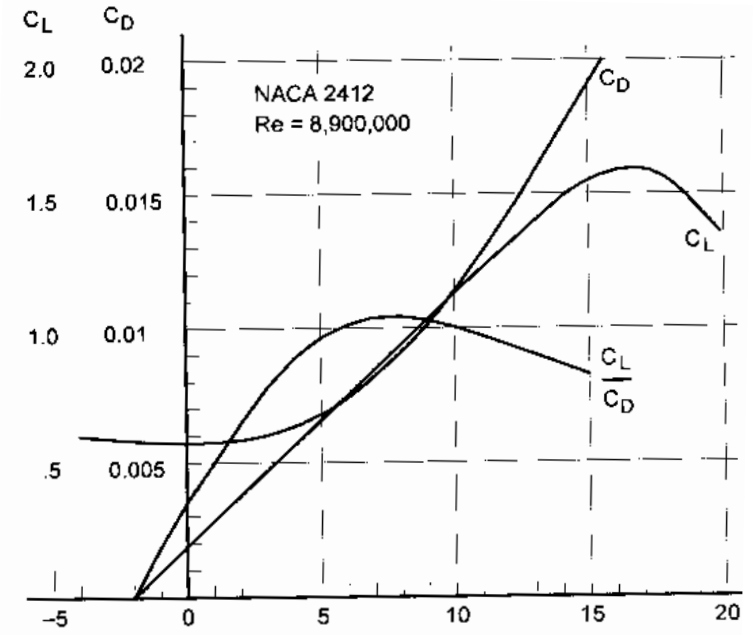


Aerodynamics
Wings

Wing Efficiency

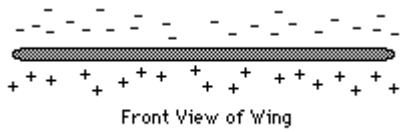
The graph shows how, for a simple wing section, Lift is maximised at an angle of attack of 16 degrees, whereas efficiency (C_L / C_D) is maximised at 7 degrees.

The question in racing is whether to go for maximum downforce, maximum efficiency or minimum drag. This depends on the car and the circuit.

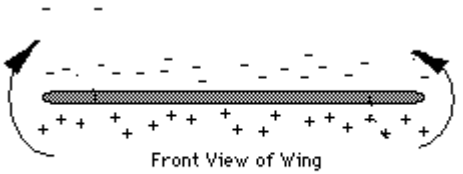


Aerodynamics
Wings

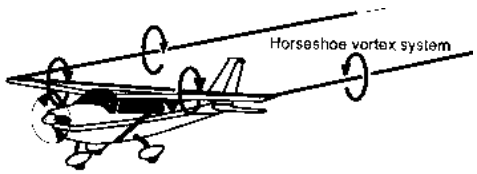
3D effects



Up to now we have looked at aerodynamic shapes in 2D. We will now look at 3D effects – which are particularly important in race car design.



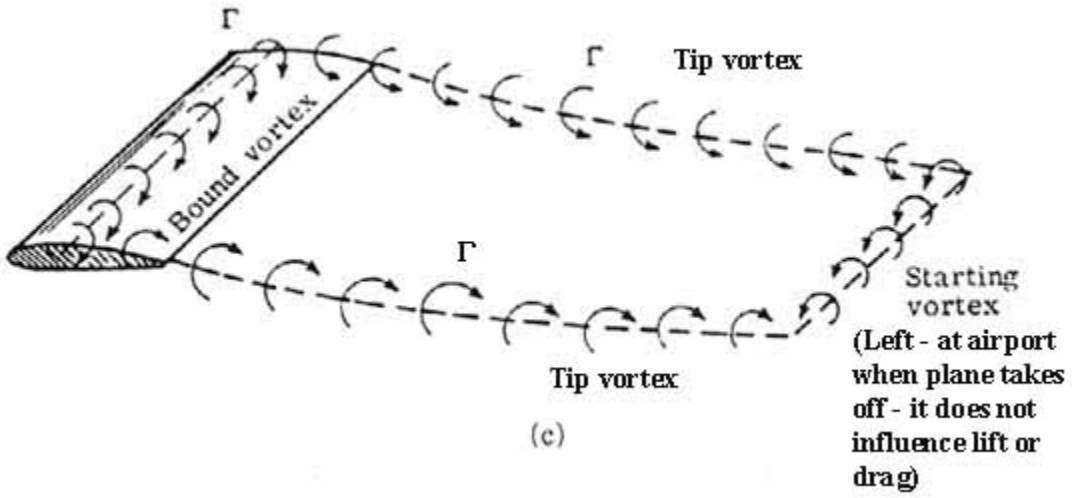
If we consider the top diagram of the front view of an aeroplane wing we can see that there is a tendency for air to flow around the wing tip from positive to negative pressure.



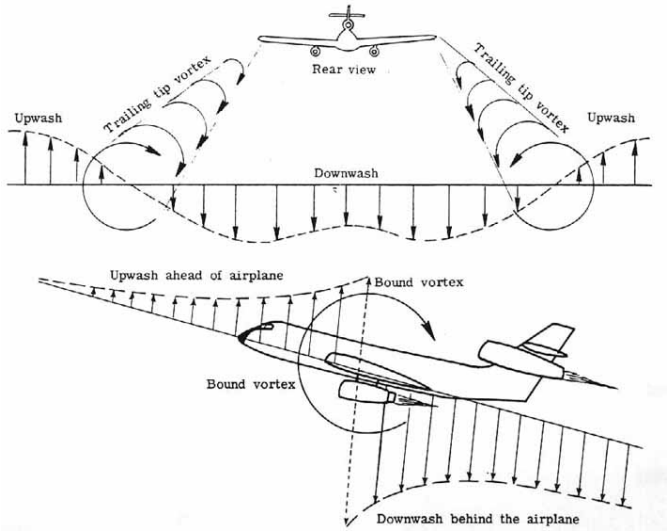
This effect generates wing tip vortices which in turn produces a downwash field on the wing. (Or an upwash field on cars)

Aerodynamics

Wings



(Left - at airport when plane takes off - it does not influence lift or drag)

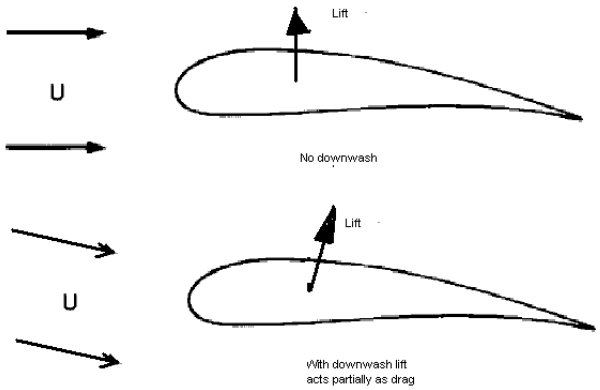


Aerodynamics
Wings

3D effects

This downwash/upwash field has several very significant effects:

- It changes the effective angle of attack of the airfoil section.
- Because the wing is now operating in an angled flowfield, the lift of the wing acts backwards. In racing cars this **induced drag** is the largest element of drag



- In addition the “Spilling” of air around the side of the wing leads to lower lift. Effectively the ends of the wing are ineffective

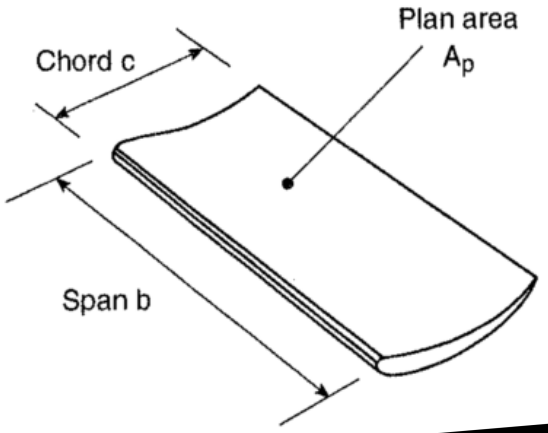
Aerodynamics
Wings

Aspect Ratio

Aspect ratio is defined as wing span/chord length. Long slender wings have a high aspect ratio while short stubby wings have a low aspect ratio.

In high aspect ratio wings the effect of the wing tip vortex is less marked. Hence gliders – which need to have low drag and high efficiency wings - tend to have high aspect ratio wings.

On the other hand, racing car wings are generally restricted in their width and tend to be very low aspect ratio.



$$\text{Aspect ratio} = \frac{b}{c}$$

$$C_L \text{ (wing)} = \frac{\text{Lift}}{\frac{1}{2}\rho V^2 A_p}$$

Calculation of the C_L of a wing

We know that we can calculate the lift of a wing using the equation below.

$$L = \frac{1}{2}\rho V^2 \times C_L \times c \times b$$

But how do we calculate the coefficient of lift for the wing it we only know its dimensions and its aerofoil section's coefficient of lift?

We can use the following equations to calculate the overall wing's coefficient of lift given the aerofoils coefficient of lift, C_1

$$AR = \frac{b}{c}$$

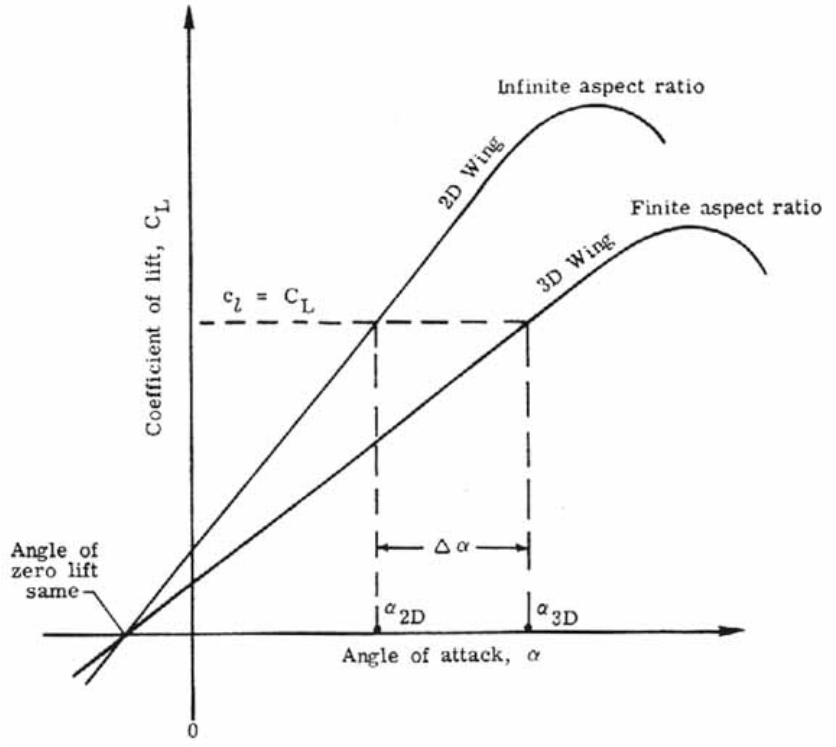
$$C_L = \frac{C_1}{1 + \frac{2}{AR}}$$

Calculation of the C_L of a wing

Equation approximates effects of pressure spill around wing tips.

$$AR = \frac{b}{c}$$

$$C_L = \frac{C_1}{1 + \frac{2}{AR}}$$



Aerodynamics
Wings

Calculation of the C_L of a wing - Example

Calculate the lift produce by the following wing -
Wing span = 1500 mm
Aerofoil Chord = 300 mm
Aerofoil Coefficient of lift = 2
Velocity = 45 m/s
Air density = 1.223 kg/m^3

$$AR = \frac{b}{c} = \frac{1500}{300} = 5$$

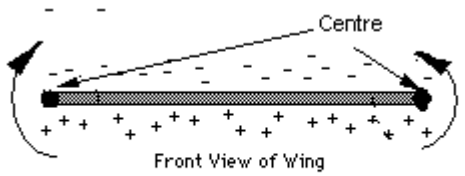
$$C_L = \frac{C_1}{1+\frac{2}{AR}} = \frac{2}{1+\frac{2}{5}} = 1.429$$

$$L = \frac{1}{2}\rho V^2 \times C_L \times c \times b = 0.5 \times 1.223 \times 45^2 \times 1.429 \times 0.3 \times 1.5 = 796.3N$$

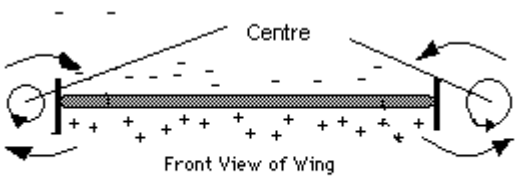
Aerodynamics
Wings

End Plates

End plates attempt to block wing tip vortices.



Because of the laws of vortices, the vorticicity caused by circulation in the wing must be preserved. I.e. the circulation vortex must go somewhere!



Probably the best way of describing endplates is that they push the wingtip vortex further out making the apparent span and aspect ratio greater. Therefore wings with endplates have higher efficiency.

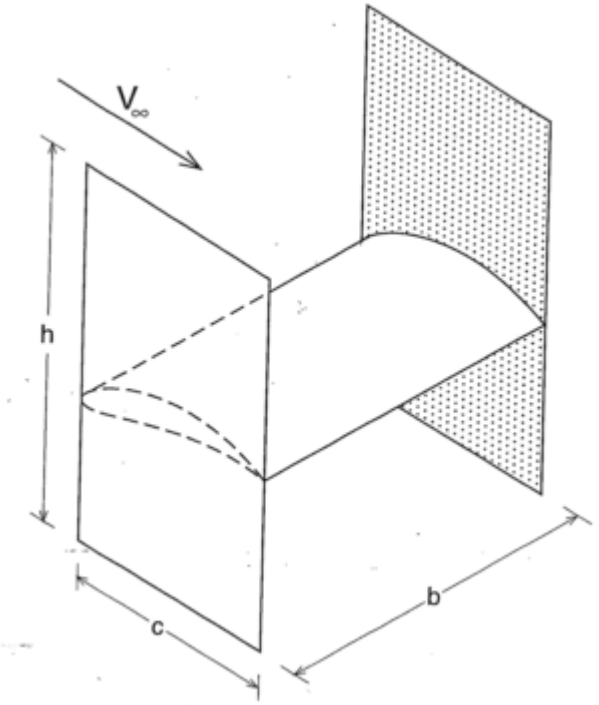
Aerodynamics
Wings

End Plates – Increase effective A.R

We can actually approximate the effective aspect ratio of a wing with endplates using the equation below.

$$AR_{effective} = AR_{actual} \times (1 + 1.9\frac{h}{b})$$

We can then use this effective aspect ratio in subsequent calculations to calculate the lift for a wing with end plates

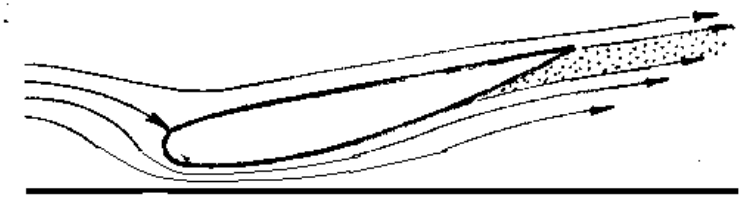


Aerodynamics
Wings

Ground Effect

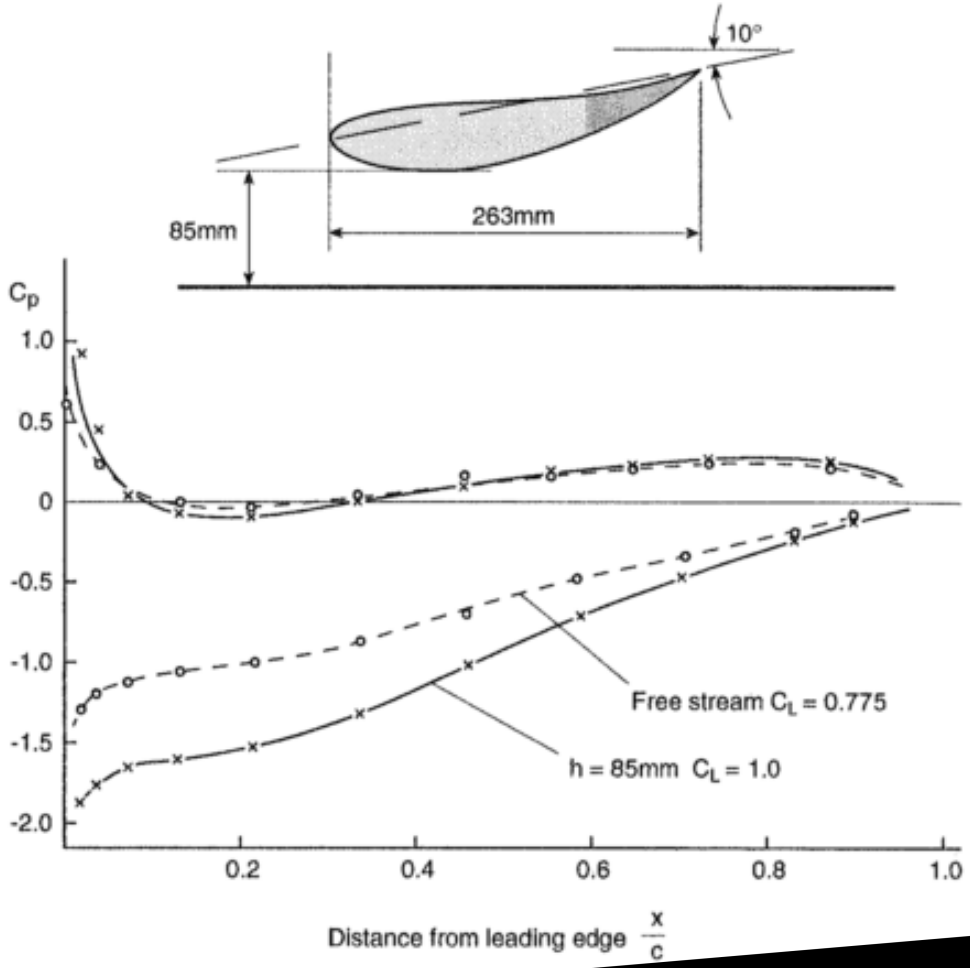
- Consider the downforce producing wing in free space
- What happens when the wing is close to the ground?
- Clearly you cannot have streamlines cutting through the ground.
- Therefore the effect is to bunch up the streamlines under the wing. This results in higher velocity under the wing and therefore , by benoullis effect, increased downforce BUT it also results in a greater adverse pressure gradient around the wing surface and therefore a greater tendency for the wing to stall.





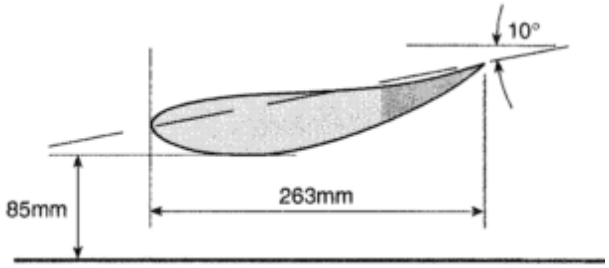
Aerodynamics
Wings

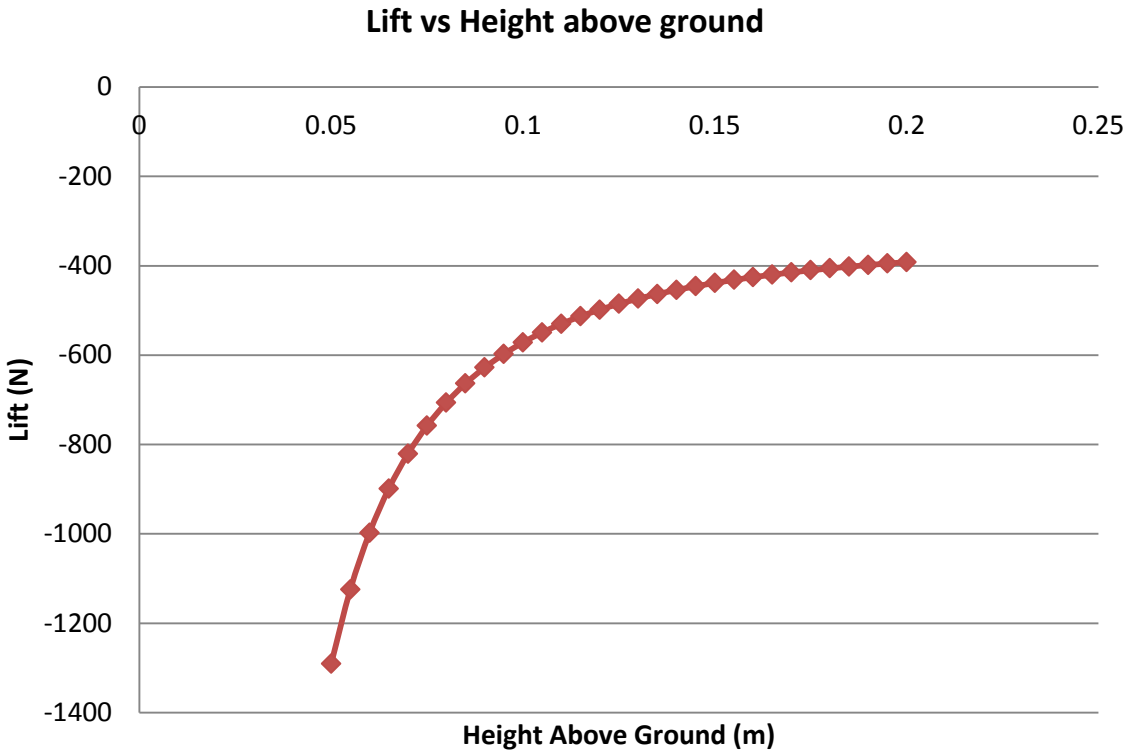
Ground Effect



Aerodynamics
Wings

Ground Effect





Aerodynamics

Wings

Ground Effects effect on lift

We can approximate the effect of ground effect on the aerofoils coefficient of lift, by using the following formula

$$C_1(ige) = C_1(oge) \times \left(1 + \frac{1}{16\left(\frac{h}{c}\right)^2} \right)$$

Where

C_1(ige) = Aerofoils coefficient of lift in ground effect

C_1(oge) = Aerofoils coefficient of lift out of ground effect

h = height of aerofoil above ground

c = chord length

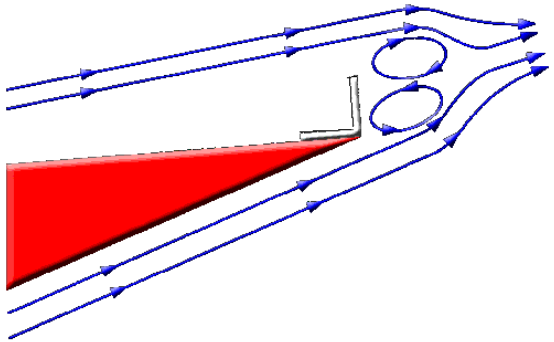
Aerodynamics
Wings

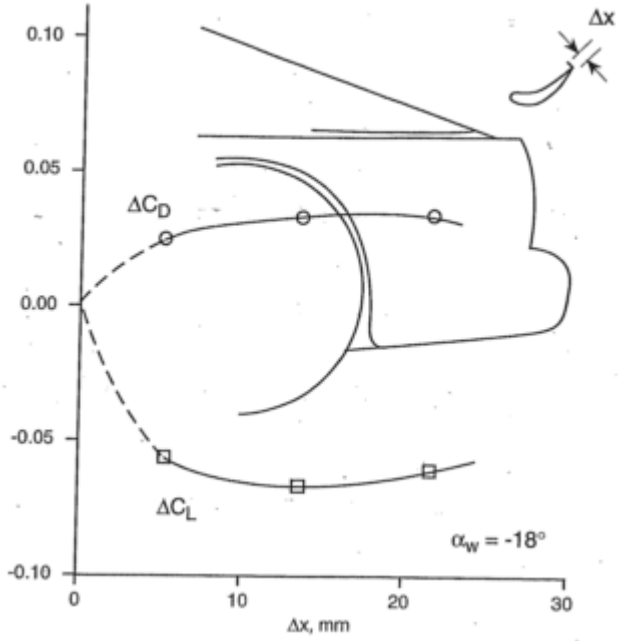
Gurney Flaps

A Gurney flap is a relatively small barrier mounted at right angles to the trailing edge of a wing.

It causes the airflow to rotate more at the trailing edge which increases lift but also helps energise the boundary layer and prevent separation. This allows higher angles of attack to be run before the wing stalls.

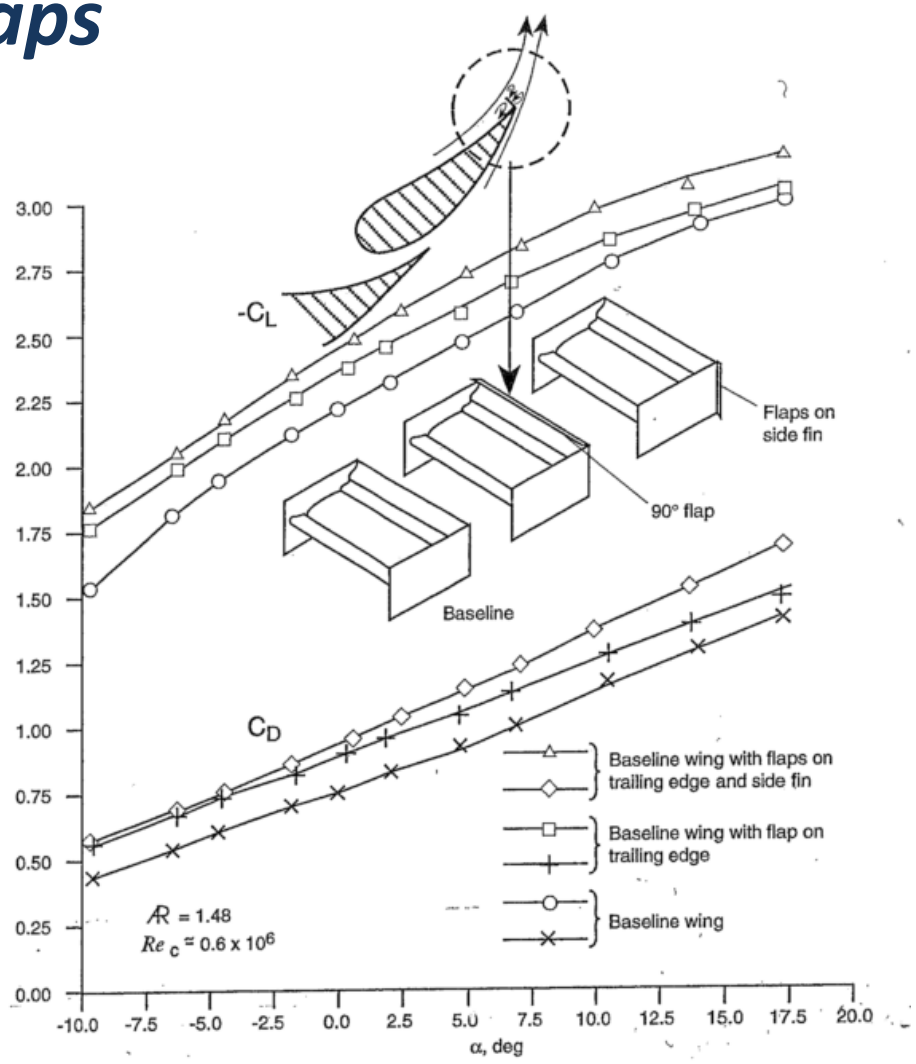
The flap has to be small enough to avoid compromising the overall shape of the wing.





Aerodynamics
Wings

Gurney Flaps



Aerodynamics

Wings

Gurney Flaps

The increase in lift due to a Gurney flap of height h, for an aerofoil with chord length c, can be approximated as

$$\Delta C_1 = 3.8 \times \left(\frac{h}{c}\right)^{0.5}$$

Aerodynamics
Wings

Example

In the previous example the wing was shown to produce 796N of lift. What would the same wing produce if it sported endplates of height 150mm and chord 300mm?

Wing span = 1500 mm
Aerofoil Chord = 300 mm
Aerofoil Coefficient of lift = 2
Velocity = 45 m/s
Air density = 1.223 kg/m^3

$$AR = \frac{b}{c} = \frac{1500}{300} = 5$$

$$AR_{effective} = AR_{actual} \times (1 + 1.9\frac{h}{b}) = 5 \times (1 + 1.9\frac{150}{1500})$$

$$AR_{effective} = 5.95$$

$$C_L = \frac{C_1}{1 + \frac{2}{AR}} = \frac{2}{1 + \frac{2}{5.95}} = 1.49$$

$$L = \frac{1}{2}\rho V^2 \times C_L \times c \times b = 0.5 \times 1.223 \times 45^2 \times 1.49 \times 0.3 \times 1.5 = 834N$$

Aerodynamics
Wings

Example

If the same wing was no longer run in free stream, but close to the ground (h=60mm) what happens to the downforce?

Wing span = 1500 mm

Aerofoil Chord = 300 mm

Aerofoil Coefficient of lift = 2

Velocity = 45 m/s

Air density = 1.223 kg/m^3

$$C_1(ige) = C_1(oge) \times \left(1 + \frac{1}{16\left(\frac{h}{c}\right)^2} \right)$$

$$C_1(ige) = 2 \times \left(1 + \frac{1}{16\left(\frac{60}{300}\right)^2} \right) = 5.125$$

$$C_L = \frac{C_1}{1 + \frac{2}{AR}} = \frac{5.125}{1 + \frac{2}{5.95}} = 3.84$$

$$L = \frac{1}{2}\rho V^2 \times C_L \times c \times b = 0.5 \times 1.223 \times 45^2 \times 4.253 \times 0.3 \times 1.5 = 2137N$$

Aerodynamics
Wings

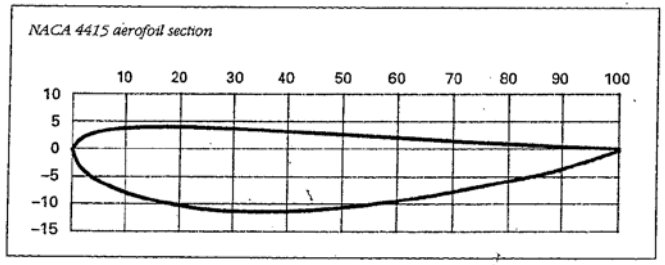
Exercise

Wing span = 1500 mm

Aerofoil Chord = 300 mm

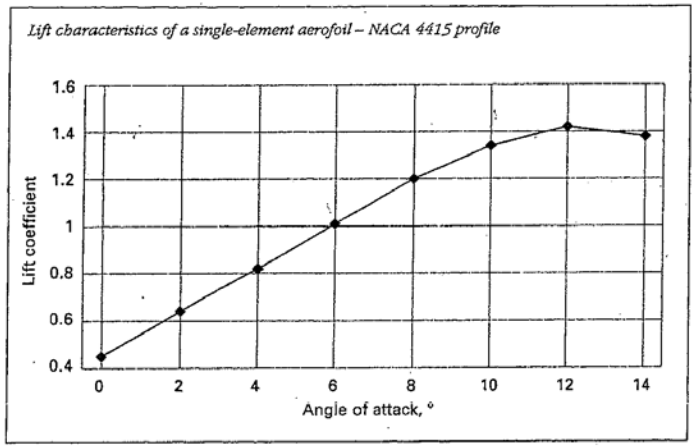
Angle of attack = 8˚

Air density = 1.223 kg/m^3



Questions:

- Calculate the co-efficient of lift of this wing out of ground effect and hence its Downforce at 150 kph with and without end plates with a height of 150mm.
- If a 5mm Gurney flap is added to the wing with end plates determine the new C_L and hence Downforce
- Assume now the wing with end plates is installed at the front of a vehicle 60mm above ground. Determine the new CL and Downforce generated.



Aerodynamics
Wings

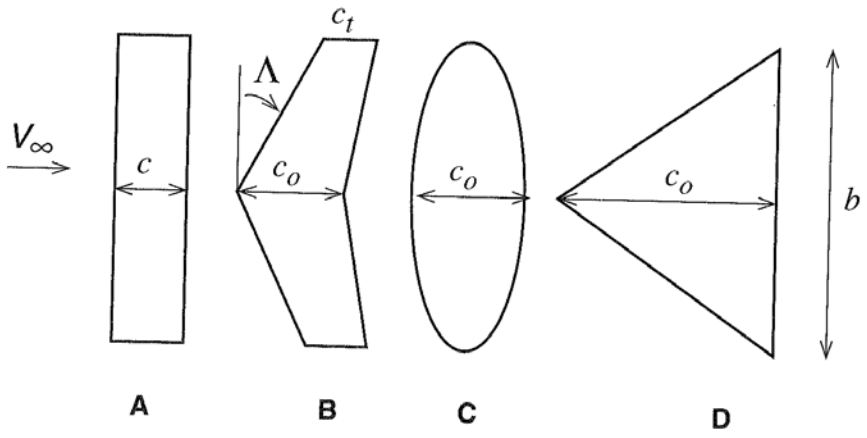
Planform Shapes

- Wing usually identifiable by aerofoil cross section and planform shape
- We've looked at influence of aerofoil shape, now lets consider planform shape
- Actual definition of aspect ratio is $AR = \frac{b^2}{S}$

  where b is the span and S is the wing area (b x c)
- For rectangular wing: $AR = \frac{b^2}{b \times c} = \frac{b}{c}$

Aerodynamics
Wings

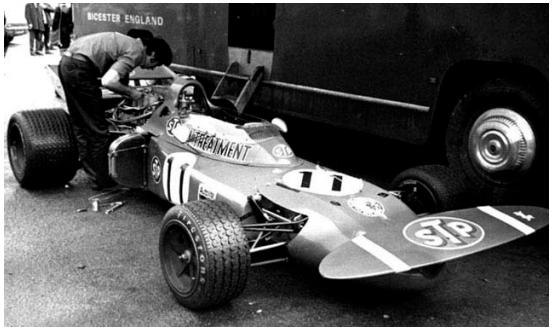
Planform Shapes

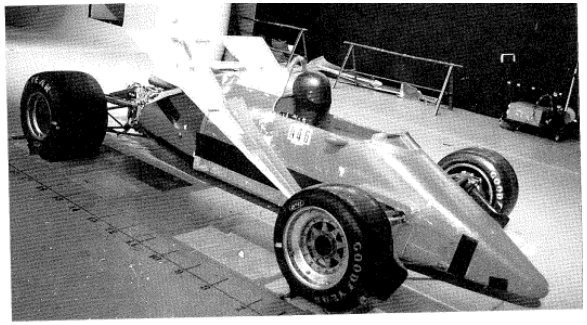


- Most common planform shapes
- The aspect ratio of each of these wings is still

$$AR = \frac{b^2}{S}$$

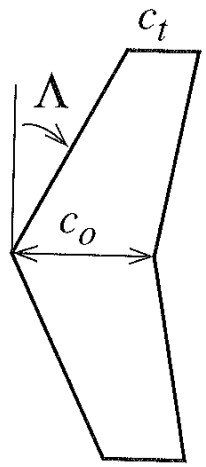
- Wing may be:
  - Rectangular
  - Swept
  - Elliptical
  - Delta





Aerodynamics
Wings

Planform Shapes



B

- Swept wings may have a wing tip chord, C_t, which is less than the root chord C_0.
- The taper ratio is simply the ratio of tip to root chord lengths.

$$\lambda = \frac{C_t}{C_0}$$

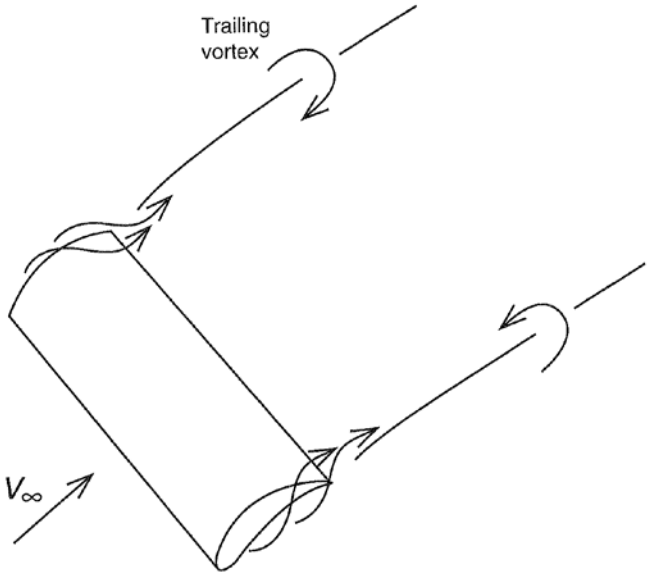
- Delta wings as found on high speed aircraft have a taper ratio of zero



Aerodynamics
Wings

Effect of planform shapes

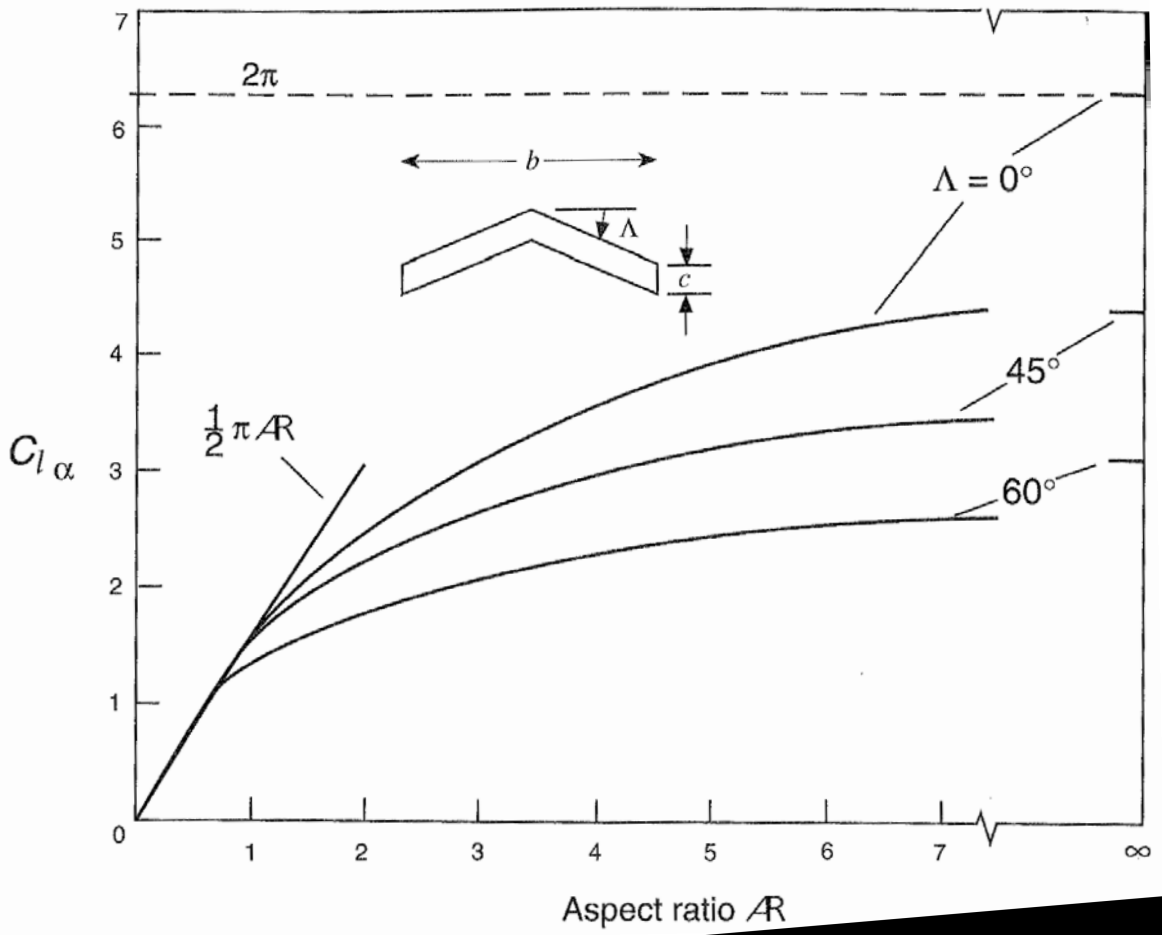
- Planform shape can affect the influence of wing tip vortices
- Small aspect ratio wings → large influence of vortices
- Large aspect ratio → small influence of vortices
- Swept wings actually reduce the coefficient of lift
- Swept wings also alter the spanwise loading



Aerodynamics
Wings

Swept wings

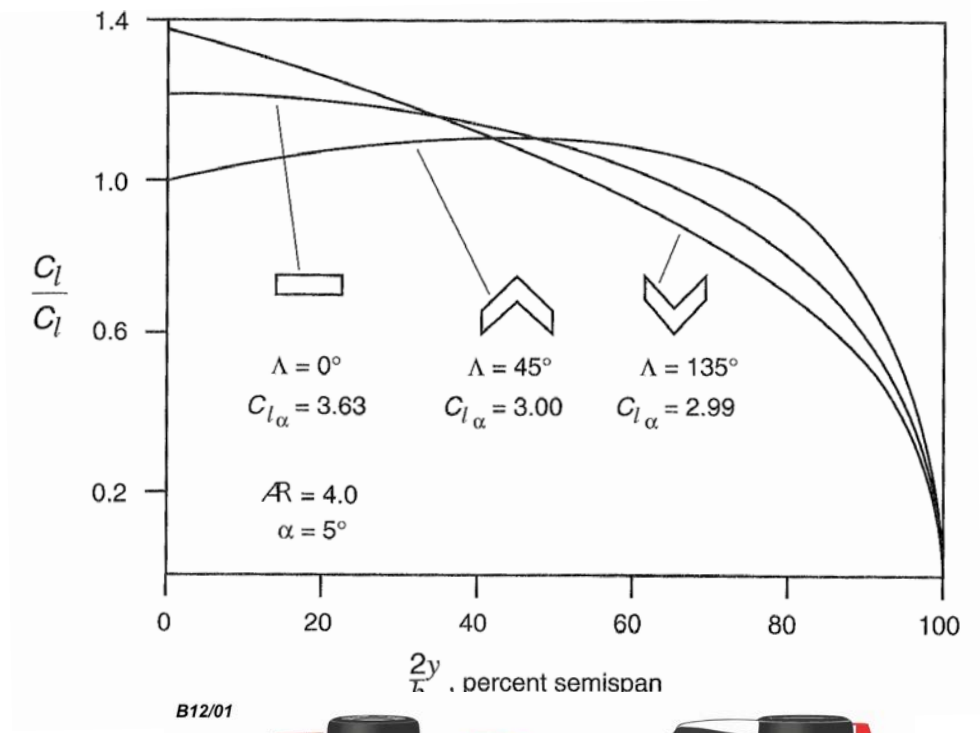
- Note: Maximum lift is for rectangular aerofoil.
- Coefficient of lift decreases as sweep angle increases

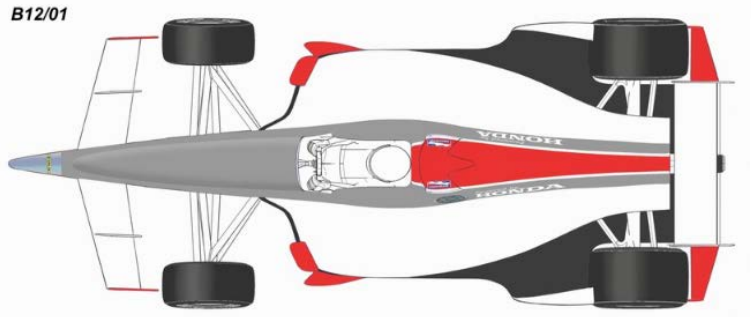


Aerodynamics
Wings

Span wise loading

- Swept wings alter the span wise loading
- **Unswept wing →**
  Maximum lift load at centre→
  Lift at tips =0
- **Swept forward wing →**
  Larger lift near root and less near tips
- **Swept backward wing →**
  Larger lift near tips

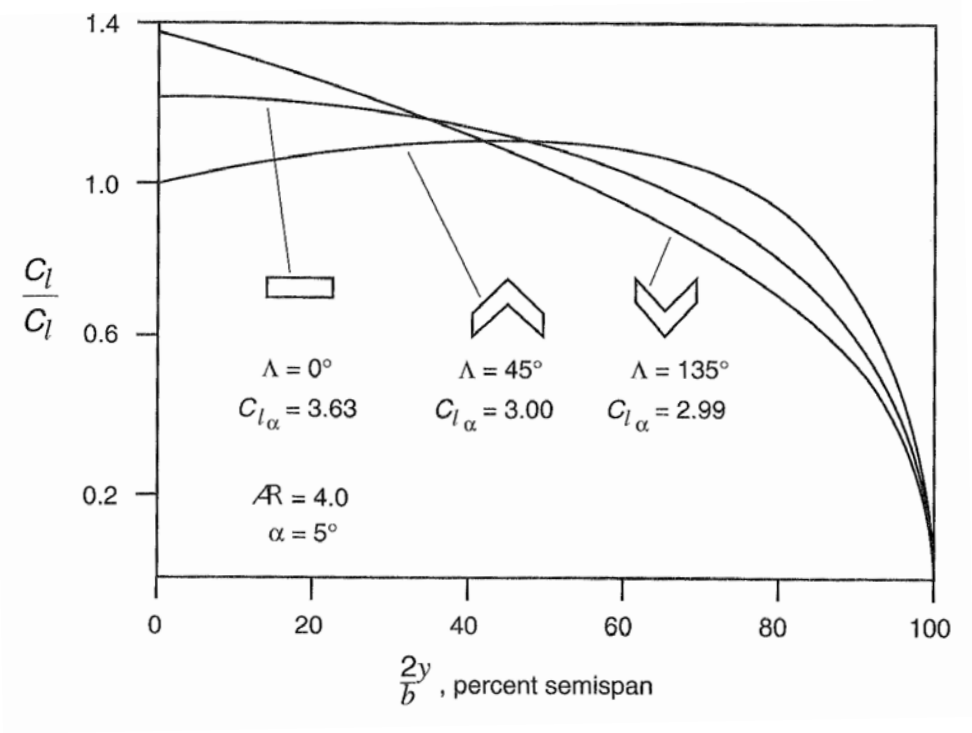




Aerodynamics
Wings

Span wise loading

- From a structural point of view
  - Root bending moments smaller for forward sweep
  - Forward sweep wings not used much as forward swept wings can become aero elastically unstable.

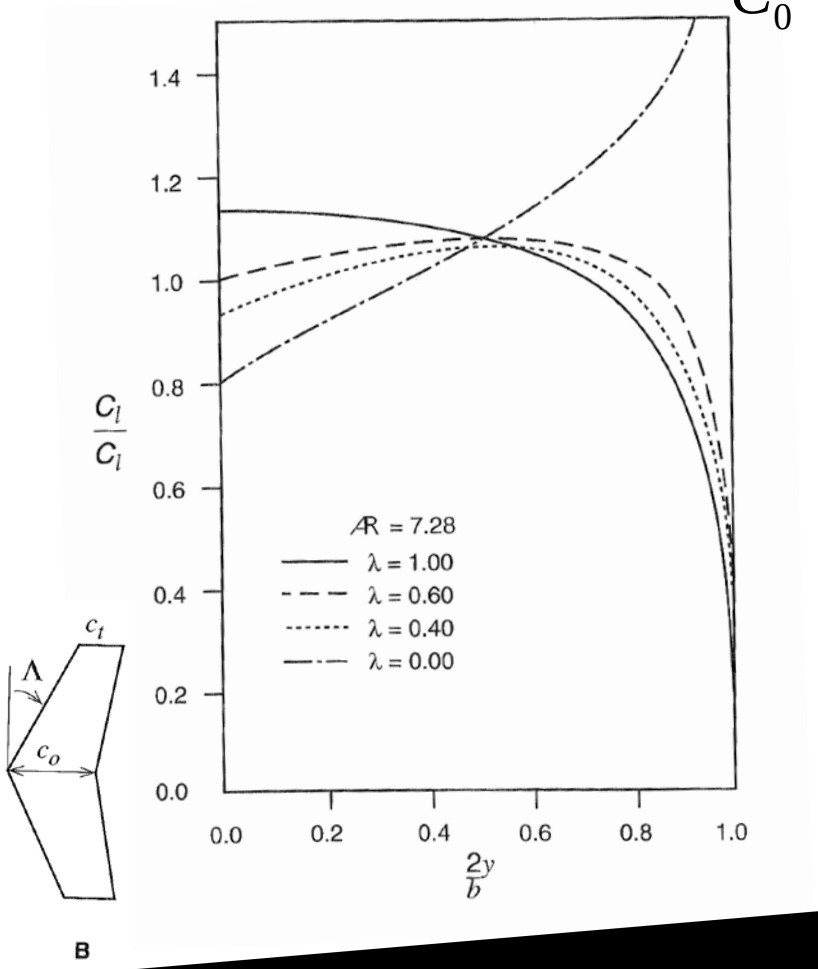


Aerodynamics
Wings

Span wise loading

$$\lambda = \frac{C_t}{C_0}$$

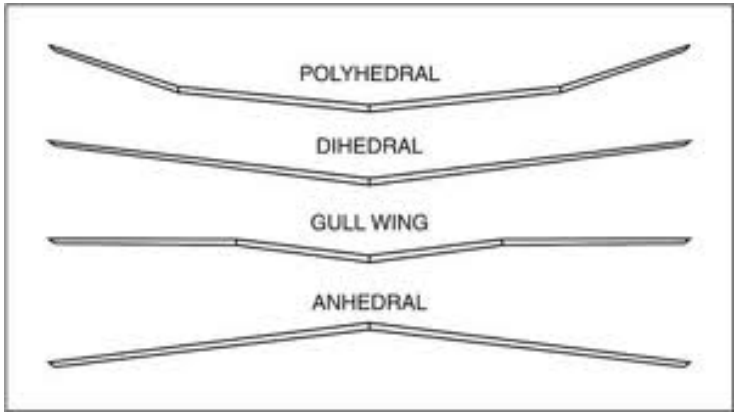
- Taper ratio also affects spanwise loading
- Local lift increases towards tip with decreasing taper ratio
- For very small taper ratios tip has a tendancy to stall
- Often wing is twisted to have smaller AoA at tip than root

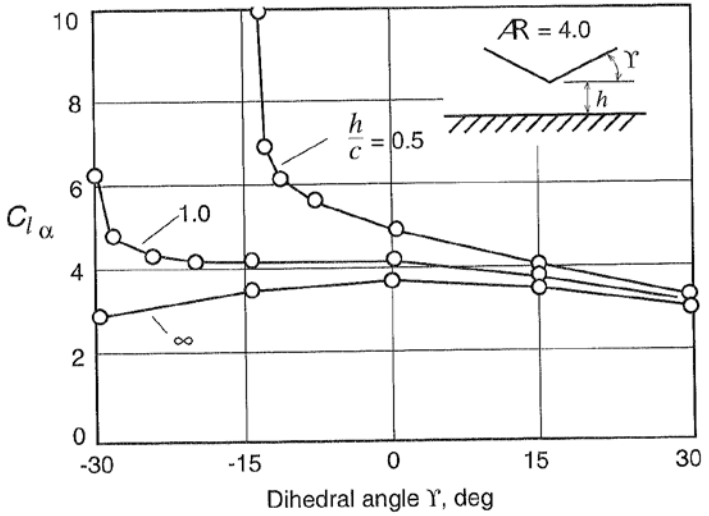


Aerodynamics
Wings

Dihedral

- The effect of wing dihedral far from the ground is similar to sweep
- Lift reduces with increasing dihederal
- This affect continues near ground for positive dihedrals
- Negative dihedrals (anhedral)
  - Increase in lift of the portion near ground is largely affected by ground effect
  - Wing tips close to ground → harder for air to slip over tips

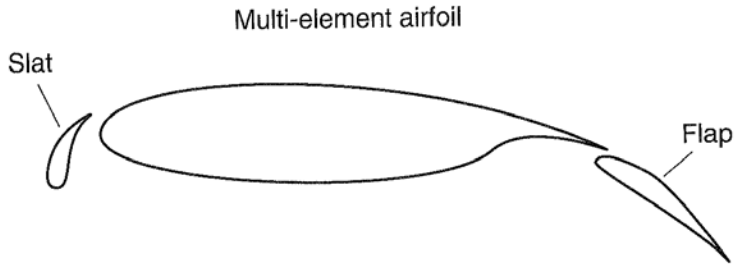




Aerodynamics
Wings

High Lift Wings

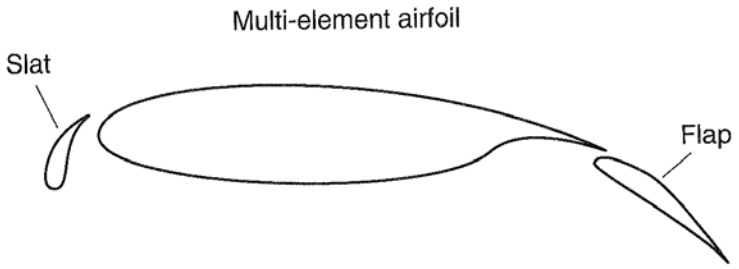
- Wings on vehicles generally designed for maximum lift
- Options include
  - Increase wing are
  - Increase camber
  - Delay flow separation by slotted flap to allow higher AoA
- Wing planform area often limited by regulations
- Multi element aerofoils used allot for maximum Downforce as they have slots to delay separation

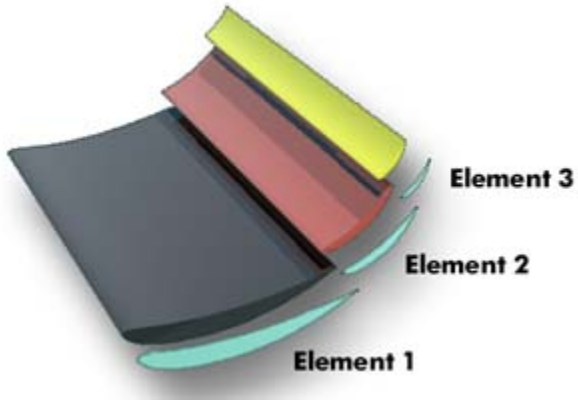


Aerodynamics
Wings

High Lift Wings

- Principle purpose behind multi-element design
  - Aerofoil camber can be increased far more than with a single element aerofoil
  - Separation delayed as boundary layer is re-energised
  - Can produce favourable interaction between wing elements resulting in combined gain in lift.

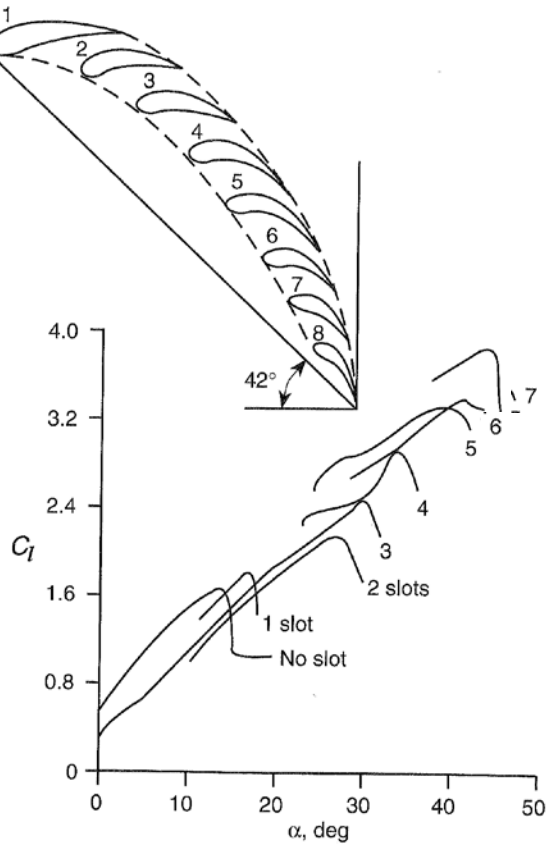




Aerodynamics
Wings

High Lift Wings

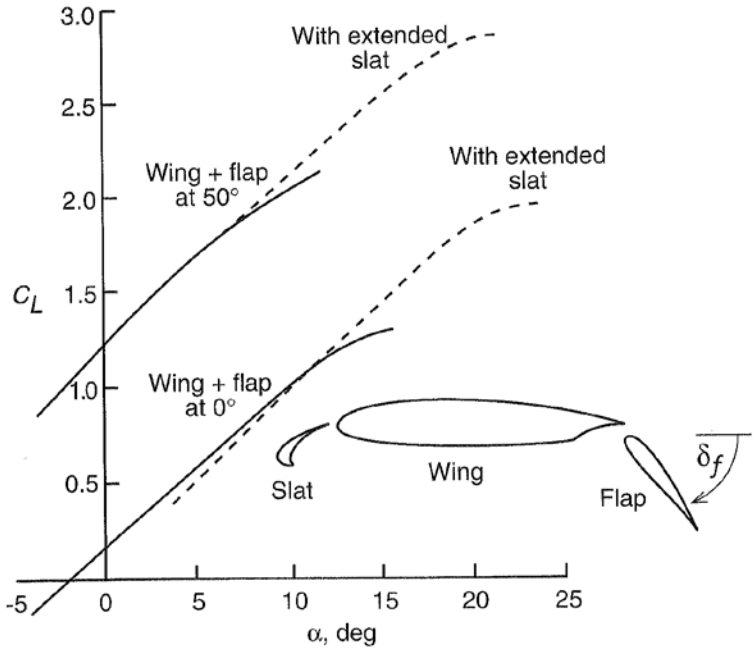
- Advantages of multi element wing realised by Handley Page
- Showed more elements = higher lift coefficient
- He broke up a RAF 19 aerofoil into different numbers of elements
- More elements mean higher AoA can be achieved prior to stall
- Higher AoA → increased lift



Aerodynamics
Wings

High Lift Wings

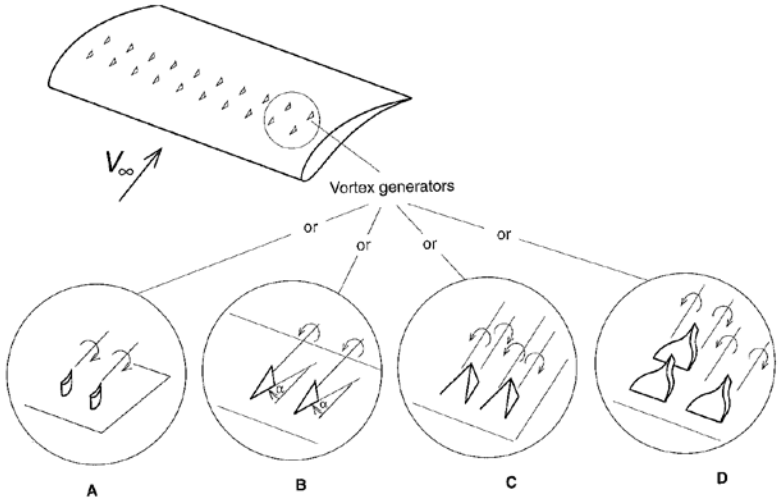
- Wings with slats and a flap is basically a special case of a 3 element wing.
- The slat results in a drooped leading edge which helps delay seperation
- Slat doesn't increase overall lift
- Slat allows higher AoA prior to stall
- Increased flap angle results in increased camber and hence lift.



Aerodynamics
Wings

Wing enhancements

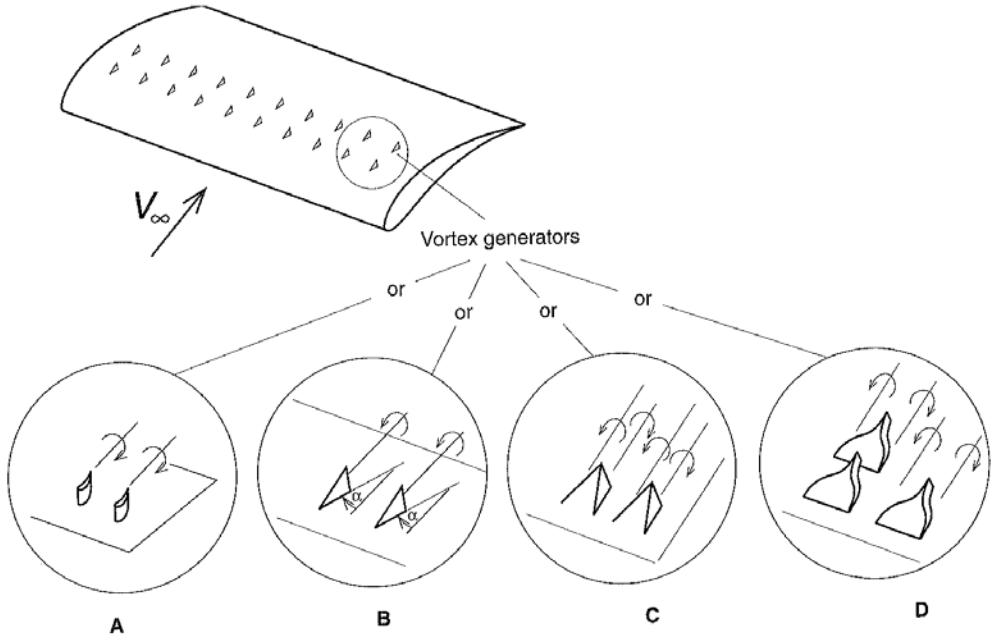
- Active devices such as jets etc banned as active devices in motorsport
- So we will look at passive enhancements
- Best know add-on is a **vortex generator**
- These may resemble small wings or more complex geometries as shown



Aerodynamics
Wings

Wing enhancements

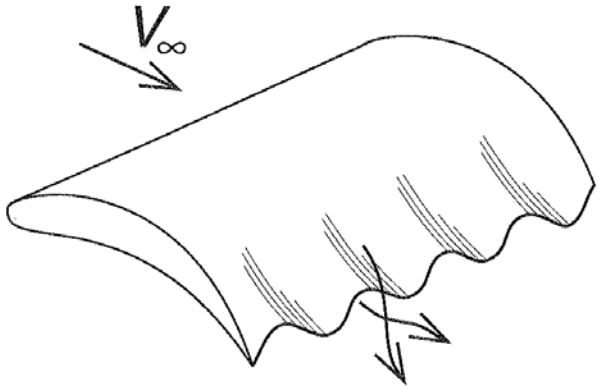
- Typically the vortex generator is a little taller than the B.L thickness
- Swirl vortices induced help add fresh momentum to the boundary layer from free stream
- Can delay seperation
- Overall effect
  - Increase lift
  - Decrease drag



Aerodynamics
Wings

Wing enhancements

- Similar advantages from wavy trailing edge device
- Mixes high pressure flow with low pressure flow at trailing edge
- Mixing reduces trailing edge separation
- Delayed separation → increased lift



Aerodynamics
Wings

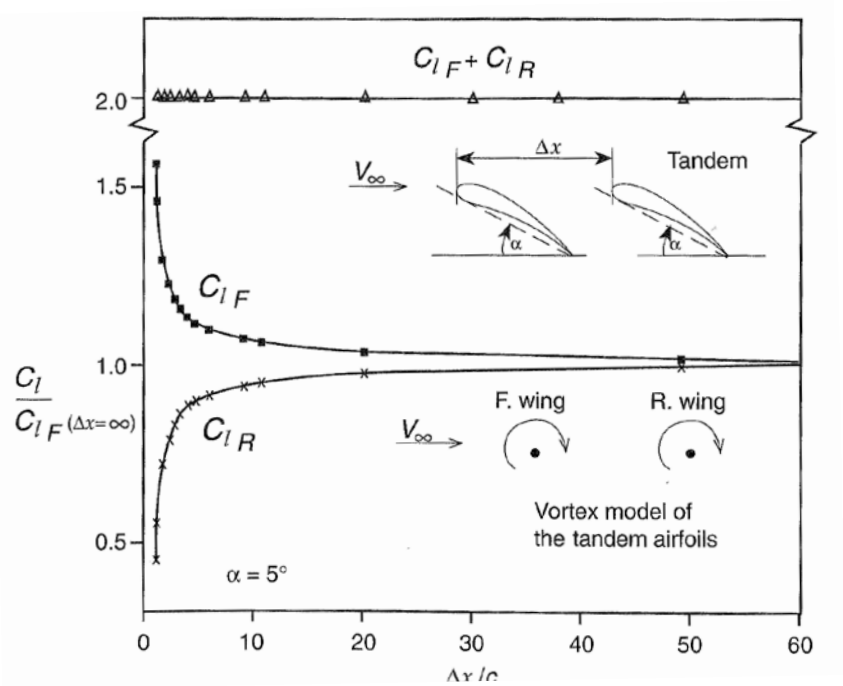
Lifting surface interaction

- When 2 aerofoils are in close proximity they affect on another's lift.

- Tandem –
  - Represent 2 aerofoils by 2 vortices

  - As Δx is decreased
    - Aft vortex induces upwash on front wing increasing lift
    - Front vortex induces downwash on rear wing decreasing lift

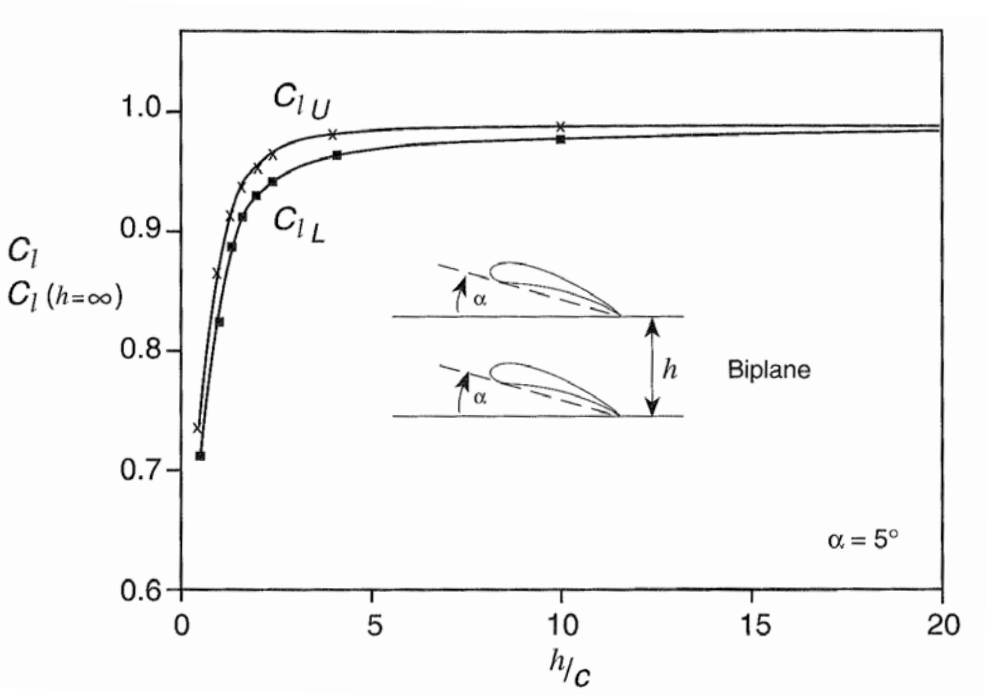
  - Combined lift remains constant



Aerodynamics
Wings

Lifting surface interactions

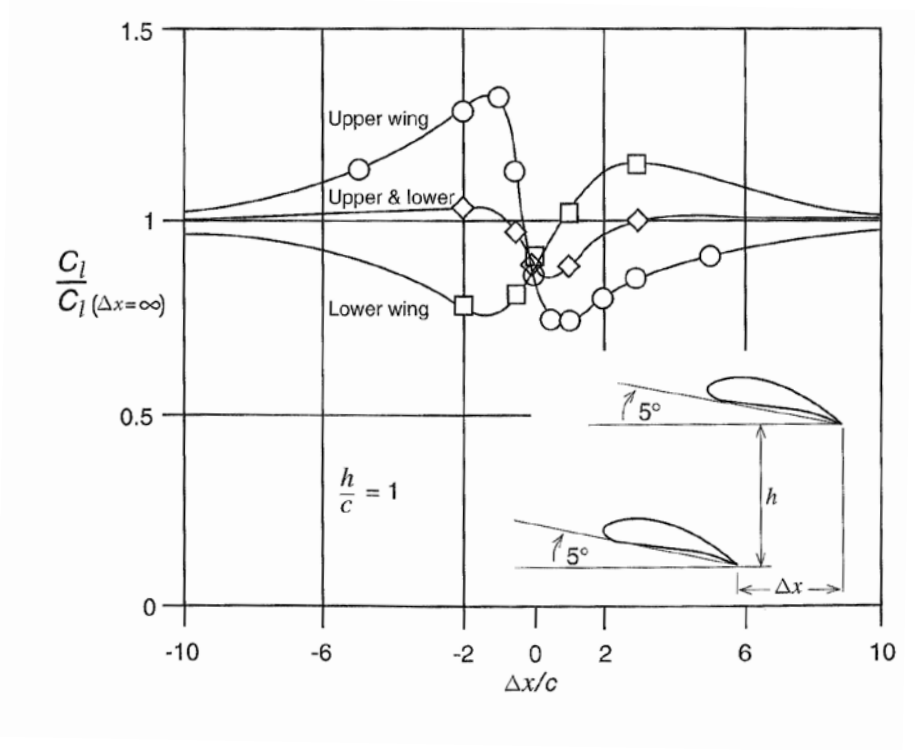
- Vertical separation (bi-plane)
  - Lift is reduced on both aerofoils
  - Combined lift therefore decreases
  - Seen in practice – Why?
    - Benefits normally result from interaction with vehicle body
    - (top element in free stream)



Aerodynamics
Wings

Lifting surface interactions

- Staggered separation
  - 2 vortex model again applies
  - Upper foil ahead of lower
    - Upper elements lift increased by upwash from lower element
    - Lower foils lift decreased by downwash of upper element
  - Opposite is true when lower element leads upper.

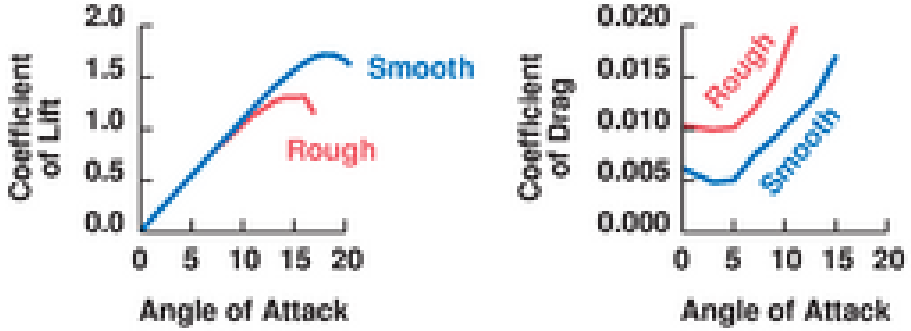


Aerodynamics
Wings

Surface Roughness of Wings

Bernoulli's principle cannot be trusted when frictional forces are at work.

Rough surfaces tend to de-energize the boundary layer, leading to separation which produces early stalling



Note at moderate and low angles of attack surface roughness has minimal any effect on the coefficient of lift.

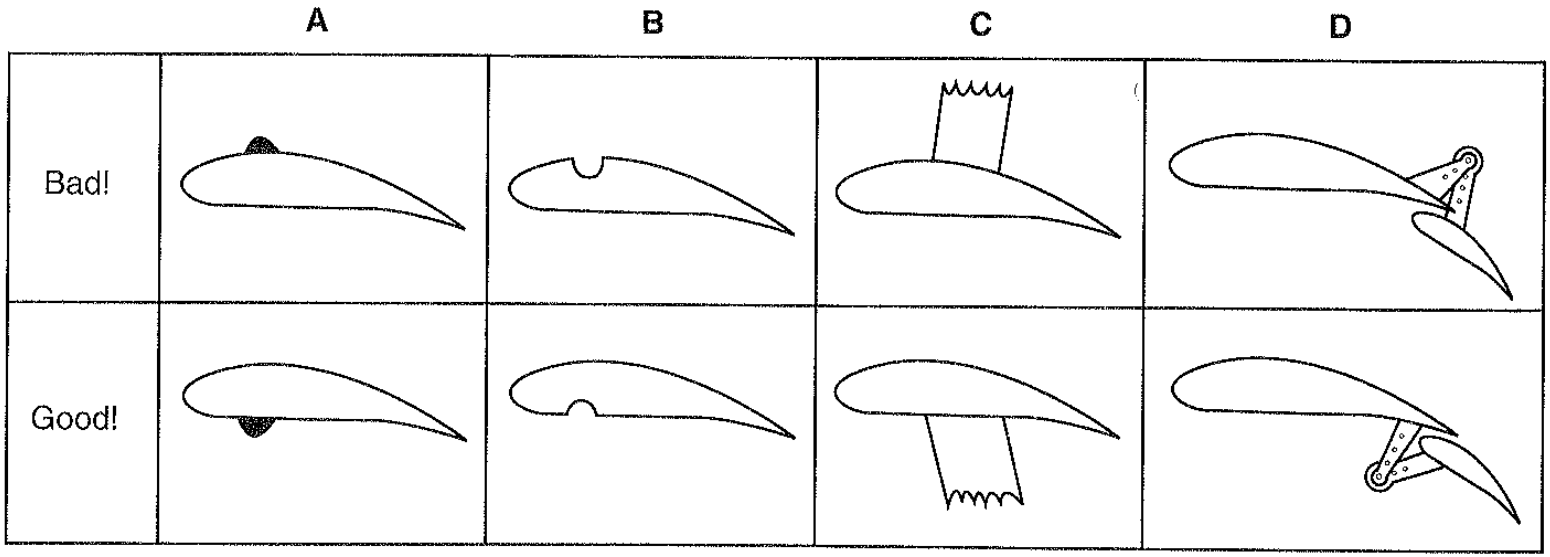
Note also that drag is increased at all angles of attack

Aerodynamics
Wings

Non aerodynamic parts on wings

– E.g Cameras, braces, imperfections on wing surface etc

- High velocity surface more susceptible/sensitive to separation
- Non aerodynamic parts should be mounted (if necessary) to low speed (high pressure) surface



What is Computational Fluid Dynamics?

- Fluid dynamics is the science of fluid motion.
- Fluid flow is commonly studied in one of three ways:

1. Simplifications of the governing equations
   → Theoretical Fluid Dynamics

2. Experiments on scale models
   → Experimental Fluid Dynamics

3. Discretize governing equations and solve by computers
   → Computational Fluid Dynamics

Aerodynamics
CFD

What is Computational Fluid Dynamics?

- Definitions of CFD

  "Computational fluid dynamics (CFD) is the use of computers and numerical techniques to solve problems involving fluid flow"

  "CFD is the simulation of fluids engineering system using modeling and numerical methods"

- *Has also been called*
  - *Cheats Frauds and Deceivers*
  - *Colourful Fluid Dynamics*
  - *Due to CFD being miss-sold to commercial companies who tried to utilise CFD without the necessary skills.*

Aerodynamics
CFD

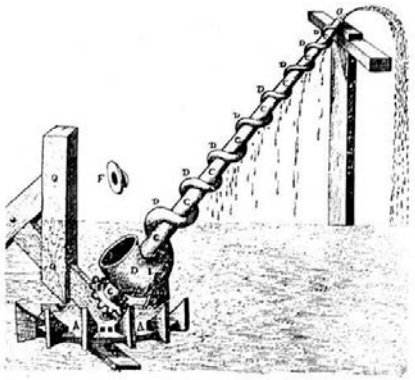
Where is CFD used?

- CFD has been successfully applied in a huge number of areas in engineering;

- **aerodynamics** (aircraft; automobiles);
- hydrodynamics (ships; submersibles);
- **engine flows** (internal-combustion and jet engines);
- turbomachinery;
- **combustion and heat transfer;**
- process engineering (chemical industry);
- wind and wave power;
- wind loading (forces and dynamic response of structures);
- hydraulics (pipe networks and channels; hydraulics structures);
- sediment transport (sediment load; scour; morphology);
- electronics (heat dissipation).
- And many more!

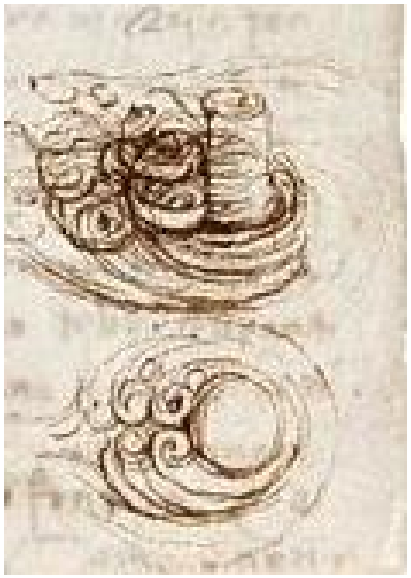
Aerodynamics
CFD

History of fluid dynamics and CFD?



Archimedes - Greece (287-212 BC).

- He initiated the fields of static mechanics, hydrostatics, and pycnometry (how to measure densities and volumes of objects).



Leonardo da Vinci - Italy (1452-1519)

- Leonardo set out to observe all natural phenomena in the visible world, recognizing their form and structure, and describing them pictorially.
- His contributions to fluid mechanics are presented in a nine part treatise that covers the water surface, movement of water, water waves, eddies, falling water, free jets, interference of waves, and many other newly observed phenomena.

Aerodynamics
CFD

History of fluid dynamics and CFD?

Isaac Newton - England (1643-1727)



- One of the most important figures in science.
- Most well known for his three laws of motion.
- His key contributions to fluid mechanics include:
  - The second law: **F=m.a.**
  - The concept of **Newtonian viscosity** in which stress and the rate of strain vary linearly.
  - The reciprocity principle: the force applied upon a stationary object by a moving fluid is equal to the change in momentum of the fluid as it deflects around the front of the object.
  - Relationship between the speed of waves at a liquid surface and the wavelength.

History of fluid dynamics and CFD?

18th and 19th century

- During this period, significant work was done trying to mathematically describe the motion of fluids.

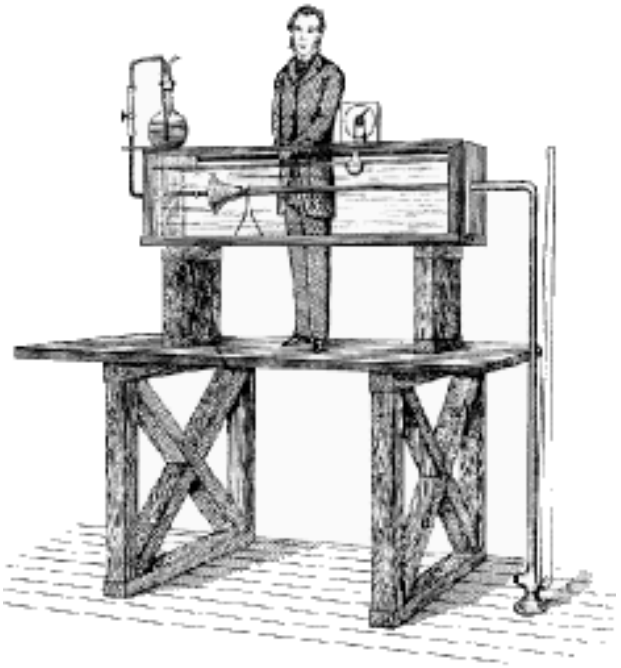
- **Daniel Bernoulli** (1700-1782) derived Bernoulli's equation.
- **Leonhard Euler** (1707-1783) proposed the Euler equations, which describe conservation of momentum for an **inviscid fluid**, and conservation of mass. He also proposed the velocity potential theory.
- Claude Louis Marie Henry **Navier** (1785-1836) and George Gabriel **Stokes** (1819-1903) introduced **viscous** transport into the Euler equations, which resulted in the Navier-Stokes equation. This forms the basis of modern day CFD.
- Other key figures were Jean Le Rond d'Alembert, Siméon-Denis Poisson, Joseph Louis Lagrange, Jean Louis Marie Poiseuille, John William Rayleigh, M. Maurice Couette, and Pierre Simon de Laplace.

Aerodynamics
CFD

History of fluid dynamics and CFD?

Osborne Reynolds - England (1842-1912)

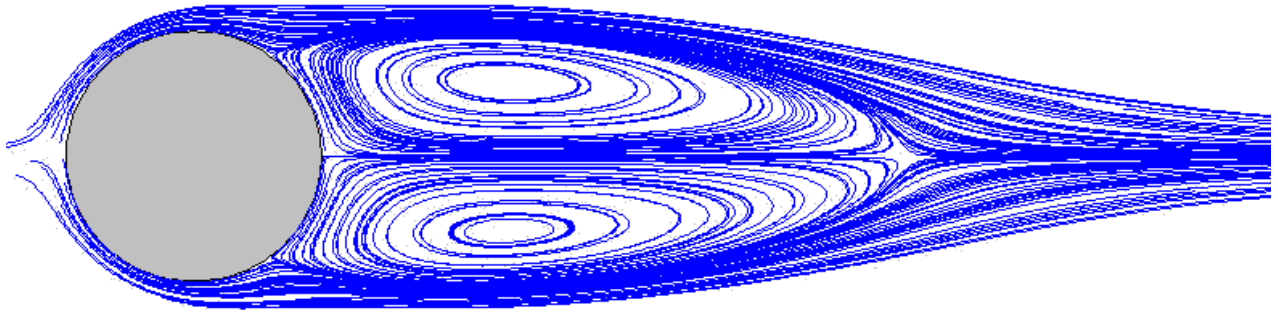
- Reynolds was a prolific writer who published almost 70 papers during his lifetime on a wide variety of science and engineering related topics.
- He is most well-known for the Reynolds number, which is the ratio between inertial and viscous forces in a fluid. This governs the transition from laminar to turbulent flow.



Aerodynamics
CFD

History of fluid dynamics and CFD?

- In the early 20th centaury much work was done on refining theories of boundary layers and turbulence.
- In 1922, Lewis Fry Richardson developed the first numerical weather prediction system. Division of space into grid cells and the finite difference approximations of Bjerknes's "primitive differential equations."
- Earliest numerical solution: for flow past a cylinder (1933).
- Kawaguti obtains a solution for flow around a cylinder, in 1953 by using a mechanical desk calculator, working 20 hours per week for 18 months, citing: "*a considerable amount of labour and endurance*."



History of fluid dynamics and CFD?

- 1960-70s
  - CFD code was continually developed primarily in academic institutions. Enhancements include better turbulence models, new meshing techniques and more accurate solvers.

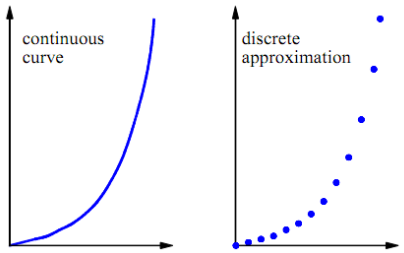
- 1980s-Present
  - Commercial CFD packages developed for industry such as;
  - Fluent (UK and US).
  - CFX (UK and Canada) now ANSYS
  - Phoenix (UK).
  - Star CD (UK).
  - Flow 3d (US).

How does CFD work?

- The fundamental elements of any CFD simulation are:

(1) The **flow field is discretised**; i.e. field variables (, *u*, *v*, *w*, *p*, …) are approximated by their values at a finite number of *points*.

(2) The **equations of fluid motion are discretised** (approximated in terms of values at points): control-volume or differential equations algebraic equations

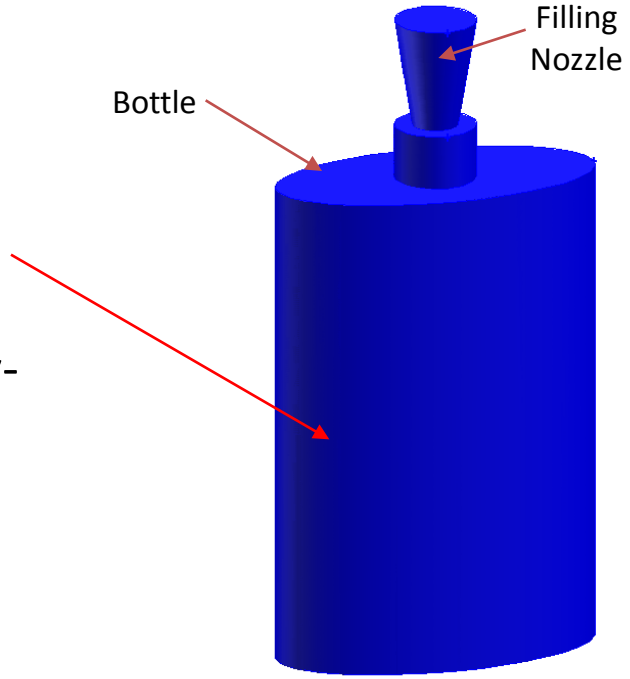


(3) The resulting **system of algebraic equations is solved simultaneously** to give values at the nodes.

Aerodynamics
CFD

How does CFD work?

- Analysis begins with a mathematical model of a physical problem.
- Conservation of matter, momentum, and energy must be satisfied throughout the region of interest.
- Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, inviscid, two-dimensional).
- Provide appropriate initial and boundary conditions for the problem.



Domain for bottle filling problem.

How does CFD work?

- The governing equations – Navier-Stokes Equations
- Basically these equations are the **conservation of energy, mass and momentum** described in a single partial differential equations for each direction

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}+\rho w\frac{\partial u}{\partial z}=-\frac{\partial \hat{p}}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right]$$

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}+\rho w\frac{\partial v}{\partial z}=-\frac{\partial \hat{p}}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right]$$

$$\rho\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}=-\frac{\partial \hat{p}}{\partial z}+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right]$$

Local acceleration

Convection

Pressure gradient

Viscous terms

Aerodynamics
CFD

How does CFD work?

- **Full set of equations:**

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0 \quad (1)$$

$$\frac{\partial \rho u}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (\rho uv)}{\partial y} + \frac{\partial (\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \nu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \quad (2)$$

$$\frac{\partial \rho v}{\partial t} + \frac{\partial (\rho uv)}{\partial x} + \frac{\partial (\rho v^2)}{\partial y} + \frac{\partial (\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \nu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \quad (3)$$

$$\frac{\partial \rho w}{\partial t} + \frac{\partial (\rho uw)}{\partial x} + \frac{\partial (\rho vw)}{\partial y} + \frac{\partial (\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z} + \nu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \quad (4)$$

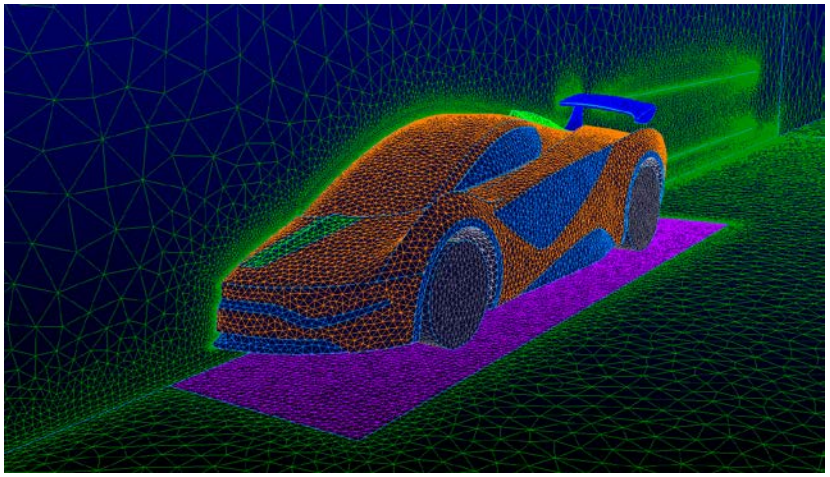
$$\frac{\partial \rho E}{\partial t} + \frac{\partial (\rho uE)}{\partial x} + \frac{\partial (\rho vE)}{\partial y} + \frac{\partial (\rho wE)}{\partial z} = -\frac{\partial pu}{\partial x} - \frac{\partial pv}{\partial y} - \frac{\partial pw}{\partial z} + S \quad (5)$$

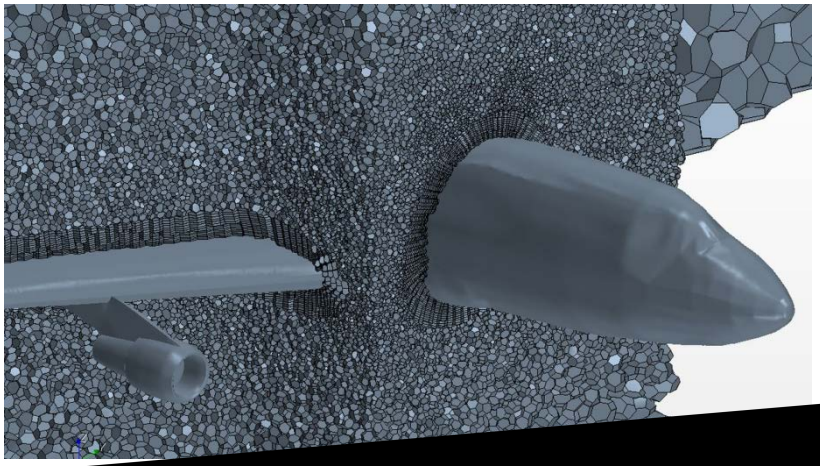
where ρ is the air density, u, v, w are the components of the air's velocity, E is measure of the air's internal energy (which allows us to compute its temperature) and p is the air pressure.

Aerodynamics
CFD

How does CFD work? Discretization

- CFD applies numerical methods to develop approximations of the Navier-Stokes equations in the fluid region of interest.
  - Governing differential equations discretised→ simpler algebraic equations.
  - Volume of interest is discretised into number of finite volumes by applying a grid or **mesh**.
  - The governing equations are formed at the **centroid** of each finite volume (finite volume method) or node (finite difference/element method)
  - System of equations are solved simultaneously to provide solution. – COMPUTER NUMBER CRUNCHING!

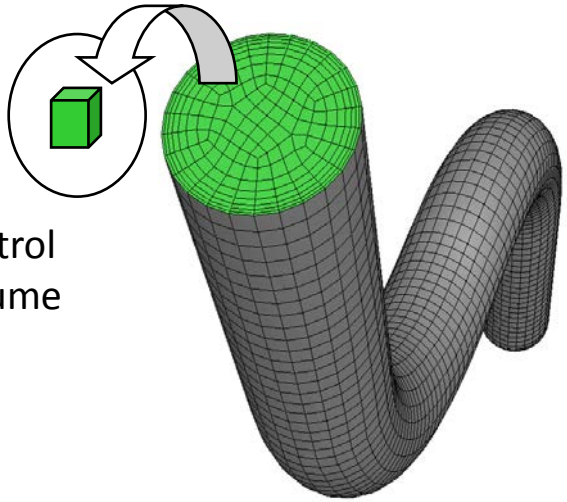




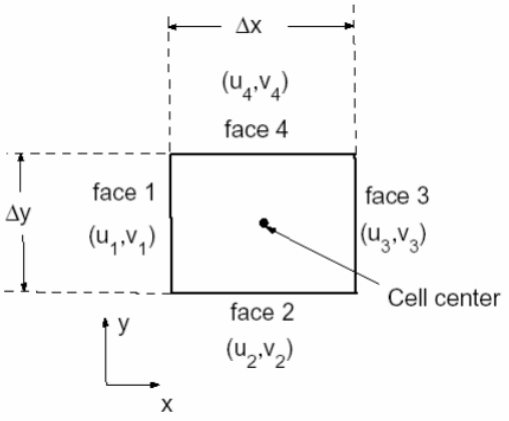
Aerodynamics
CFD

How does CFD work? Discretization

- Equations solved at centroid (centre) of each control volume (element).
- Fluxes calculated at each face so velocity and pressure information is passed onto next control volume.

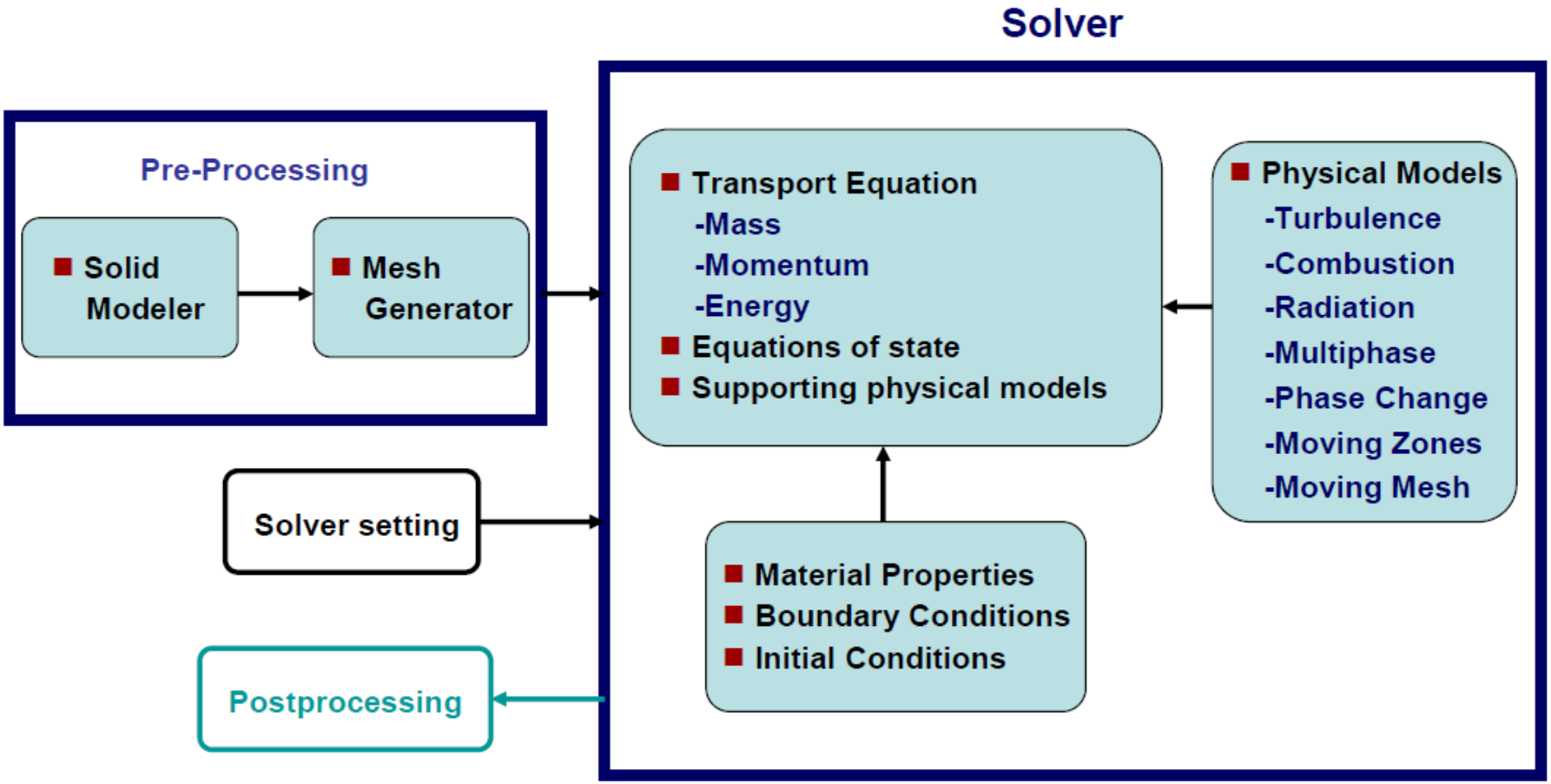


Fluid region of pipe flow discretized into finite set of control volumes (mesh).



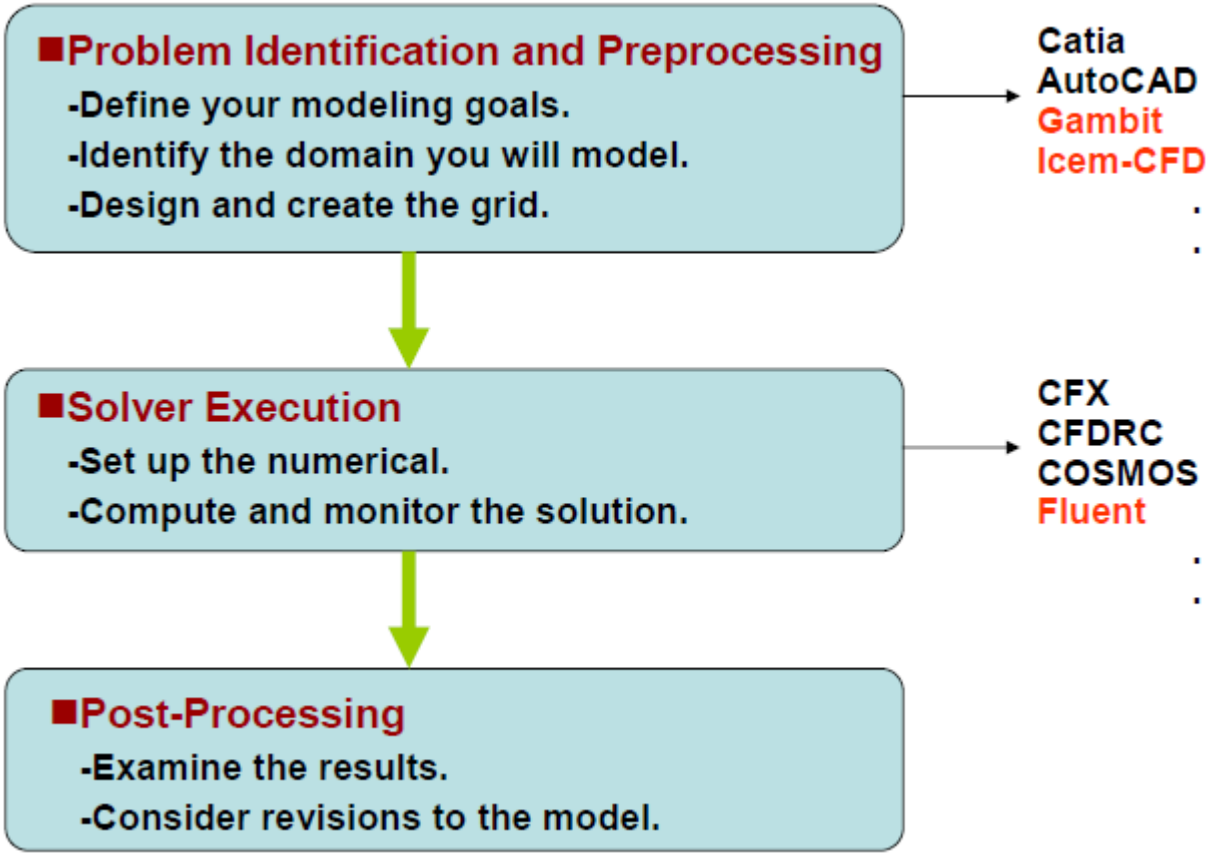
Aerodynamics
CFD

Stages in a CFD Simulation



Aerodynamics
CFD

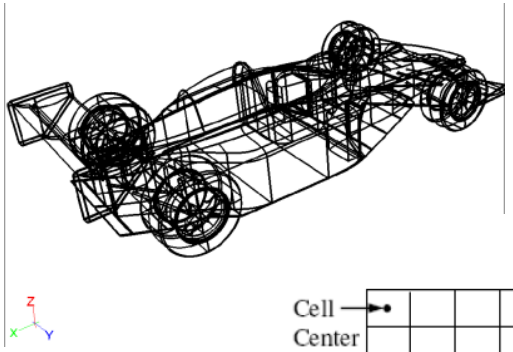
Stages in a CFD Simulation

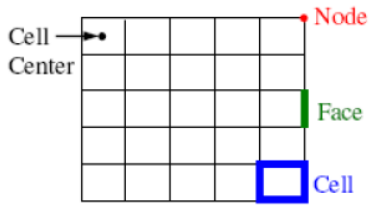


Aerodynamics
CFD

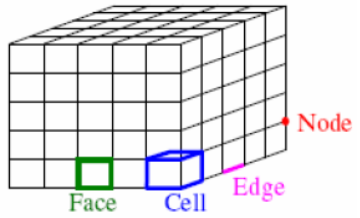
Setting up the model (Pre-processing) – Generating the mesh



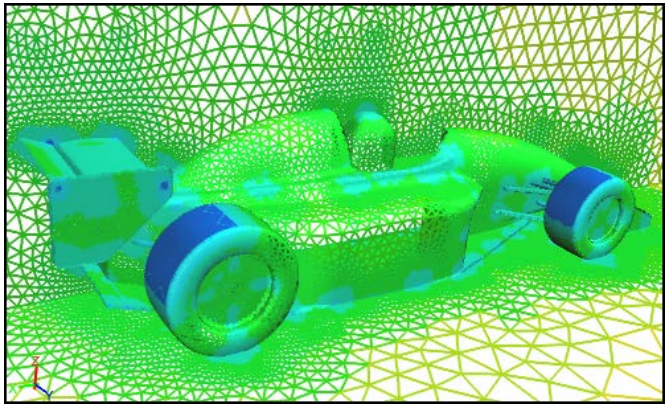


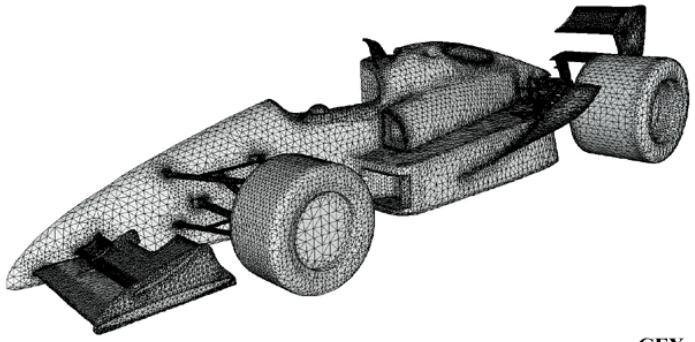


Simple 2D Mesh



Simple 3D mesh



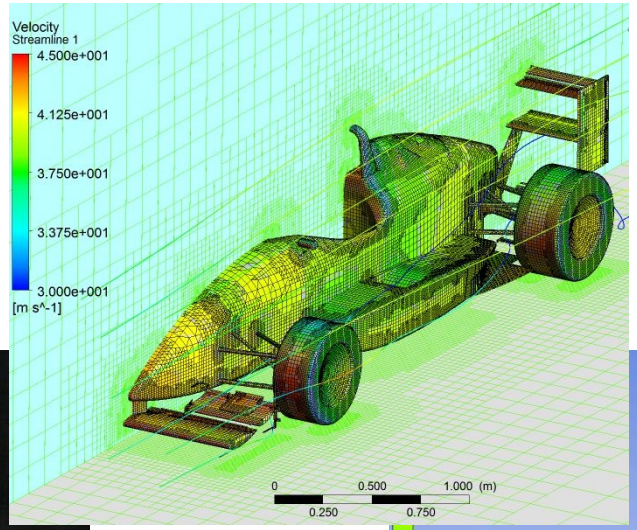

CFX

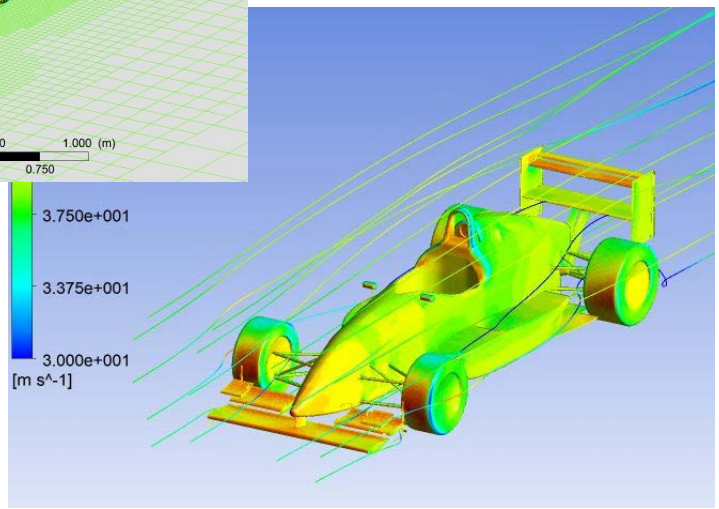
Aerodynamics

CFD

Setting up the model (Pre-processing) – Generating the mesh

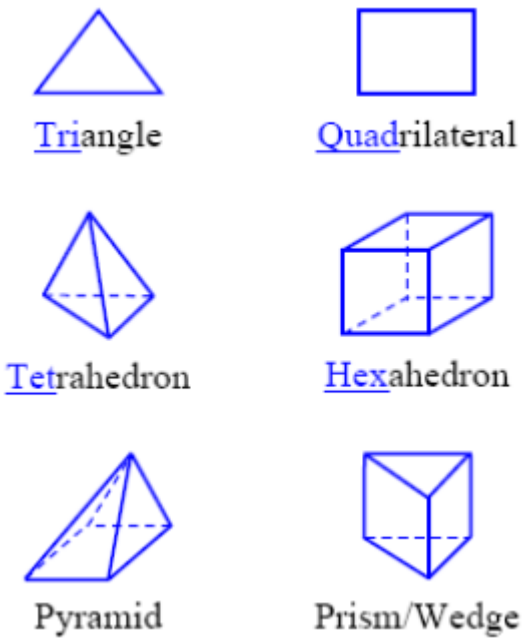






Mesh Considerations

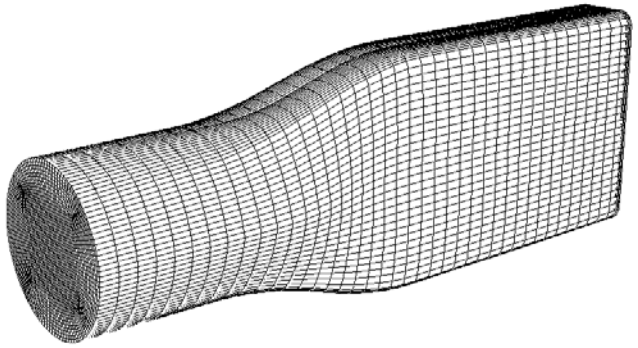
- What shape should the control volumes (elements) be? Should you use a quad/hex grid, a tri/tet grid, a hybrid grid?
- What degree of grid resolution is required in each region of the domain? (Needs to be fine enough to capture local flow effects)
- How many cells are required for the problem? (More cells = longer computation time)
- Will you use adaption to add resolution? (Moving/changing mesh)
- Do you have sufficient computer memory?



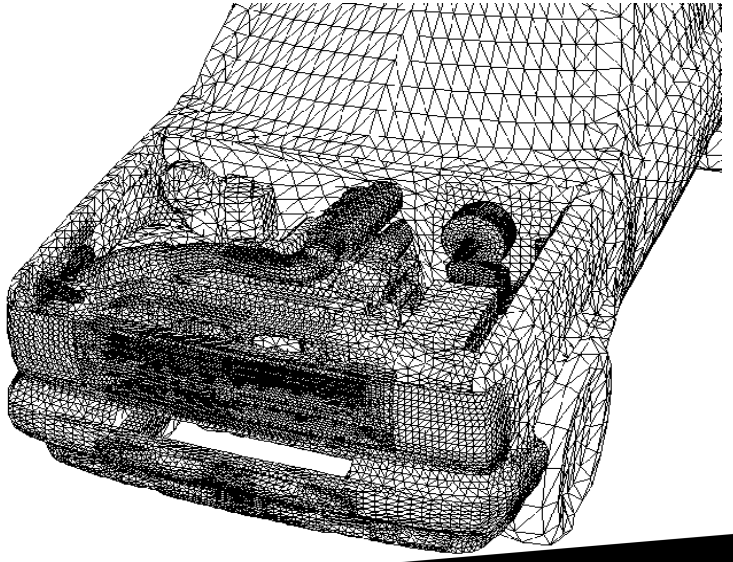
Aerodynamics
CFD

Mesh Considerations

- The two most commonly used element types are the tetrahedral and hexahedron elements.

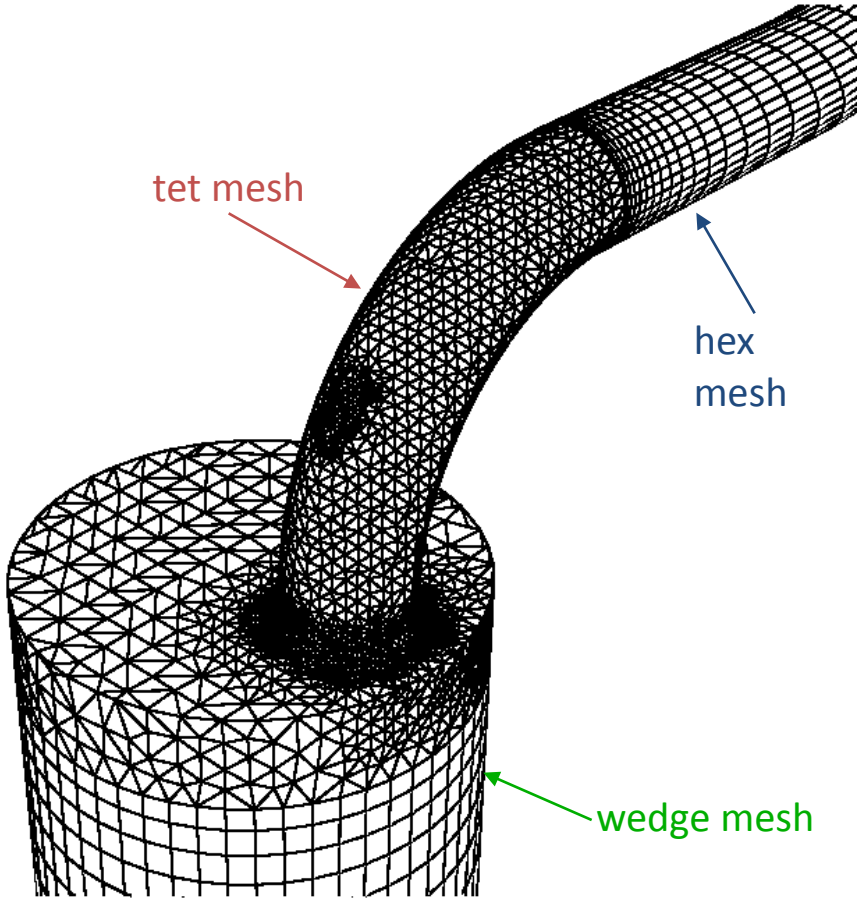


- But which one should you use?
  - For simple geometries, quad/hex meshes can provide high-quality solutions with fewer cells than a comparable tri/tet mesh.
  - For complex geometries, quad/hex meshes show no numerical advantage, and you can save meshing effort by using a tri/tet mesh



Mesh Considerations

- You may also employ both element types i different regions of the flow field to benefi from both element types advantages
- Specific regions can be meshed with different cell types.
- Both efficiency and accuracy are enhanced relative to a hexahedral or tetrahedral mesh alone.
- See valve port mesh.

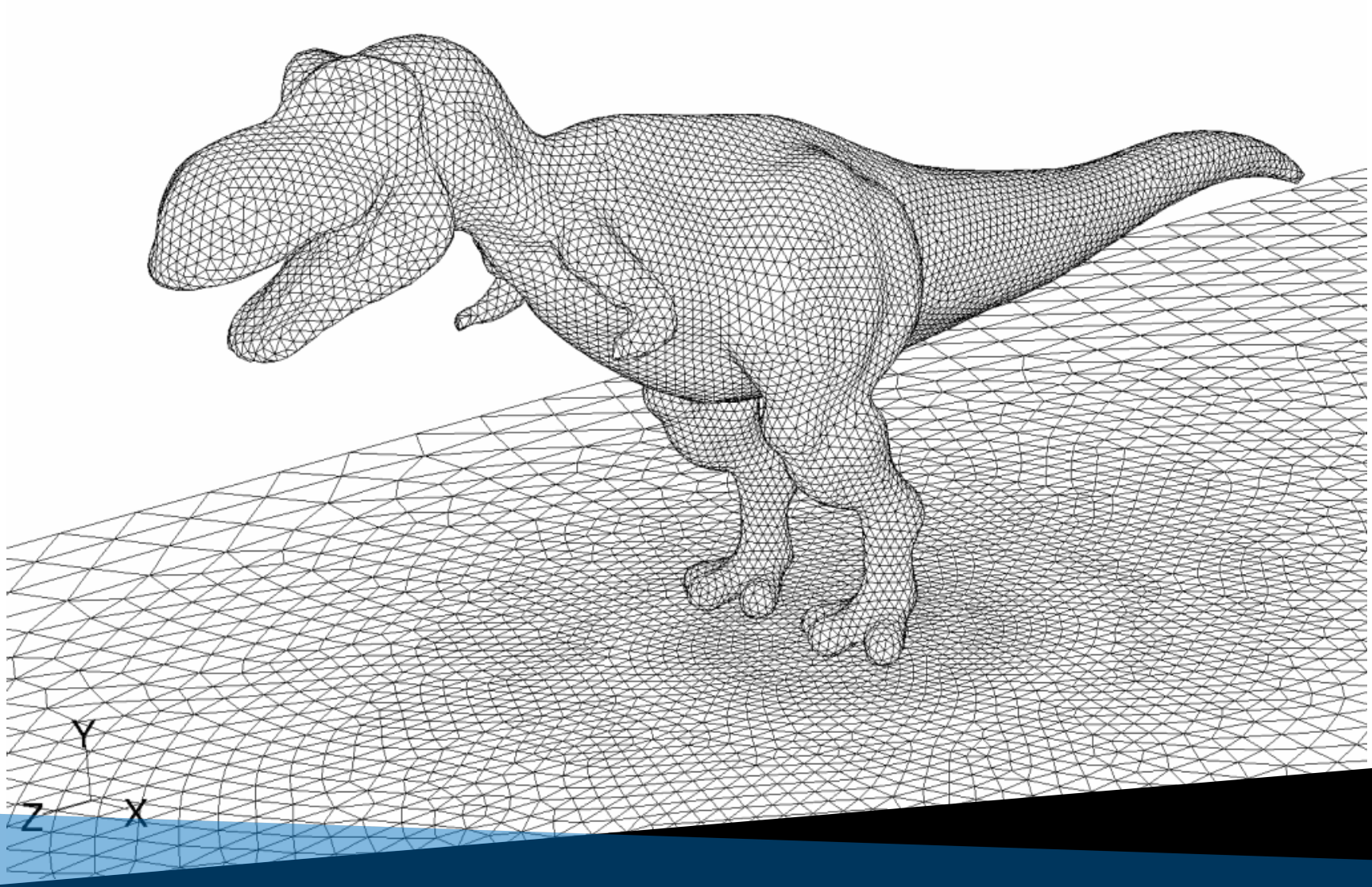


Hybrid mesh for an
IC engine valve port

Aerodynamics

CFD

A Complex 3D Mesh



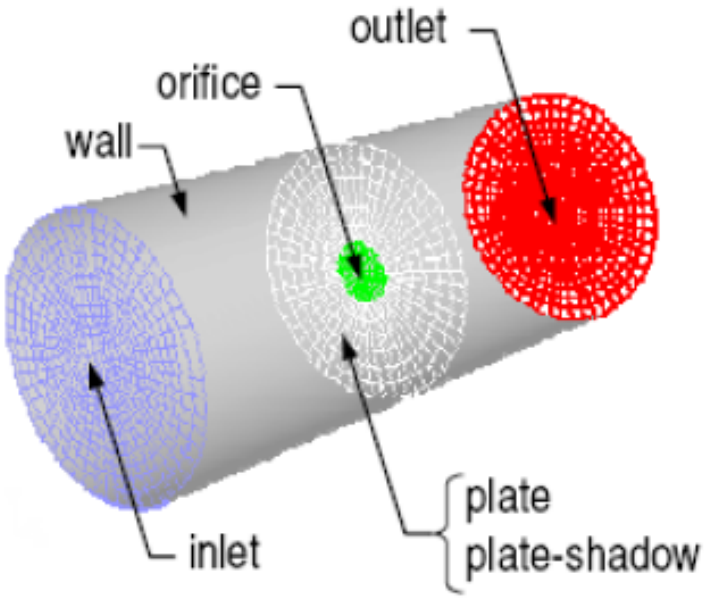
Setting up the model (Solver Setup) – Boundary Conditions

- For a given problem, you will need to:
  - Select appropriate physical models.
  - Turbulence, combustion, multiphase, etc.
  - Define material properties.
    - Fluid.
    - Solid.
    - Mixture.
  - Prescribe operating conditions.
  - Prescribe boundary conditions at all boundary zones.
  - Provide an initial solution! (Comes from boundary conditions)
  - Set up solver controls.
  - Set up convergence monitors.

Aerodynamics
CFD

Setting up the model (Solver Setup) – Boundary Conditions

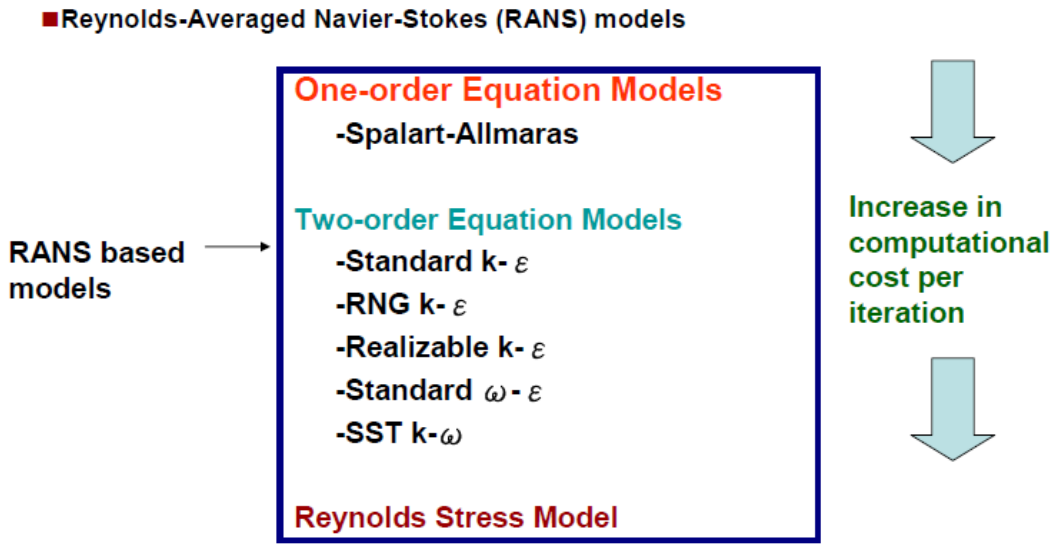
- **External faces**
  - -General ~ pressure inlet , pressure outlet
  - -Incompressible ~ velocity inlet, outflow
  - -Compressible ~ mass flow inlet , pressure far-field, mass flow outlet
  - -Other ~ wall, symmetry, axis, periodic
- **Cell Zones**
  - -Fluid
  - -Solid
  - -Heat exchanger
- **Interface**
  - -wall, interior.....



Aerodynamics
CFD

Setting up the model (Solver Setup) – Turbulence Models

- Most flows around vehicles (or inside) will have some turbulent elements.
- We therefore must either have a mesh which is fine enough to pick up the smallest scales of turbulence (direct numerical simulation) which is very expensive in terms of computing, or approximate the turbulent flow by use of a turbulence model.



Aerodynamics
CFD

Computing the solution (Solving)

- The discretized conservation equations are solved iteratively.
- A number of iterations are usually required to reach a converged solution.
- Convergence is reached when:
  - Changes in solution variables from one iteration to the next are negligible.
  - Residuals provide a mechanism to help monitor this trend.
  - Overall property conservation is achieved.
- The accuracy of a converged solution is dependent upon:
  - Appropriateness and accuracy of the physical models.
  - Grid resolution and independence.
  - Problem setup.

Aerodynamics
CFD

Visualising the results (Post-processing)

- Visualization can be used to answer such questions as:
  - What is the overall flow pattern?
  - Is there separation?
  - Where do shocks, shear layers, etc. form?
  - Are key flow features being resolved?
  - Are physical models and boundary conditions appropriate?
  - Numerical reporting tools can be used to calculate quantitative results, e.g:
    - Lift, drag, and torque.
    - Average heat transfer coefficients.
    - Surface-averaged quantities.

Visualising the results (Post-processing)

- Common visualization tools include;
  - Fluid Velocity Vectors
  - Fluid Velocity Contours
  - Fluid Pressure Contours
  - Pressure Vectors on specified surface
  - Pathline and particle trajectory plots. (Streamlines)
  - XY plots.
  - Animations.

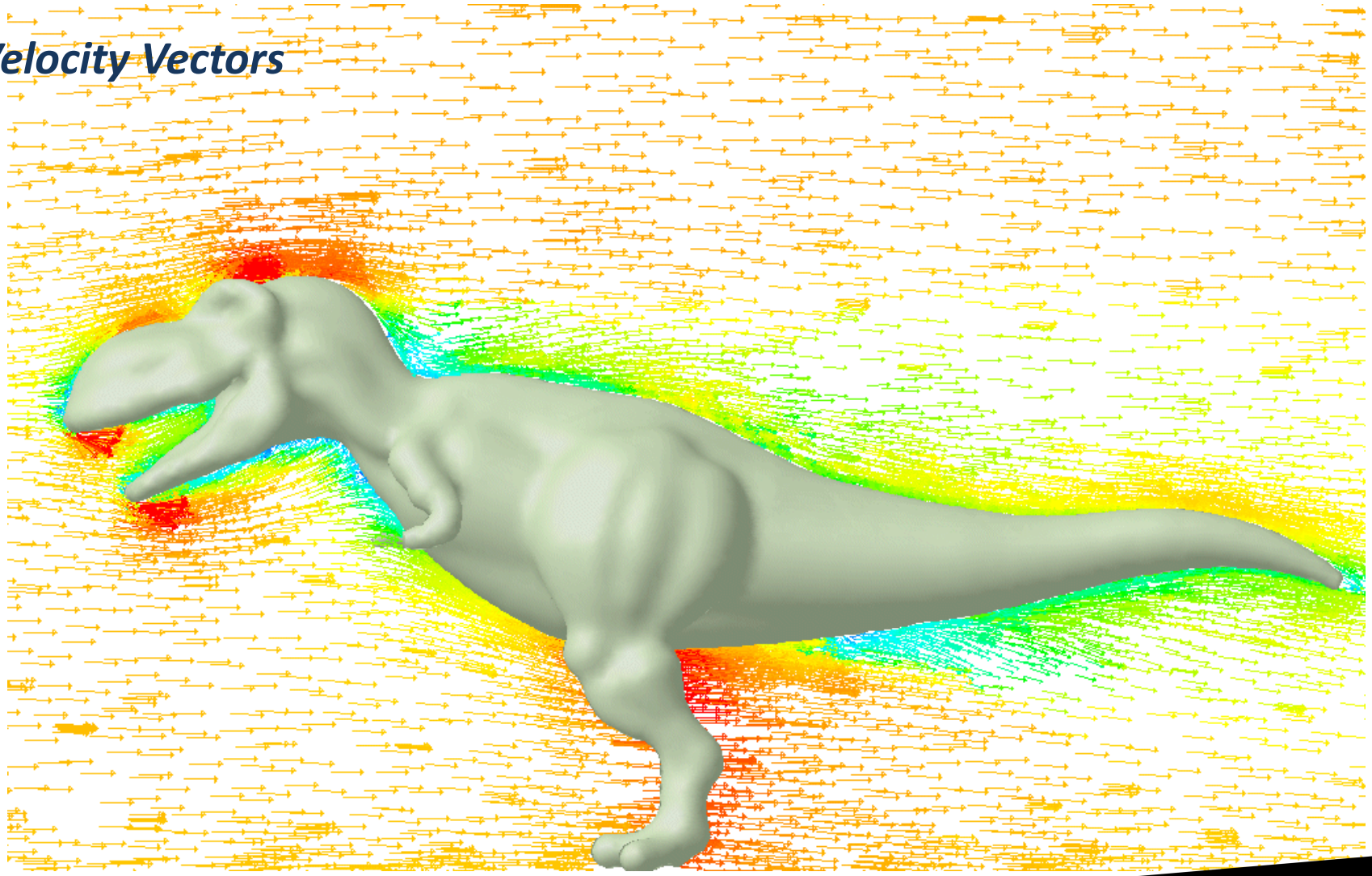
- Numerical reporting tools:
  - Flux balances.
  - Surface and volume integrals and averages.
  - Forces and moments.

Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

Aerodynamics

CFD

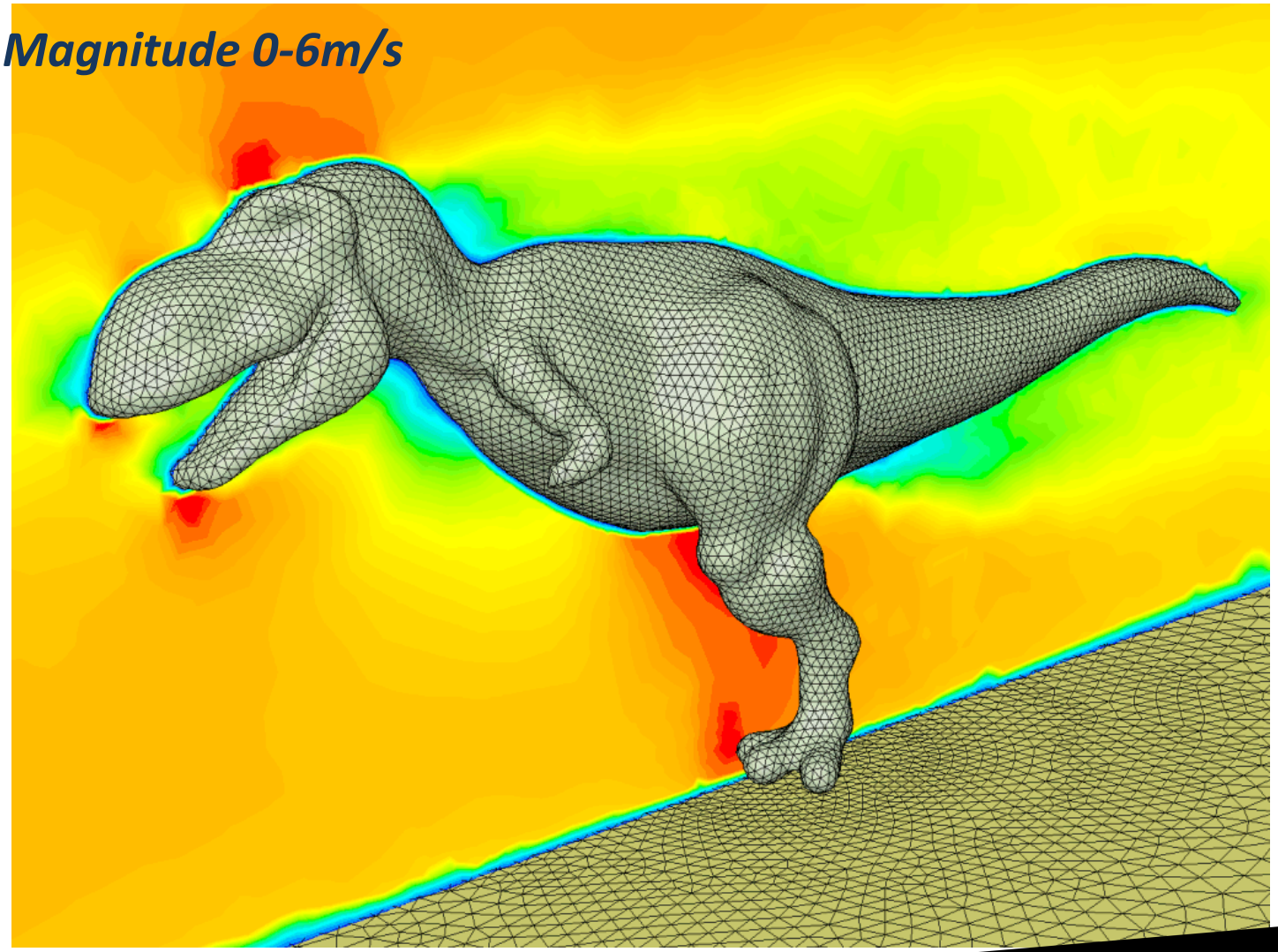
Velocity Vectors



Aerodynamics

CFD

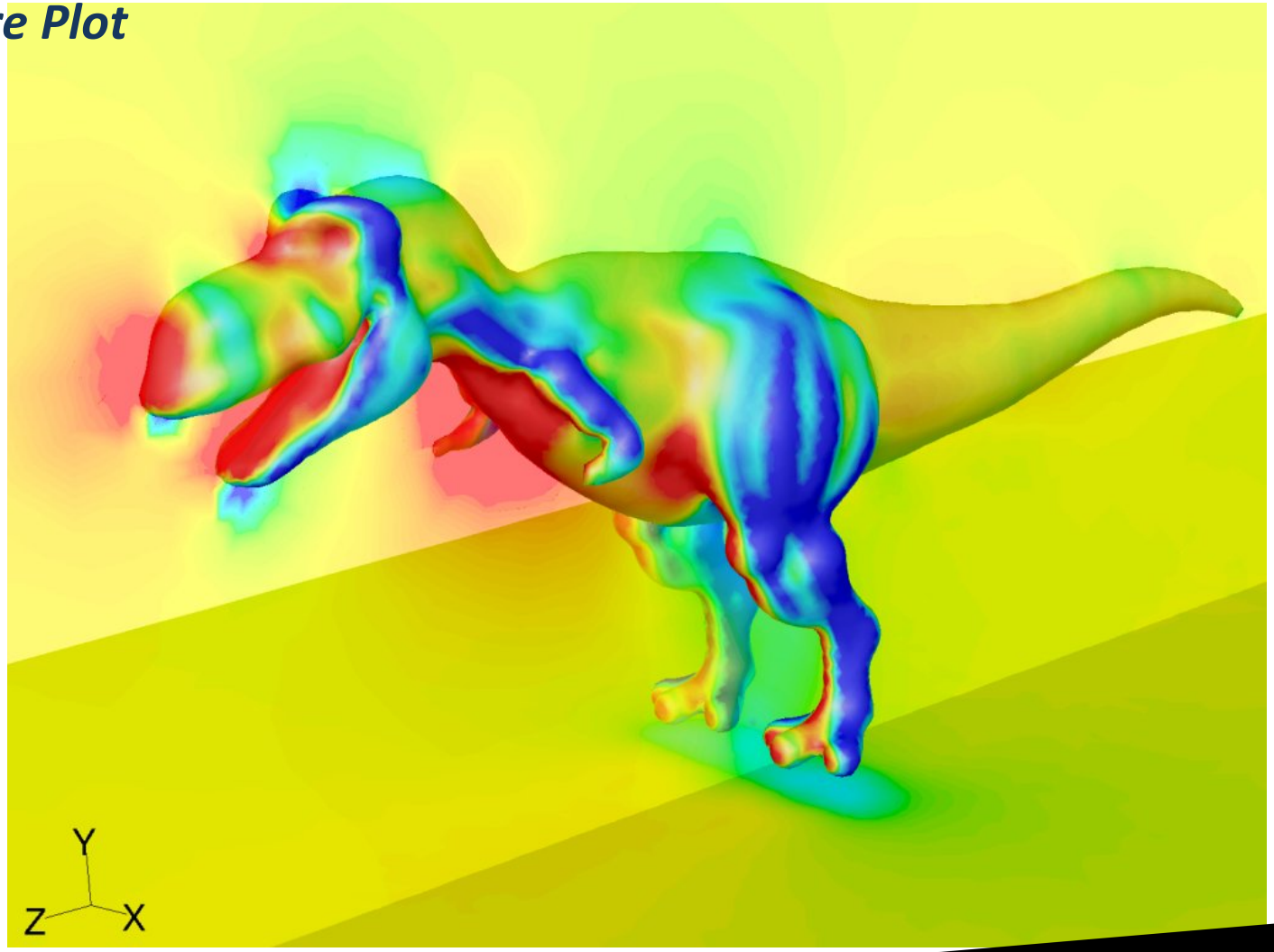
Velocity Magnitude 0-6m/s



Aerodynamics

CFD

Pressure Plot



Forces from the Dinosaur simulation

- Drag force: 17.4 N.
- Lift force: 5.5 N.
- Wind velocity: 5 m/s.
- Air density: 1.225 kg/m^3.
- The dinosaur is 3.2 m tall.
- It has a projected frontal area of A = 2.91 m^2.
- The drag coefficient is:

$$C_D = \frac{Drag\ Force}{\frac{1}{2}\rho v^2 A} = \frac{17.4}{0.5*1.225*25*2.91} = 0.11$$

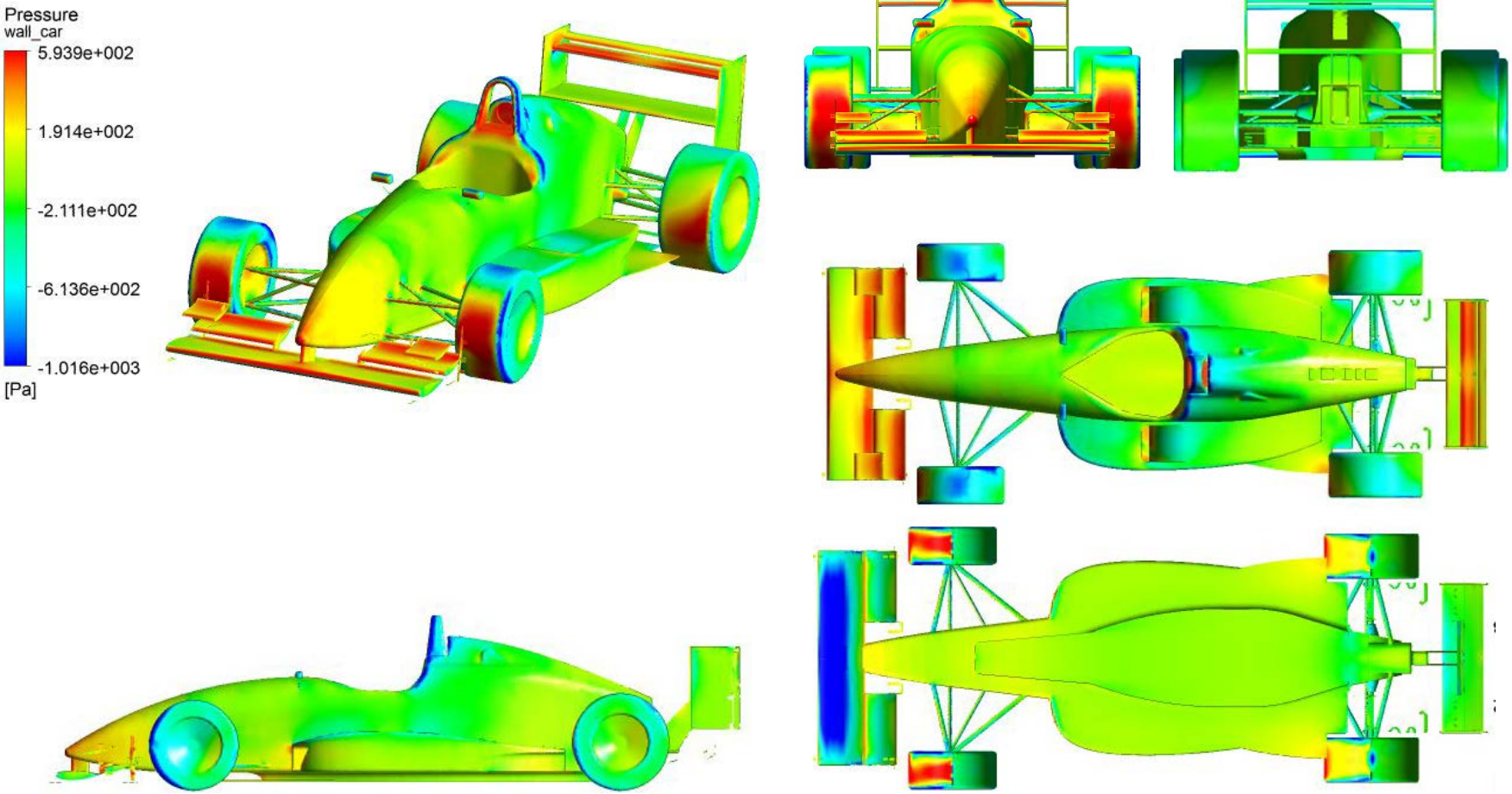
- This is pretty good compared to the average car!

- The streamlined back of the dinosaur resulted in a flow pattern with very little separation.

Aerodynamics

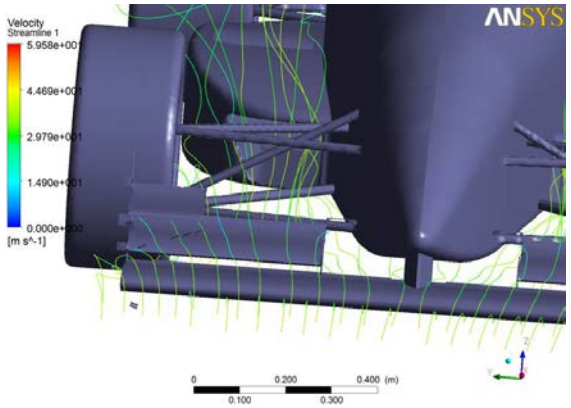
CFD

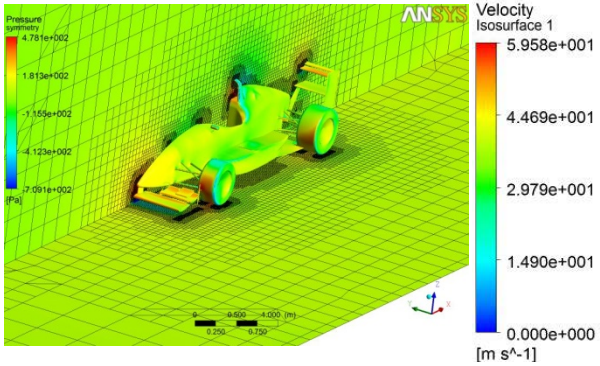
Pressure Plot

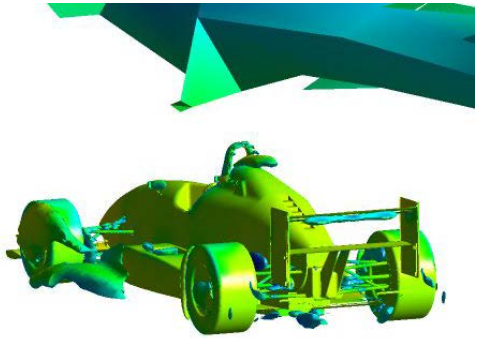


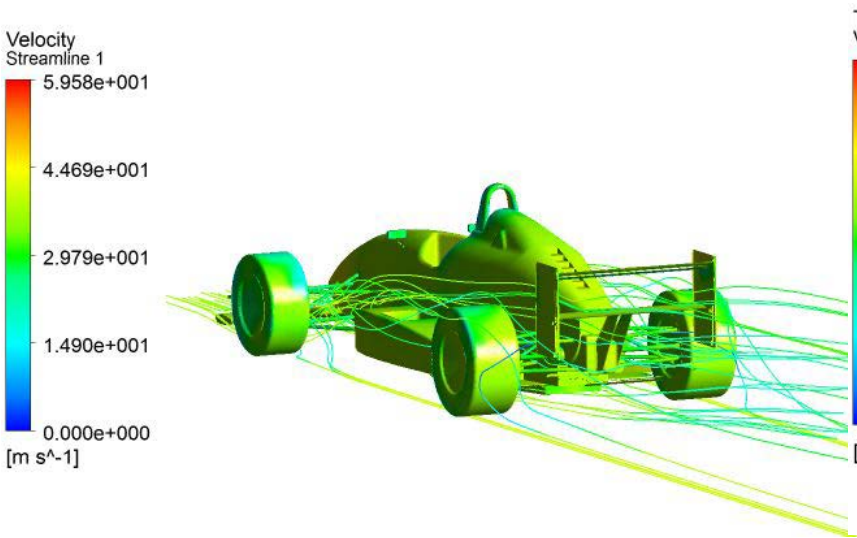
Aerodynamics
CFD

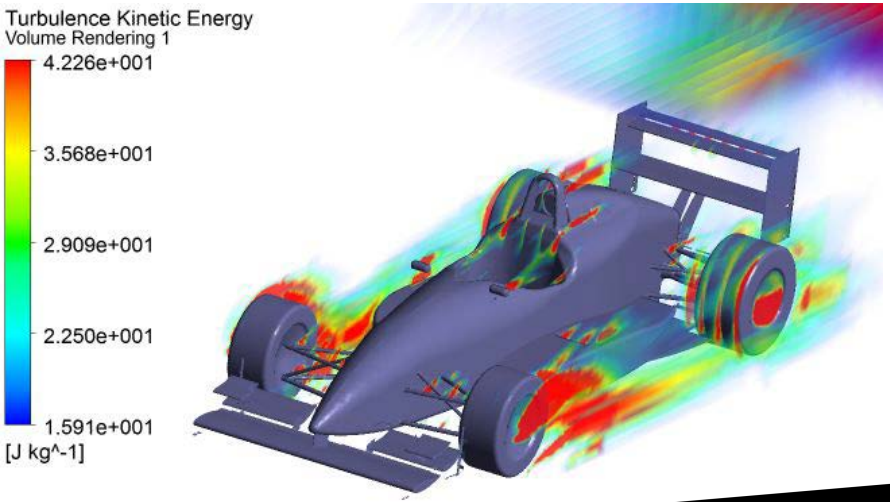
Other Plots











Consider improvements to simulation

- Are physical models appropriate?
  - Is flow turbulent?
  - Is flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?
  - Are boundary conditions correct?
- Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values reasonable?
- Is grid adequate?
  - Can grid be adapted to improve results?
  - Does solution change significantly with adaption, or is the solution grid independent?
  - Does boundary resolution need to be improved?

Verification and Validation of CFD Model

- ***Verification*** is defined as:

  *The process of determining that a model implementation accurately represents the developer's **conceptual description** of the model and the solution to the model.*

- **Validation** is defined as:

  *The process of determining the degree to which a model is an accurate representation of the **real world** from the perspective of the intended uses of the model.*

Verification of CFD Model

- Verification assessment examines
  1) If the computational models are the correct implementation of the conceptual models
  2) If the resulting code can be properly used for an analysis.

- The strategy is to identify and quantify the errors in the model implementation and the solution. The two aspects of verification are the ***verification of a code*** and the ***verification of a calculation***. The objective of verifying a code is *error evaluation ,* that is, finding and removing errors in the code. The objective of verifying a calculation is *error estimation*, that is determining the accuracy of a calculation.

- Verification has also been described as ***solving the equations right***. It is intended to concern itself more with mathematics rather than engineering. It is intended to look for errors in the programming and implementation of the models.

Validation of CFD Model

- Validation has also been described as *"**solving the right equations**"*. It is not possible to validate the entire CFD code. One can only validate the code for a specific range of applications for which there is experimental data. Thus one validates a model or simulation. Applying the code to flows beyond the region of validity is termed prediction.

- Validation examines if the conceptual models and computational simulation agree with real world observations. The strategy is to indentify and quantify error and uncertainty through comparison of simulation results with experimental data.

- The approach to Validation Assessment is to perform a systematic comparison of CFD simulation results to experimental data from a set increasingly complex cases.

Aerodynamics
CFD

Advantages of CFD

- Relatively low cost.
  - Using physical experiments and tests to get essential engineering data for design can be expensive.
  - CFD simulations are relatively inexpensive, and costs are likely to decrease as computers become more powerful.

- Speed.
  - CFD simulations can be executed in a short period of time.
  - Quick turnaround means engineering data can be introduced early in the design process.

- Ability to simulate real conditions.
  - Many flow and heat transfer processes can not be (easily) tested, e.g. hypersonic flow.
  - CFD provides the ability to theoretically simulate any physical condition.

Advantages of CFD

- Ability to simulate ideal conditions.
  - CFD allows great control over the physical process, and provides the ability to isolate specific phenomena for study.
  - Example: a heat transfer process can be idealized with adiabatic, constant heat flux, or constant temperature boundaries.

- Comprehensive information.
  - Experiments only permit data to be extracted at a limited number of locations in the system (e.g. pressure and temperature probes, heat flux gauges, LDV, etc.).
  - CFD allows the analyst to examine a large number of locations in the region of interest, and yields a comprehensive set of flow parameters for examination.

Limitations of CFD

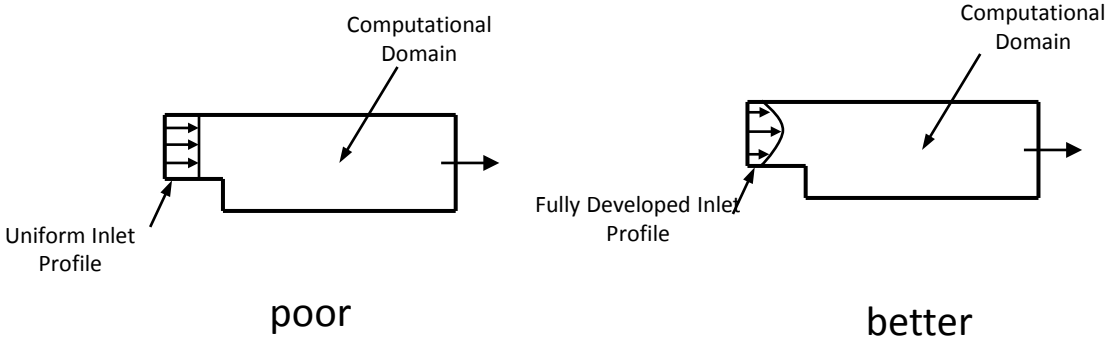
- Physical models.
  - CFD solutions rely upon physical models of real world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.).
  - The CFD solutions can only be as accurate as the physical models on which they are based.

- Numerical errors.
  - Solving equations on a computer invariably introduces numerical errors.
  - Round-off error: due to finite word size available on the computer. Round-off errors will always exist (though they can be small in most cases).
  - Truncation error: due to approximations in the numerical models. Truncation errors will go to zero as the grid is refined. Mesh refinement is one way to deal with truncation error.

Aerodynamics
CFD

Limitations of CFD

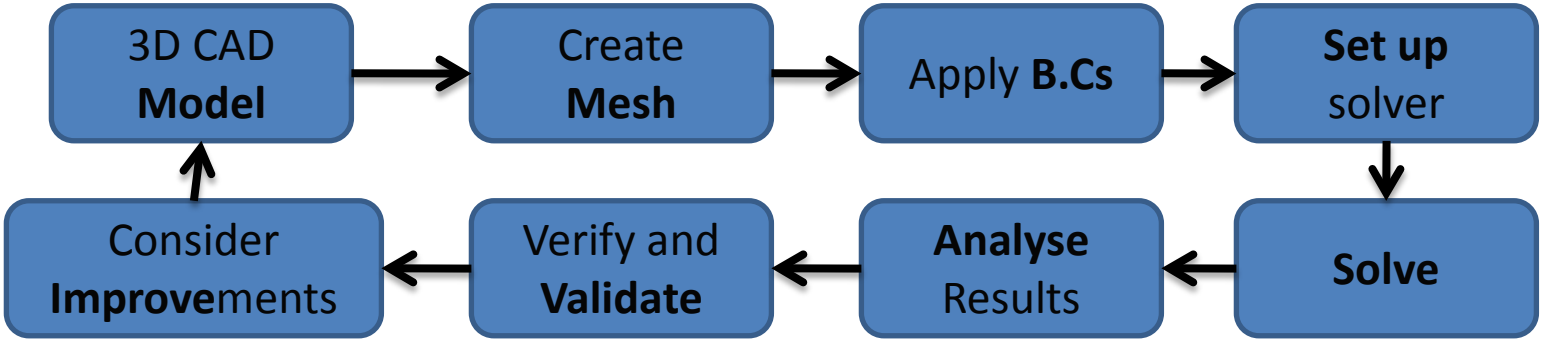
- Boundary conditions. **GARBAGE IN = GARBAGE OUT**
  - As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model.
  - Example: flow in a duct with sudden expansion. If flow is supplied to domain by a pipe, you should use a fully-developed profile for velocity rather than assume uniform conditions.



Aerodynamics
CFD

Summary

- CFD is a method to numerically calculate heat transfer and fluid flow.
- Currently, its main application is as an engineering method, to provide data that is complementary to theoretical and experimental data. This is mainly the domain of commercially available codes and in-house codes at large companies.
- The process of running a CFD simulation can be summarized by the flow diagram below.



Aerodynamics
Windtunnels

Windtunnel introduction

There are no simple theoretical methods for predicting the aerodynamic characteristics of a whole vehicle.

Although CFD has many advantages, it also has significant limitations.

It is normally necessary to use wind tunnel and road testing as well.

Ferrari's Wind Tunnel shown on right.



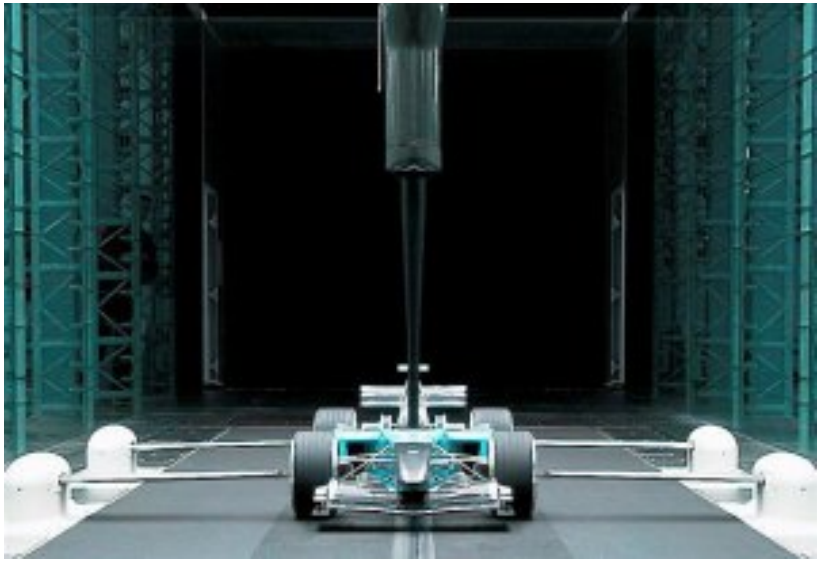
Aerodynamics
Windtunnels

Wind tunnels originally developed for aeronautical use

Specialised automotive tunnels evolved, which have distinguishing features from an aeronautical tunnel.

Two types of tunnel

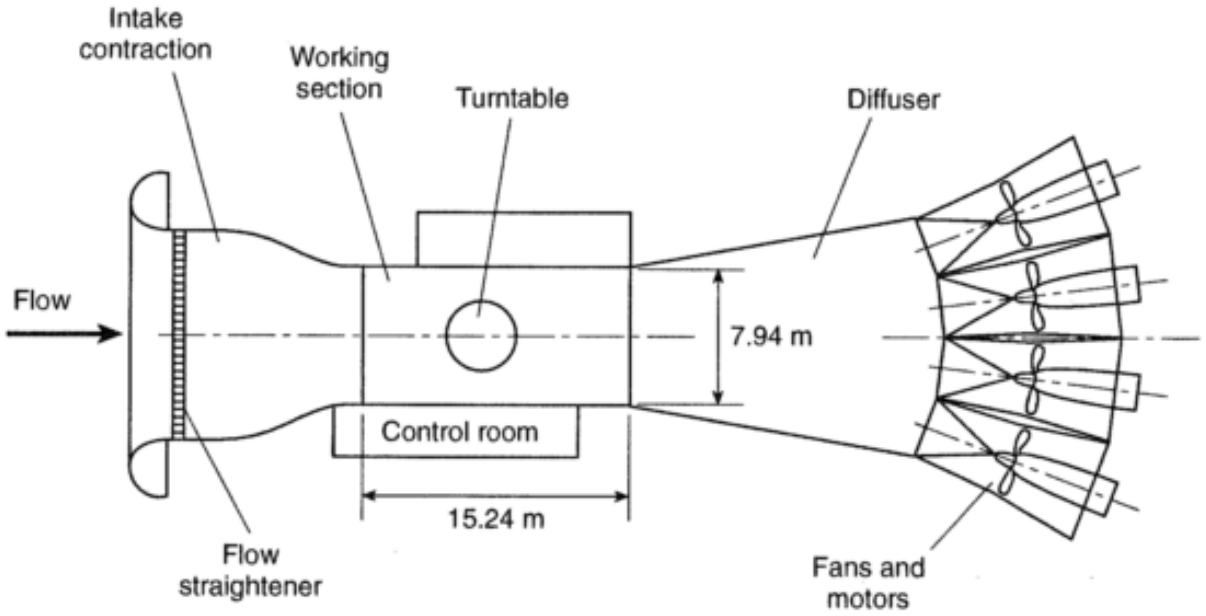
1. Open-return
2. Closed-return



Aerodynamics
Windtunnels

Wind Tunnel Types - Open

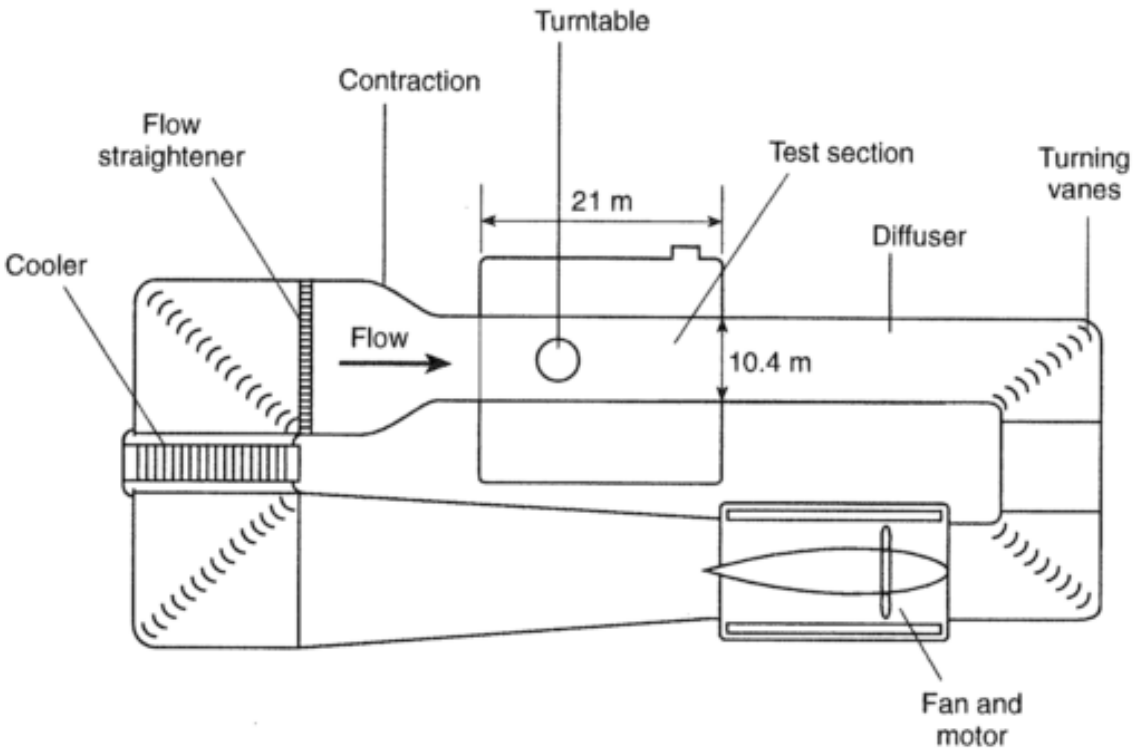
- Effectively a tube which is open at both ends
- Air is drawn and returned to surroundings



Aerodynamics
Windtunnels

Wind Tunnel Types - Closed

- Air circulates around a continuous closed circuit.



Aerodynamics

Wind Tunnel Types - Comparison

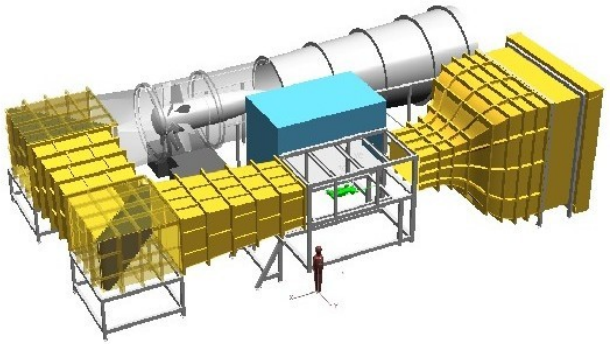
	Advantages	Disadvantages
Open-return	• **Compact** • Uses **open space** of enclosing building as return circuit • Air **doesn't heat** up much as has time to cool in large space	•**KE wasted** •**Power** required is more than closed return tunnel •Air starts from rest at atmospheric pressure. When sped up **pressure decreases** below atmospheric pressure. (Working section must be well sealed from surroundings
Closed-return	•KE not wasted •Power less than open-return type. •Atmospheric pressure can be maintained	•Heat produced by friction with walls can cause temperature rise affecting results. •Larger than open-return

- Both tunnel types capable of producing equally good results
- Closed-return type is usually preferred by automotive manufacturers

Aerodynamics
Windtunnels

Wind Tunnel Features

Both tunnel types have the following features:



- Contraction – Air Speeded up
- Working Section – Flow over test specimen
- Diffuser – Flow slowed down again

Contraction ratio is the ratio between the areas of the inlet to outlet of the contraction.

Large contraction ratio → improves steadiness and uniformity of flow in working section

Aerodynamics
Windtunnels

Wind Tunnel Features

Tunnels often utilise turning vanes to screens help reduce turbulence levels.

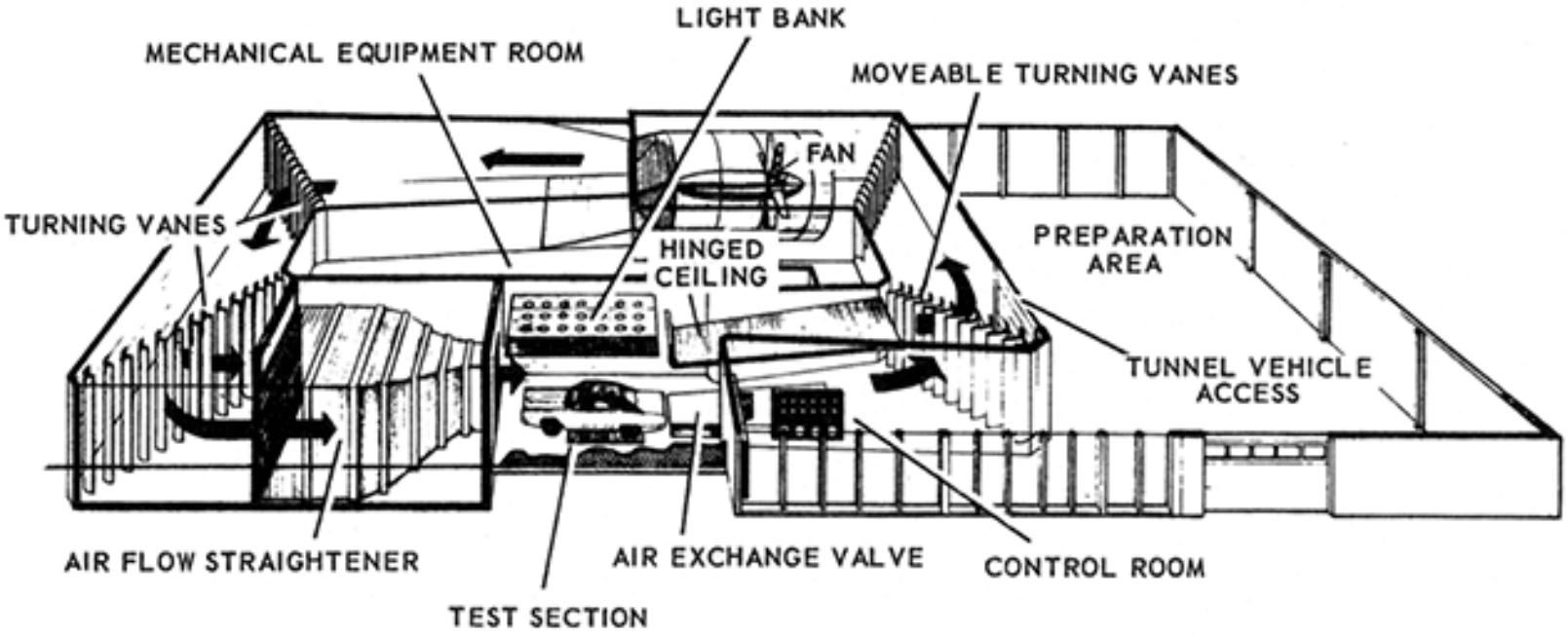
Turbulence intensity is almost always greater in a wind tunnel than on a road (in a calm day)

This is one of the primary causes of discrepancy between wind tunnel and track results.

Tunnels are often characterised by the cross sectional area of the working section. GM's tunnel has 5.5 x 10.4m, whilst MIRA's measures 4.5 x 8m

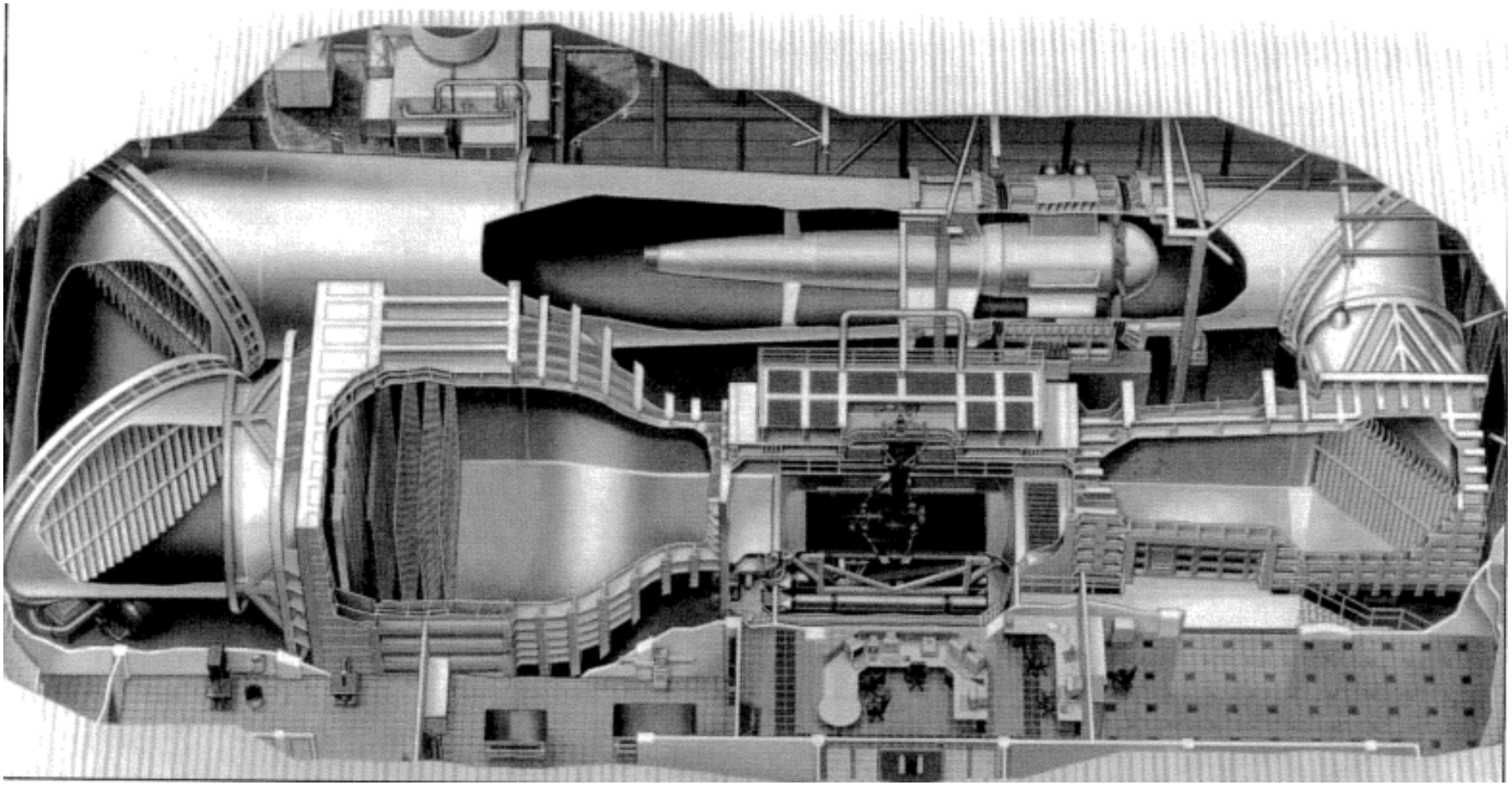
Aerodynamics
Windtunnels

Example Wind Tunnel



Aerodynamics
Windtunnels

F1 Wind Tunnel



Aerodynamics
Windtunnels

Wind Tunnel Sources Of Error

Wind tunnels have been used for aeronautical applications for many years. However different tunnels produce slightly different results due to errors.

Primary sources of error include

1. Scaling or **Reynolds number effect**
2. Errors due to the fact that the **road moves** relative to the car whereas wind tunnel floor stationary
3. **Blockage**
4. Failure to model **fine detail** accurately
5. Failure to model flow through **ventilation and cooling**
6. Difficulties in **measuring forces**

Aerodynamics
Windtunnels

Scale Or Reynolds Number Effect

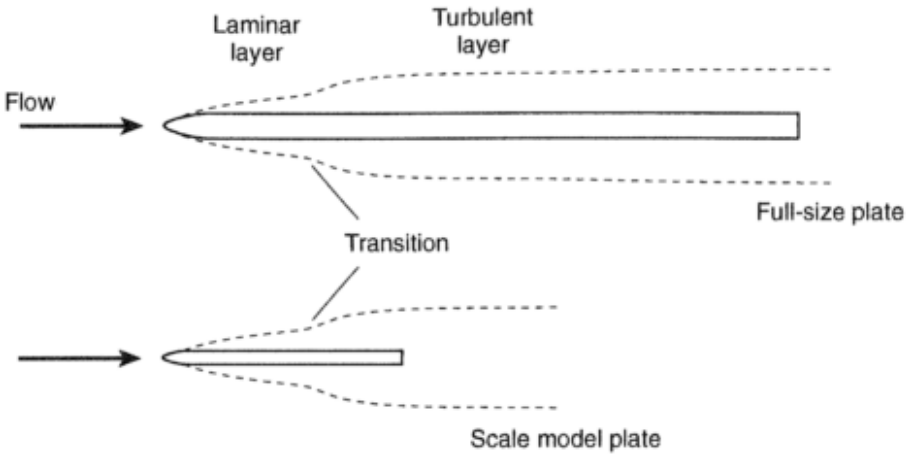
- Two plates placed parallel to flow direction.

- Each plate represents full scale car and scale model.

- Transition from laminar to turbulence at same distance along plates.

- Scale model has greater proportion of laminar boundary layer (Lower drag)

- **Drag per unit area is not the same on the scale model as the full size plate.**



Aerodynamics
Windtunnels

Tackling the Reynolds Number Effect

• As the title suggests, the effect is due to the Reynolds number being different for each plate.

$$\mathrm{Re} = \frac{\rho V l}{\mu} model = \frac{\rho V l}{\mu} \; full - scale$$

•We must change the density, viscosity or velocity of the flow for the scale model to achieve the same Reynolds number as the full size plate.

Aerodynamics
Windtunnels

Tackling the Reynolds Number Effect

$$\mathrm{Re} = \frac{\rho V l}{\mu} model = \frac{\rho V l}{\mu} full - scale$$

• Unfortunately to model a real vehicle at 100mph with a 10% scale model you would need to use a flow velocity of 1000mph which is well over the speed of sound → Flow compressible → Nothing like real life flow.

•Reynolds number can be raised by pressurising the tunnel to increase density → Wind tunnel is extremely expensive pressure vessel!

•Viscosity can also be decreased by cryogenic cooling – although this is even more expensive.

•Often the cheapest solution is to build full size tunnel!

Aerodynamics
Windtunnels

Small Scale Wind Tunnels

- It is often acceptable to utilise small scale tunnels and models with mismatched Reynolds numbers for comparative work.

- The effect of using a wrong Reynolds number is not normally that large as long as the number is above a certain critical value.

- Car shapes C_D values rarely show dramatic change with Reynolds number.

- Common to use ¼ scale models.

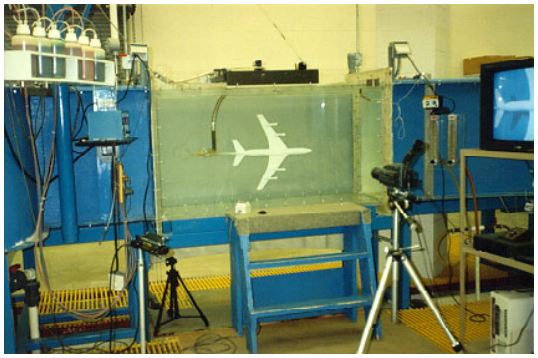
- For small scale tunnels you must endeavour to match Reynolds numbers as far as possible, so most are run at maximum velocity!

Aerodynamics
Windtunnels

Small Scale Wind Tunnels

- Matching Re's can also be achieved by changing working fluid and hence both viscosity and density.

- Water tunnels are used for high speed, small scale flow regimes.





$$\mathrm{Re} = \frac{\rho V l}{\mu} model = \frac{\rho V l}{\mu} fullscale$$

- Surfaces may also be finished so they induce boundary layer growth to match full scale (roughening)

Aerodynamics
Windtunnels

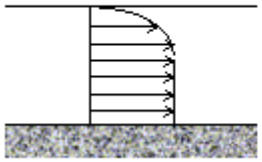
The Moving Road Problem

• Under real world conditions in still air there is no relative motion between the ground and the air flow.

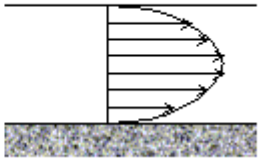
•Thus no boundary layer is formed on the road surface.

•In a wind tunnel there is relative motion and a boundary layer forms on the floor of the tunnel with the model immersed in it.

a) Velocity profile, Vehicle on road



b) Velocity profile, Vehicle in wind tunnel

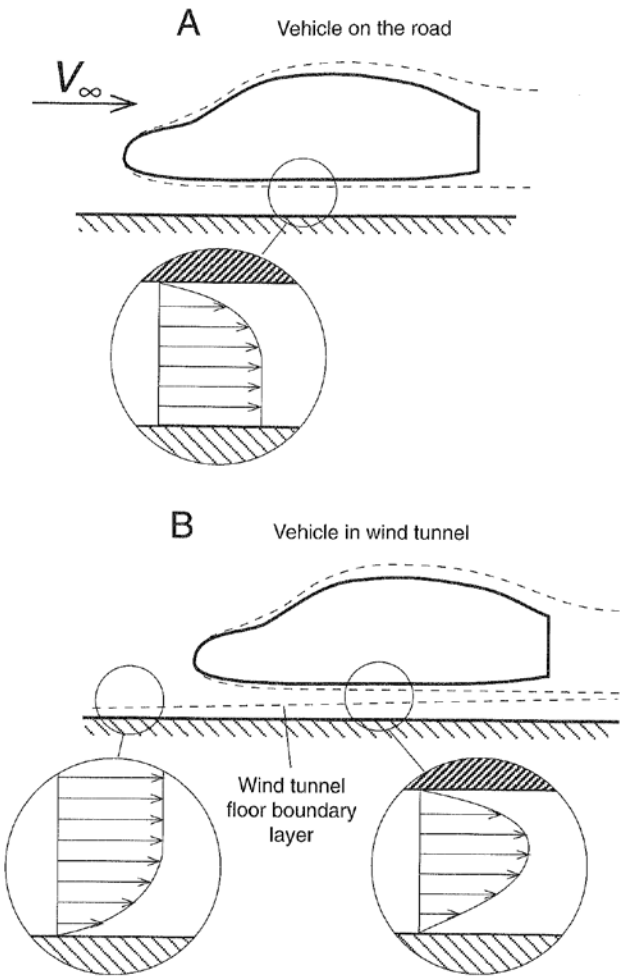


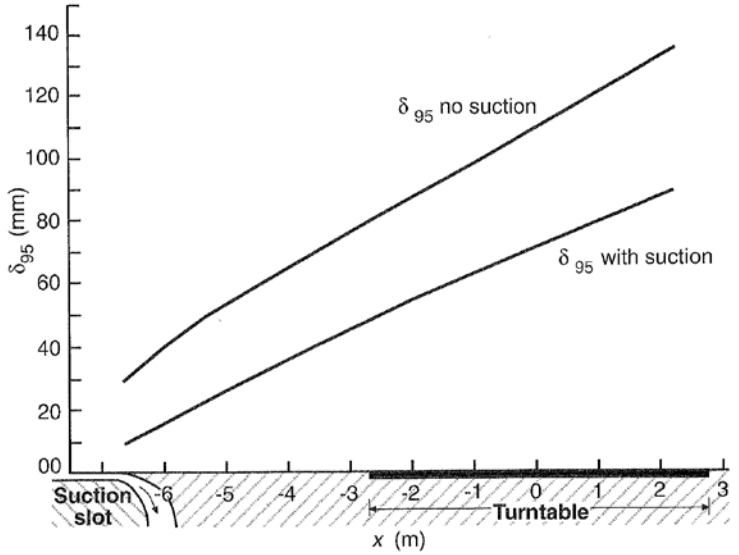
•Commonly employed methods to reduce the effect include

- •Use of ground board
- •Use of moving belt
- •Use of porous floor

Aerodynamics
Windtunnels

The Moving Road Problem





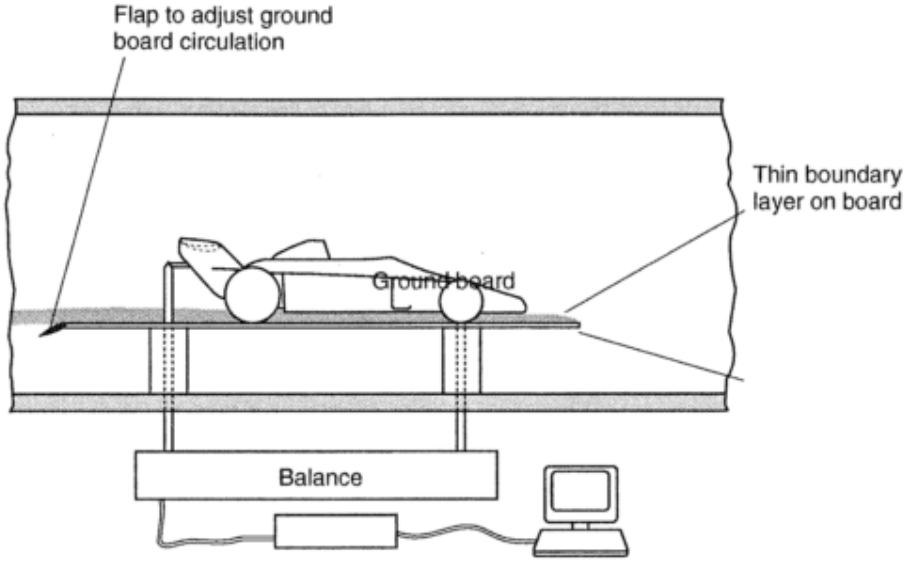
Aerodynamics
Windtunnels

The Moving Road Problem – Ground Board Method

- Simplest method which can yield quite satisfactory results.

- Board mounted well clear from tunnel floor.

- Board starts just before the vehicle so boundary layer has little time to form and hence is thinner



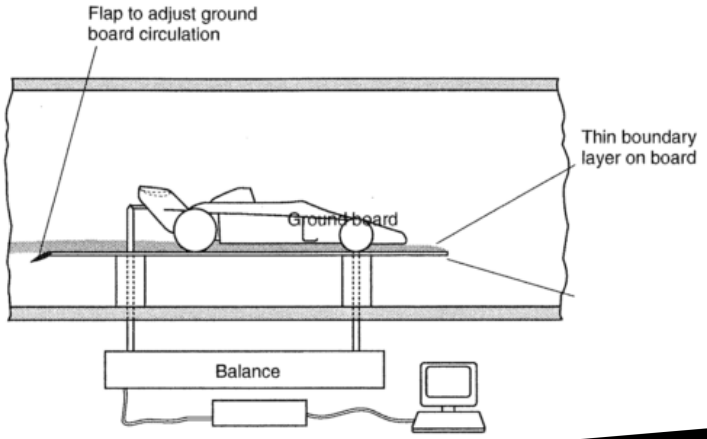
Aerodynamics
Windtunnels

The Moving Road Problem – Ground Board Method

• Thin boundary layer can be compensated for by raising vehicle by boundary layer thickness.

• Important to ensure that blockage above ground board doesn't induce circulation – Equalise blockage or add small flap at trailing edge of ground board.

•It has been found that the downstream length of ground board can affect the results due to influence on the wake, and a length of around 3 x model width is required.



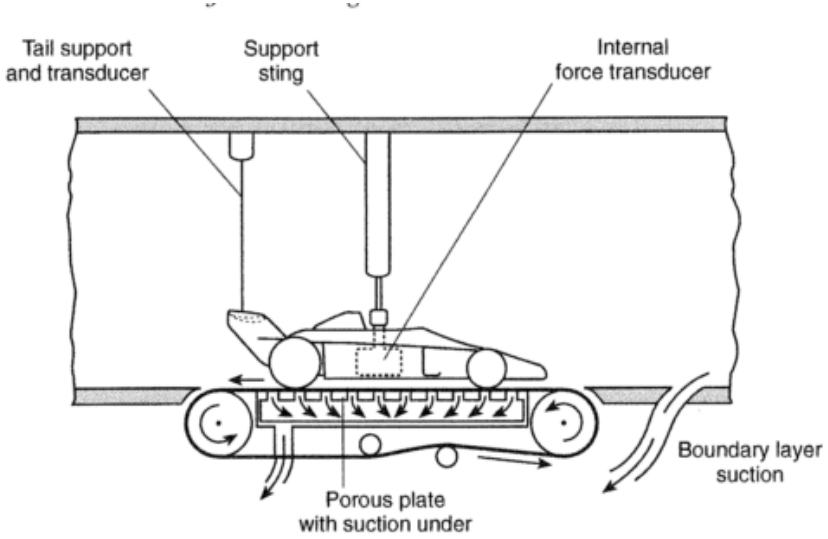
Aerodynamics
Windtunnels

The Moving Road Problem – Moving Belt Method

- Moving belt replicates the fact there is no relative velocity between air and floor.

- Boundary layer is fully removed by suction just upstream of belt.

- Belt usually supported by plate and kept in contact with plate by means of suction. Vibrations and bellowing of the belt must be controlled carefully.



Aerodynamics
Windtunnels

The Moving Road Problem – Moving Belt Method

•If the belt brakes **damage** can be caused to very expensive models.

• Models cannot me mounted from underneath and require a '**sting**' which may cause some flow interference.



• Rotating wheels will have some **rolling resistance** which must be separated from the aerodynamic drag.

• Difficult to prevent **reactions** on wheels upsetting lift measurements(Normally wheels mounted independently of rest of vehicle)

• Moving belt not as critical for large ride height vehicles → More important for race cars with low ride heights.

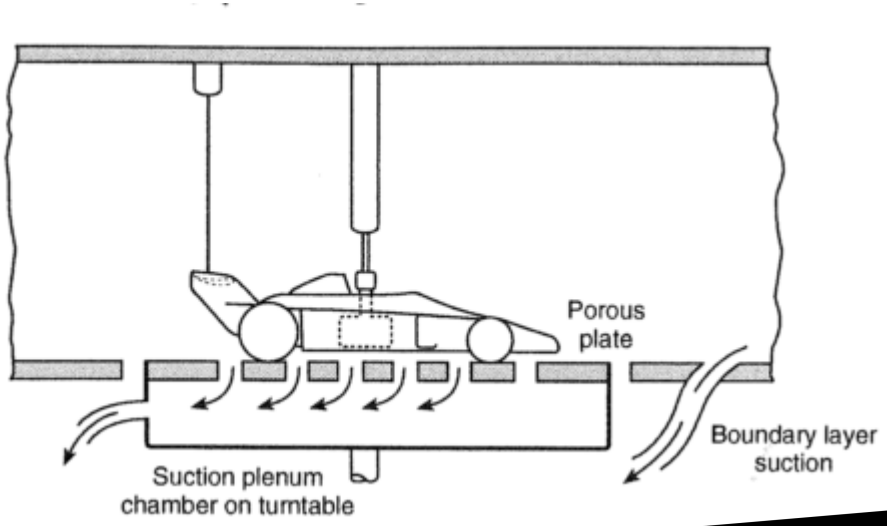
Aerodynamics
Windtunnels

The Moving Road Problem – Porous Floor method

- Another method to reduce the floor boundary layer effect is the porous floor method.

- The floor B.L is removed by suction through a slot upstream of the working section.

- A new B.L is prevented from growing by means of suction from the porous floor section



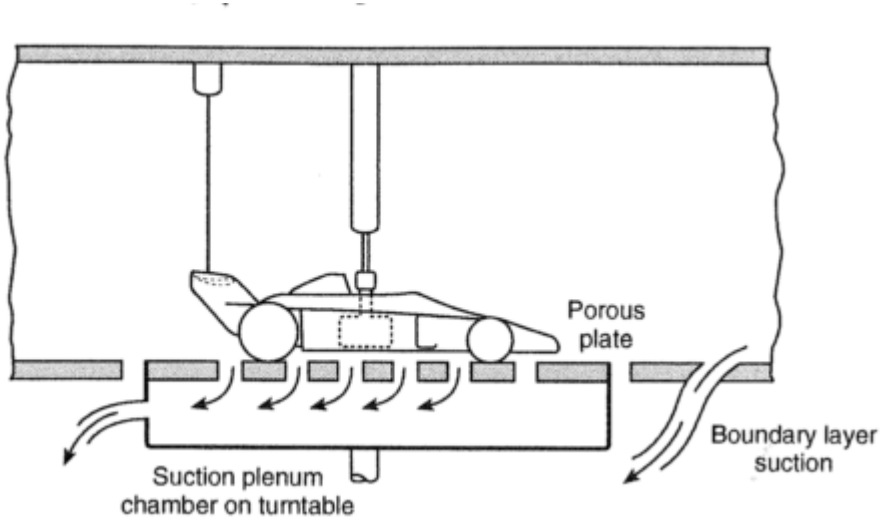
Aerodynamics
Windtunnels

The Moving Road Problem – Porous Floor method

- Measurements can be obtained which are very close to those from the moving belt method and road tests.

- However forces are sensitive to amount of suction

- Mechanically simple. Easy to yaw vehicle.



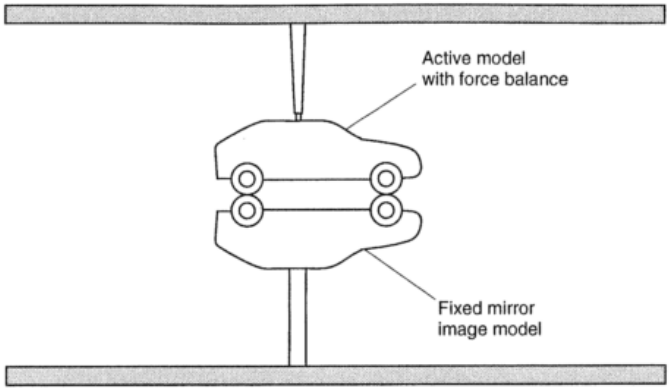
Aerodynamics
Windtunnels

The Moving Road Problem – Mirror Image Method

• A further method is to suspend the model in a tunnel with a mirror image model below.

• Because of symmetry the axis of the line dividing the two models must be a stream line and thus there is no B.L effect.

• Rarely used as two models are expensive and flow is rarely actually symmetrical due to turbulence effects.

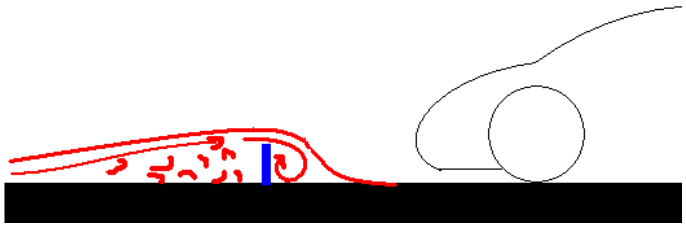


Aerodynamics
Windtunnels

The Moving Road Problem – Boundary Layer Trips

• Another simple method is to apply a 90 degree fence perpendicular to the ground, upstream of the vehicle.

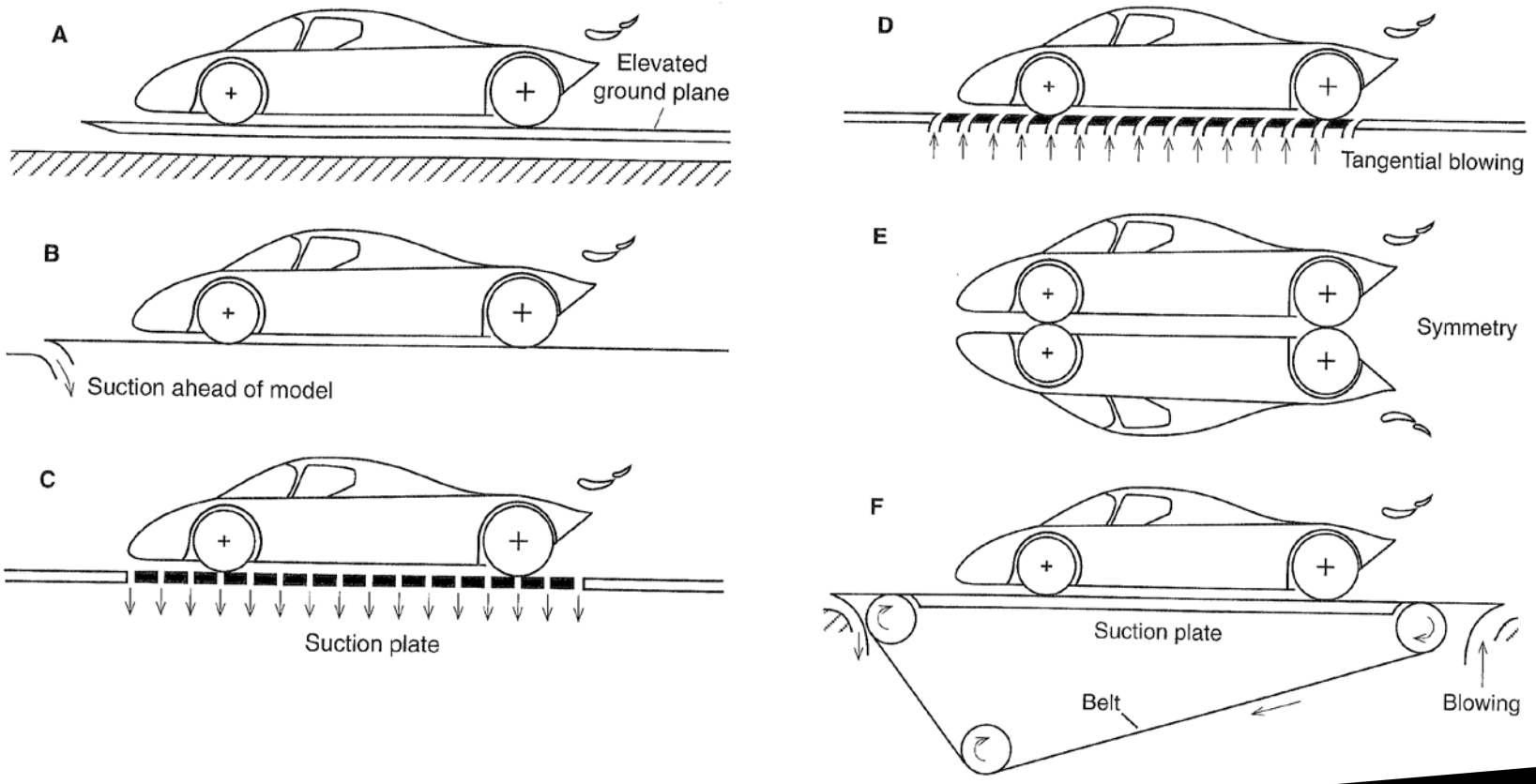
•This fence trips the boundary layer an helps reattach the flow before it reaches the vehicle





Aerodynamics
Windtunnels

The Moving Road Problem – Summary



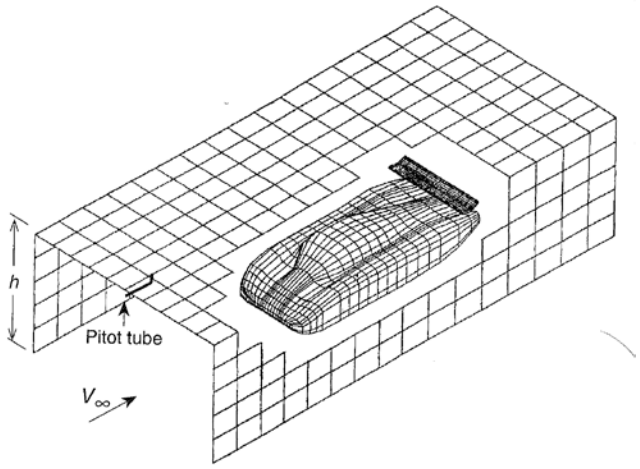
Aerodynamics
Windtunnels

Blockage Effects

- The blockage effect is due to a model being placed in a given volume.

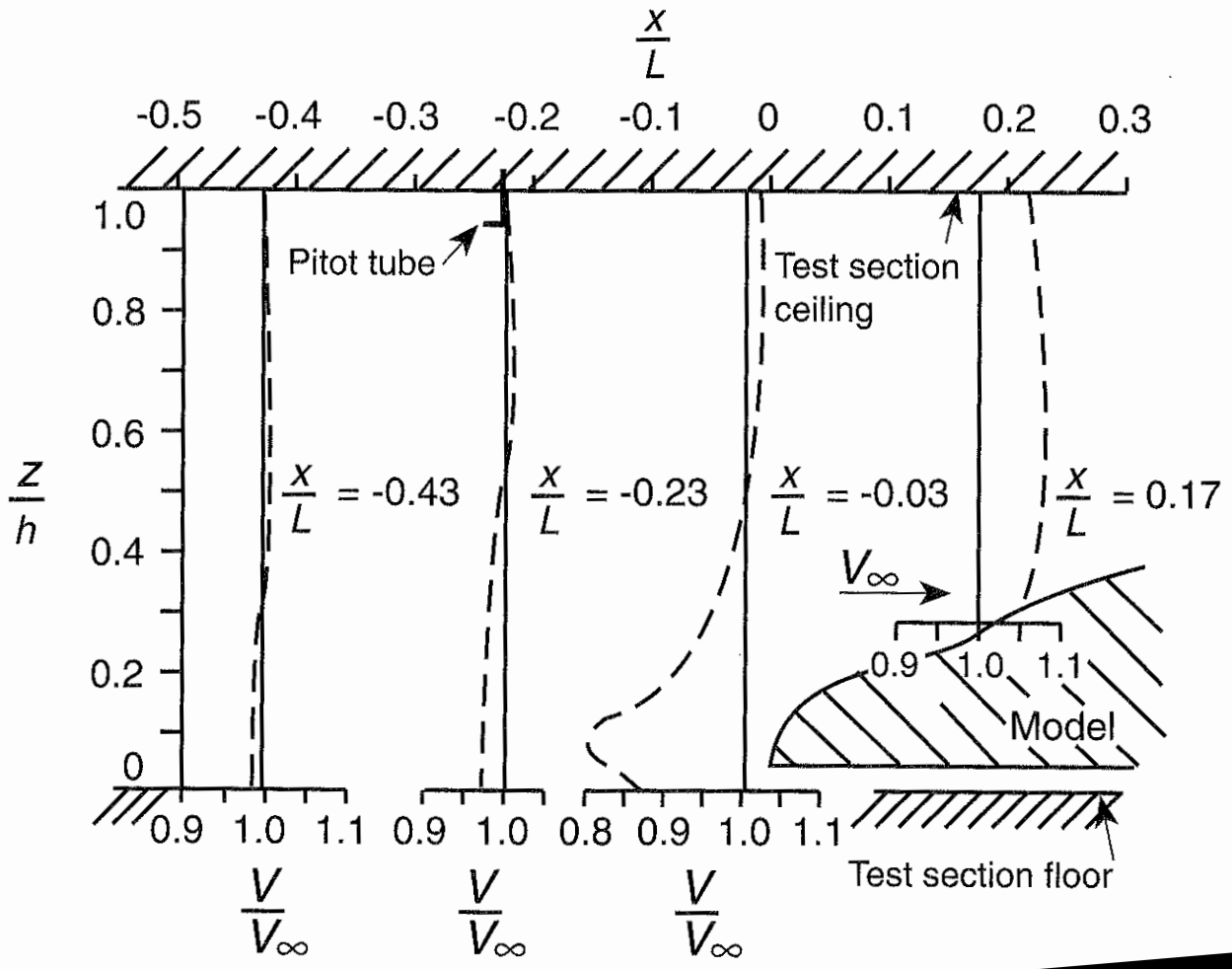
- Due to continuity of mass, the flow velocity must increase around the model if the CSA before the vehicle is the same as that around the vehicle.

- This increased velocity will lead to increased readings for lift and drag and therefore compensation must be applied.



Aerodynamics
Windtunnels

Blockage Effects



Aerodynamics

Blockage Effects

- A crude blockage correction:
  - Continuity equation $\rho \times Area \times speed = constant$
  - Density assumed constant $Area \times speed = constant$
  - If working section area is S and model area is A

$$V_{true} \times (S - A) = V_{indicated} \times S$$

Where V_{true} is the actual speed around model and $V_{indicated}$ is the upstream flow speed

Aerodynamics
Windtunnels

Blockage Effects

$$V_{true} \times (S - A) = V_{indicated} \times S$$

$$C_D = \frac{D}{\frac{1}{2}\rho V_{true}^{\ 2} \times A}$$

$$C_D = \frac{D}{\frac{1}{2}\rho V_{indicated}^{\ 2} \times (S/(S-A))^2 \times A}$$

- Ignoring second order terms

$$C_D = C_{D\,indicated}(1 - 2A/S)$$

Aerodynamics
Windtunnels

Blockage Effects – Open Sections

- Shaped tunnels for specific shapes can reduce error.
- Open sections are also used so that wall effects do not effect streamlines.
- Much smaller blockage factors result in open section tunnels.

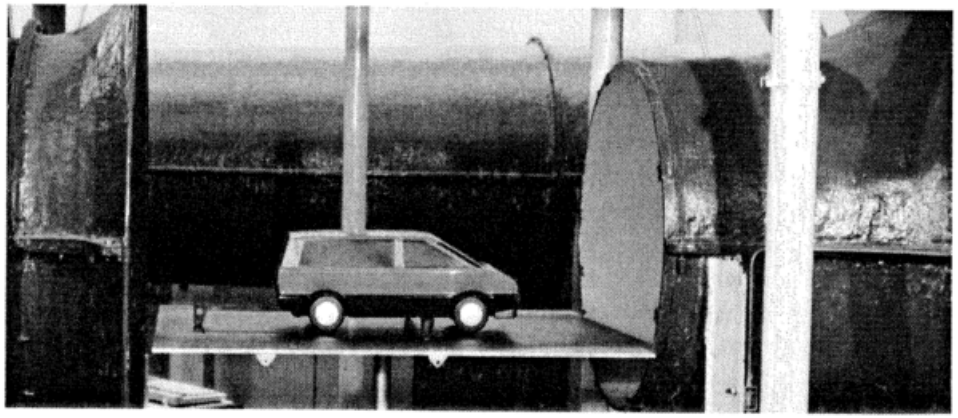


Fig. 11.9 An open jet wind-tunnel with a model mounted over a ground board.

Aerodynamics
Windtunnels

Blockage Effects – Slotted Walls



- Some tunnels use slotted walls.
- The working section walls and roof are not continuous surfaces but made up of series of slats with slots.
- Slotted section is surrounded by plenum chamber
- Slots help reduce the blockage facto considerably.

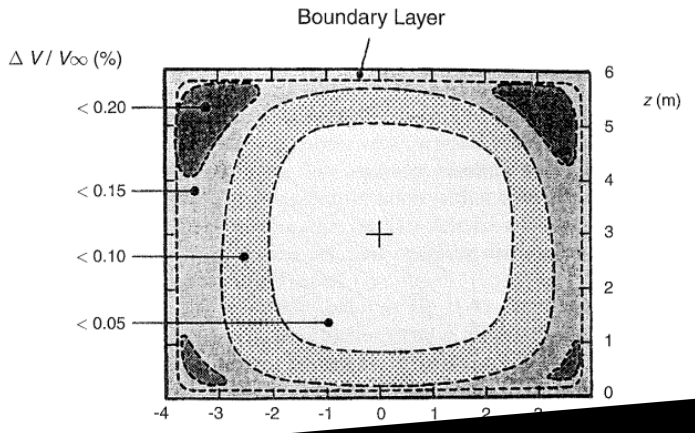


Fig. 3-39. Variation in the test-section free-stream velocity in the center of the German-Dutch wind tunnel test section. Courtesy of DNW.

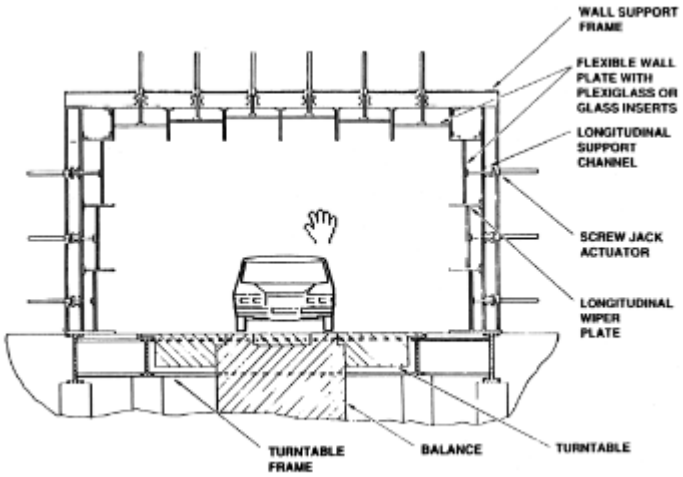
Aerodynamics
Windtunnels

Blockage Effects – Adaptive Walls



- Some tunnels use adaptive walls.

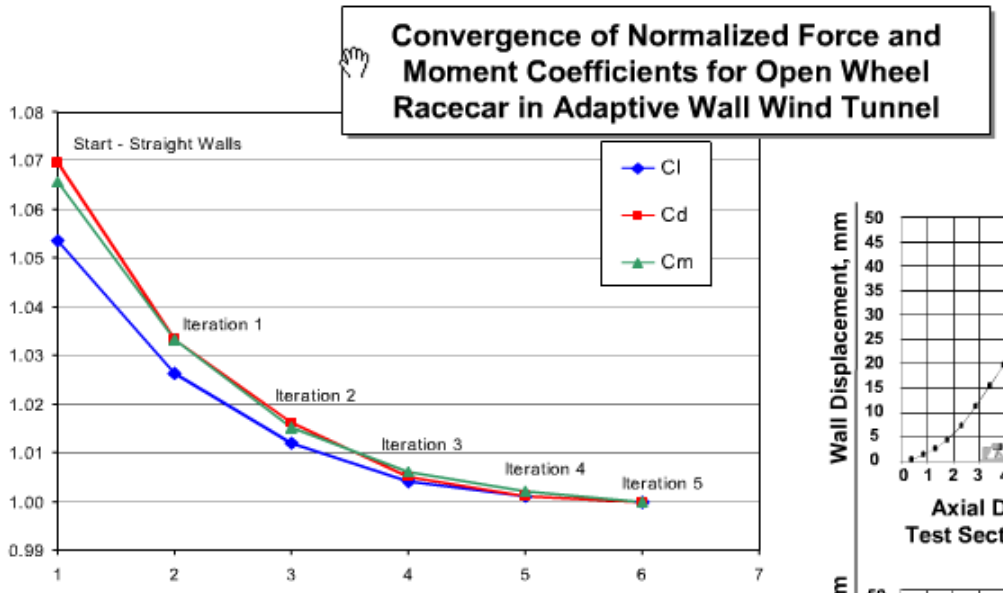
- Monitor flow and adapt automatically to reduce blockage effect.

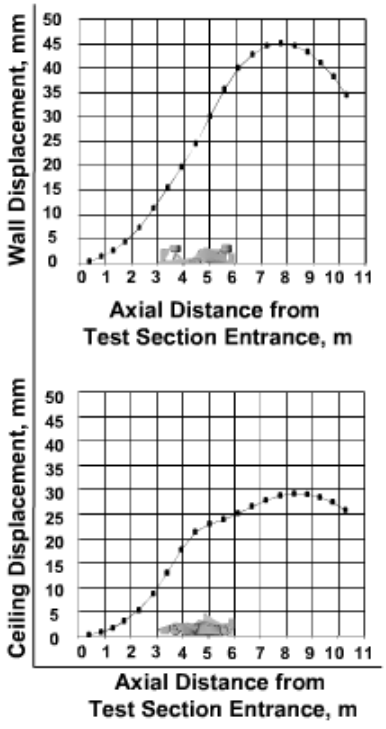




Aerodynamics
Windtunnels

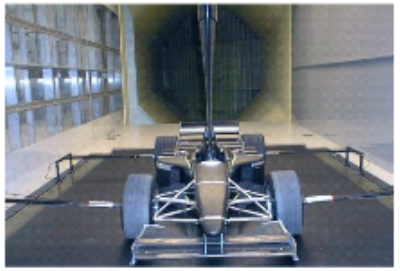
Blockage Effects – Adaptive Walls





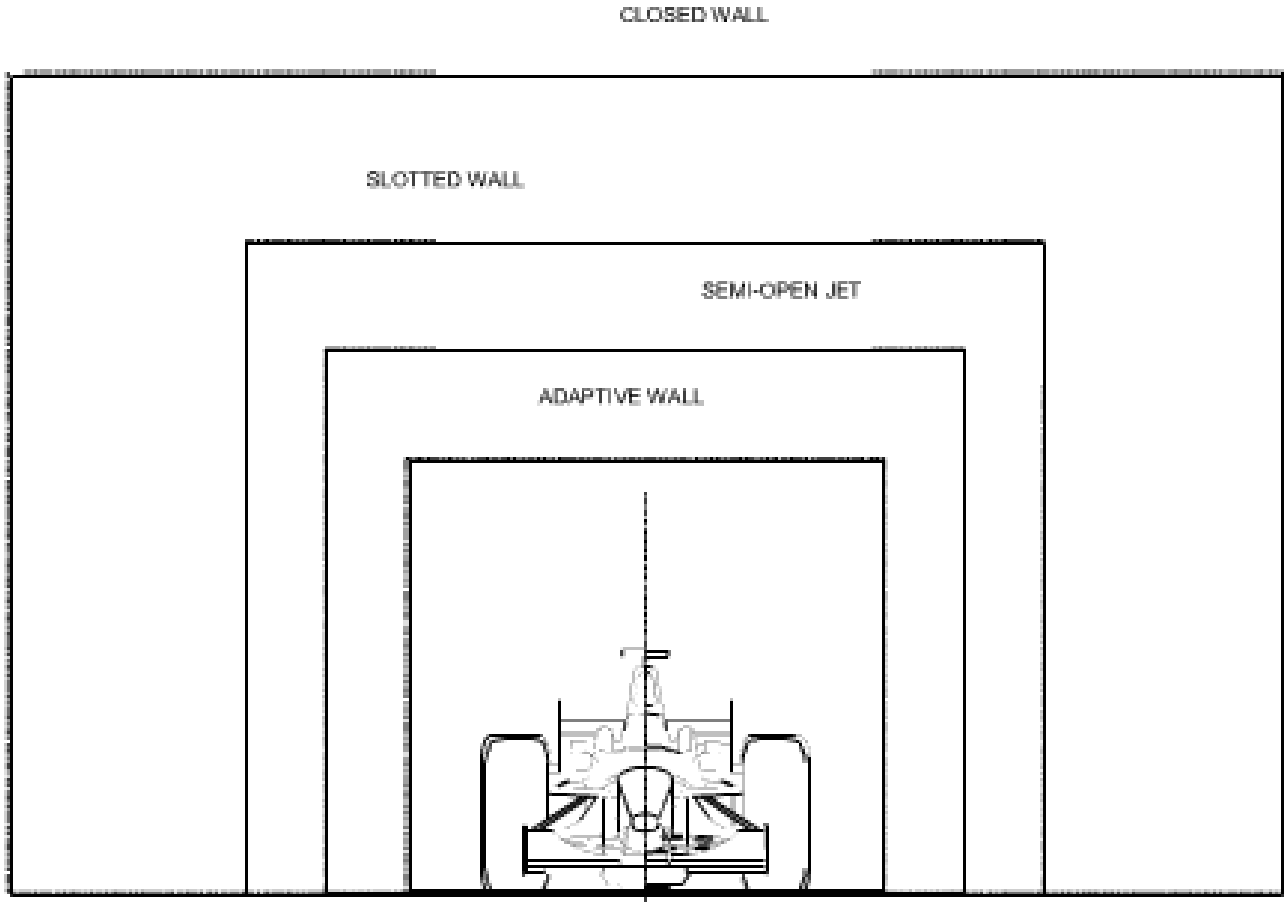






Aerodynamics
Windtunnels

Blockage Effects – Adaptive Walls



Test Section sizing based on typical blockage criteria

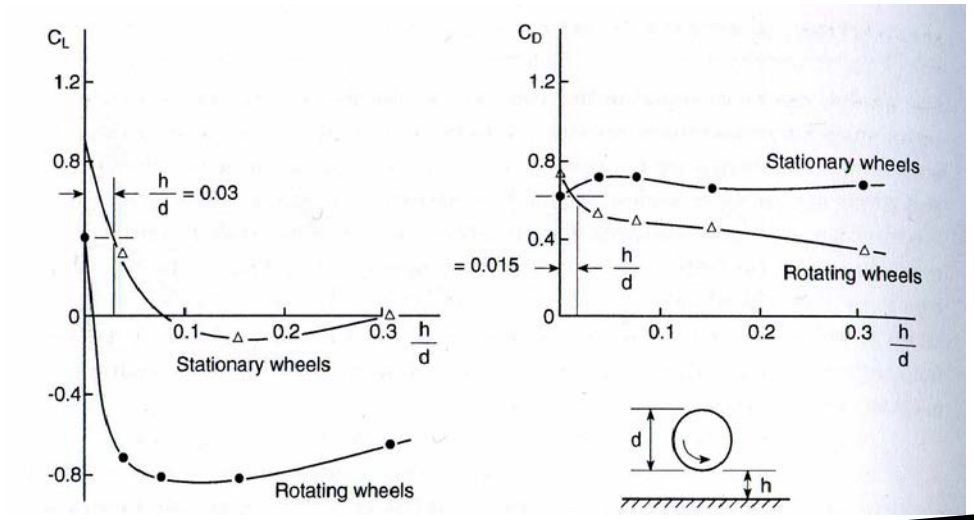
Aerodynamics

Rotating Wheel Problem

• On a fixed ground, wheels obviously cannot be rotated whilst still in contact with the ground.

•Rotating wheels can be used but the clearance gap required can affect lift measurements.

• It can be shown that fixed wheels with a clearance of between 1.5 and 3% of the wheel diameter gives roughly the same lift and drag as rotating wheels



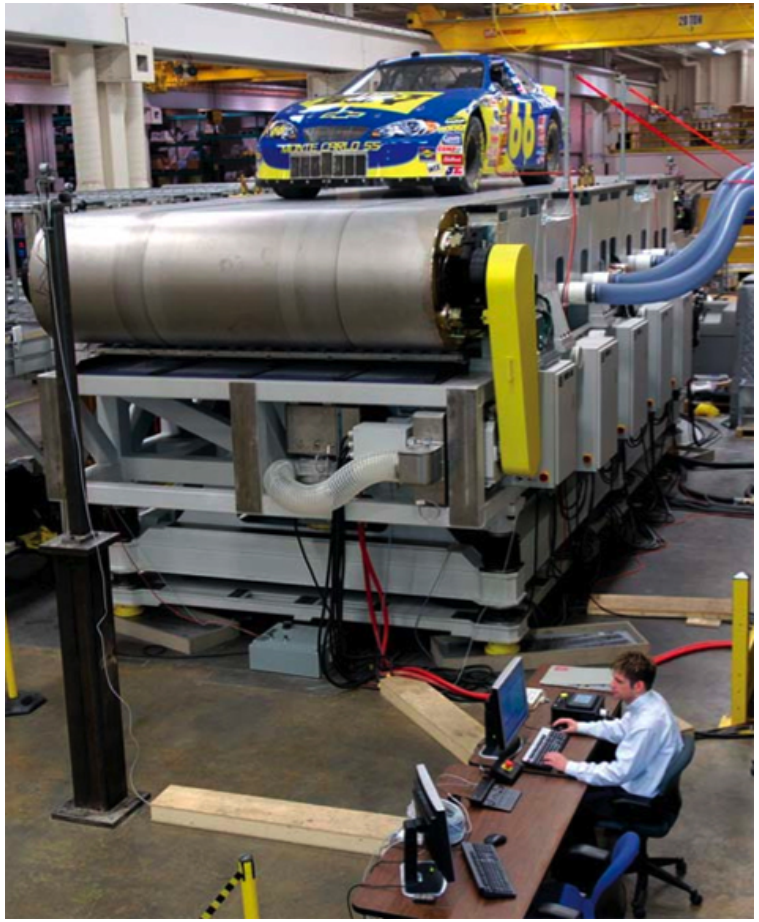
Aerodynamics
Windtunnels

Rotating Wheel Problem

• When a moving belt floor is used, wheels can be rotated at the correct speed via contact.

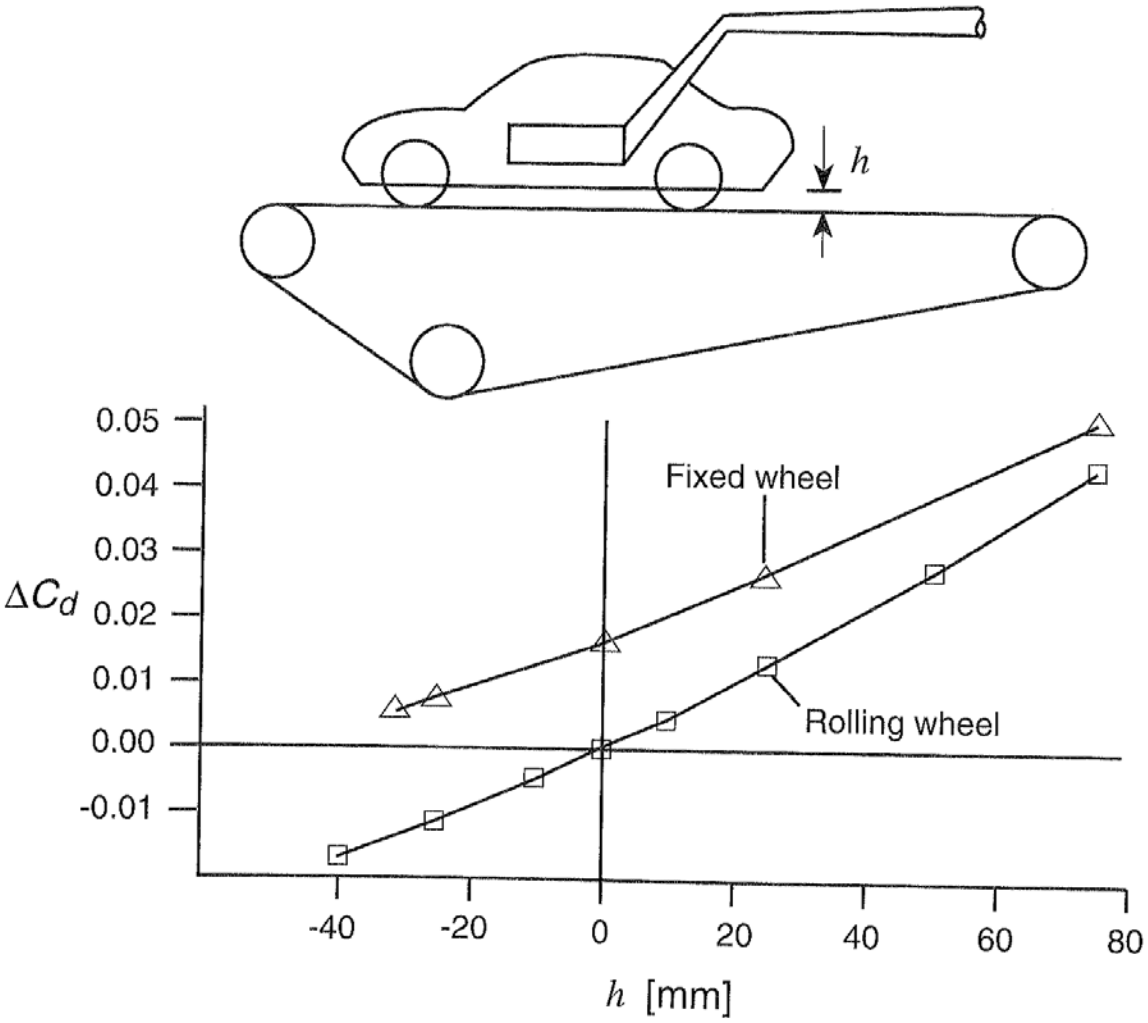
• Rolling resistance of wheel must be measured with zero air flow and then subtracted during tests.

•Very good for simulating tyres as distortion is replicated.



Aerodynamics
Windtunnels

Rotating Wheel Problem



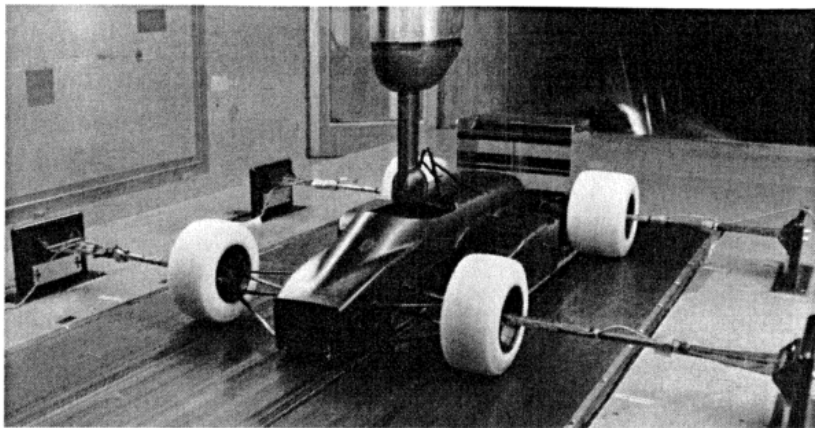
Aerodynamics
Windtunnels

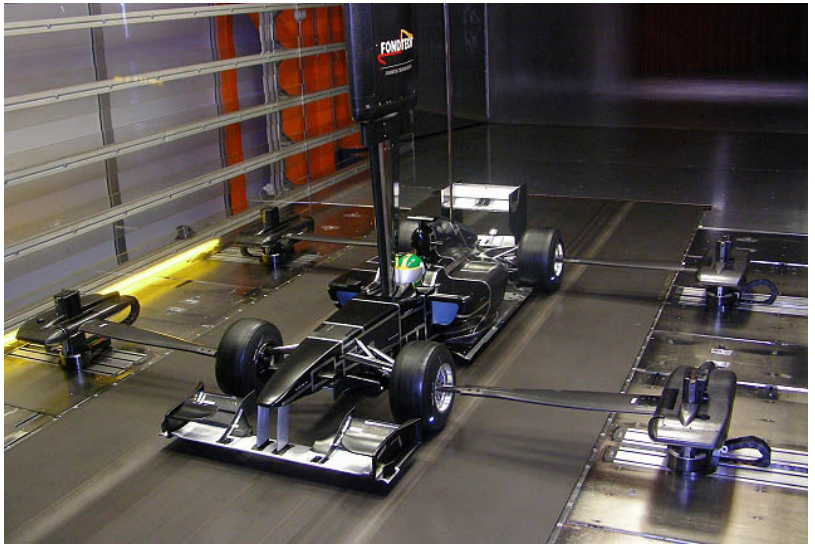
Force Measurement

- Forces are usually measured by stings which support the model in the wind tunnel

- Stings measure the lift, drag and aerodynamic moments such as yaw and pitch moments

- Pressure pads can also be used to measure the wheel loads but must be have a small enough area to not pick up the pressure in the flow.

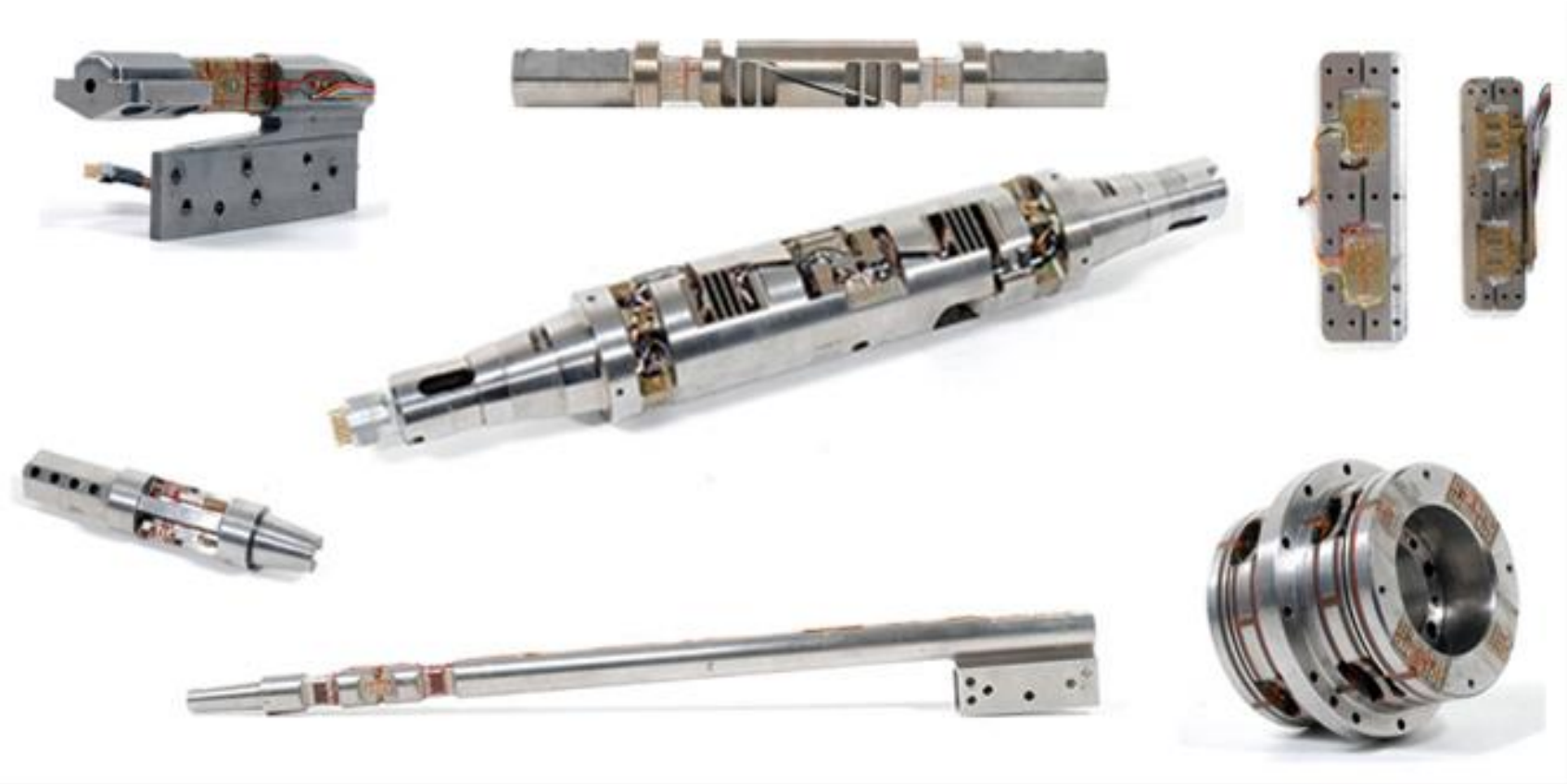




Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

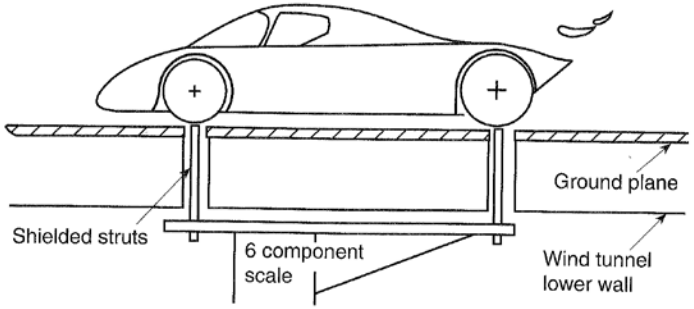
Aerodynamics

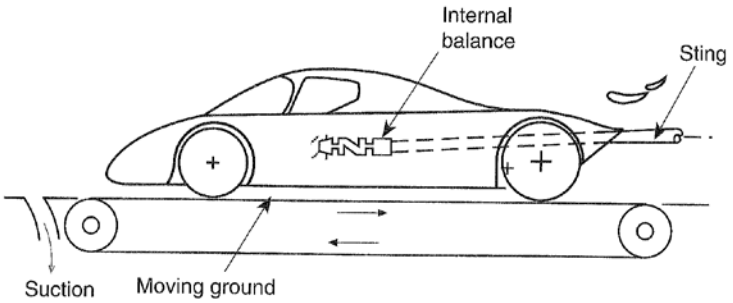
Windtunnels

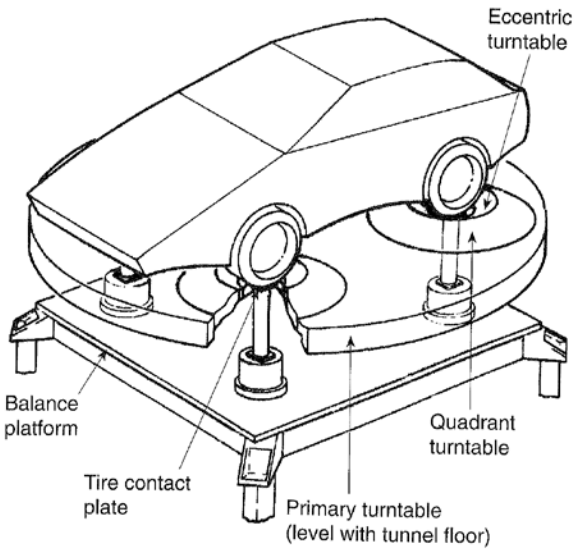


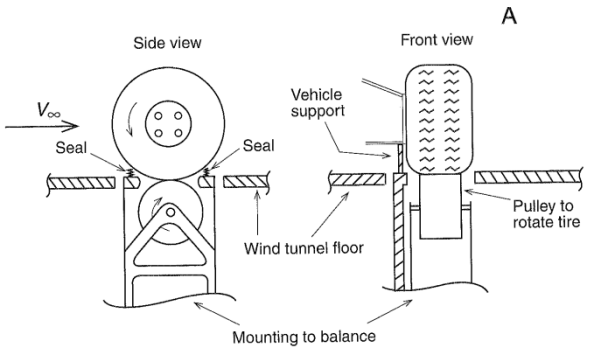
Aerodynamics
Windtunnels

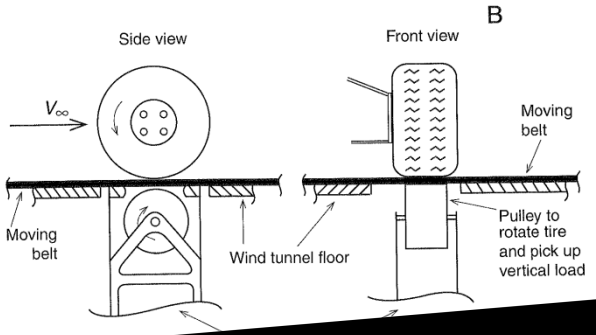
Force Measurement – Mounting Vehicles





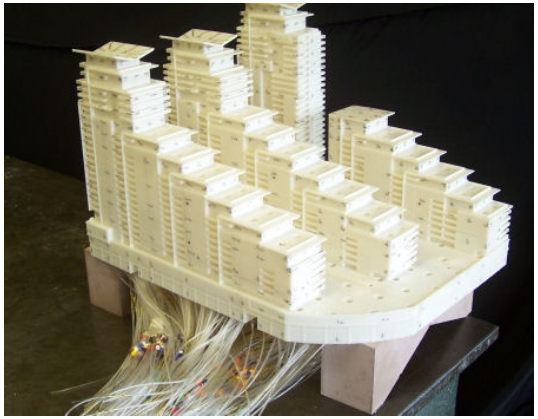






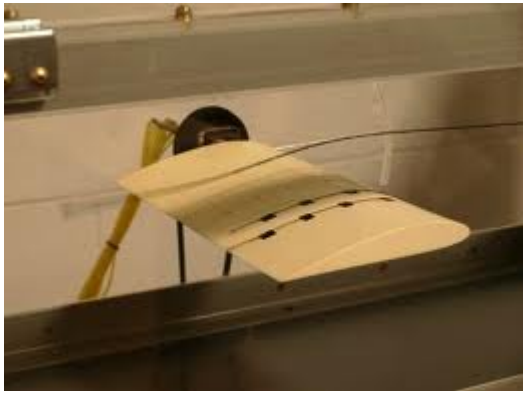
Aerodynamics
Windtunnels

Pressure Measurement



• Pressure taps are often used to measure pressure on a given aerodynamic surface.

•Pressure taps are small holes on the surface which are attached to pressure transducers via tubes.



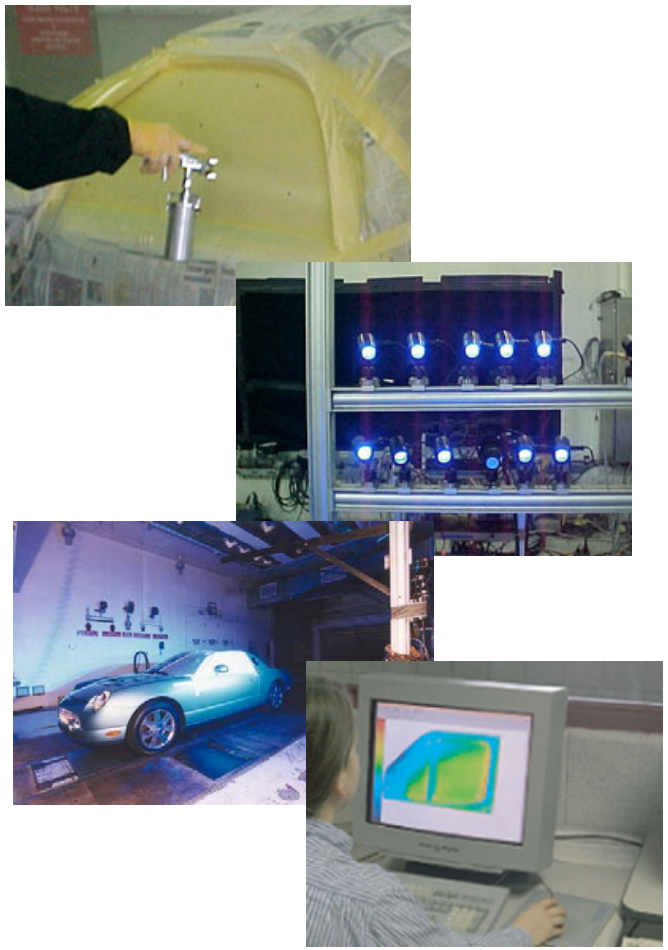
•Pressure transducers change pressure into voltage signals and these can be monitored so that the pressure can be calculated.

• Normally a large number of pressure taps are required and transducers are expensive so a scanning transducer is used. A scanning transducer can look at a number of pressure taps in turn and log the voltage at each.

Aerodynamics
Windtunnels

Pressure Measurement

- Pressure sensitive paint –
  - Paint applied to surface
  - Light source illuminates surface
  - Camera photographs with tunnel off and on
  - Computer produces pressure map from captured images



Aerodynamics
Windtunnels

Velocity Measurement

- Hot wire anemometry –
  - Moving fluid over hot-wire cools wire
  - Hold wire temp constant and measure required energy to do so
  - As flow velocity increases, power required increases

- Laser doppler anemometry (velocimetry) (LDV)–
  - 2 beams of light focused at one point in flow
  - Particles which pass beams scatter light according to speed
  - Fluid velocity not directly measured so particle velocity must be similar to fluid velocity

- Particle image velocimetry (PIV) –
  - Photographic measurement technique
  - Particle photographed twice a small time period apart
  - Distance particle covers during known period allows calculation of velocity

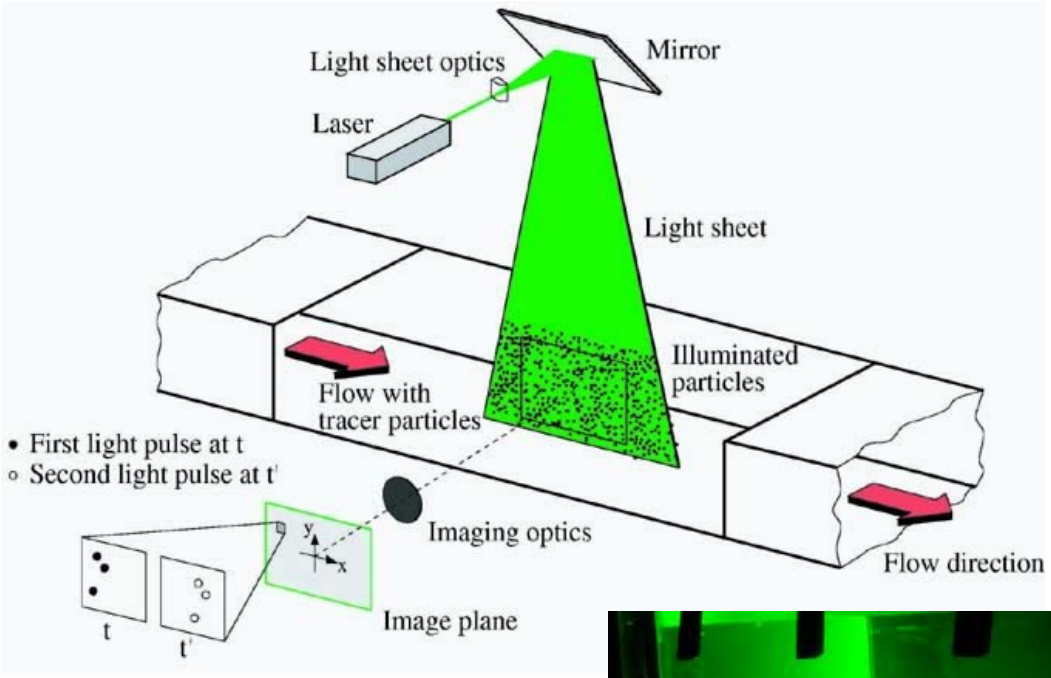
Aerodynamics
Windtunnels

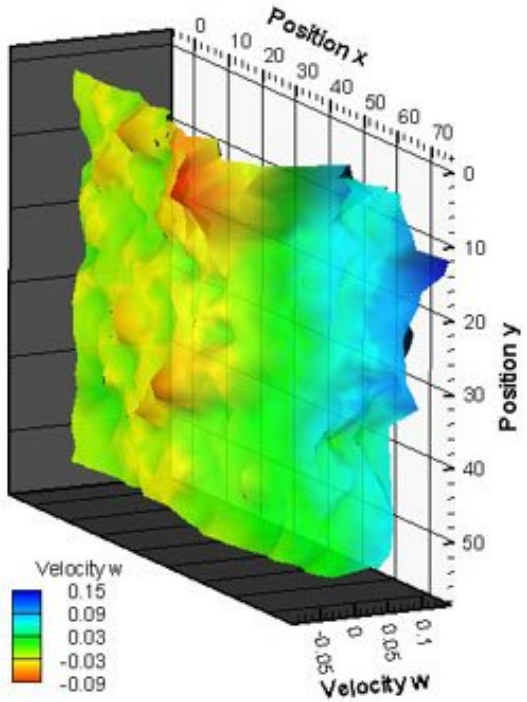
Comparison of techniques

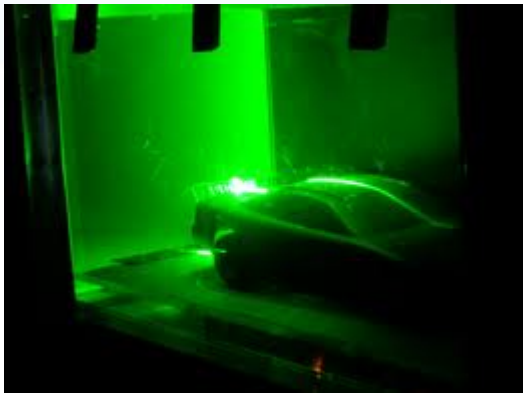
Method	Key Features	Advantages	Disadvantages
Pressure – Manometers, gauges, transducers	All types measure pressure difference	•Simple to source and get results	•Measure difference in pressure only. •Limited number of points measures
Pressure – Pressure sensitive paint	Paint applied to surface and photographed	•Pressure over entire surface •Non intrusive	Expensive Slow response Coating degrades with time Temperature sensitive
Velocity – Hot-wire anemometry	Use of fine wire kept at constant temperature.	•Accurate when calibrated •High frequency response	Intrusive probe Requires constant calibration Wire breaks often
Velocity – Laser Doppler Anemometery	Beams of light illuminate particles in flow	•Accurate •Non-intusive	Expensive Flow needs seeding Very few measurement positions Optical access needed
Velocity Particle image velocimetry	Photographs of particles determine distance travelled.	•Velocity at many positions measured simultaneously •Accurate •Non-intrusive	Flow needs seeding Expensive Optical access needed

Aerodynamics
Windtunnels

Comparison of techniques : PIV



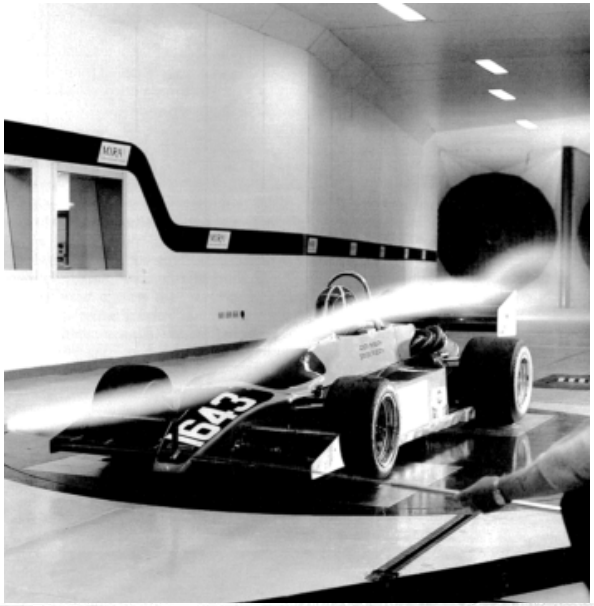




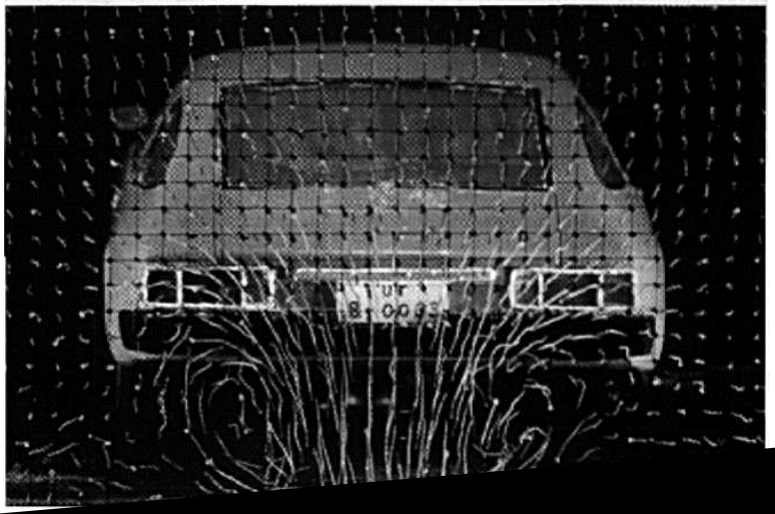
Aerodynamics
Windtunnels

Flow Visualisation

- Smoke Visualisation –
  - Does have it's uses, although is limited.
  - Does give an idea where separation, turbulence, recirculation and re attachment occurs.
  - Hard to use in closed return tunnels



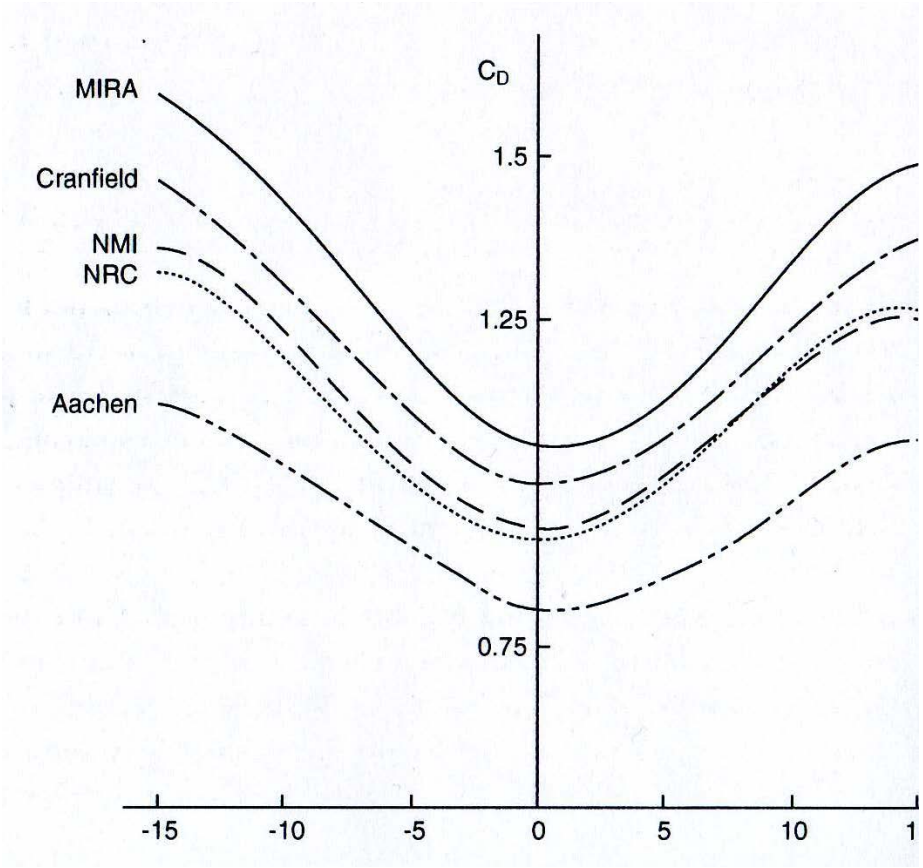
- Flow visualisation paint may also be applied to aerodynamic surfaces which accumulates at separation points.



- Wool tufts allow an inexpensive and reliable visualisation technique to assess whether flow is laminar or turbulent

Aerodynamics
Windtunnels

Comparative Tests





Aerodynamics
Windtunnels

Summary

- Dedicated features of an Automotive wind tunnel
  - It should ideally be large enough for a full size vehicle
  - A moving belt should be provided (or a means to reduce Reynolds number effect)
  - Circumferentially slotted working section or adaptive walls help reduce blockage effect

Aerodynamics
Windtunnels

CFD V Experimental methods

Area	Computational Methods	Experimental methods
Capability	•Software used for all flow types •Turbulence rarely fully resolved •Enables unsafe situations to be modelled •Quick geometry variation	•Seen as 'real thing' •Exact simulation if full scale situation used •Scale effects can alter results
Accuracy	•Depends on algorithm used and computer resources •Mesh and boundary condition dependant	•Should be correct within experimental error
Detail	•All variables calculated at every point or cell •Variables integrated to find totals	•Easy to find overall properties •Difficult and expensive to instrument •Limited measurement points
Cost	•Relatively low cost hard and software •Time and care required •Trained operators required	•Instrumentation expensive •Data is limited for cost •Time and care required
Time	•Solutions can take long time •Problem and mesh dependant	•Calibration and set-up time consuming •Results come relatively quickly once set-up

Aerodynamics
General Vehicle

Whole Body Aerodynamics

- We will examine generic body shapes for vehicles
- These can be quite different as shown on the right!
- **Sedan Based** – GTO, Touring, NASCAR – Only minor aerodynamic modifications allowed from passenger car.
- **Enclosed Wheel** – Prototypes, Lemans – Designers dream – Aerodynamic rules very open.
- **Open Wheel** – Indy, F1, F2 – Tight regulations, generally 2 wings front and rear, narrow bodies may have under floor tunnels.









Aerodynamics
General Vehicle

Whole Body Aerodynamics

- Typical objectives for good aerodynamic design :
  - Reduction of drag
  - Increase downforce
  - Decrease sensitivity to yaw, roll and pitch

Table 2.3 Lift and Drag Coefficients for Several Road Vehicles

	Year/Make	C_d	C_l	Remarks
Sedans	1973 Opel Record	0.47	0.36	Ref. 2.3
	1980 Peugeot 305 GL	0.44	0.44	Ref. 2.4, p. 128
	1986 Subaru XT	0.29–0.31	0.10	Ref. 2.5
Sports cars	Porsche 911 Carrera	0.38–0.39	NA	Ref. 2.6, p. 198
	1982 Camaro Z28	0.37–0.38	NA	Ref. 2.6, p. 198
	1982 Corvette	0.36–0.38	NA	Ref. 2.6, p. 198
Race cars	1990 Mazda GTO (rear deck spoiler)	0.51	–0.44	Ref. 2.7
	1991 Mazda GTO (rear wing)	0.48	–0.53	Ref. 2.7
	1973 Porche 917/30	0.57	–1.04	Ref. 2.8 p. 170
	1985 generic prototype	0.74	–1.79	Ref. 2.9
	generic Porsche 962 C	0.80	–4.80	Ref. 2.10
	1992 Mazda RX-792P	0.70	–3.80	Ref. 2.11
	1992 Nissan P35, C	0.50	–3.00	Say so
	1983 generic F-1, no sidepods	1.07	–0.99	Ref. 2.12
	1987 March Indy	1.06	–1.71	Ref. 2.13
	1991 Penske PC20, high downforce	1.11	–3.33	Ref. 2.14
	1991 Penske PC20, speedway	0.740	–2.073	Ref. 2.15
	1992 Galmer G92, high downforce	1.397	–3.688	Ref. 2.15
	1992 Galmer G92, speedway	0.669	–1.953	Ref. 2.15
	2000 Ferrari F-1 (high downforce)	0.99	–2.85	Ref. 2.22
	2002 Dodge NASCAR	0.39	–0.42	Ref. 3.13
	2004 Lola Indy car (high downforce)	0.96	–2.75	NA
	2004 Audi R8	0.54	–2.60	NA

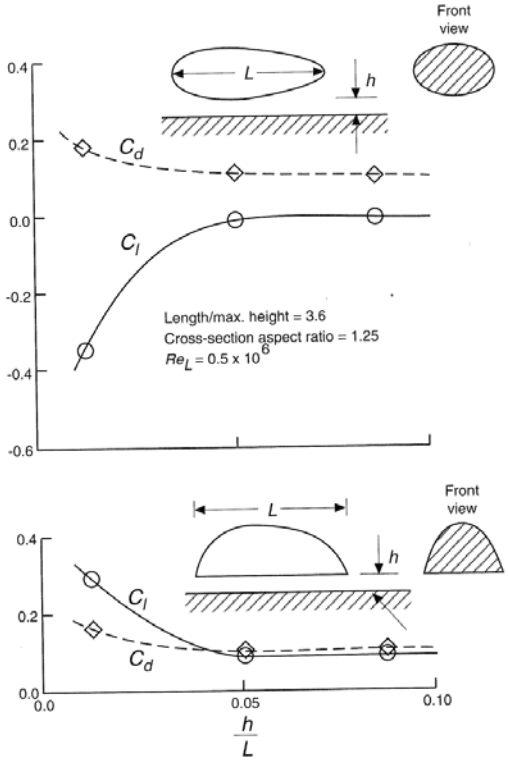
NA = not available

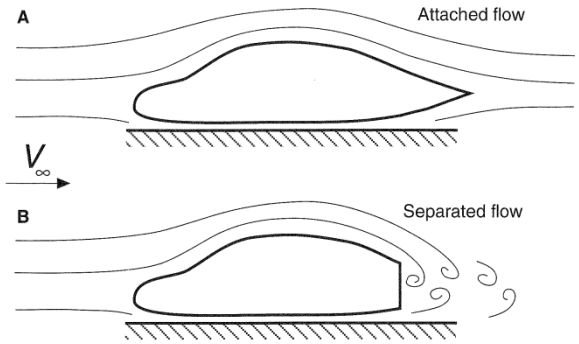
Aerodynamics
General Vehicle

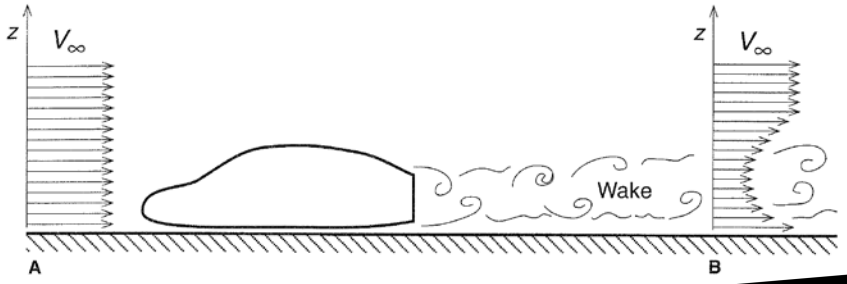
Generic Vehicle Shapes

- Positive or negative lift may be generated by a body in close proximity to the ground depending on shape

- Drag is primarily a result of a blunt rear end shape resulting in local separation.



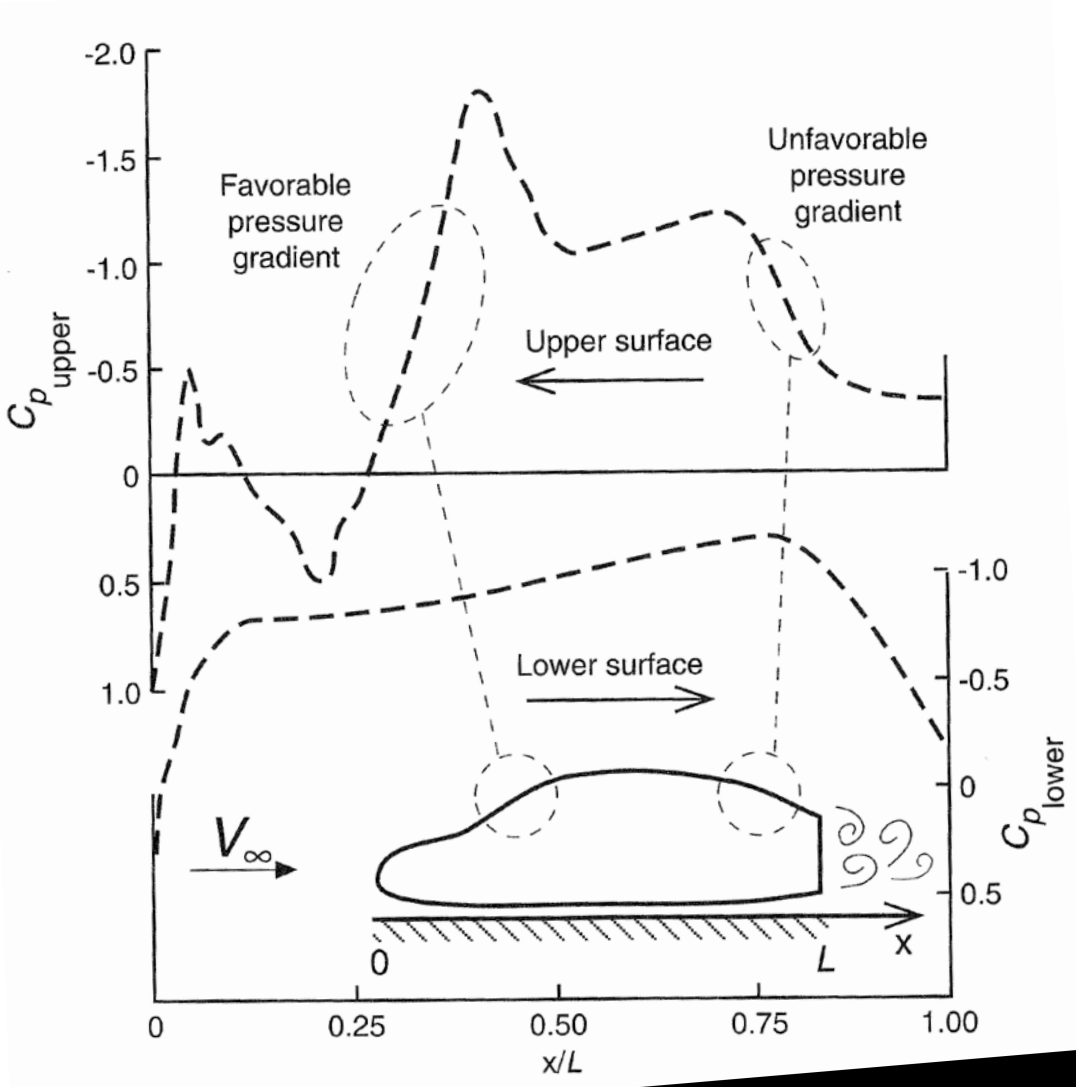




Aerodynamics
General Vehicle

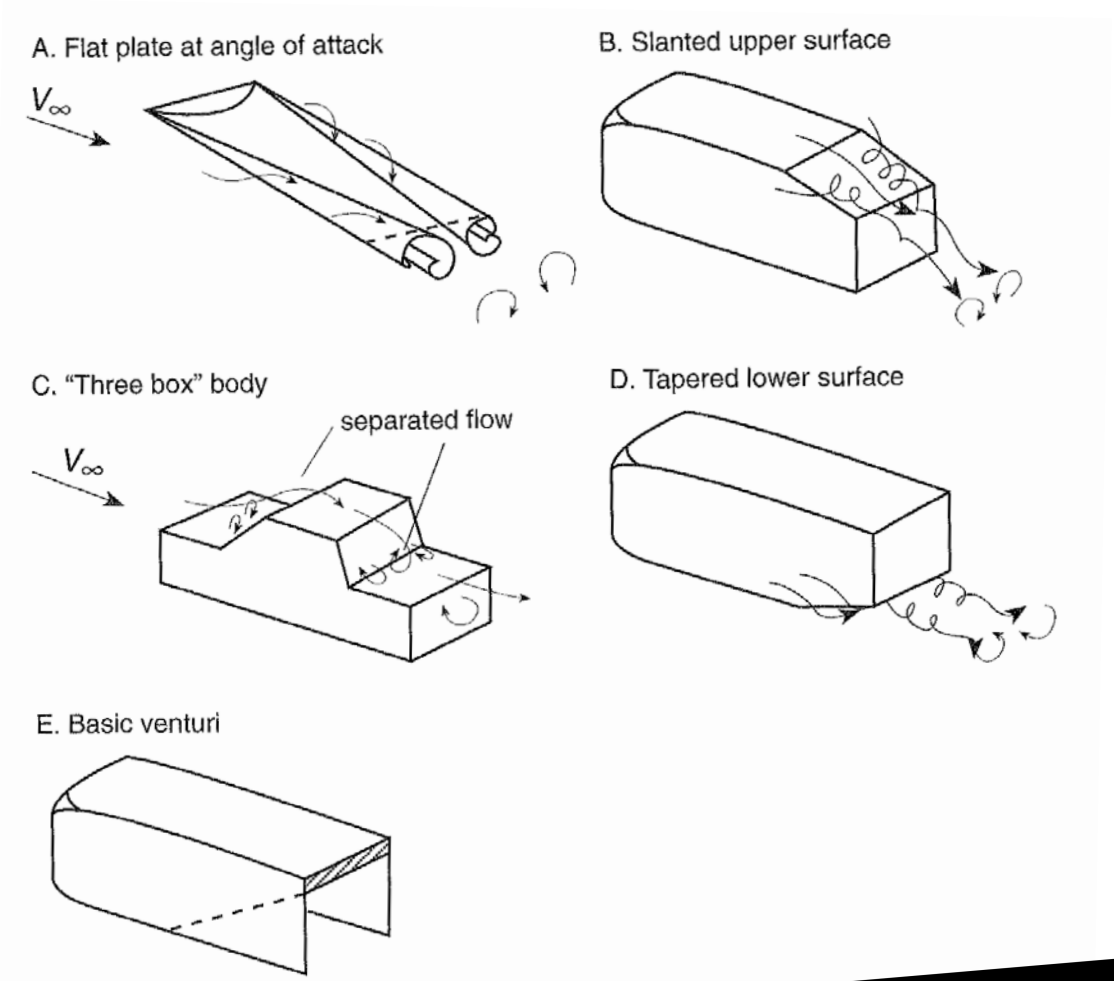
Generic Vehicle Shapes

- Pressure distribution of a generic vehicle body near the ground
- Notice unfavourable pressure gradient may cause separation
- Total lift calculated by integrating pressures over upper and lower surfaces



Aerodynamics
General Vehicle

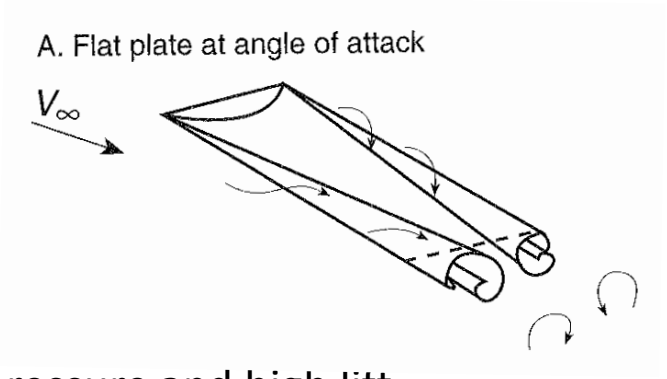
Generic Vehicle Shapes – typical separated flow patterns



Aerodynamics
General Vehicle

Generic Vehicle Shapes – Slanted Rear Upper Surface

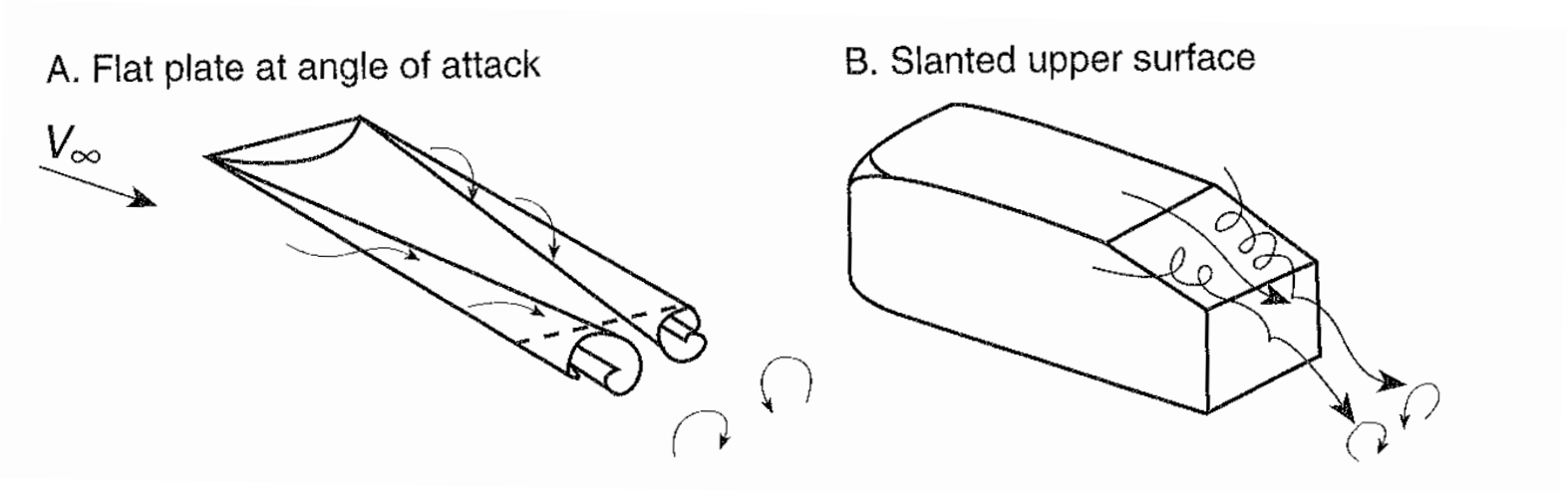
- Vehicle bodies often are not highly streamlined shapes and may have separation of flow towards rear end of vehicle.
- Consider Flat thin plate in flow at >10 degree AoA
- Side edge vortices dominate nearby flow field.
- Vortices induce high velocity which results in low pressure and high lift.
- Effect occurs at a slant angle of $10^\circ < \theta < 30^\circ$



Aerodynamics
General Vehicle

Generic Vehicle Shapes – Slanted Rear Upper Surface

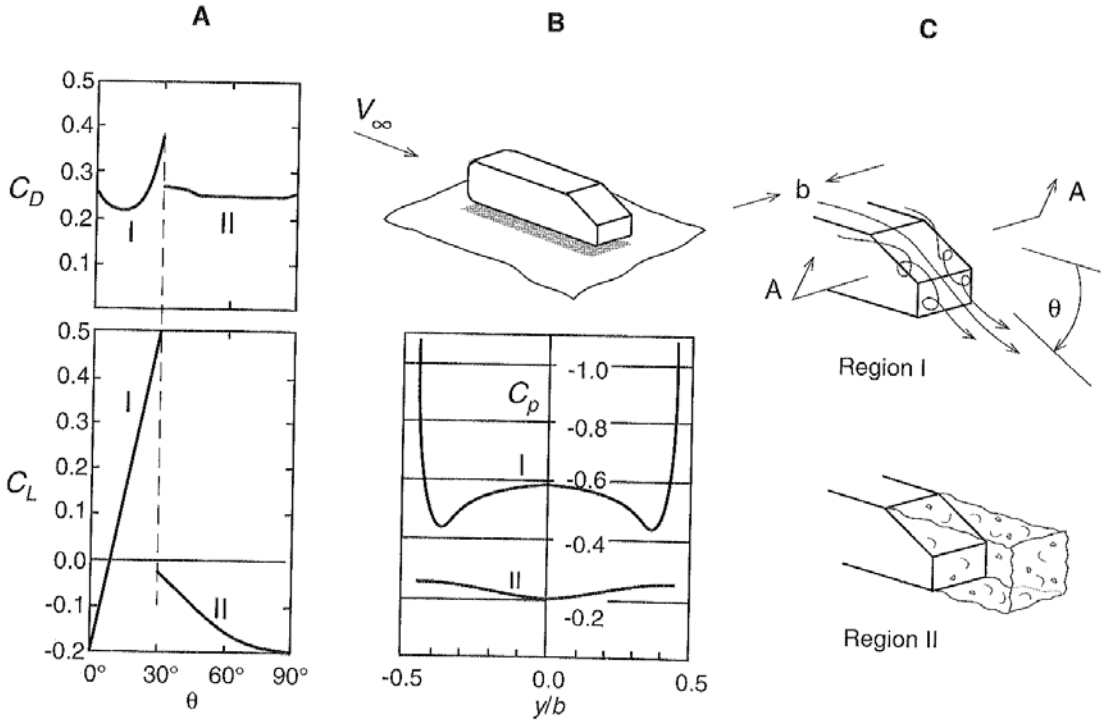
- A similar effect occurs when slanting the rear upper surface of a body.
- Lift will be produced by slanted surface vortices



Aerodynamics
General Vehicle

Generic Vehicle Shapes – Slanted Rear Upper Surface

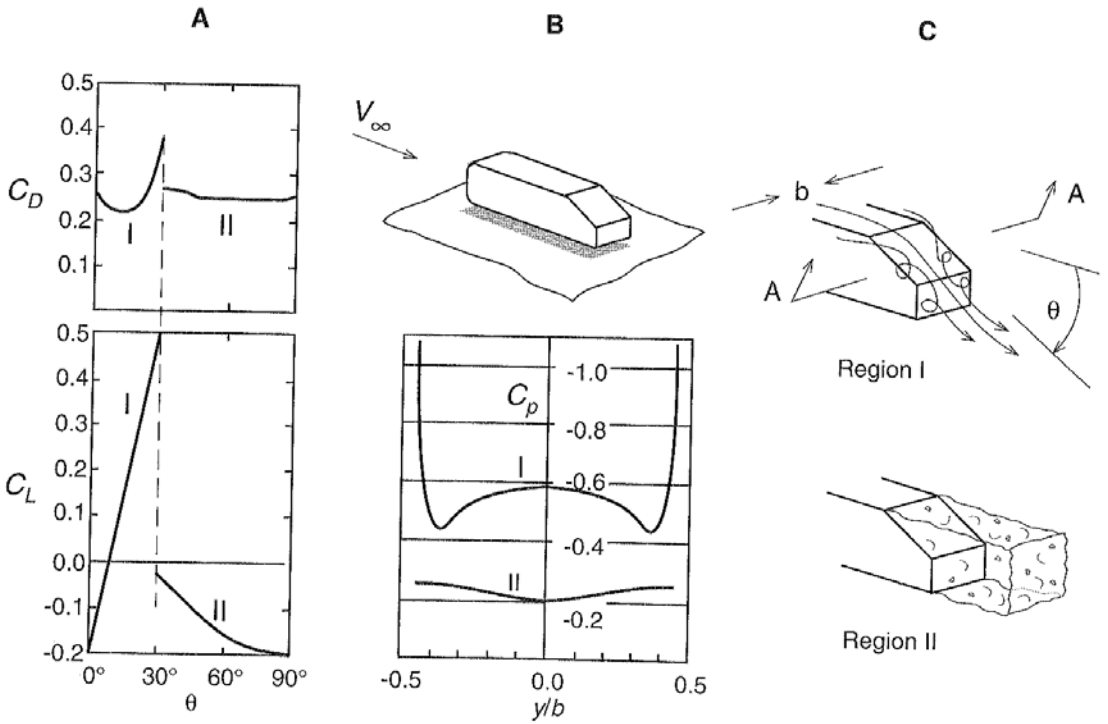
- **A –** Lift and drag versus AoA
- **B** - Lateral pressure distribution across section AA
- Mode 1 (10-30˚) –
  - 2 side vortices attach flow near centreline
  - Lift created
  - Large negative pressure peaks near vortices



Aerodynamics
General Vehicle

Generic Vehicle Shapes – Slanted Rear Upper Surface

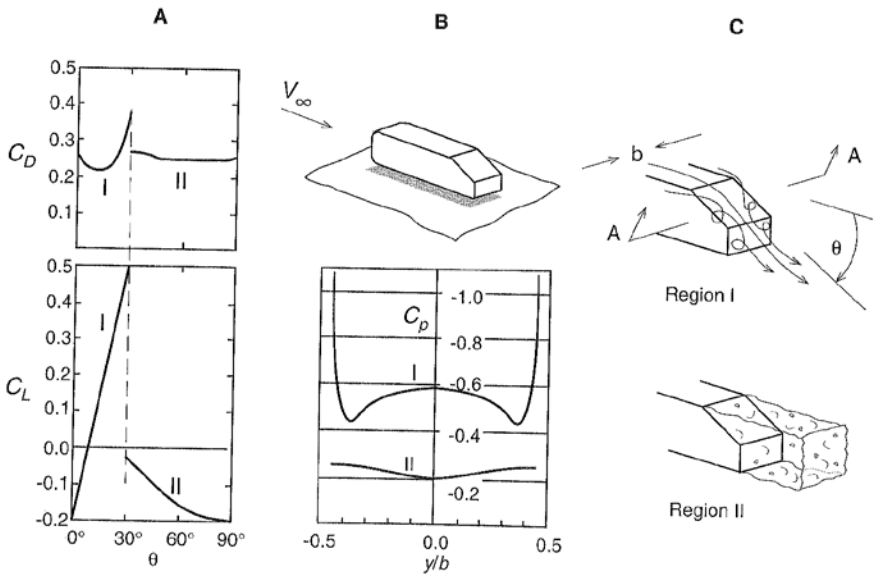
- Mode 2 (>30˚) –
  - Flow separation
  - Flat pressure distribution
  - Drop in lift
- As slant angle is increased
  - Lift increases up to 30˚.
  - Above 10˚ rearward projection of lift increases drag



Aerodynamics
General Vehicle

Generic Vehicle Shapes – Slanted Rear Upper Surface

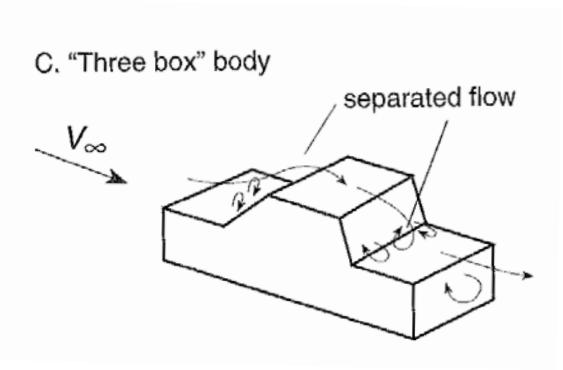
- Most important observation:
  - Above 30˚ vortex structure breaks down
  - Drag and lift contribution of slanted surface decreased
- Hatchback design – Rear window should either be less that 25 or more than 35˚ slant. Away from critical value for consistency.
- Note at θ=0˚, we have a basic body, which will have negative lift due to ground effect.



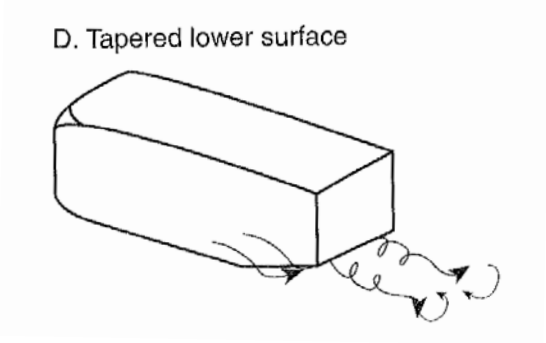
Aerodynamics
General Vehicle

Generic Vehicle Shapes – typical separated flow patterns

- 3 Box type ‘sedan’ body –
  - Separation bubble with recirculating flow observed at bottom of windscreen
  - Separation bubble also observed at bottom of rear window



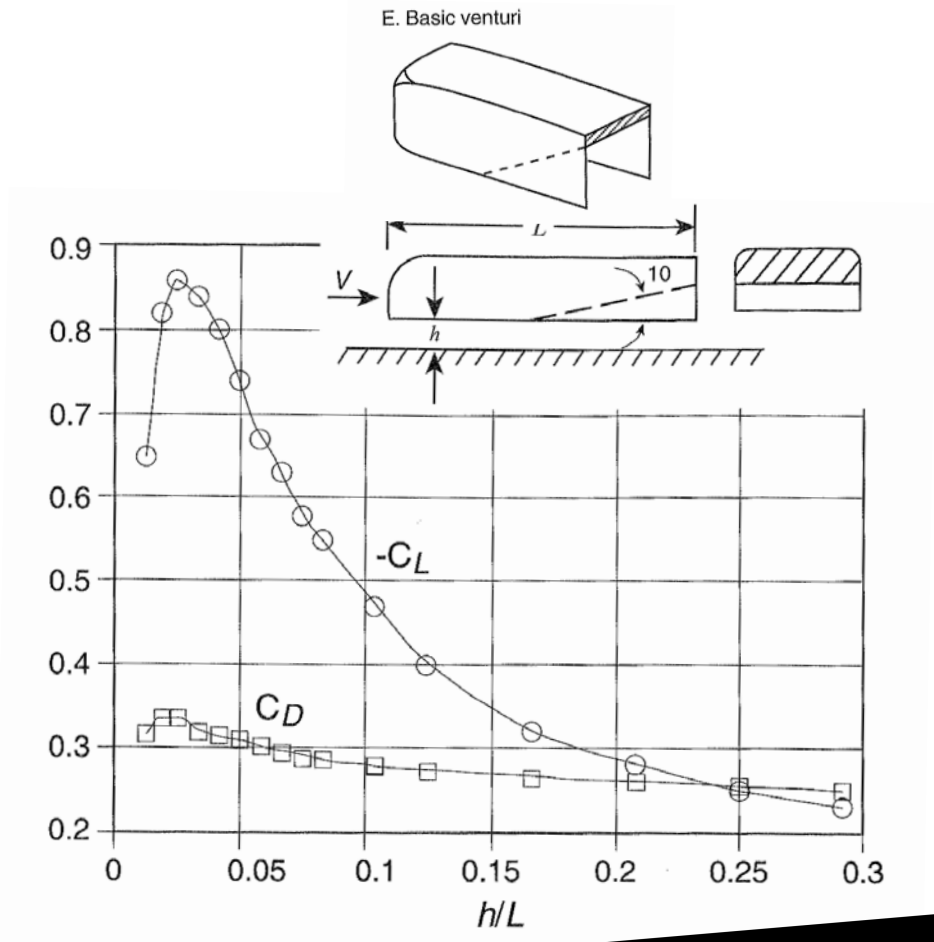
- Bodies with a slanted rear lower surface
  - Yield a similar result as the slanted upper body case, where a lift force will be created by vortices.
  - Downforce increased up to a slant angle



Aerodynamics
General Vehicle

Generic Vehicle Shapes – typical separated flow patterns

- If side fins are added :
  - Create Venturi
  - High downforce can be achieved
  - Only moderate increase in drag
  - Downforce increases with decreasing ground clearance

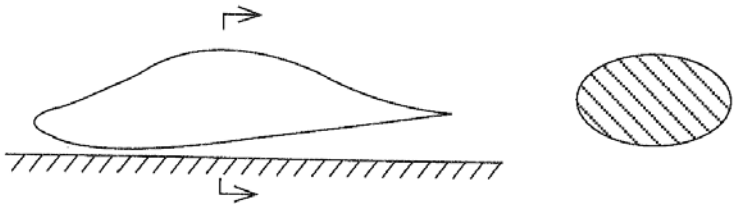


Aerodynamics
General Vehicle

Car orientated aerodynamic shapes

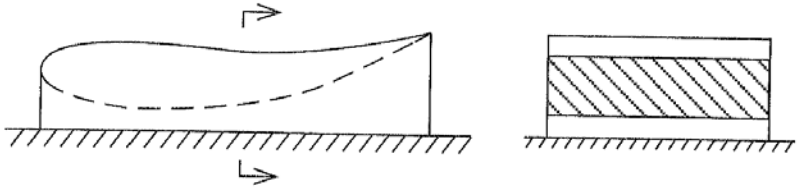
- Shapes which can generate downforce without large drag.

A. Low-drag body



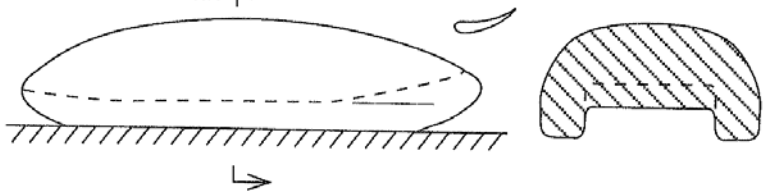
- Encourage laminar flow.
- Generates some lift/downforce

B. Wing in ground effect



- Inverted wing in ground effect
- Side fins increase downforce considerably
- Large downforce, moderate drag

C. Basic catamaran

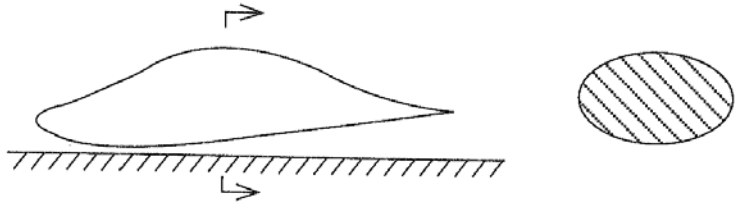


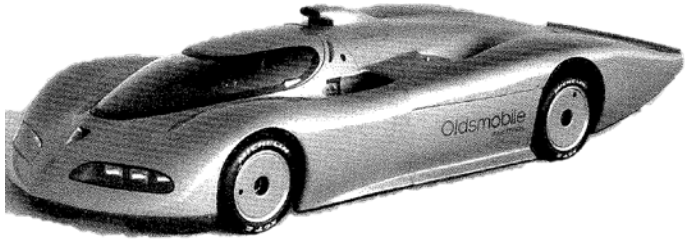
- More realistic car shape ‘catamaran’
- Momentum of free stream channelled under body to reduce separation at rear
- High downforce, low drag

Aerodynamics
General Vehicle

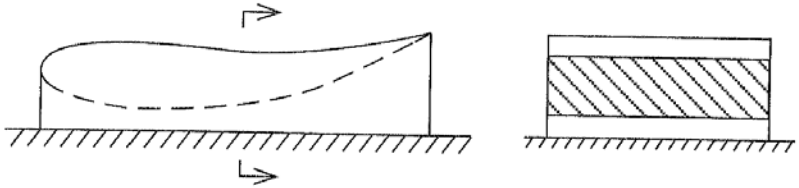
Car orientated aerodynamic shapes

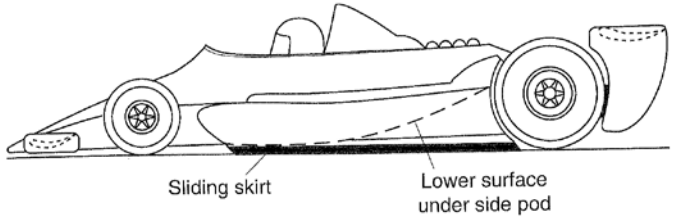
A. Low-drag body



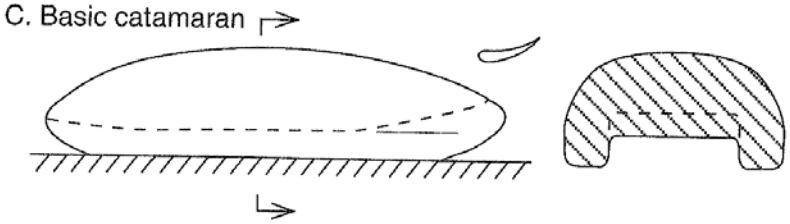


B. Wing in ground effect





C. Basic catamaran

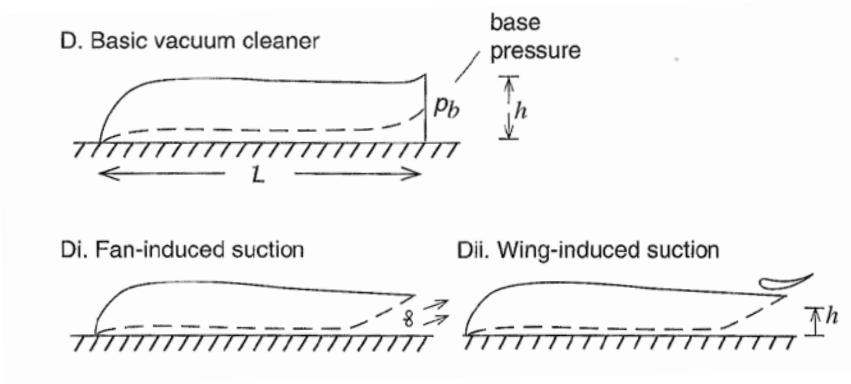




Aerodynamics
General Vehicle

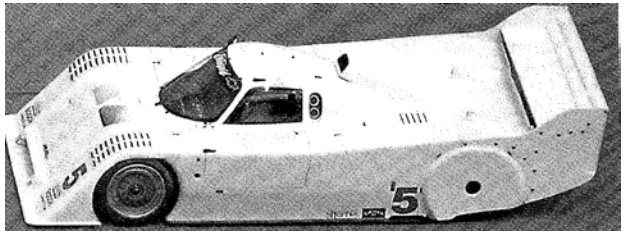
Car orientated aerodynamic shapes

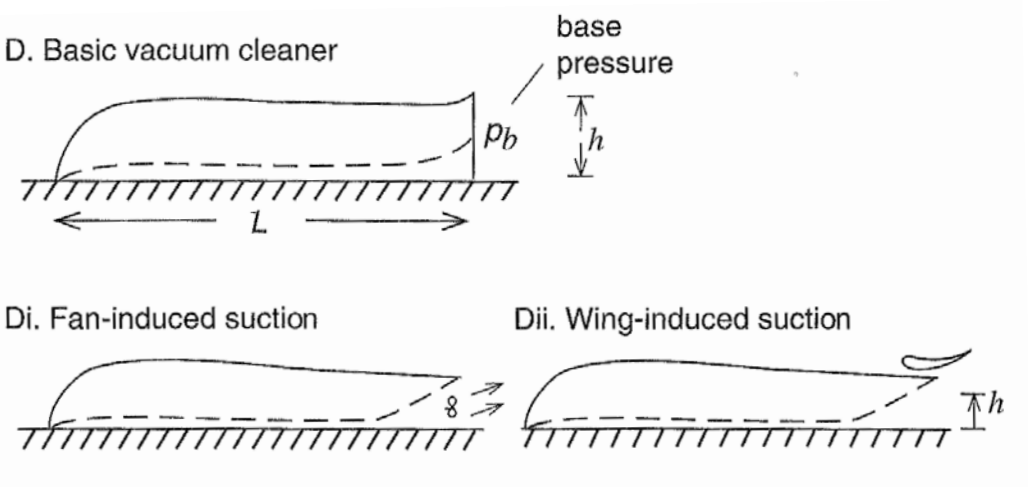
- Vacuum Cleaner concept
  - Seal gap at front and sides of vehicle
  - Airflow under vehicle minimised
  - Negative pressure on lower surface
  - Downforce produced
  - Downforce may be increased by addition of spoiler or fan!
  - Lift/downforce can be approximated by length/height

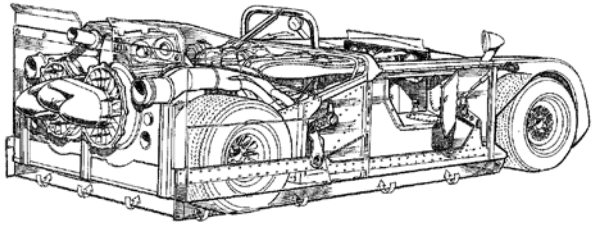


Aerodynamics
General Vehicle

Car orientated aerodynamic shapes



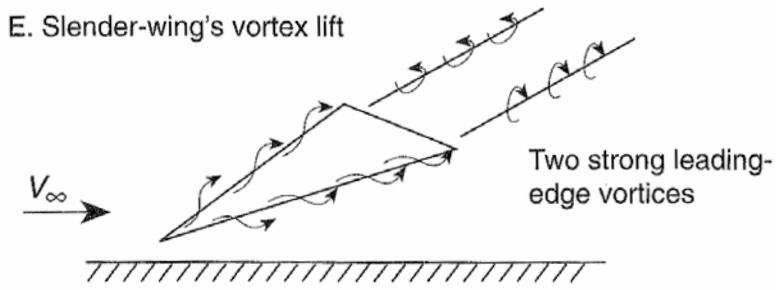


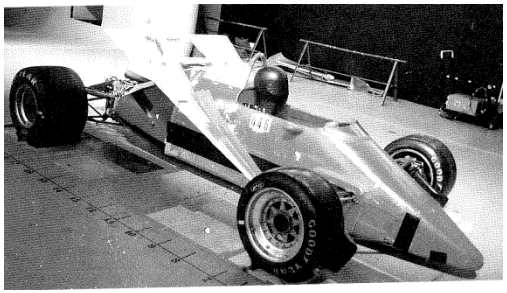


Aerodynamics
General Vehicle

Car orientated aerodynamic shapes

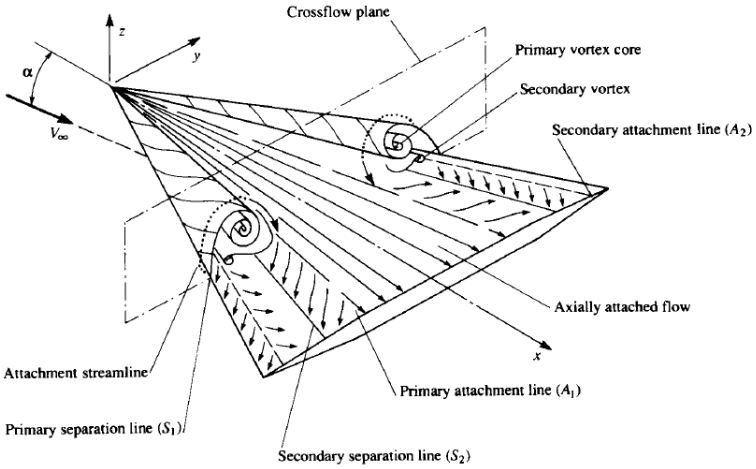
- Vortex generated lift on slender wings may also be used on race vehicle body design
  - Inverted delta wing can create downforce of the order C_L ~ -1.3
  - Disadvantage is vortex wake reduces effectiveness of any rear wing when placed downstream of delta wing.

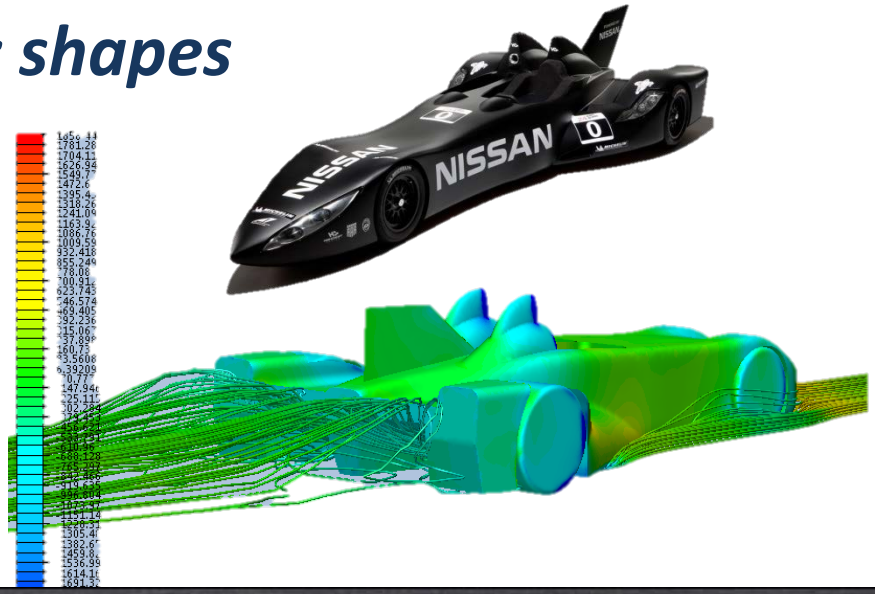




Aerodynamics
General Vehicle

Car orientated aerodynamic shapes



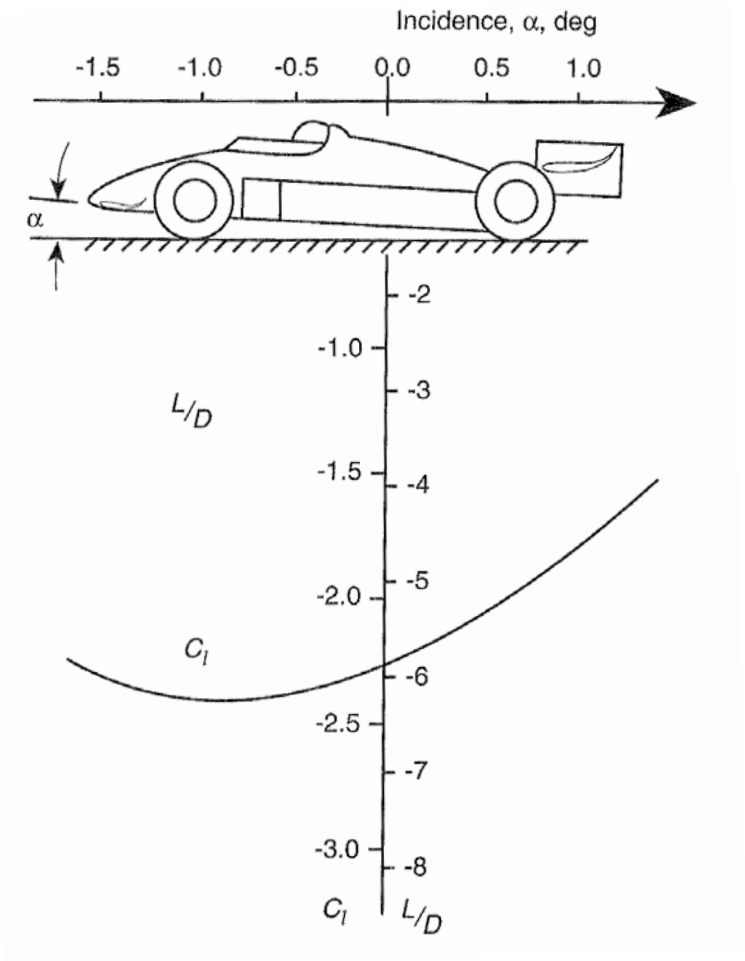




Aerodynamics
General Vehicle

Vehicle Attitude - Pitch

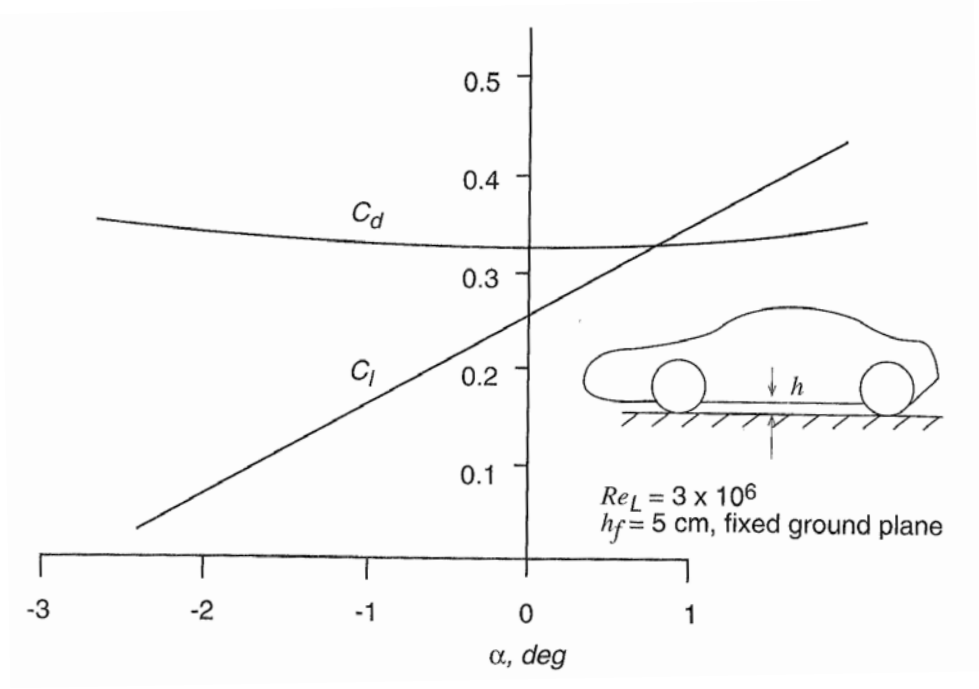
- Pitch of a vehicle can significantly effect overall lift.
- Generally like a wing with an AoA.
- Vehicle shown has low ground clearance so at incidences of less than -1˚ the viscous effects block the under body flow and therefore there is no increase in downforce beyond this point.
- A small negative incidence is desirable as this compensates for boundary layer growth



Aerodynamics
General Vehicle

Vehicle Attitude – Pitch

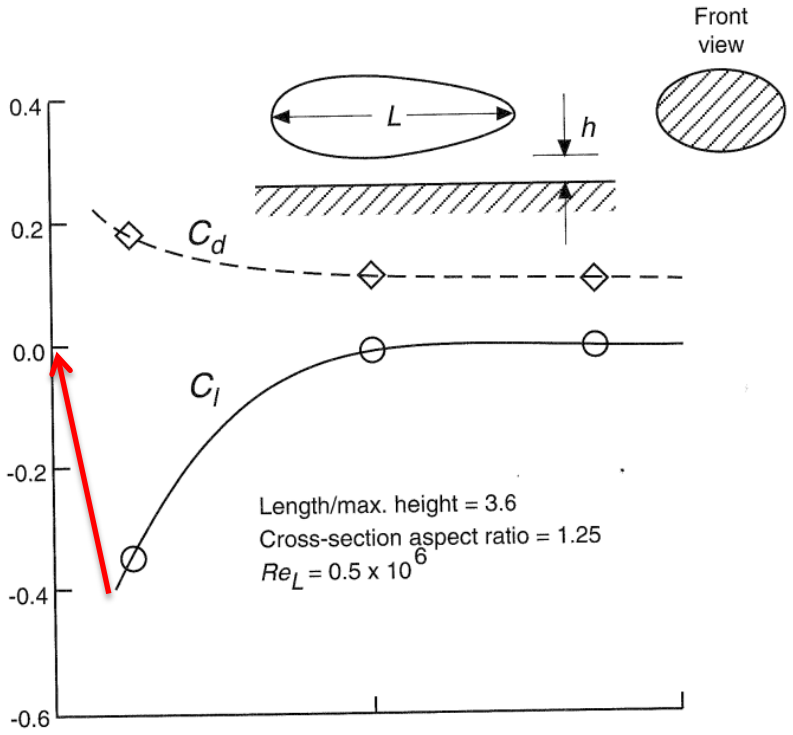
- Vehicle with larger ride height
  - Doesn't suffer from the effect of the viscous blocking the underfloor flow.
  - Higher incidence angle favourable.



Aerodynamics
General Vehicle

Vehicle Attitude - Ride Height

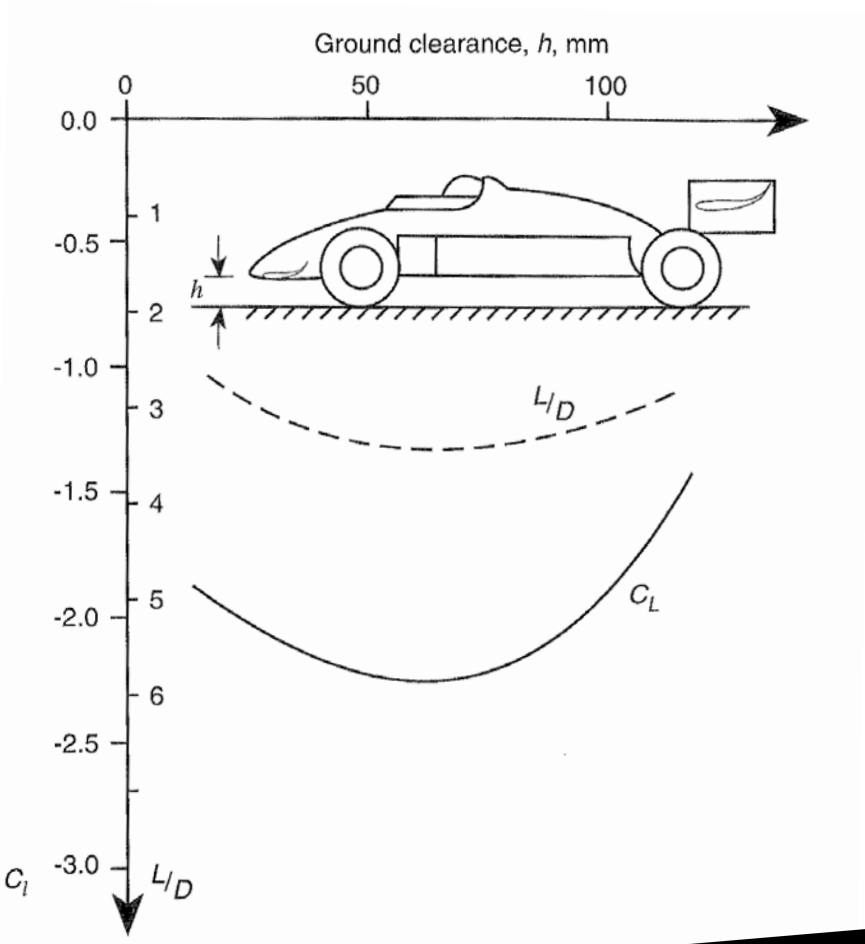
- As ride height is decreased, downforce is generally increased until a critical limit is reached
- As described by previous data of wings and bluff bodies in ground effect.



Aerodynamics
General Vehicle

Vehicle Attitude - Ride Height

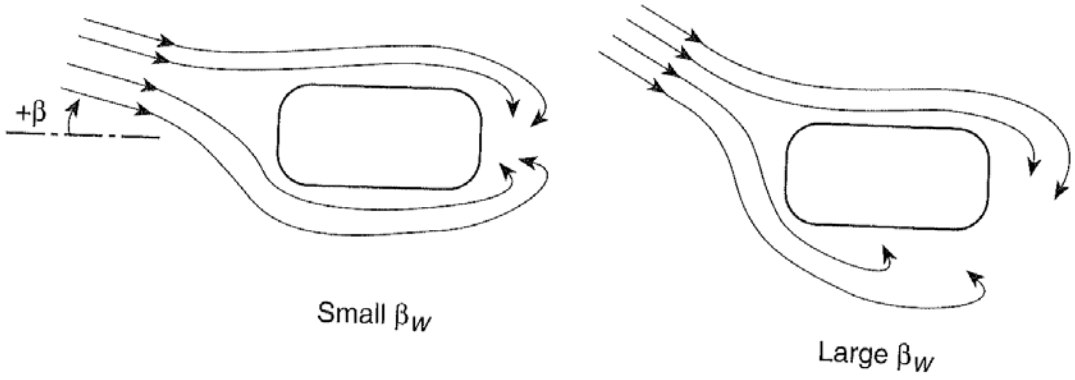
- Graph on right shows how lift varies with ride height for an open wheel vehicle with underfloor tunnels.
- Interesting that maximum lift is at around h=60mm.....



Aerodynamics
General Vehicle

Vehicle Attitude - Yaw

- Vehicle yaw can have a dramatic effect on the flow fields around a vehicle, and therefore the resulting drag and lift.

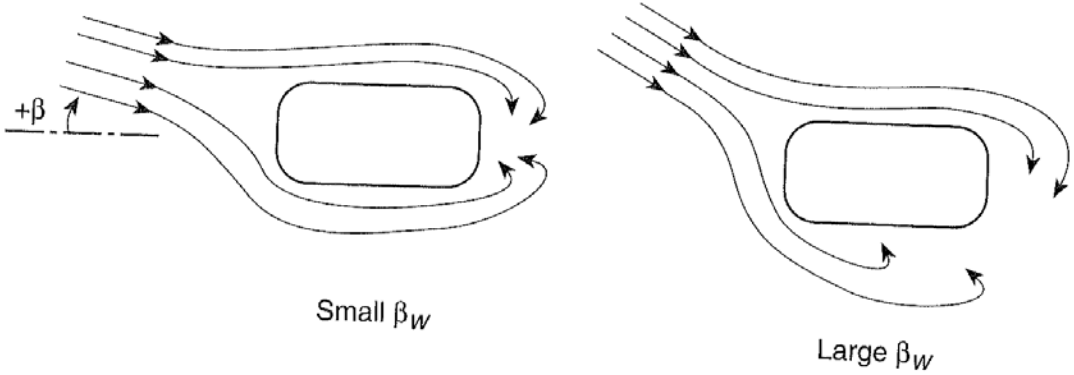


Aerodynamics
General Vehicle

Vehicle Attitude - Yaw

- If flow has a small angle (near-zero slip) of incidence to the vehicle
  - flow remains attached on the vehicles sides
  - pressure distribution resembles that of a thick aerofoil.

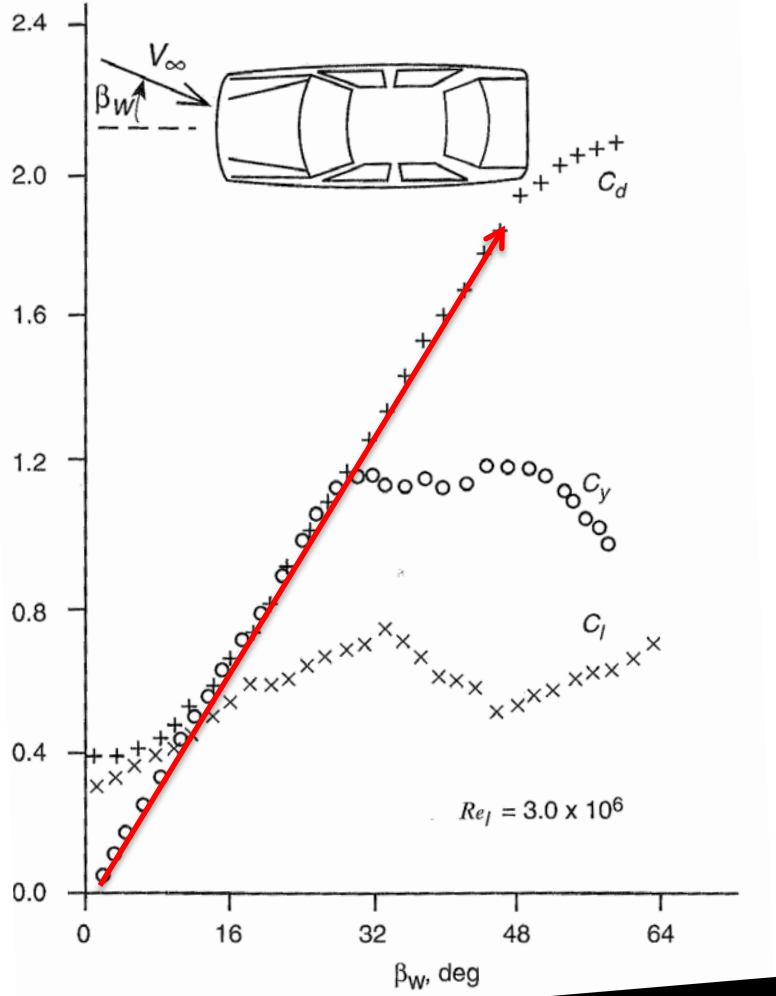
- Larger angle of incidence
  - flow separates from leeward side of the vehicle.
  - This separation combines with separation at rear of vehicle to produce much larger separation bubble.



Aerodynamics
General Vehicle

Vehicle Attitude - Yaw

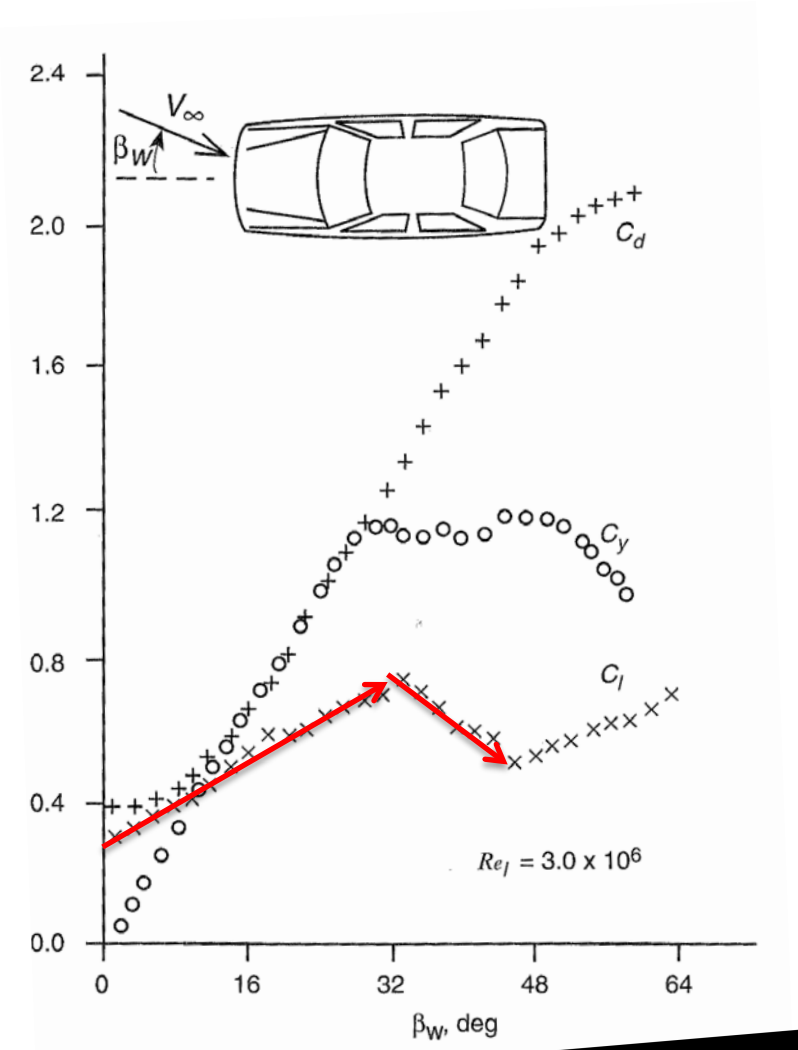
- Large separation bubble results in large zone of low pressure
- Increased angle of incidence also increases effective frontal area
- These combine and result in sharp increase of drag with increase in angle of incidence



Aerodynamics
General Vehicle

Vehicle Attitude - Yaw

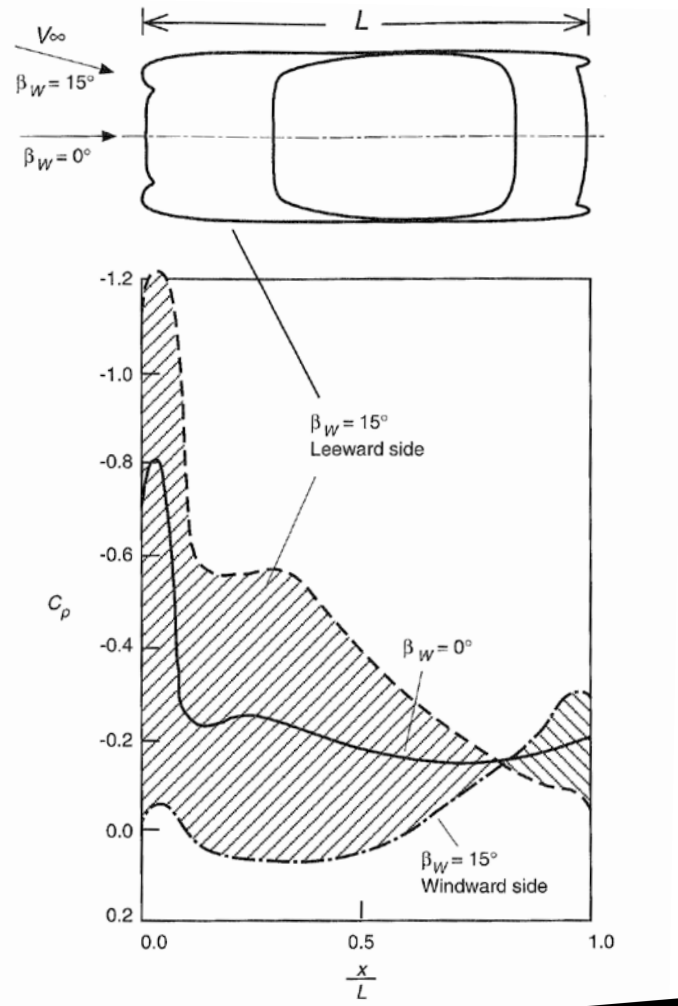
- Separation bubble resulting from yaw generally also increases the velocity above the vehicle
- Small increase in lift produced
- At very large angles of incidence separated flow at rear window is fed momentum by side flow and can cause a reduction of lift.



Aerodynamics
General Vehicle

Vehicle Attitude - Yaw

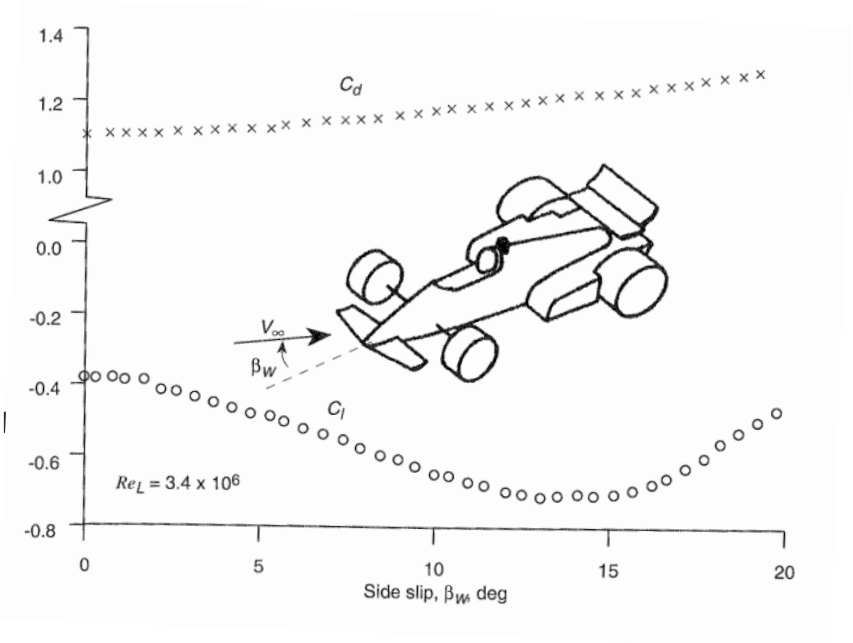
- Diagram shows co-efficient of pressure on each side of the vehicle at
  - 0 degrees yaw
  - 15 degrees yaw



Aerodynamics
General Vehicle

Vehicle Attitude - Yaw

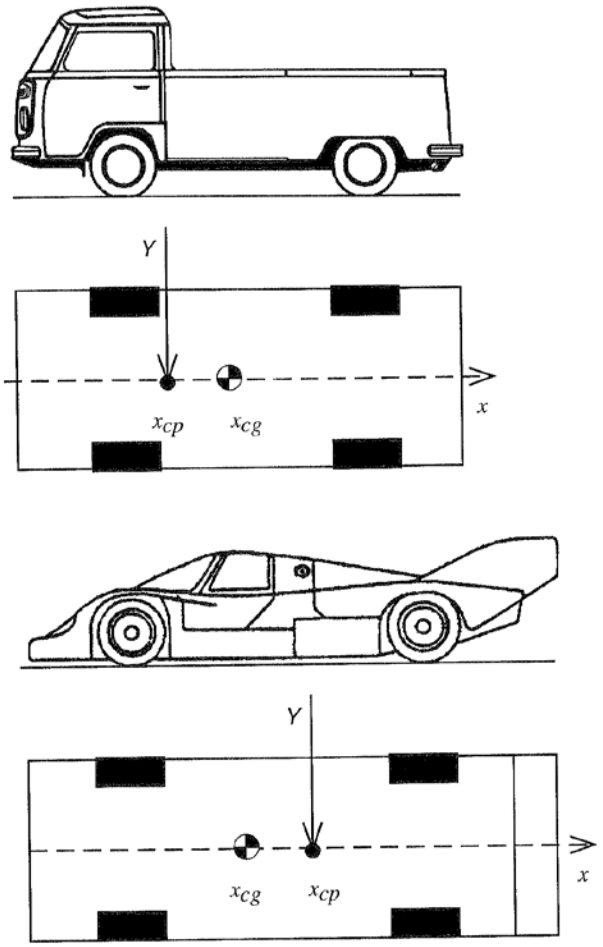
- Data for race cars sparse in open literature
- Some data for a generic flat-bottomed race car shown.
- Generally as angle of incidence increases drag again increases
- However effect is less pronounced as separatio is less as air is not blocked between wheels
- Lift decreases with increasing angle of incidenc due to increases flow under car



Aerodynamics
General Vehicle

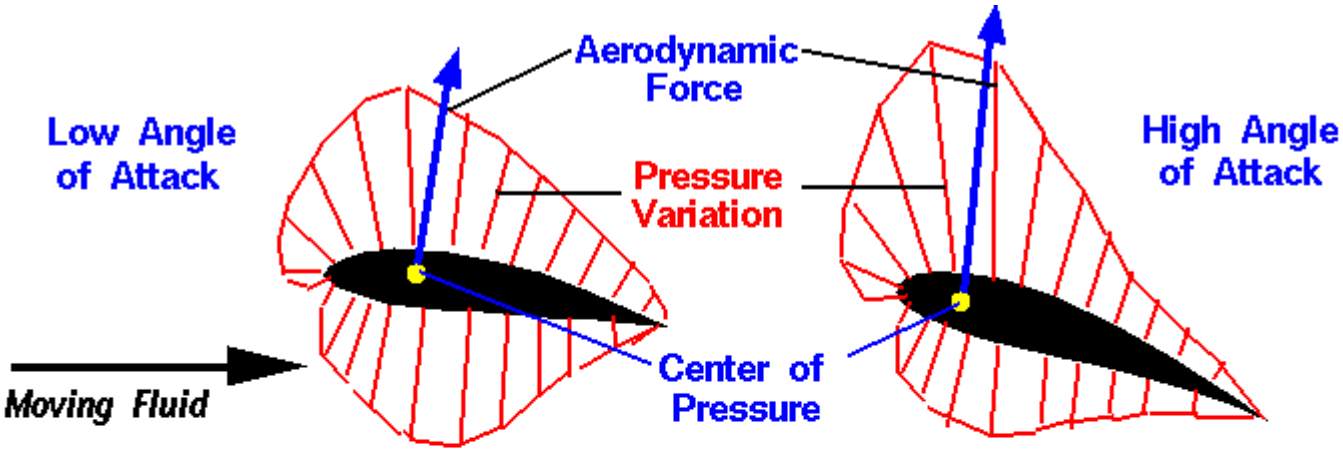
Centre of pressure

- CoP is analogous to centre of gravity
- Point at which you can assume your overall aerodynamic loads act on vehicle.
- Position is a function of the vehicle's geometric shape.
- Passenger cars often have a centre of pressure forward of CoG
- Relative positions only become important at higher speeds where aerodynamic forces are significant.



Aerodynamics
General Vehicle

Centre of pressure



Center of Pressure is the average location of the pressure.
Pressure varies around the surface of an object. P = P(x)

$$cp = \frac{\int x\ p(x)\ dx}{\int p(x)\ dx}$$

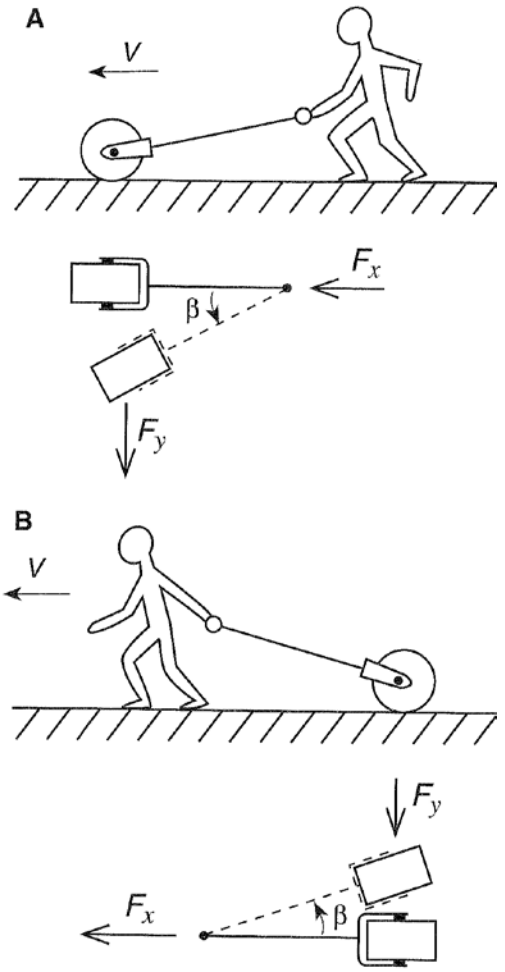
Aerodynamic force acts through the center of pressure.
Center of pressure moves with angle of attack.

Aerodynamics
General Vehicle

Centre of pressure

- If the centre of pressure is forward of the CoG
  - Small disturbance will result in slip angle of incidence
  - Angle of incidence will increase lateral aerodynamic load in direction of slip and increase slip further

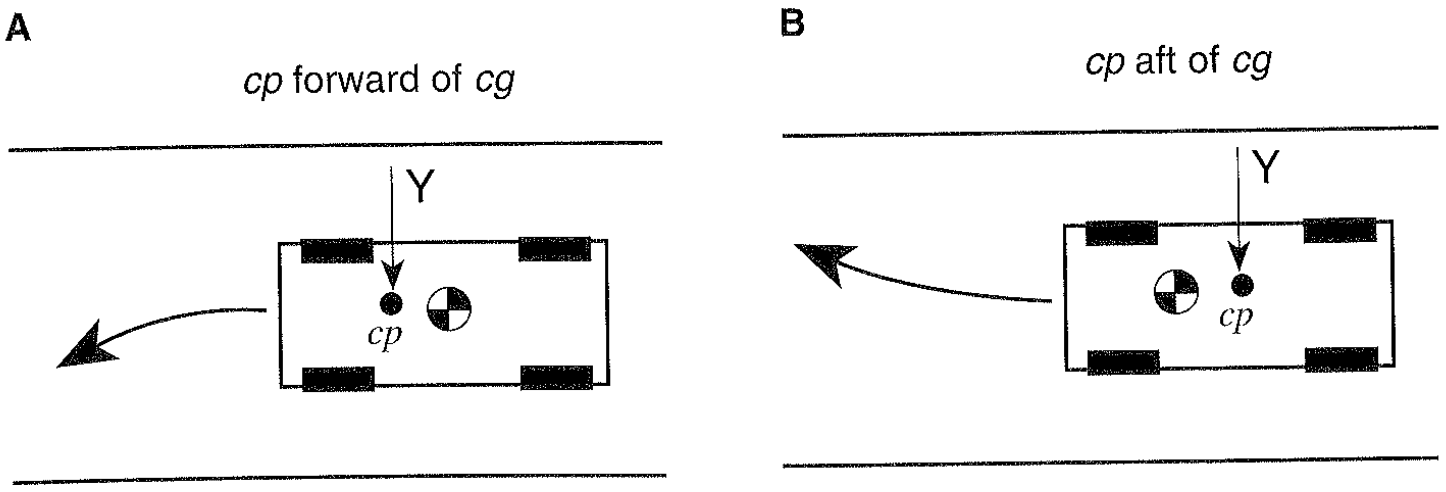
- If CoP is behind CoG
  - Small disturbance will again cause slip angle of incidence
  - Aerodynamic force will be created in opposite direction



Aerodynamics
General Vehicle

Centre of pressure

- Diagram below shows vehicle response to



Aerodynamics
General Vehicle

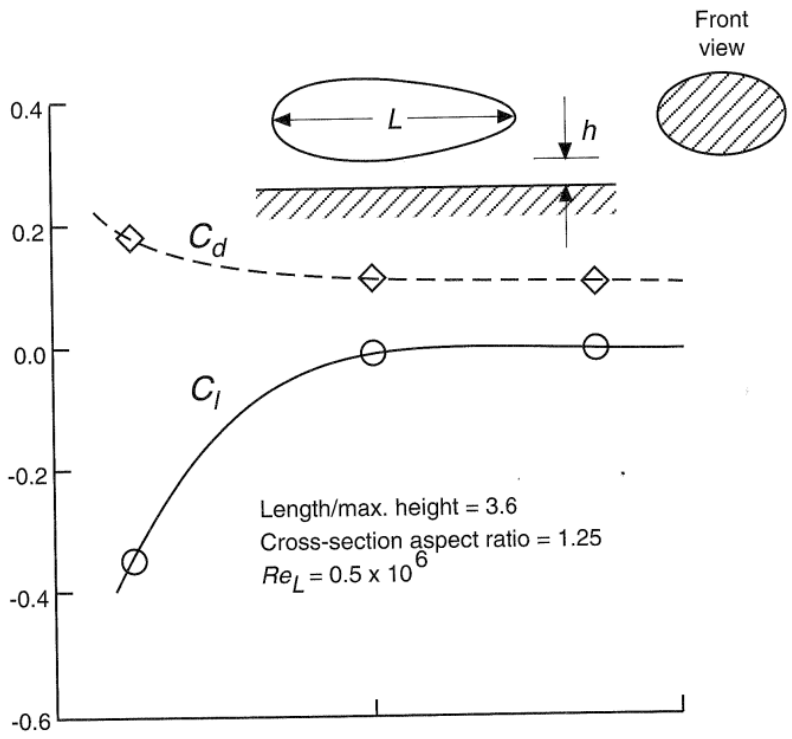
Centre of pressure

- For a vehicle to be inherently stable the CoP must be behind the CoG.
- The distance between the CoP and CoG is called the ***static margin***
- The larger this static margin the larger the moment of the aerodynamic forces
- Car is more stable with larger static margin

Aerodynamics
Underbody

Underbody Flow

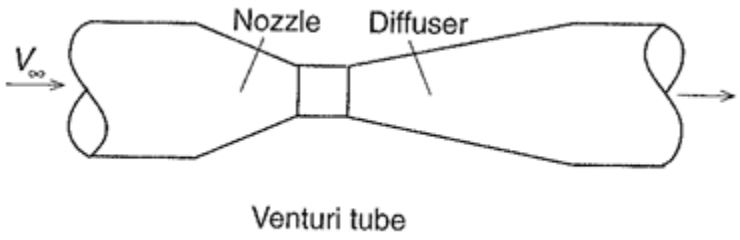
- The air flow under a vehicle is affected by the vehicles proximity to the ground
- This may result in advantages or disadvantages with regards to performance
- The disadvantage of high drag on rough under bodies was noticed first.
- We have already seen that advantages in terms of lift can result



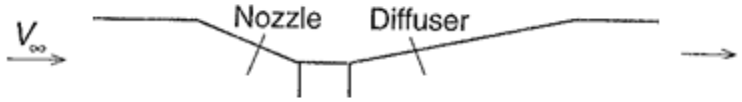
Aerodynamics
Underbody

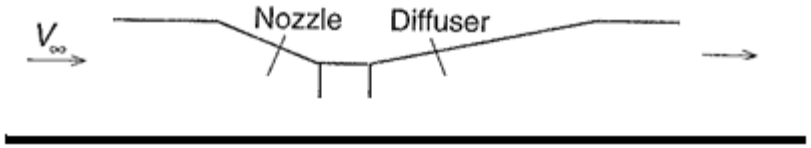
Underbody Flow

- Looking generally at a venturi tunnel.
    - Inlet :
        - Initial CSA decreases in nozzel
        - Initial velocity increases
        - Initial pressure decreases
    - Throat :
        - Decreased CSA
        - Increased Velocity
        - Reduced pressure
    - Outlet :
        - CSA Increases
        - Velocity decreases
        - Pressure increases back to original



Venturi tube



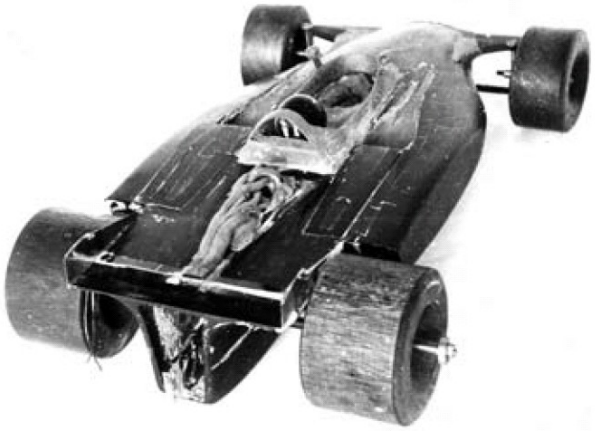


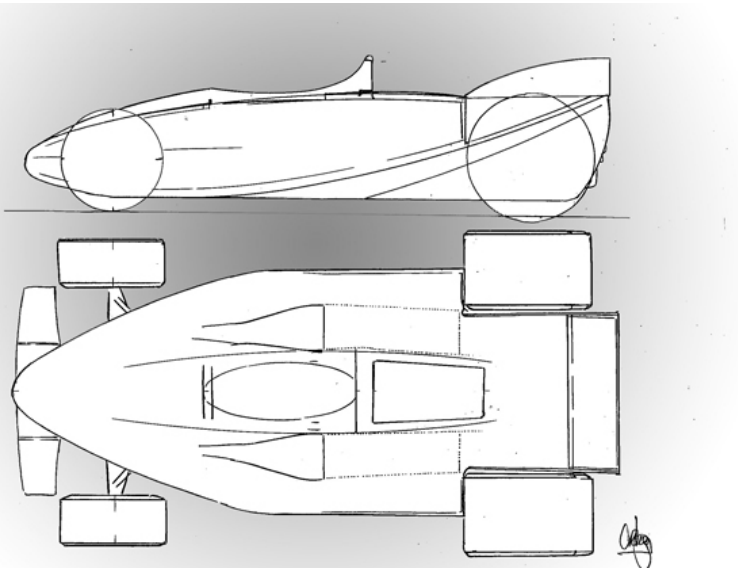
Aerodynamics
Underbody

- During the mid to late 60's F1 teams recognised that they could use shaped floors to create downforce.

- BRM's Peter Wright was the first to start developing a car with an aerodynamic shape to make use of these findings.

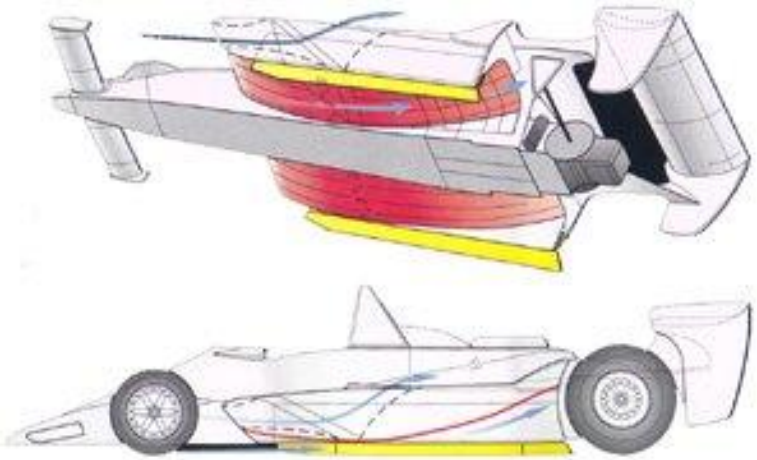
- Car effectively had half a venturi tunnel to produce a lower pressure on underside.





Aerodynamics
Underbody

- Lotus took this forward a step by adding venturi tunnel shaping to their type 78 and adding skirts.



- The Lotus was very competitive and the type 79 was dominant.

- Brabham again took a step forward by adding a fan to help reduce the pressure under the car further.

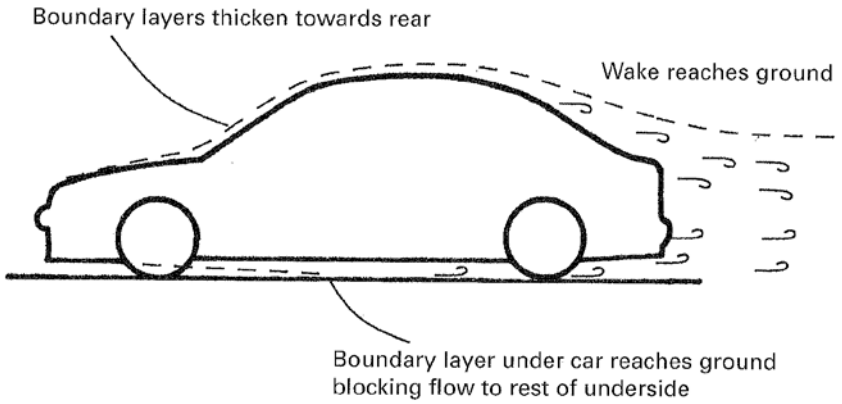


- This was banned as a movable aerodynamic device.

Aerodynamics
Underbody

Passenger Vehicles

- Passenger cars often have rough undersides.
- This often results in
  - High turbulence
  - High drag
  - Thick boundary layer



- The thick boundary layer builds up at the front of the vehicle due to high turbulence – rapid boundary layer growth

Aerodynamics
Underbody

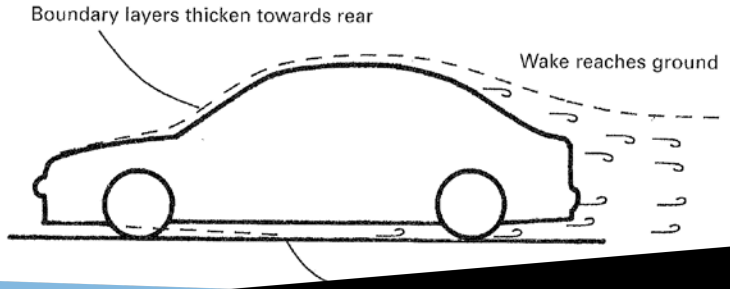
Passenger Vehicles

• The thick BL slows the flow of air under the vehicle increasing pressure and therefore lift.

• Combined with the lift generated by the top surface – poor situation!

• Decreasing ride height would actually increase lift as BL increasingly would slow the airflow!

• Increasing ride height would decrease lift as air velocity would increase decreasing pressure. CoG would be raised.



Aerodynamics
Underbody

Passenger Vehicles

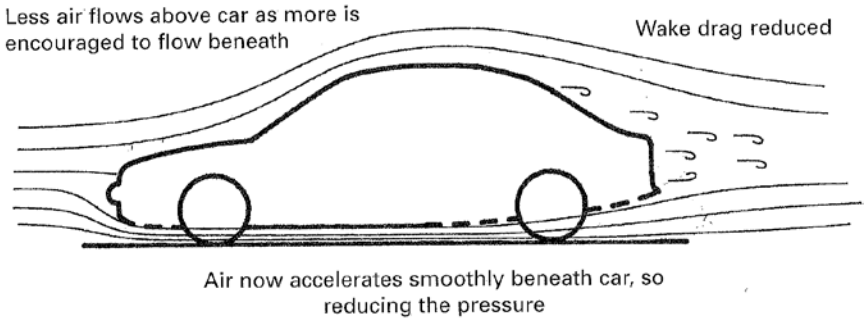
- To reduce lift on underside you should aim to smooth airflow under vehicle.

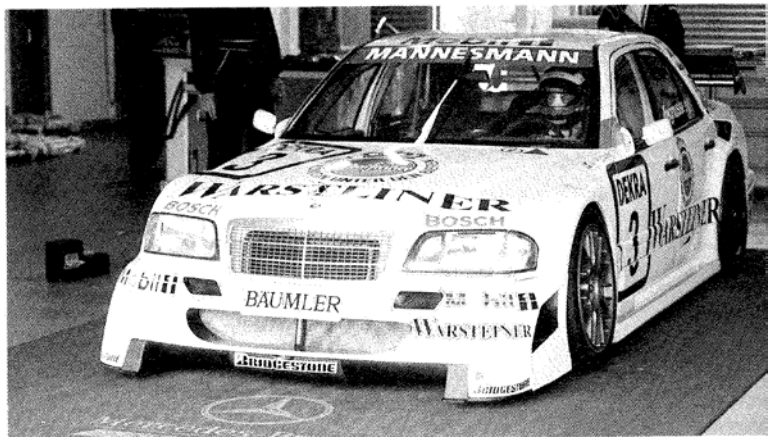
- Smooth air flow means the BL tends to be thinner and build slower

- Thin BL results in greater flow under vehicle and lower pressure.

- The reduced turbulence also results in less turbulent wake and therefore less drag.

Figure 5–2 *Idealistic shaping of the underside.*

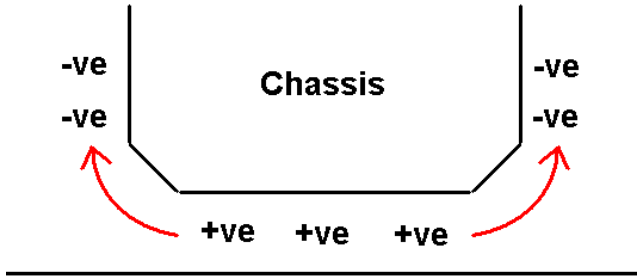




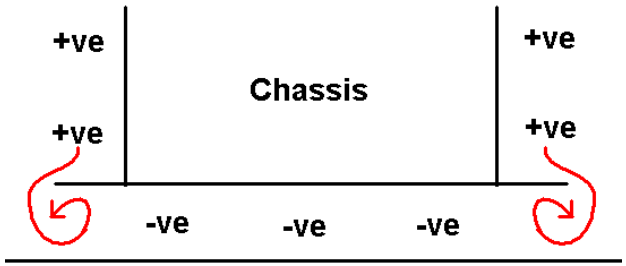
Aerodynamics
Underbody

Vehicle Sill Shapes -

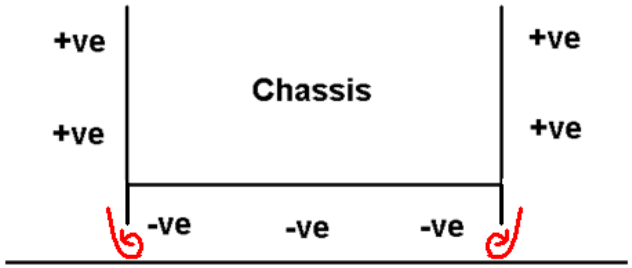
- The shape of vehicle sills can be designed to minimise lift on the bottom of the vehicle. (Maximise downforce)



- If a vehicle has a positive pressure on the underfloor, sills encourage high pressure air to bleed to low pressure regions.



- If vehicle has low pressure region under floor, bleeding of higher pressure from vehicle sides can be discouraged



- Vortices can be induced to help ‘block’ gaps.

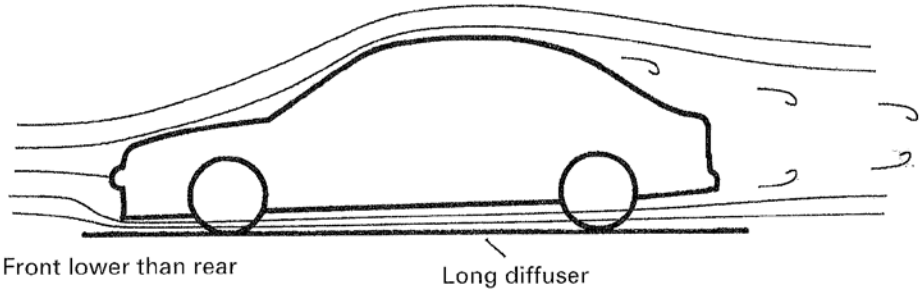
Aerodynamics
Underbody

Flat Floors

- Vehicles with smooth flat floors can encourage clean air flow and hence limit BL growth on the underside of the vehicle.

- If the vehicle also has a negative angle of incidence, or rake, a simple venturi tunnel can be formed.

Figure 5–3 *Flat underside with inclined rake.*



Inclined underside acts like a simple venturi easing the air-flow along beneath car

Aerodynamics
Underbody

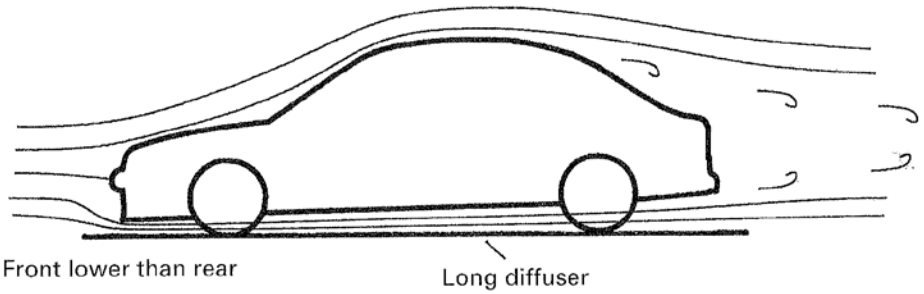
Flat Floors

- The airflow will speed up as it flows under the front of the vehicle (the throat) and this results in a drop in pressure here.

- As the air flows along the long diffuser its velocity drops back to almost free stream and therefore the pressure increases to free stream.

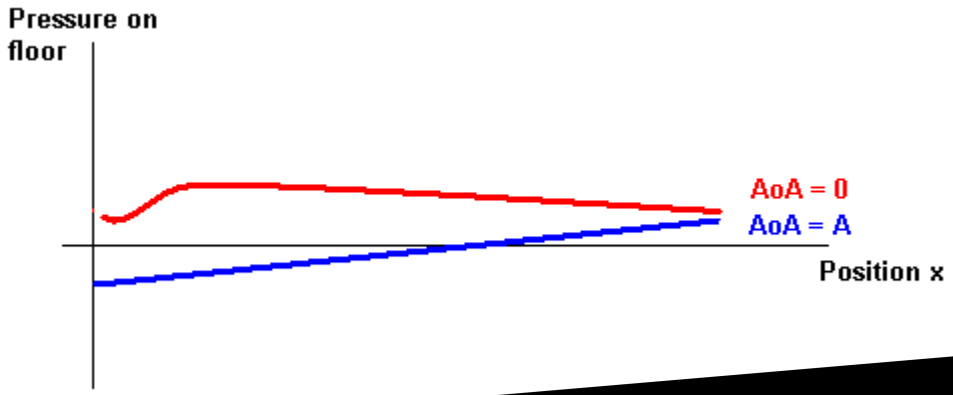
- This limits the resulting drag at the rear of the vehicle.

Figure 5–3 *Flat underside with inclined rake.*



Inclined underside acts like a simple venturi easing the air-flow along beneath car





Aerodynamics
Underbody

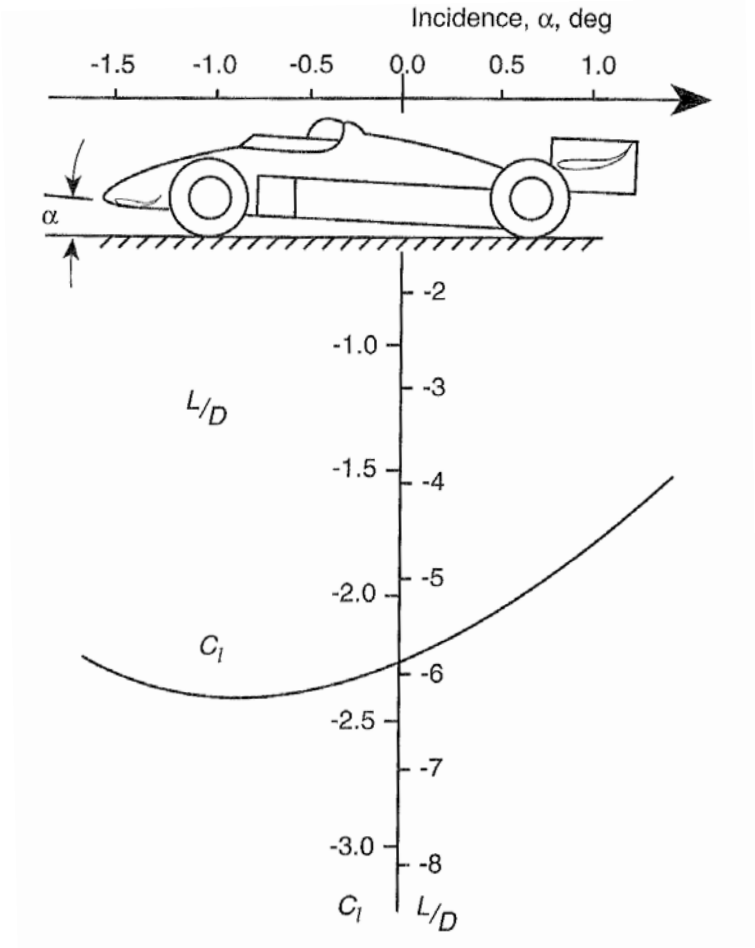
Flat Floors

The graph on the right shows a vehicle at various angles of incidence.

It is clear that a negative angle of incidence results in an increase in negative lift to a certain point.

This point is where boundary layer slows the flow under the vehicle either due to increase in thickness or reduction of front gap.

An angle of incidence of 1 or 2 degrees can result in a significant increase in negative lift.



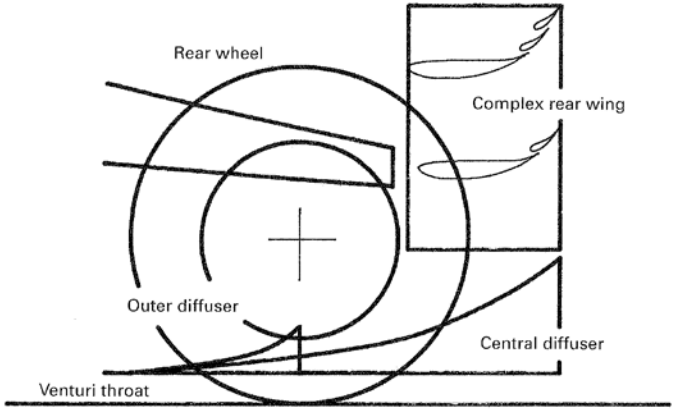
Aerodynamics
Underbody

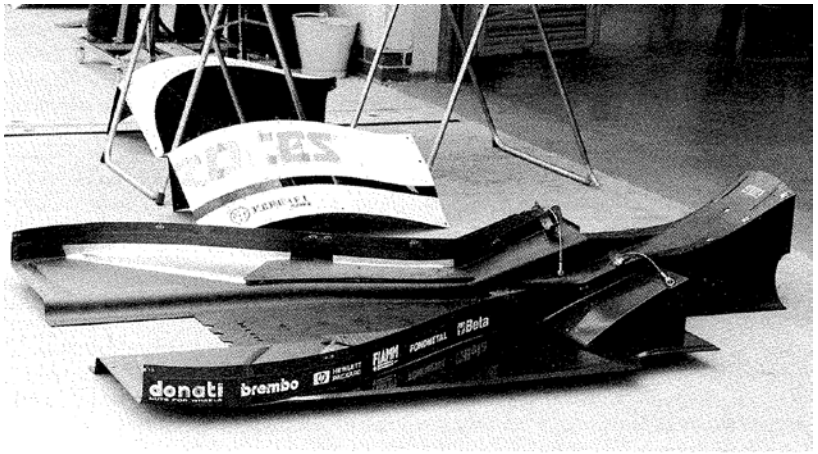
Diffusers

Flat floors are often augmented with a diffuser.

A diffuser can generally be defined as an area, often at the rear of the car, incorporating an increased angle of incidence to the forward part of the under floor.

The floor on a vehicle then effectively becomes the throat of our venturi and the diffuser is the divergence in which air decelerates.





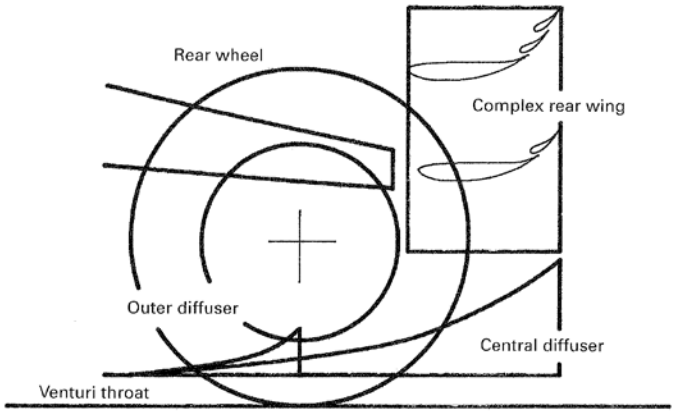
Aerodynamics
Underbody

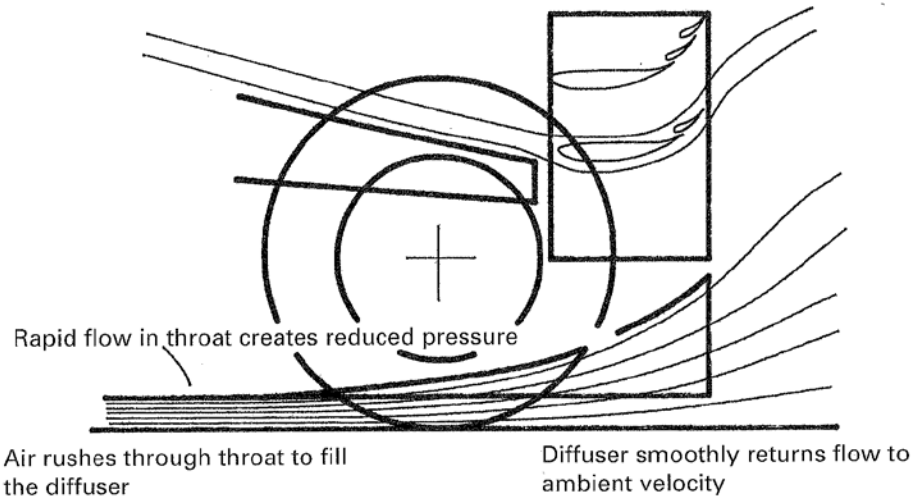
Diffusers

The purpose of the diffuser is to

- Help **drive /compel air flow** under the vehicle (the expansion must be filled with air and so air accelerates through throat)

- Bring that **air flow back to free stream** velocity and pressure as smoothly as possible as it reaches the vehicle wake.

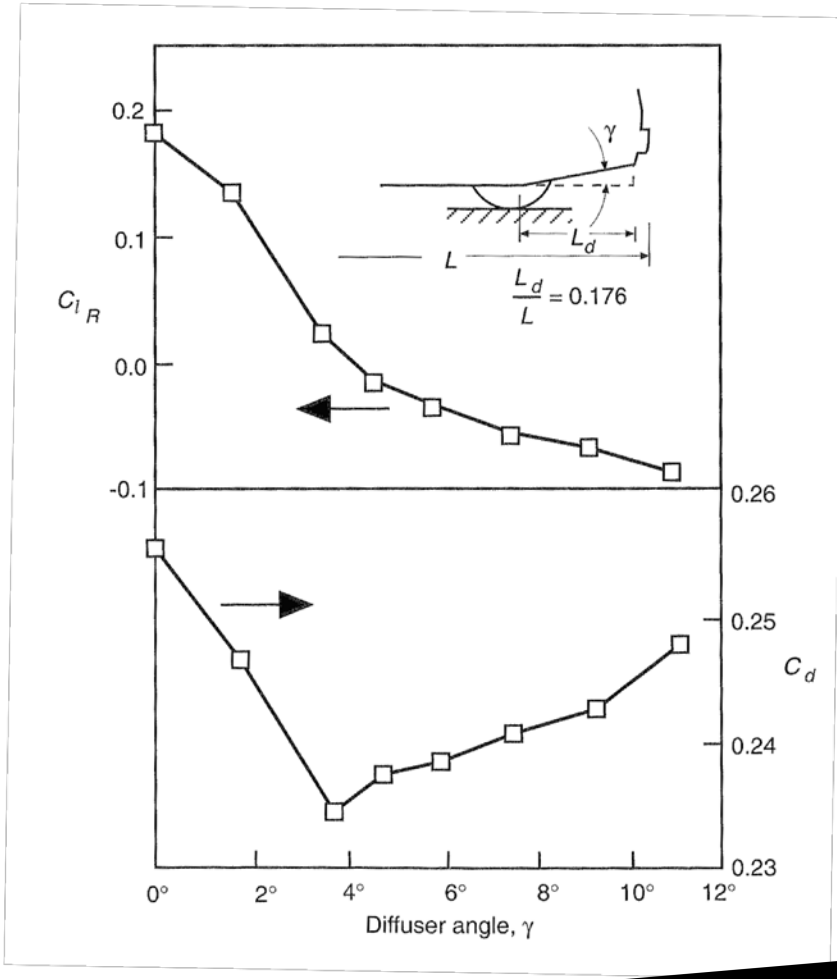




Aerodynamics
Underbody

Diffusers

- As the air is brought back to free stream velocity and pressure **progressively**
  - The adverse pressure gradient is minimised
  - Separation is minimised.
- To achieve this the diffuser should
  - Have a gradual change of angle of incidence to avoid separation of flow from roof or sides.



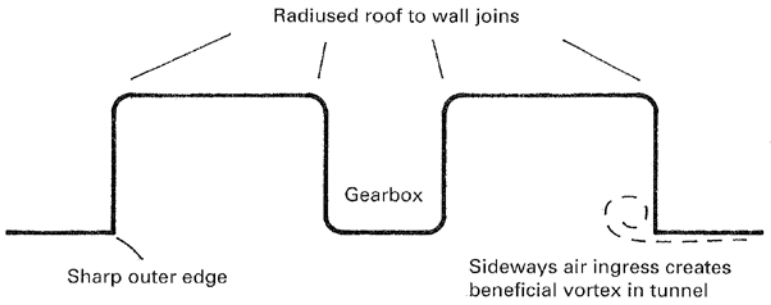
Aerodynamics
Underbody

Diffusers – Vortices.

Diffuser design is a complex area.

Vortices may be used to help keep flow attached.

For instance on the right, as flow enters the low pressure region from the sides of the vehicle the sharp lower edges induce a vortex at the corners of the diffuser which help keep the diffuser flow attached to walls.



Aerodynamics
Underbody

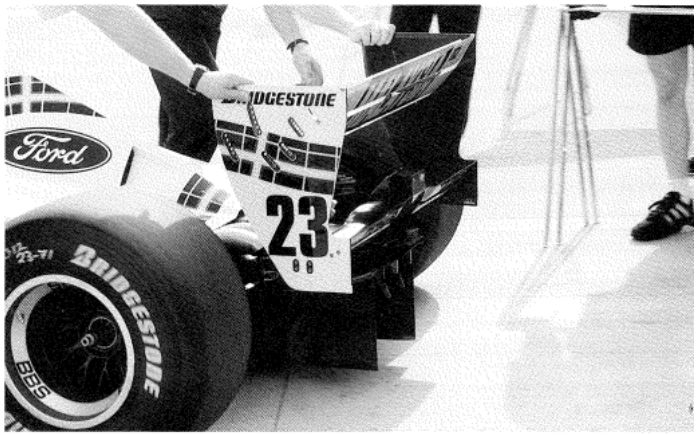
Diffusers – Wing interactions

Some diffusers have very steep divergences.

This is often due to the imposed aerodynamic regulations.

These diffusers often rely heavily on the interaction with the rear wing.

The rear wing can be used to drive flow in the diffuser. It can basically act as a second flap in a large scale ‘wing’

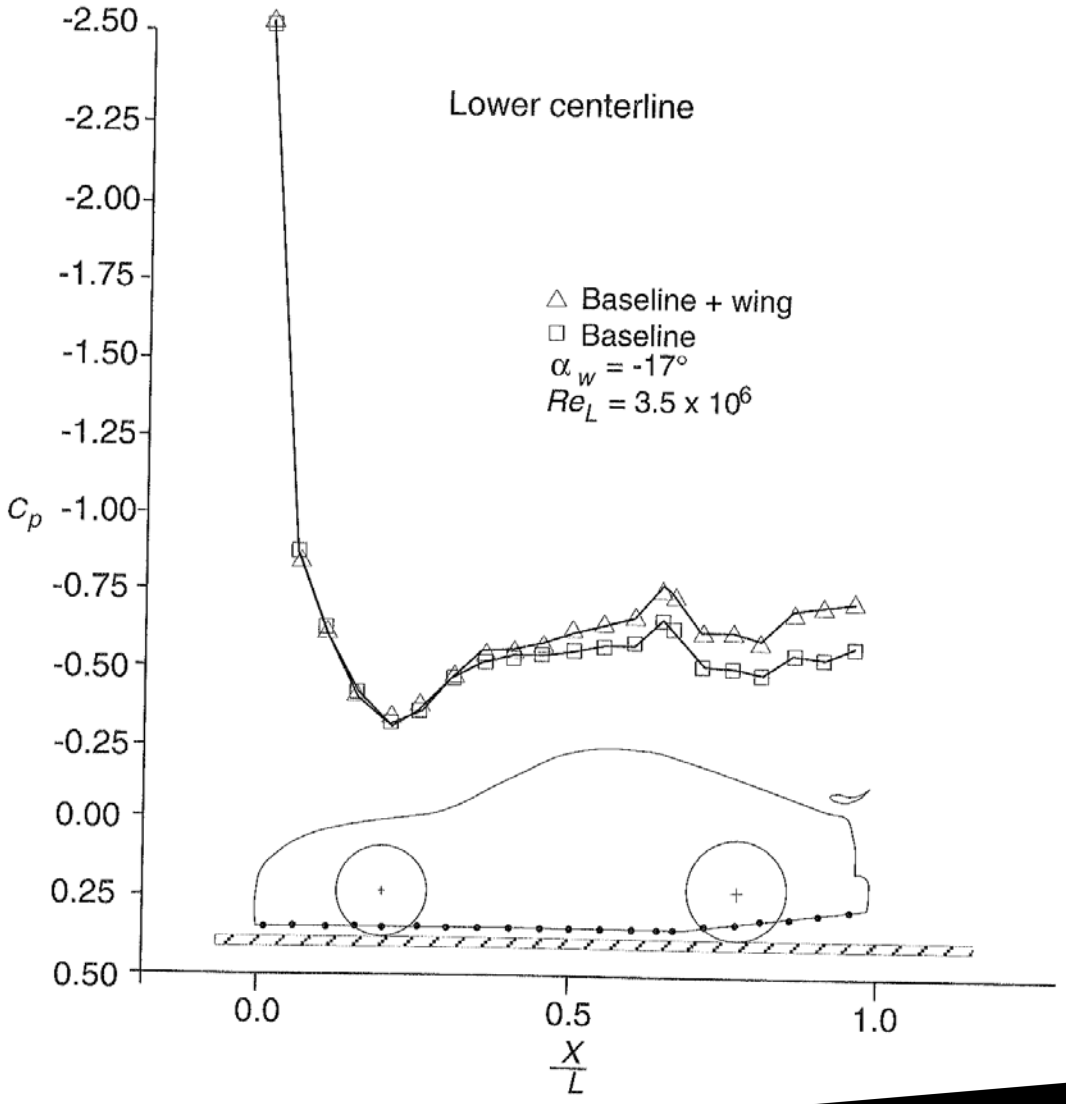




Aerodynamics
Underbody

Diffusers – wing interactions

- Wing helps flow stay attached.
- Influence extends well forward of the rear axle.
- Result is significant when integrated over the area.



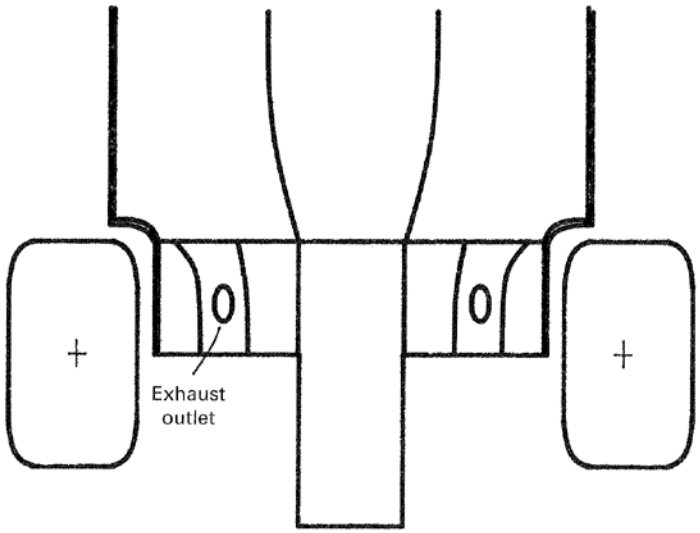
Aerodynamics
Underbody

Diffusers – Exhaust Blowing

Exhaust blowing refers to the practice of positioning the exhaust outlets in the diffuser.

The principle is that the high velocity exhaust cases are injected into the diffuser to re-energise the boundary layer and help keep flow attached increasing downforce.

The result is obviously inconsistent as exhaust flow is dependant on engine speed.

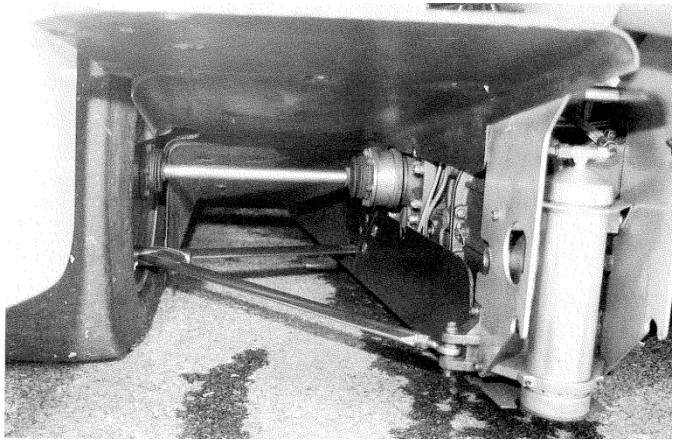


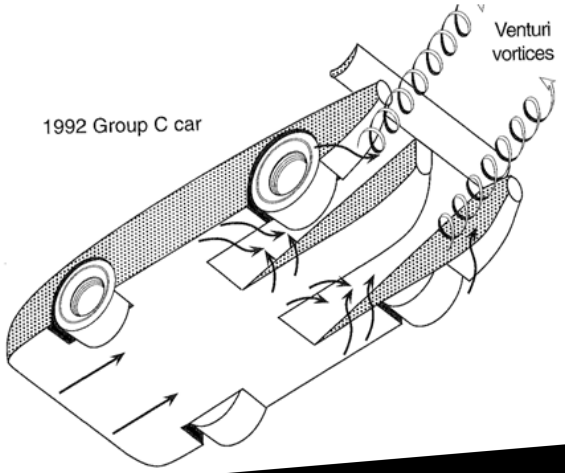
Aerodynamics
Underbody

Tunnels – Sports Car

Some vehicle often have to have 2 tunnels each side of the monocoque for packaging reasons.

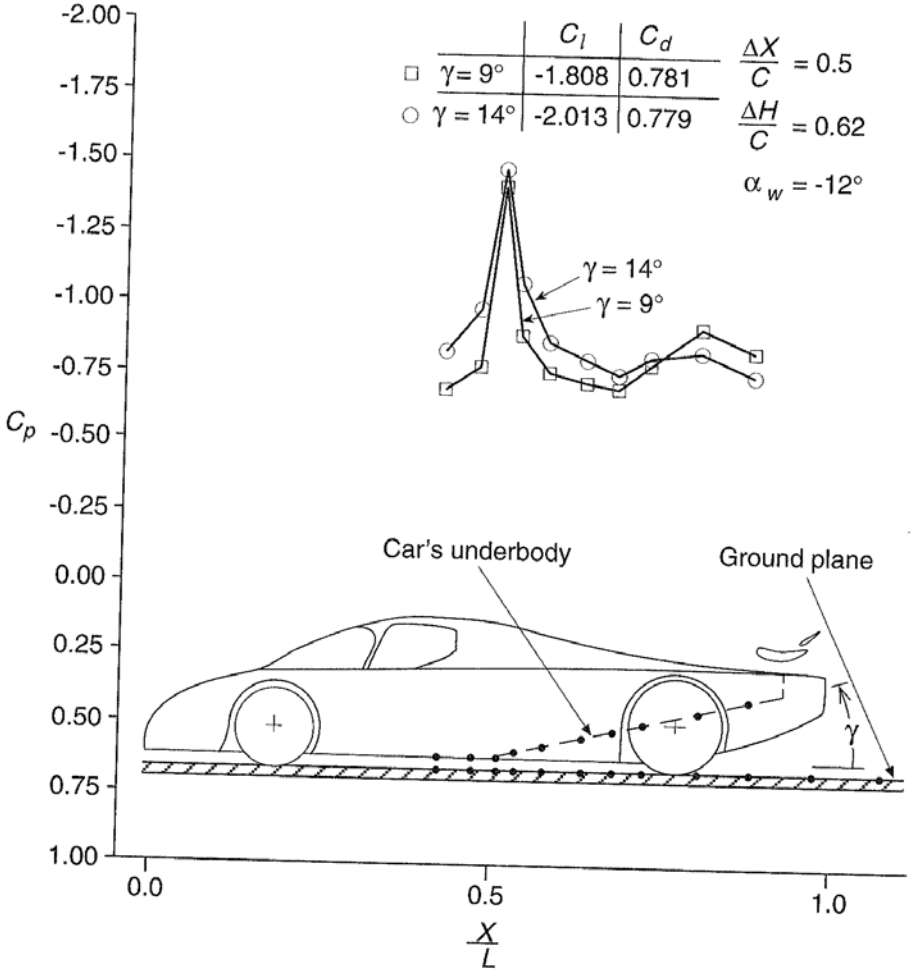
However sports cars still often use the tunnels as diffusers to again accelerate the air ahead of them and then gradually bring the air back to free stream velocity and pressure.





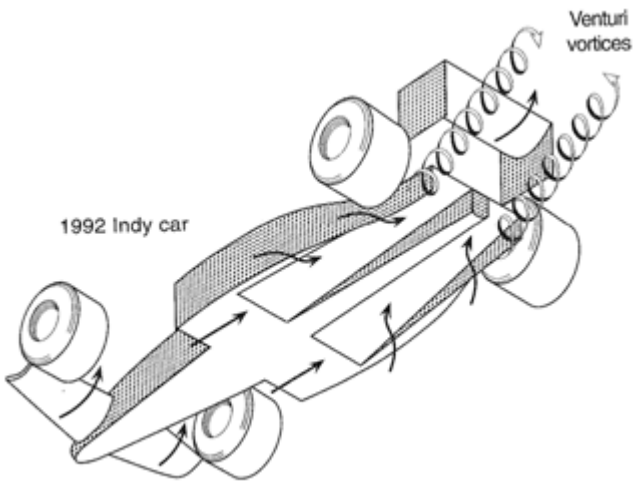
Aerodynamics
Underbody

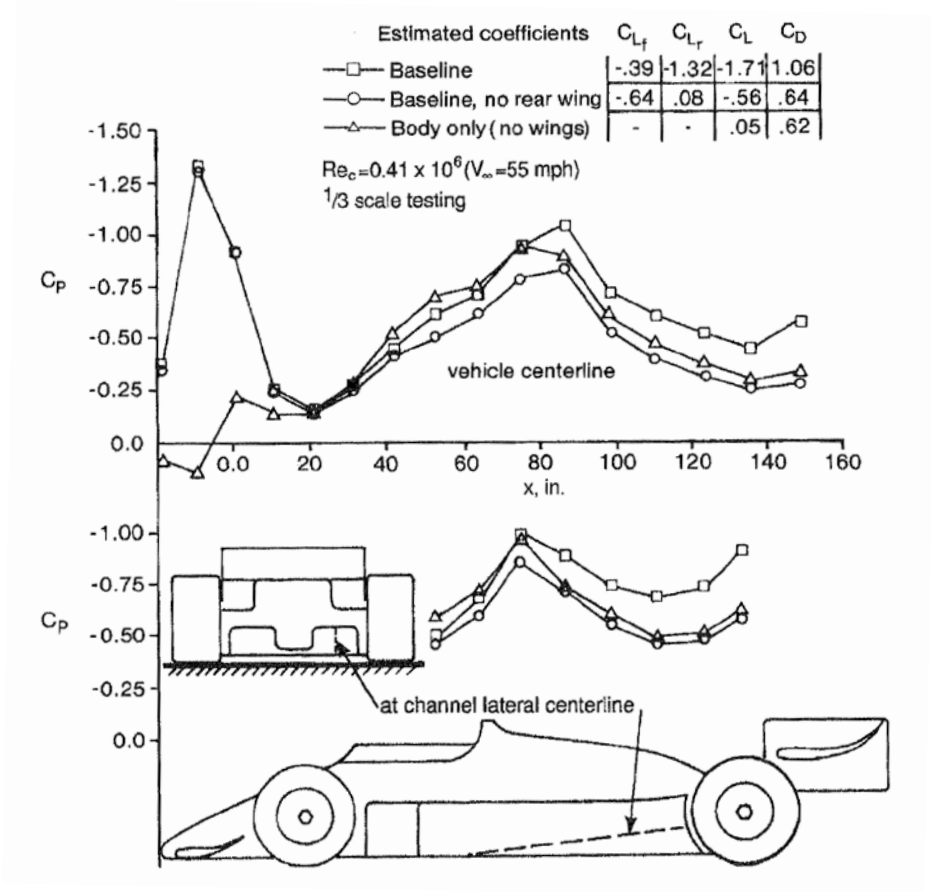
Tunnels – Sports Car



Aerodynamics
Underbody

Tunnels – Single Seater





Aerodynamics
Underbody

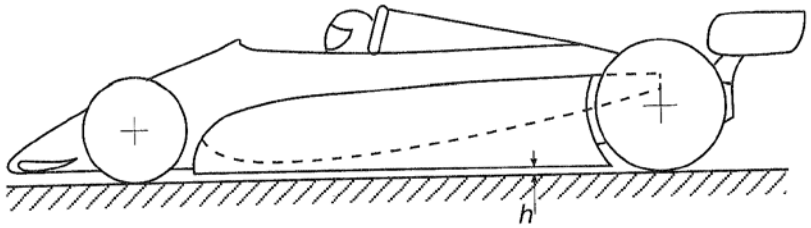
Tunnels – Single Seater

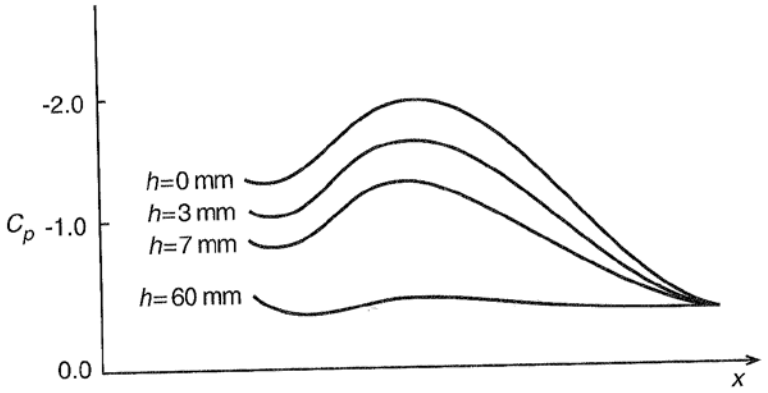
Some vehicles have large sections of the under floor specifically shaped to create large venturi tunnels.

A nozzle helps pack air through a small throat followed by a long diffuser section.

This results in a low pressure under the surface and minimal drag.

The closer such a floor gets to the ground the more pronounced the effect, upto a point where the BL blocks the flow through the throat.





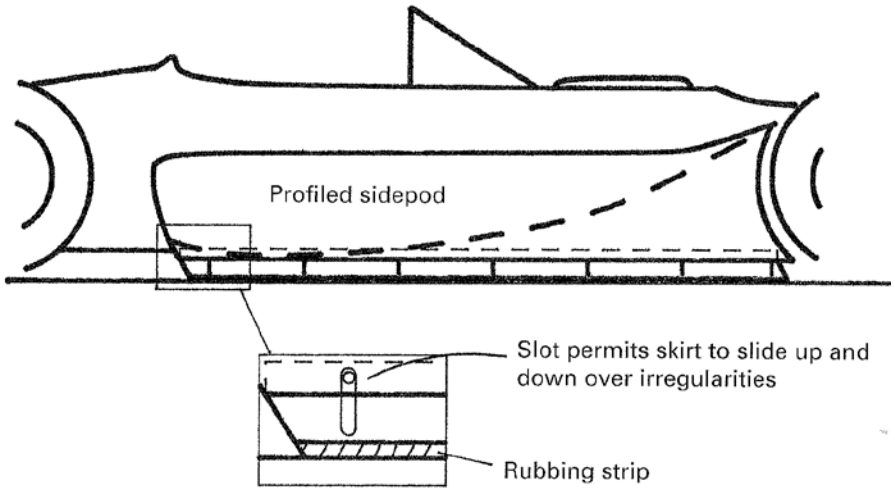
Aerodynamics
Underbody

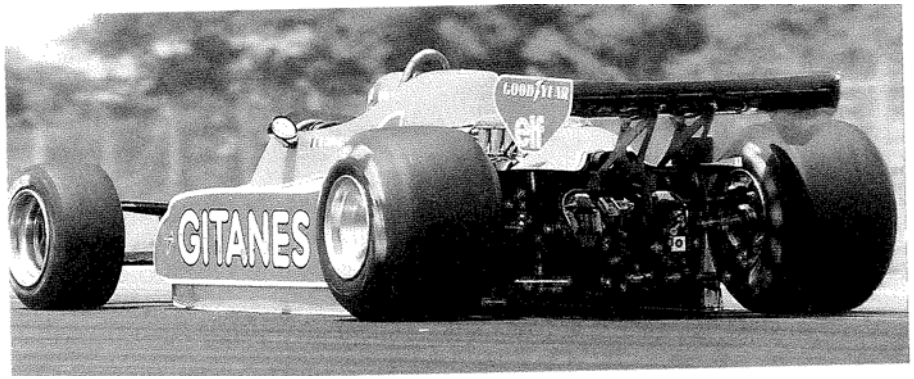
Tunnels – Single Seater - skirts

Vehicles sporting venturi tunnels used to also employ side skirts.

These were small ‘curtains’ which sealed the gaps at the sides of the tunnels to the ground.

This helped limit the amount of high pressure air which could bleed in from the sides of the car and hence maximise the negative pressure.

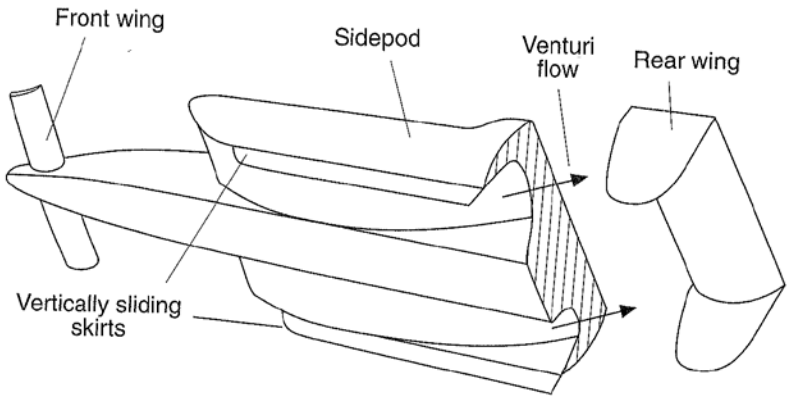


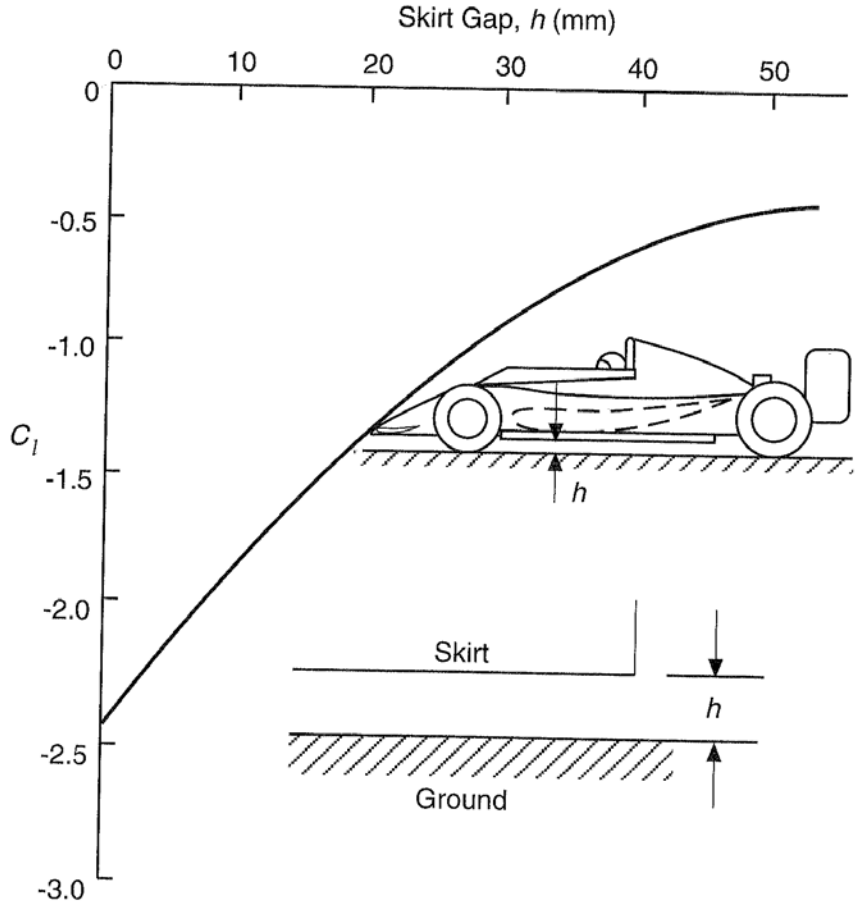


Aerodynamics
Underbody

Tunnels – Single Seater – skirts

These skirts were banned in motorsport as they resulted in vehicle which were so effective that cornering speeds were becoming unsafe.





Aerodynamics
Underbody

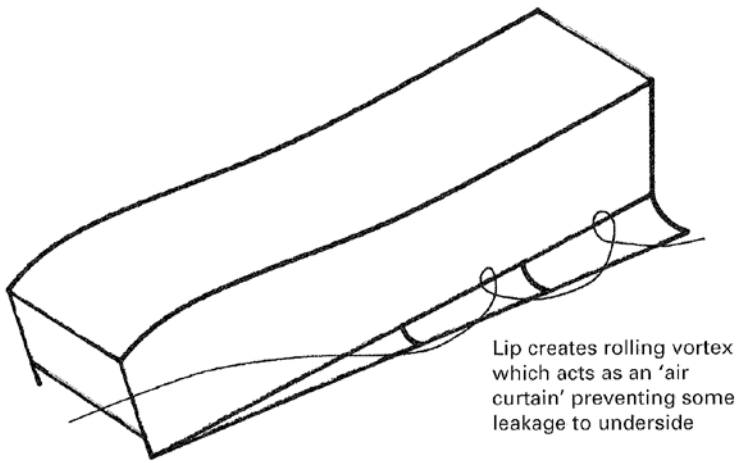
Tunnels – Sidepod shaping to replace skirts.

Like sills on passenger vehicles, single seater sidepods can be shaped to reduce bleeding of the high pressure to the underside of the vehicle.

A lip specifically designed to induce a vortex which acts as a curtain to seal the gap between the floor and ground.

This method is not as pronounced as skirts but is worthwhile.

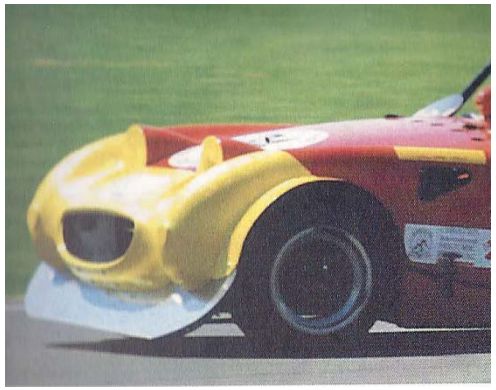
Figure 5–11 *Schematic of a sidepod with a profiled outer lip in place of a skirt.*

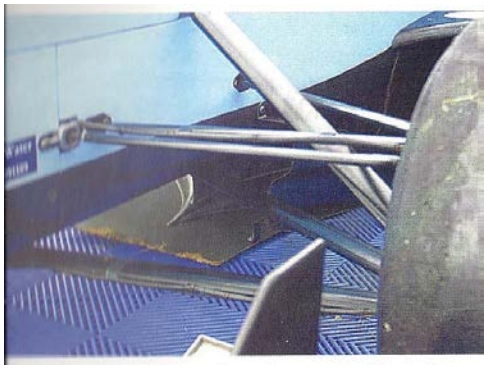


Aerodynamics

Spoilers and Airdams

- Many aerodynamic appendages appear or race vehicle most of which try to either,
  - Increase downforce
  - Reduce drag
  - or both!
- Appendages may include
  - Front spoilers (Air dams)
  - Rear spoilers
  - Splitters
  - Wings
  - Dive planes
  - Vortex generators





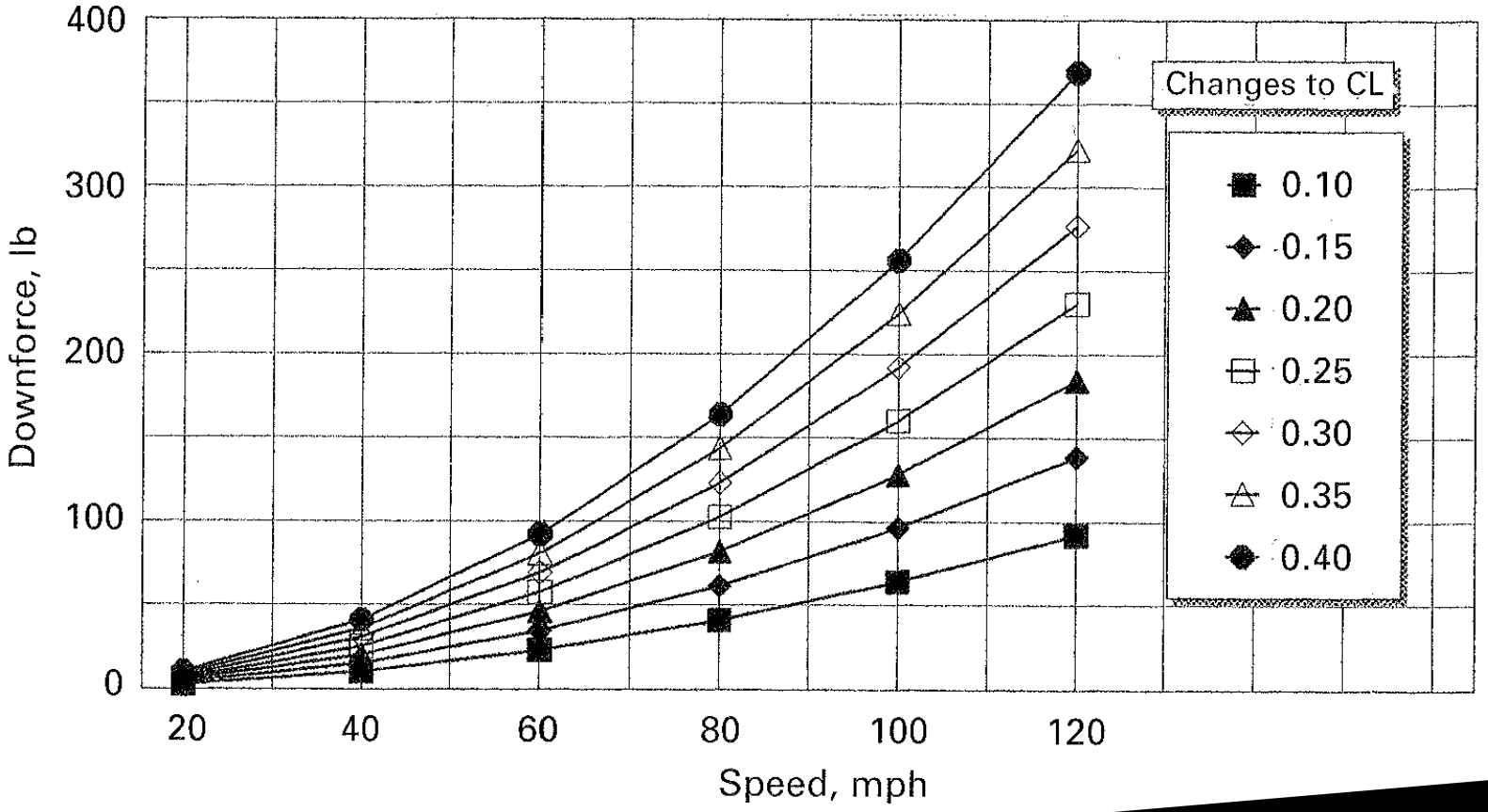
Difference between spoilers and wings it that spoilers have no ‘gap’ between body and under surface – no circulation

Aerodynamics

Spoilers and Airdams

Spoilers

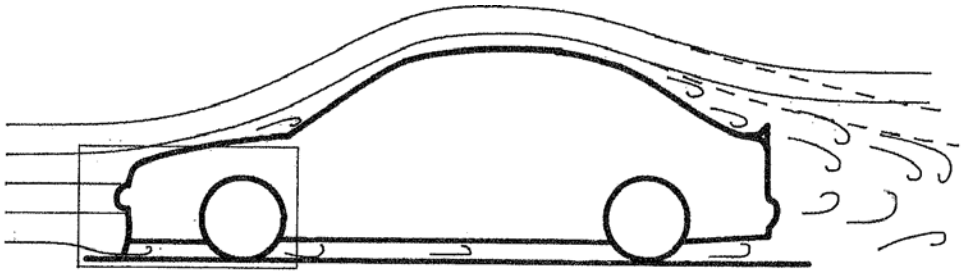
•Note small changes to lift coefficients can yield significant changes in lift forces produced.



Aerodynamics
Spoilers and Airdams

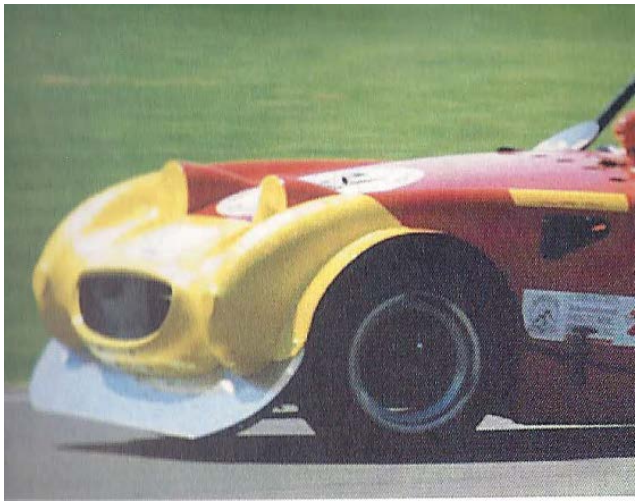
Airdams

• Airdams may also be referred to as front spoilers



• Usually of the form of a front ‘bib’

• Simple air dam is a rigid, near vertical sheet bolted to the lower front body



• Work on the principle that the front air dam reduces the air flow under the vehicle.

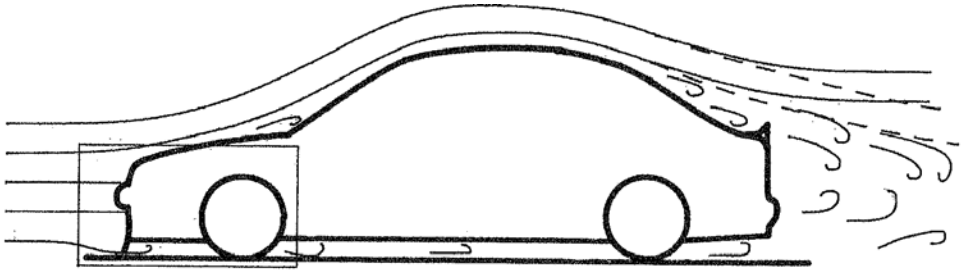
• Opposite to venturi tunnels!!

Aerodynamics

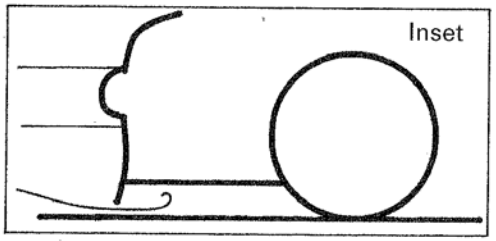
Spoilers and Airdams

Airdams

• Assuming bib touches floor and sides of vehicle have skirts, air flow under vehicle is eliminated and diverted over vehicle:



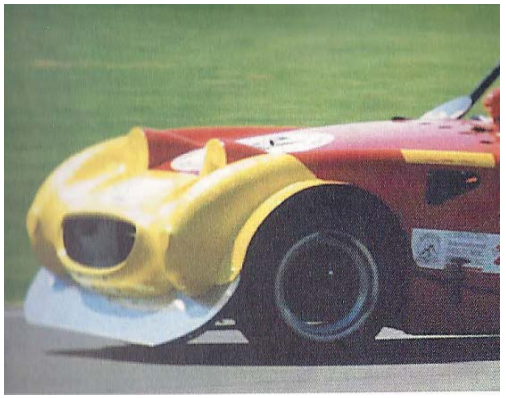
Airdam reduces amount of air flowing under car, and creates wake-like low pressure beneath front



•***Drag*** –

- • Vehicles with flat floors will see increase in drag due to increased frontal area.

- •Vehicles with rough floor will generally see an overall decrease in drag due to reduction of turbulence under vehicle, even though frontal area is increased.

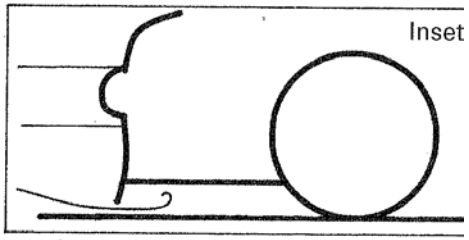


Aerodynamics
Spoilers and Airdams

Airdams

- ***Downforce***

  - Wake region behind car is low pressure due to turbulence
  - Low pressure propagates under vehicle.
  - Low pressure acts on plan area of floor.
  - Downforce/lift reduction results.



  - Wake is used to remove pressure under vehicle.

  - This effect is greater than the change in pressure due to the decrease in flow velocity under the vehicle caused by the expansion after the air dam

Aerodynamics
Spoilers and Airdams

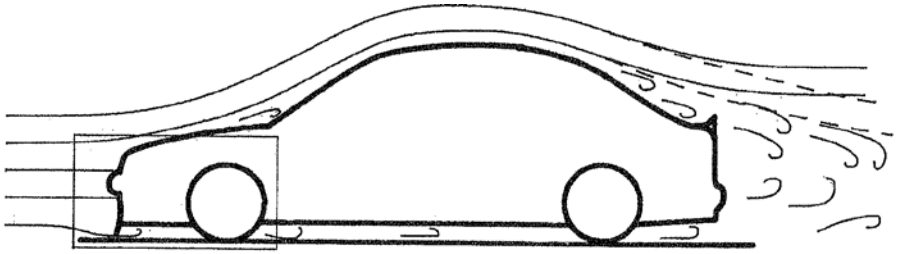
Airdams

•In practice air dams will have a clearance height (regulations) and hence some air will flow under vehicle.

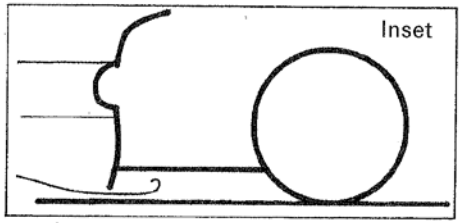
• Skirts or sills will also further reduce effect due to spill,

• Air dam can also increase flow in engine compartment and aid cooling, due to low pressure below.

• Road car, 300mm ground clearance, when 100mm front spoiler added



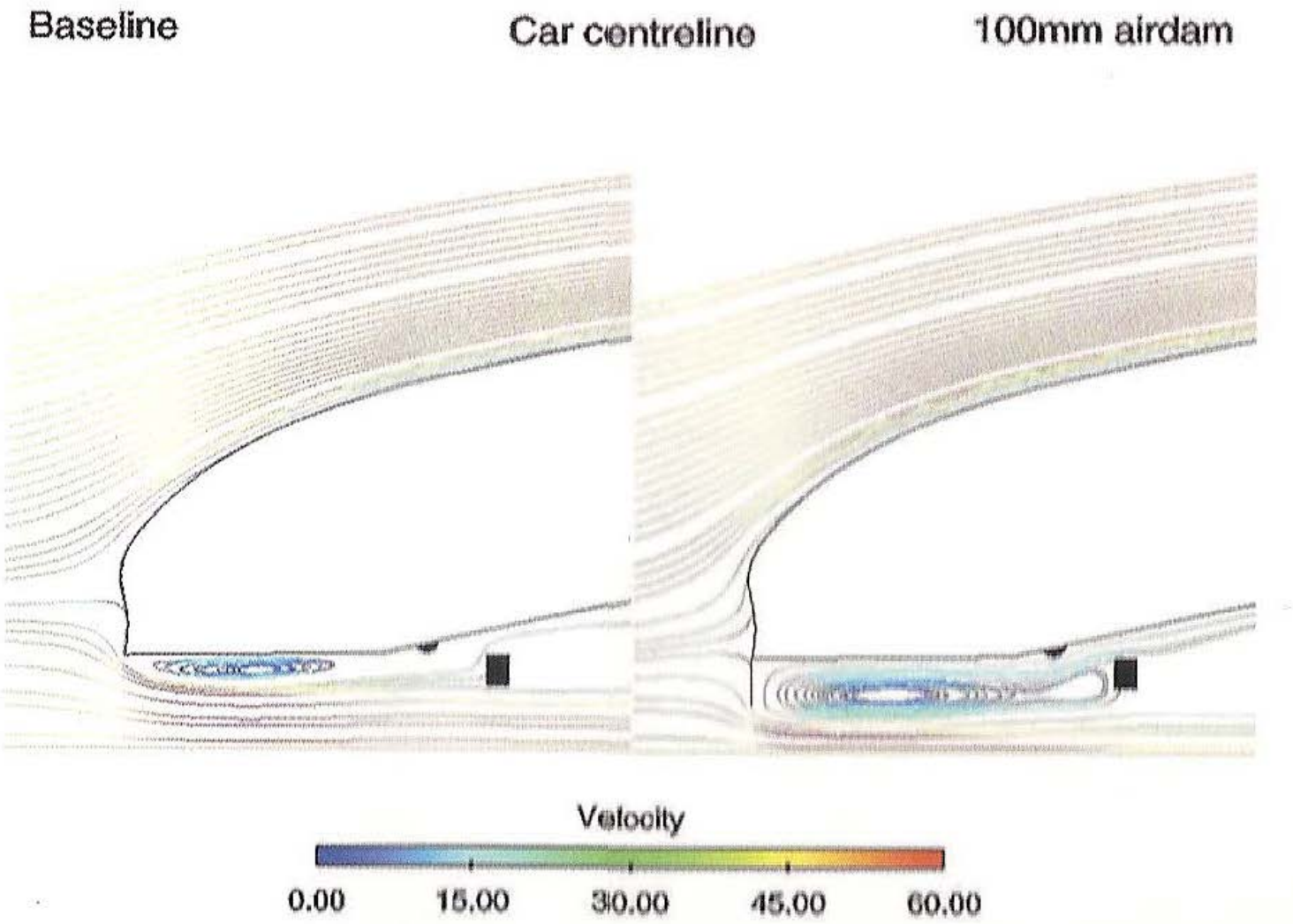
Airdam reduces amount of air flowing under car, and creates wake-like low pressure beneath front



δC_L =-0.21 (~500N @100mph) and δC_D = -0.04

Aerodynamics

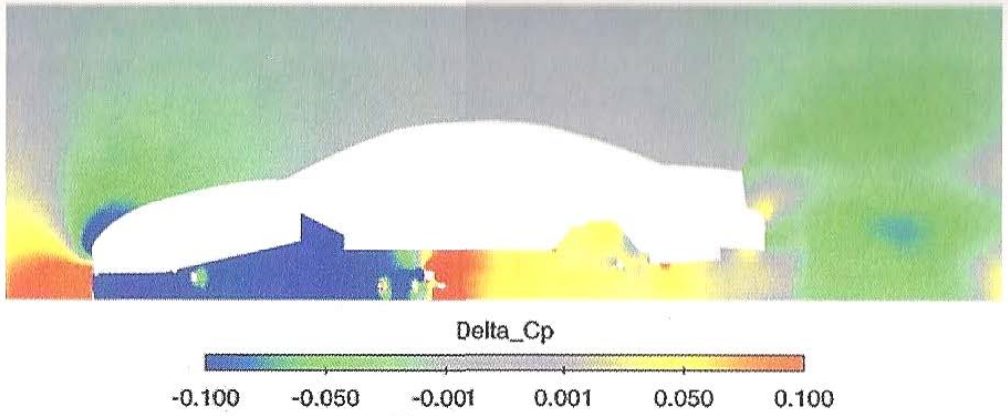
Spoilers and Airdams



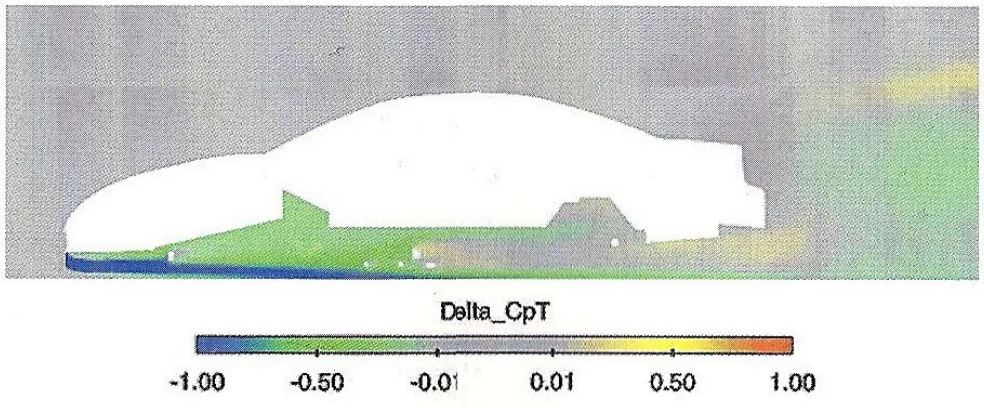
Aerodynamics

Spoilers and Airdams

The effect of adding an airdam on static pressure along the car centreline.



The effect of adding an airdam on total pressure along the car centreline.

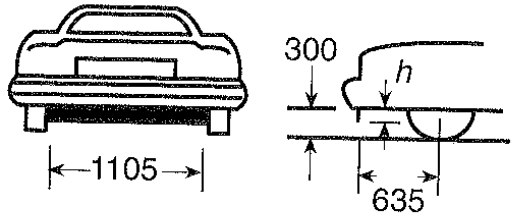


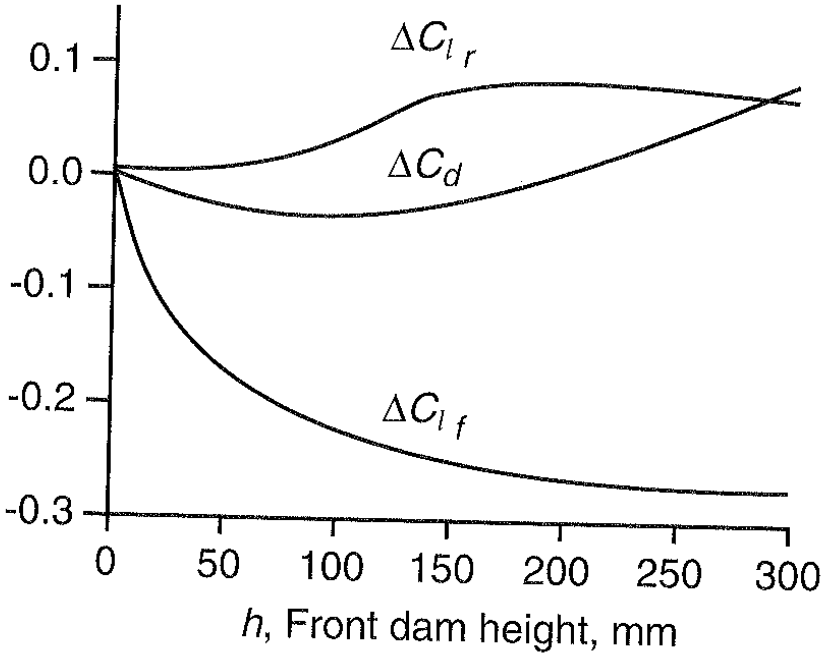
Aerodynamics

Spoilers and Airdams

Airdams

- As the graph on the right shows an increase of air dam size
  - Reduces lift / increases downforce
  - Initially decreases drag due to reduced turbulence
  - Drag then increases due to increasing frontal area.
  - Lift at rear of the car is affected as under body flow is effected.





Aerodynamics
Spoilers and Airdams

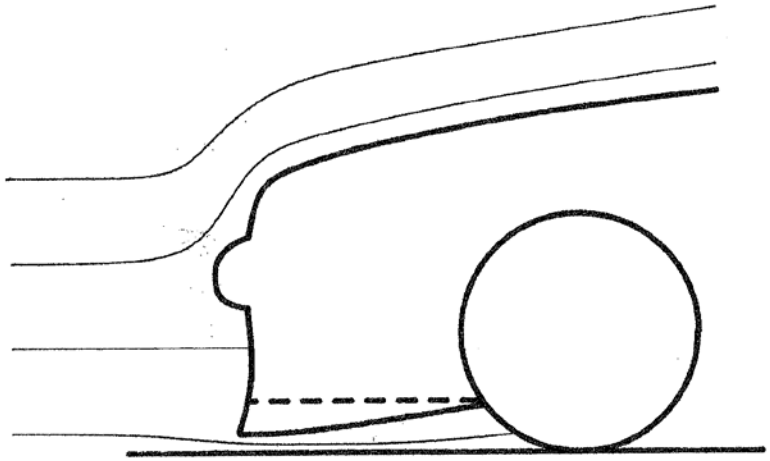
Airdams

• Air dams have a small car-to-ground gap s
flow accelerates in this region.

• If a diffuser is employed behind the air da

  • Accelerated flow's reduced pressure can act on a near horizontal surface
  • Downforce is produced
  • Drag acting on airdam is minimised

• Diffuser airdams often encourage some air flow.....



Smooth, fast flow = low pressure

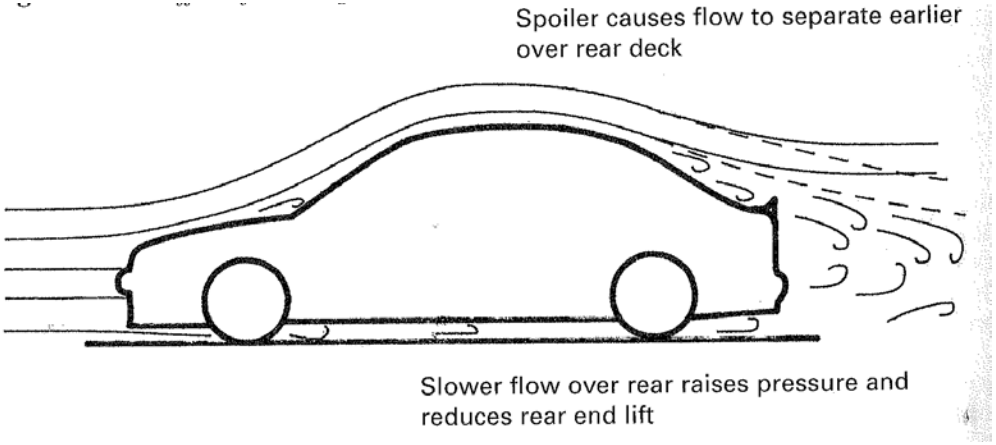


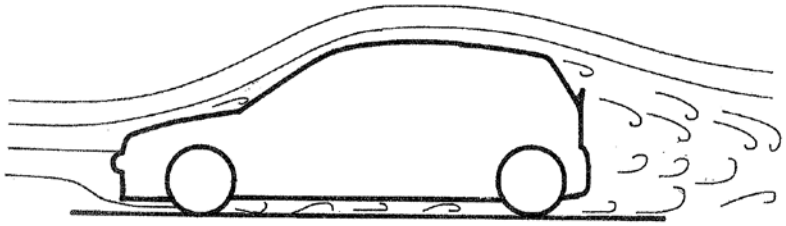
Aerodynamics

Spoilers and Airdams

Spoilers

- Simple spoilers can also be implemented at the rear of a vehicle.
- Again such spoilers can increase downforce.
- Fast laminar, low drag airflow on top of the vehicle usually results in lift.
- Spoilers 'spoil' this airflow and slow it down,
- This minimises this low pressure or can even create positive pressure





Aerodynamics

Spoilers and Airdams

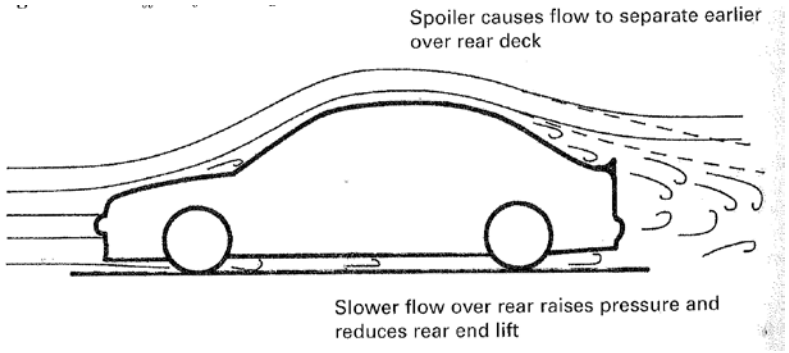
Spoilers

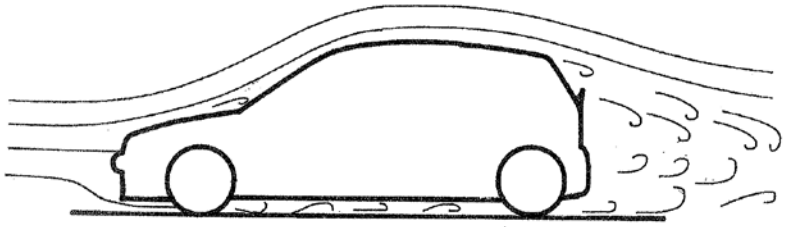
•A rear spoiler causes flow separation (or earlier) on the rear upper surface.

• The flow in the separation bubble is at low velocity and hence increased pressure.

• Increased pressure acts on horizontal surface reducing lift.

• On hatchback arrangement shown there is no horizontal surface in front of spoiler → no downforce is produced.



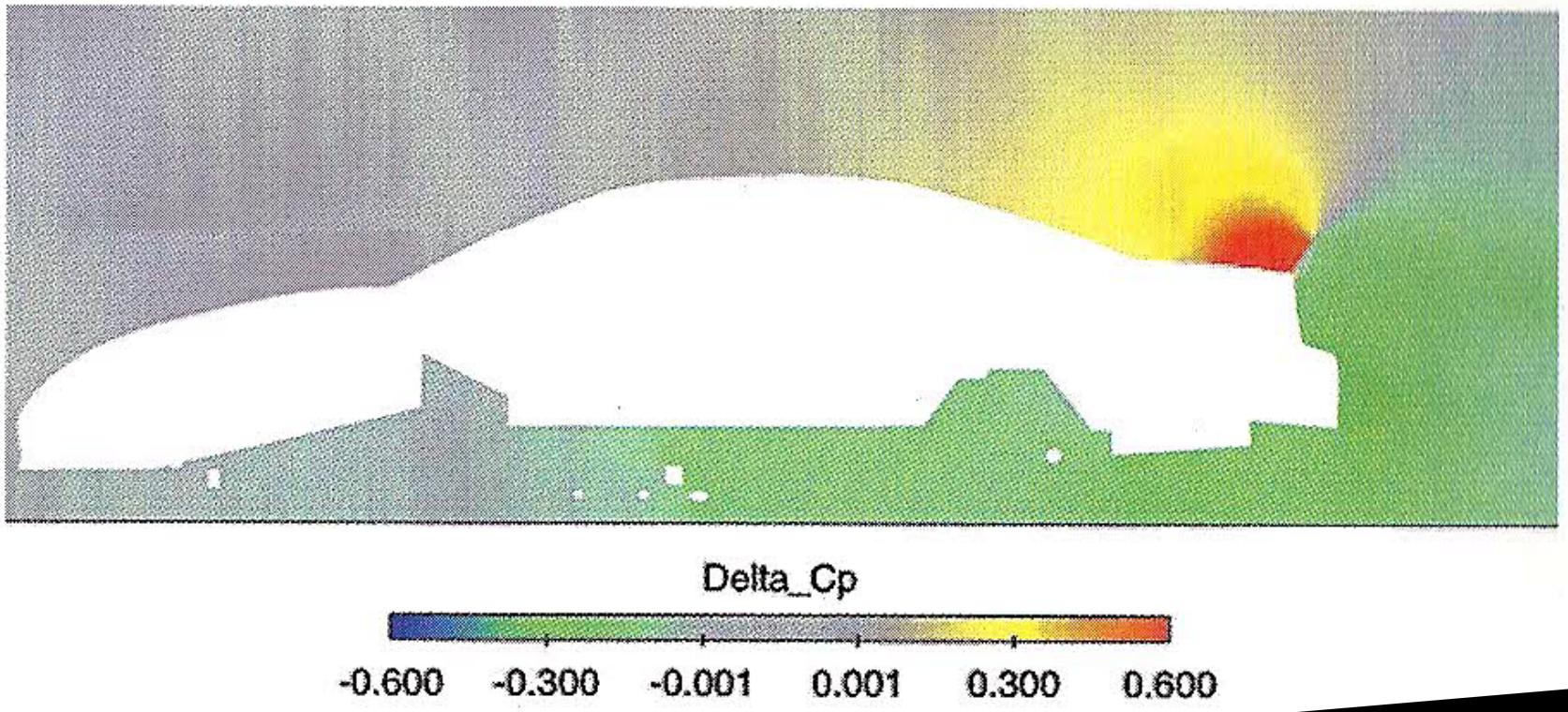


Aerodynamics

Spoilers and Airdams

Spoilers

•The picture below shows the change in pressure resulting from the application of a rear spoiler



Aerodynamics
Spoilers and Airdams

Spoilers

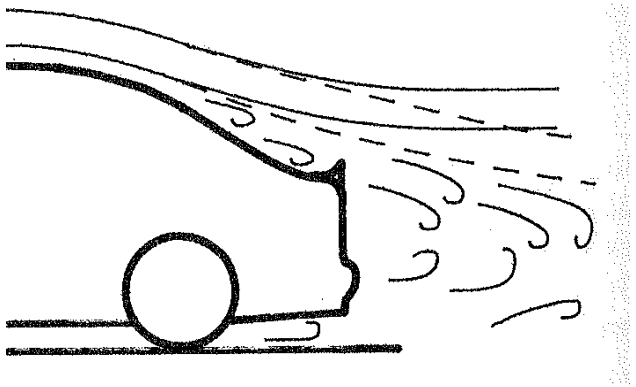
• Some spoilers often result in negligible increase in drag

• Up to a point!

• They are already positioned in a low speed area of separation and hence drag is low.

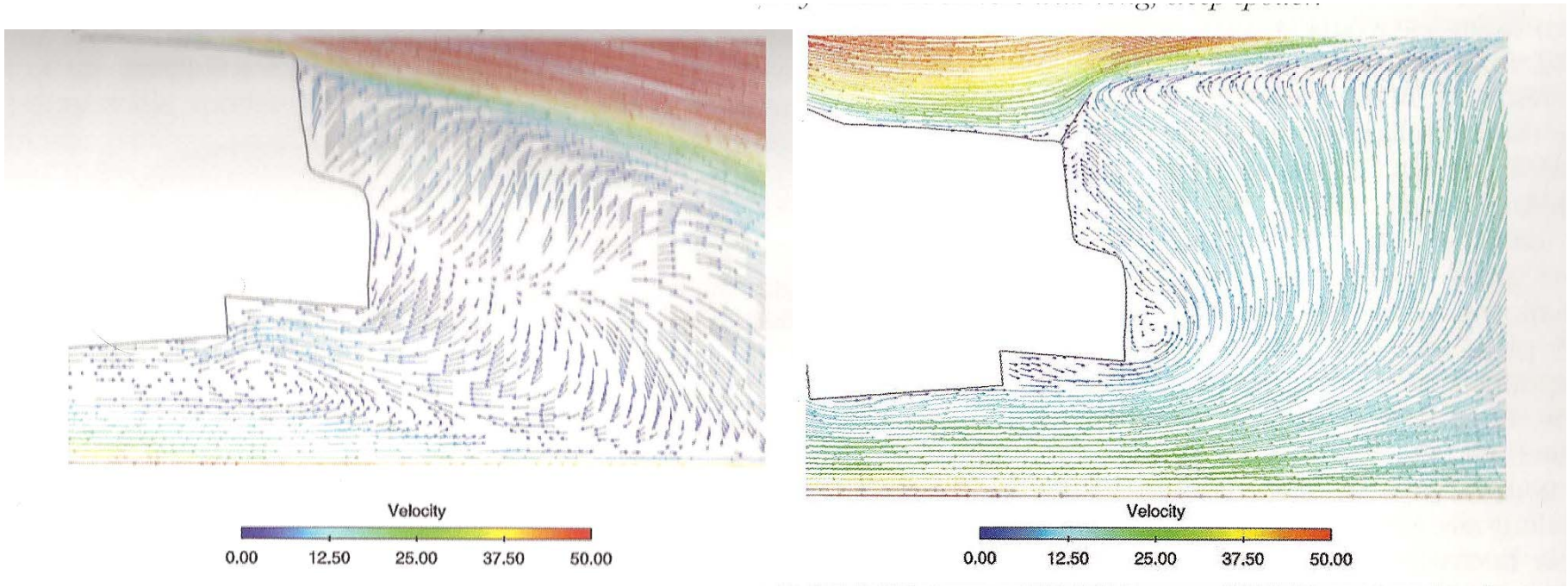
• Next slide also shows wake behind the vehicle can be significantly effected.

• A reduction in drag can be achieved with a correctly sized and positions rear spoiler.



Aerodynamics
Spoilers and Airdams

Spoilers



• Note that rear spoiler actually reduces turbulence in wake and results in increased pressure behind vehicle.

Aerodynamics

Spoilers and Airdams

Spoilers

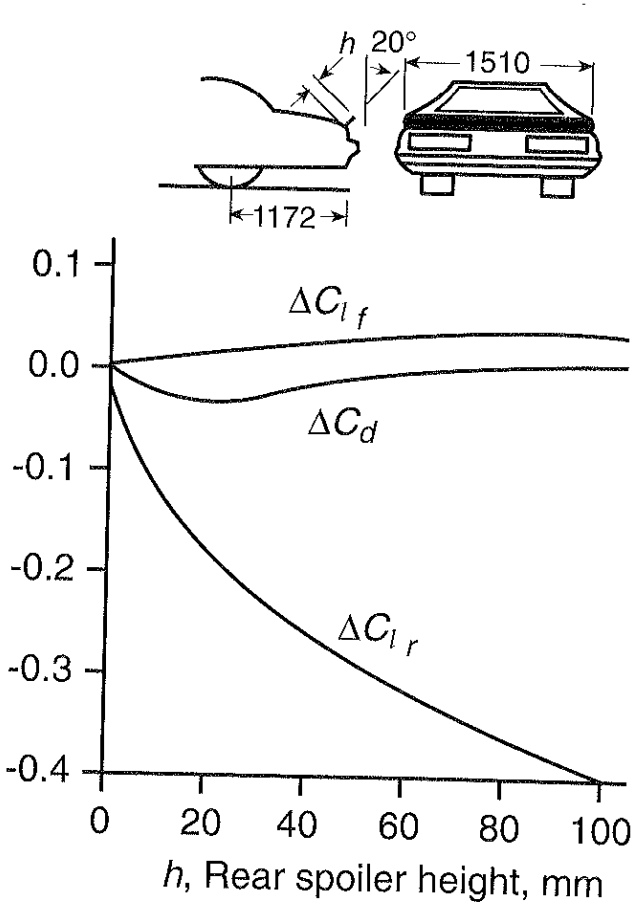
• The graph on the right shows how rear spoiler height affects downforce and drag of a vehicle.

• Notice that rear lift is reduced with increasing spoiler height, but at a falling rate.

• Change in drag is fairly small but some reduction achievable.

• Front lift only slightly affected.

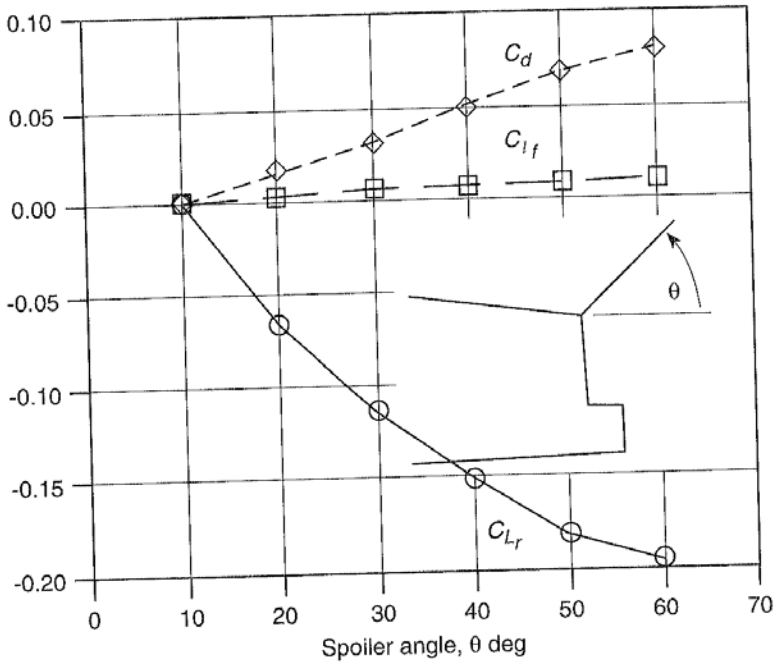
• A change of -0.2 for C_L results in 58Kg of downforce at 100mph.

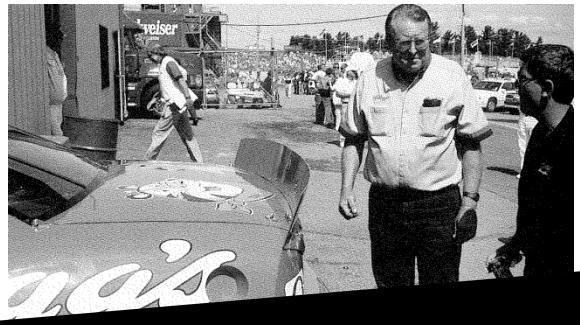


Aerodynamics
Spoilers and Airdams

Spoilers

- The inclination of the rear spoiler also affects the lift and drag results.
- Increasing inclination, increases downforce
- but drag is increased also.
- Significance of drag increase depends on vehicle application.





Aerodynamics
Spoilers and Airdams

Splitters



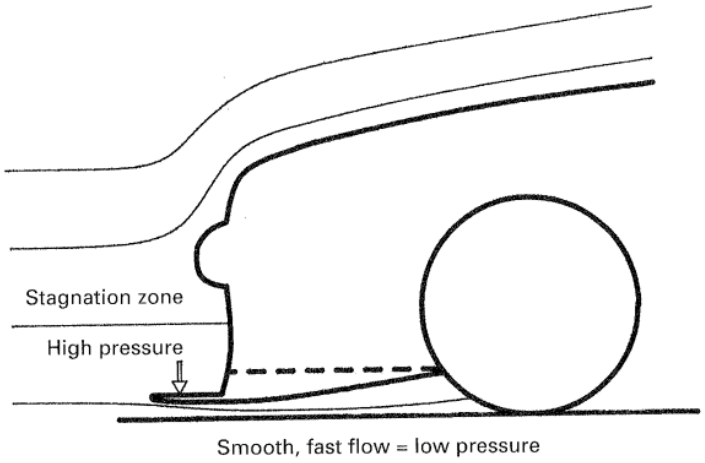
• Splitters are a ‘crude’ aerodynamic device but is actually very efficient.

• Can create significant downforce with no increase in drag!

• At the front of most vehicles, the air slows before it is forced around the vehicle.

•This stagnation zone produces a high pressure usually resulting in drag.

• If a horizontal surface is placed below this high pressure region a vertical force is produced and hence downforce can be achieved.



Aerodynamics
Spoilers and Airdams

Splitters

- If a splitter is used in conjunction with an air dam it will often assist with the function of that air dam.

- The splitter will generally reduce the spillage of air from the stagnation zone to the volume under the car.

- Effectively the splitter has the same affect as increasing air dam height.

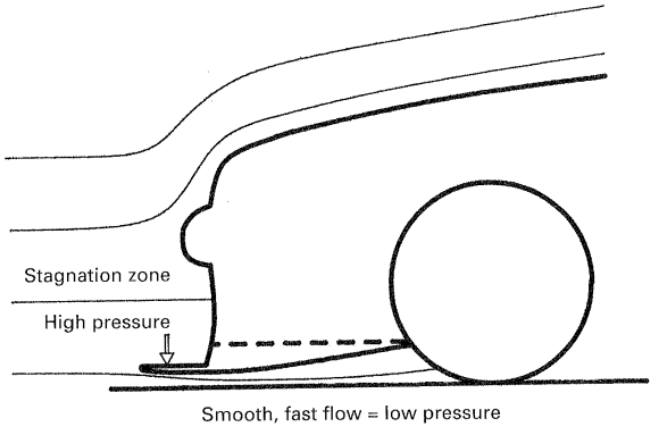
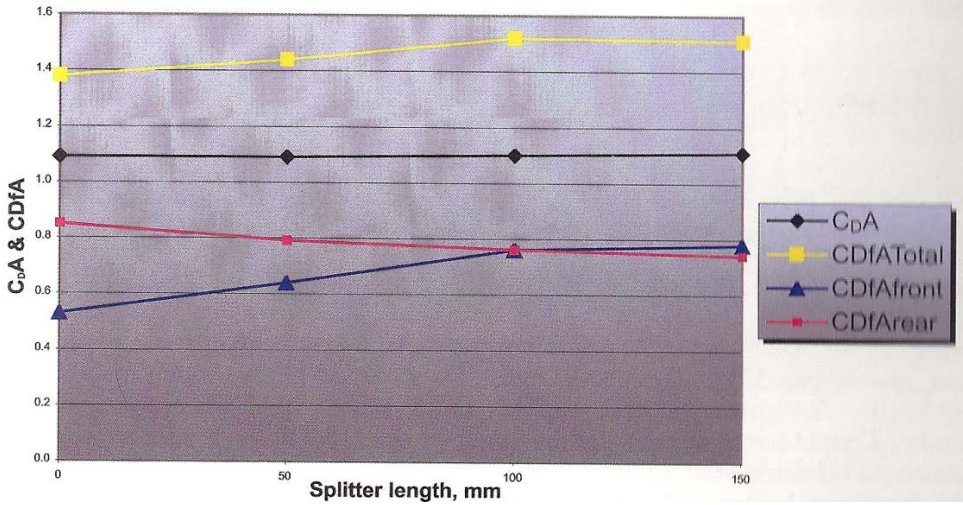


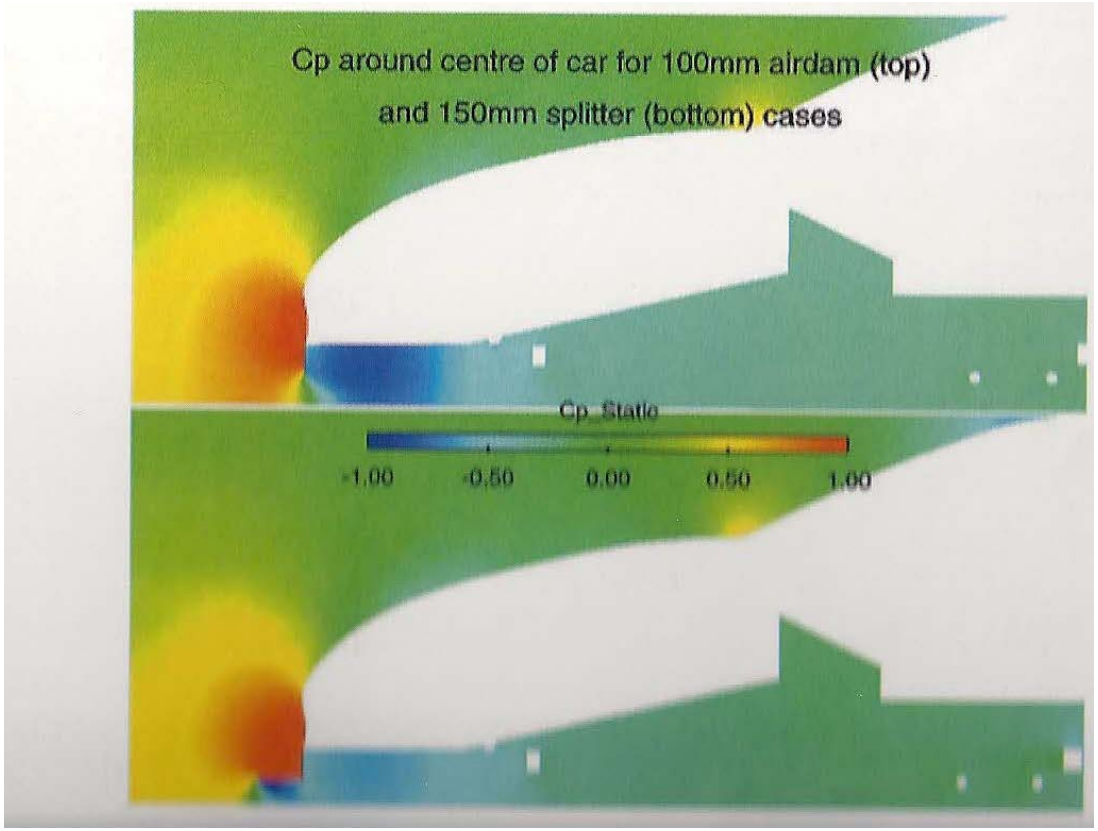
Figure 4-10 *The effect of splitter length on downforce and drag.*



Aerodynamics
Spoilers and Airdams

Splitters

• The pictures blow show the pressure distributions with and without a splitter.



Aerodynamics
Spoilers and Airdams

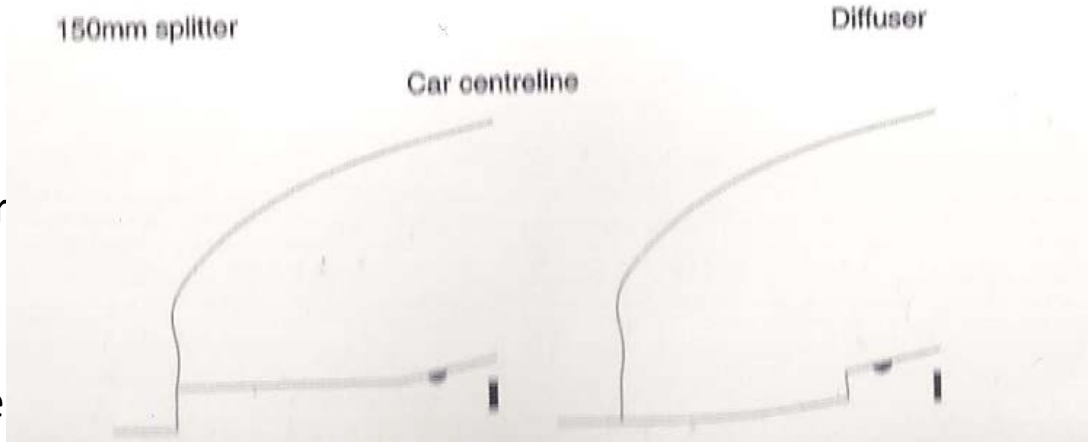
Splitters

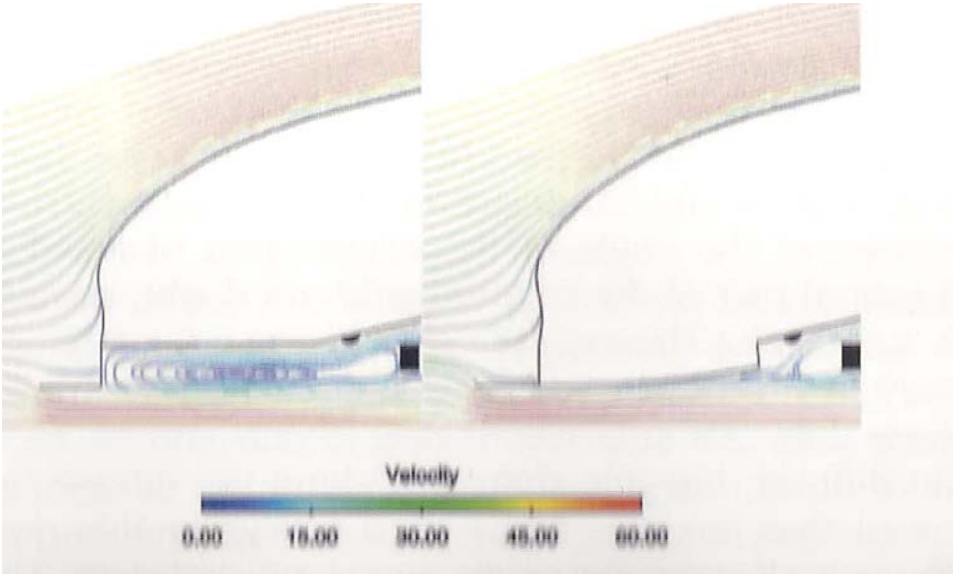
- Splitters may also be employed with air dams which sport a diffuser

- The advantages are again demonstrated by the images on the right.

- The increased velocity under the diffuser produces a lower pressure

- This lower pressure will also act on the underside of the splitter.

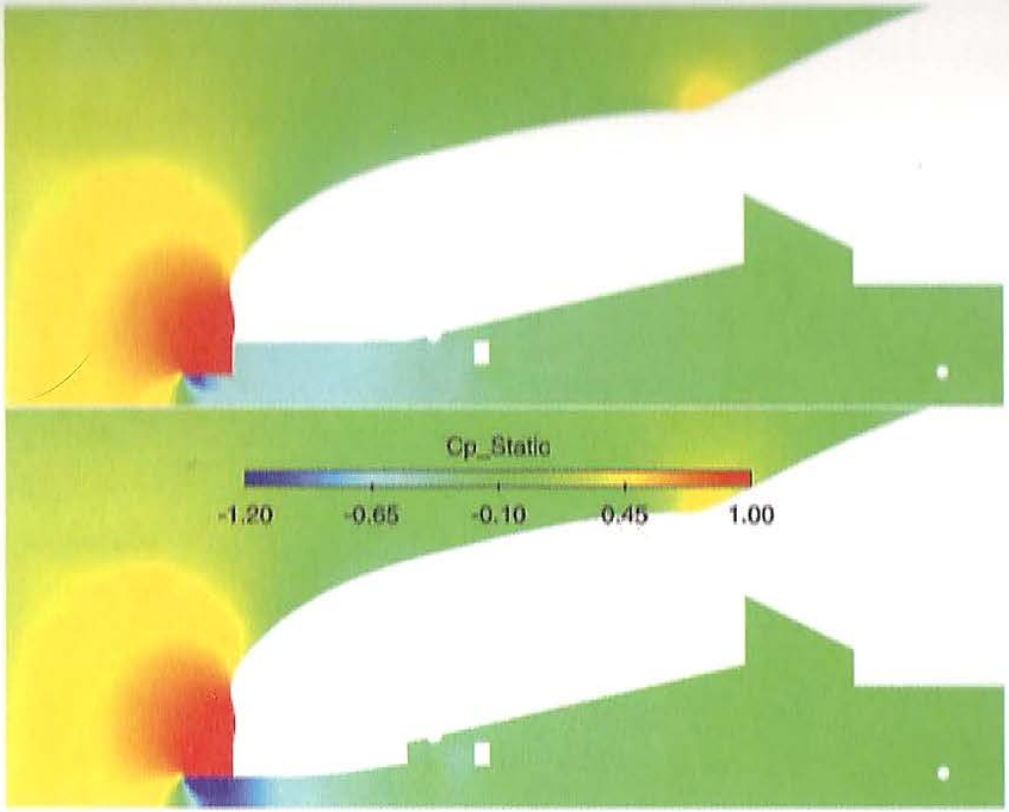




Aerodynamics
Spoilers and Airdams

Splitters

Figure 4-14 *Static pressures are different in the forward underbody with and without the diffuser.*



Aerodynamics

Spoilers and Airdams

Splitters

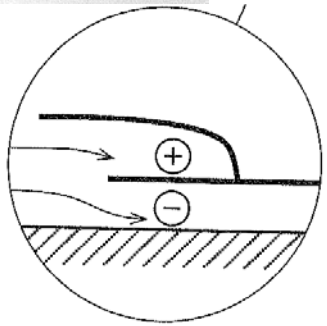
•Splitters may also be applied to open wheel vehicles.

• Under chassis splitters make use of the stagnation zone on the underside of tubs.

• Often help reduce the lift produced by the stagnation pressure acting on the underside of the tub.

• The splitter will also increase low pressure under body by reducing spillage or air from high pressure region.

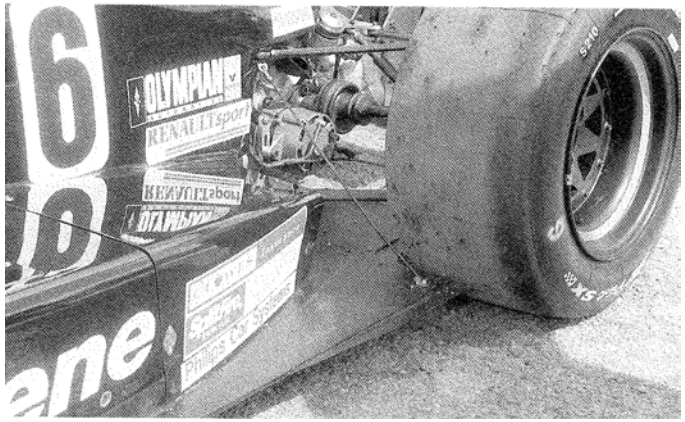




Aerodynamics

Spoilers and Airdams

Splitters

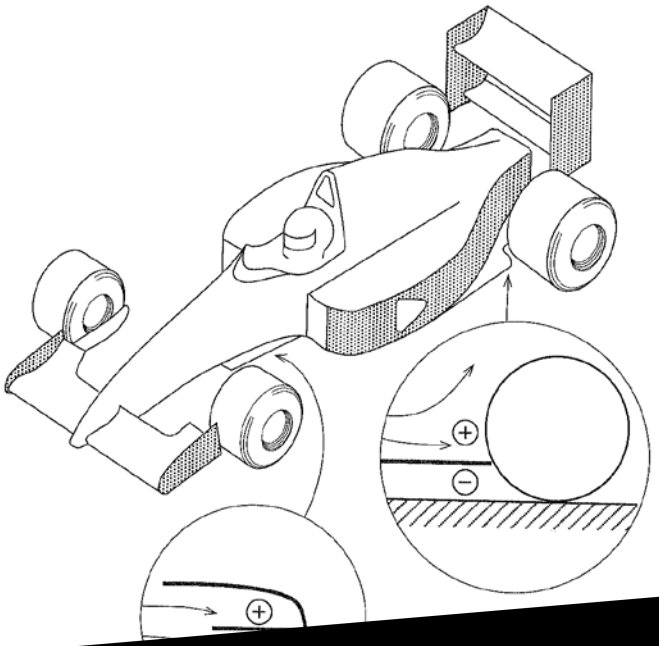


- Another form of splitter is that of the tyre shelf.

- Often employed on open wheel cars

- Make use of the stagnation pressures in front of the rear wheels

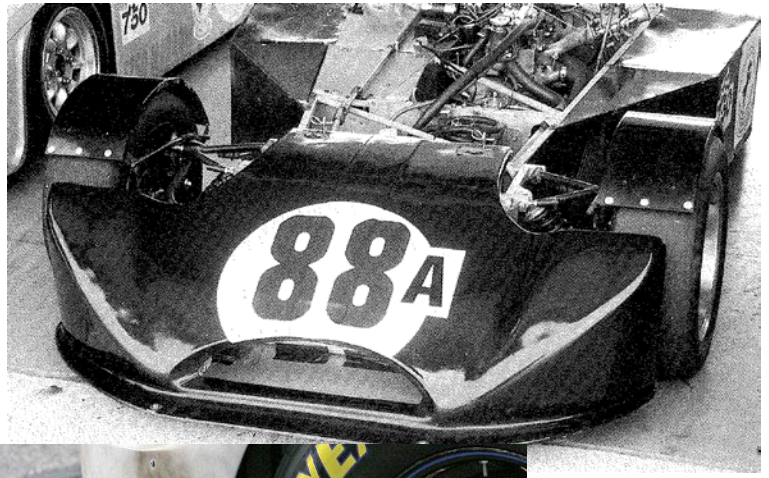
- Horizontal surface provided to yield vertical force from pressure differential across them.



Aerodynamics

Spoilers and Airdams

Splitters







- As splitters are very efficient they are often applied to many race vehicles.

- Easy to install as all that's required is a flat surface which is stiff enough to resist the aerodynamic loads.

- Some splitters require supports to withstand the resulting loads.

- These may be in the form or cables or solid mounts.

Aerodynamics
Other Devices

• Race cars aerodynamic packages become more complex as designs try to maximise downforce and minimise drag within the confines of the regulations.

• Some examples of other appendages not previously presented, which are often applied to race cars are

- Barge boards
- Vortex generators
- Dive Planes
- NACA Ducts
- Louvers

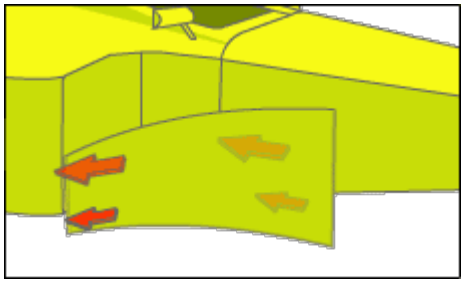




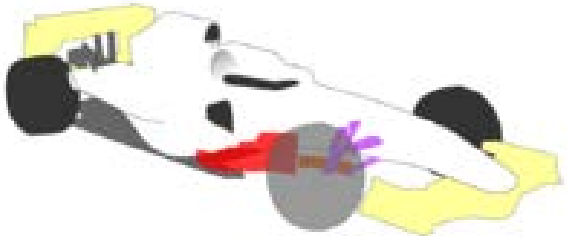


Aerodynamics
Other Devices

Barge Boards



• Bargeboards are aerodynamic devices which take the form of a curved vertical plane situated longitudinally between the front wheels and side pods.



• They are held away from the chassis with struts and generally have a trapezoidal type side profile.

• The original purpose of a bargeboard was to act as flow conditioners to :



- •Shield the radiator ducts from the turbulent wake of the front wheels.
- • Smooth and redirect high drag turbulent flow away from and laminar flow towards aerodynamic devices down stream.
- • Reduce drag of exposed side pods.

Aerodynamics
Other Devices

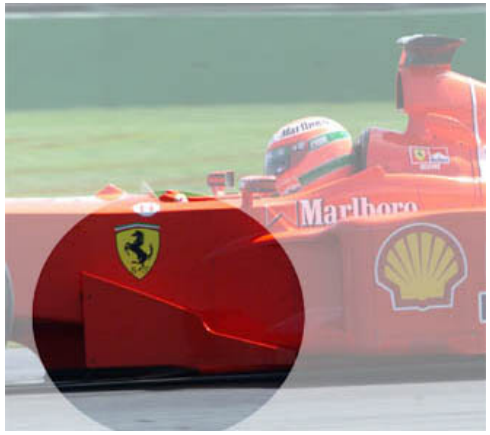
Barge Boards

- In recent years the barge boards role has partially changed.

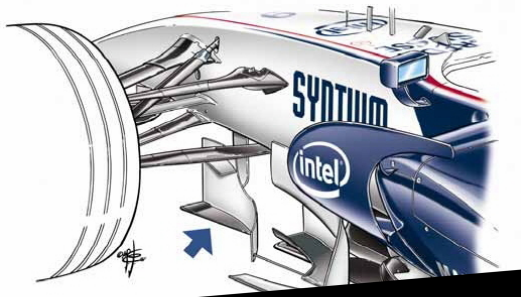
- As well as yielding the previously mentioned benefits bargeboards are increasingly being utilised as vortex generators.

- By careful design barge boards can shed very helpful vortices.

- These vortices can be used to
  - Re-direct air flow
  - Re-energise airflow (by taking energy from free stream)



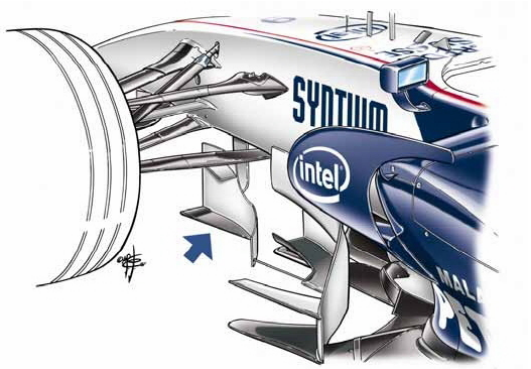




Aerodynamics
Other Devices

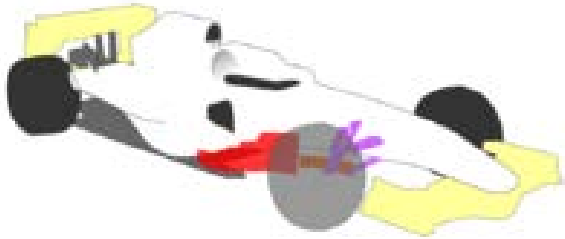
Barge Boards

• The upper downwards sloping edges can can shed a large vortex which



- Smoothes and directs flow around side pods.
- Interacts with various flip-ups downstream
- Aids sealing of chassis to ground helping reduce spillage of pressure to under body.

• The bottom edge of the bargeboard can also shed vortices that energize the airflow to the under body.



•This helps delay flow separation and allows the use of more aggressive diffuser profiles.

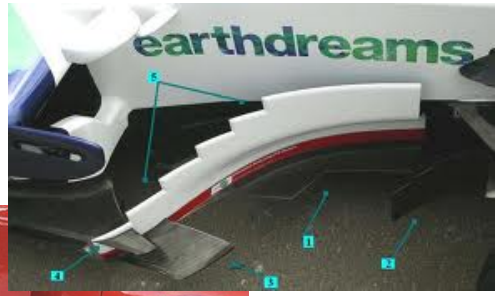
Aerodynamics
Other Devices

Barge Boards

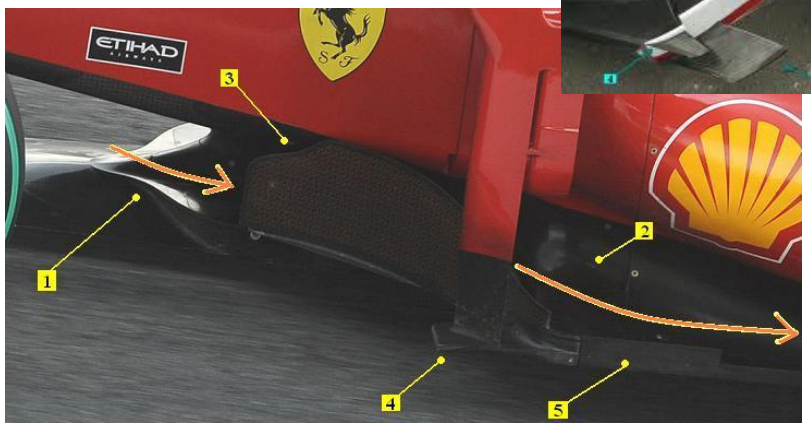


- In recent years, these two edges have acquired relatively large serrations or protruding tabs to increase and shed controlled vortices.

- In f1 however, the regulations changed which reduced the size and complexity of barge boards reducing their effects.







Aerodynamics
Other Devices

Turning Vanes



• Turning vanes are often used in front of barge boards to condition and redirect the flow behind the front wing.

• Generally a similar structure to barge boards but smaller in size.

• Turning vanes are also used in other areas on a vehicle to control flow, such as

- Under a raised front chassis
- Around the leading edge of side pods
- As part of front wing end plates







Aerodynamics
Other Devices

Vortex Generators

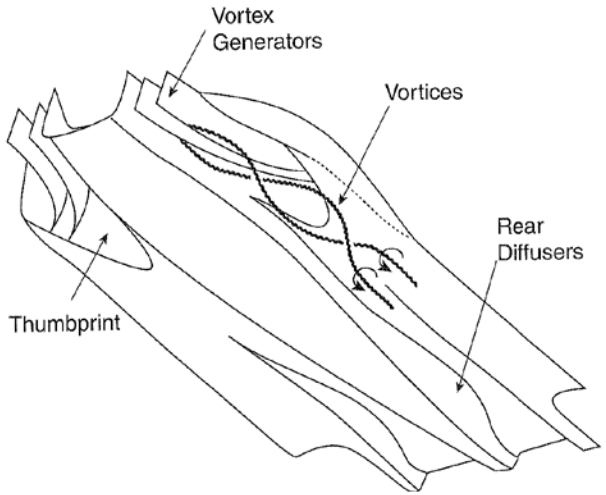
• As well as vortex generators on wing surfaces, specific VGs may be applied in different areas on a vehicle

• On the right vortex generators take the form of two flat curved plates on the under body. (They could be straight and also work)

• These plates create tip vortices which snake under the body.

• Due to the high speed around the vortex core low pressures surround the vortex snake

• Low pressure → Downforce



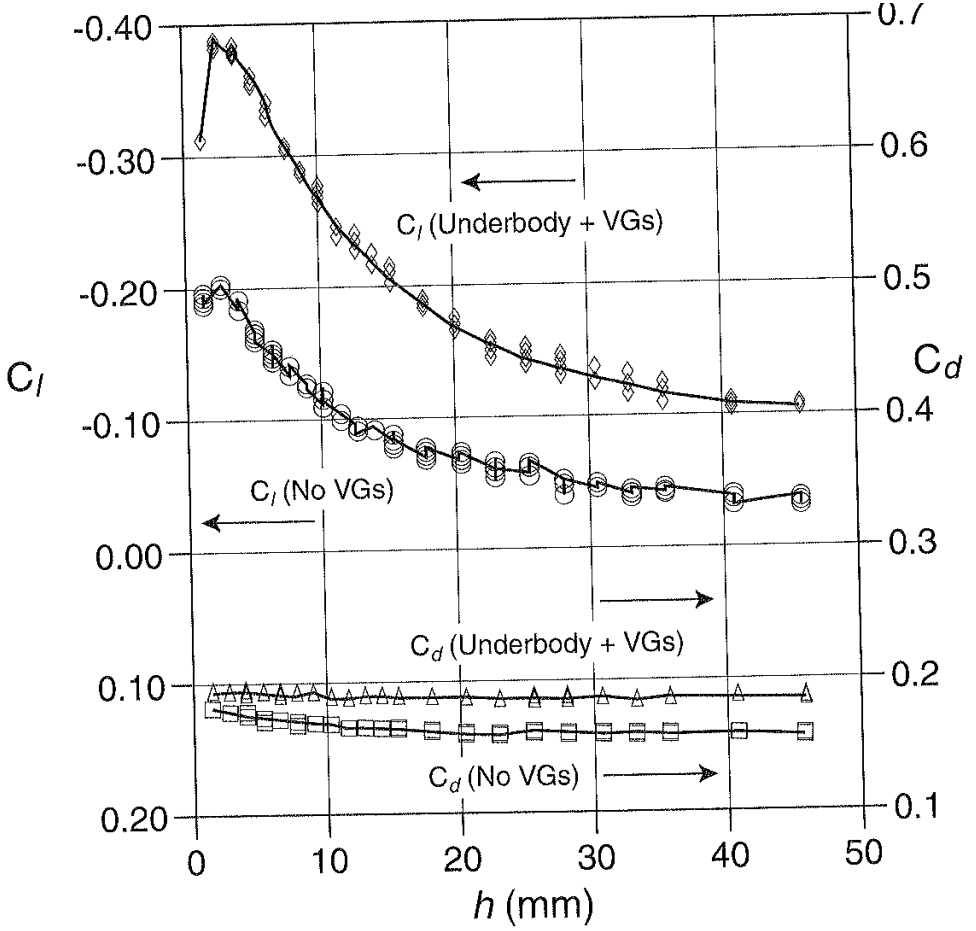
Aerodynamics
Other Devices

Vortex Generators

- The combined drag and downforce contribution of the under body with VG shape is show on the right versus ride height.

- In general both downforce and drag increase with decreasing ride height.

- This is explained by the fact that as ground clearance reduces the vortices ‘untangle’ and are brought closer to the under body.



Aerodynamics
Other Devices

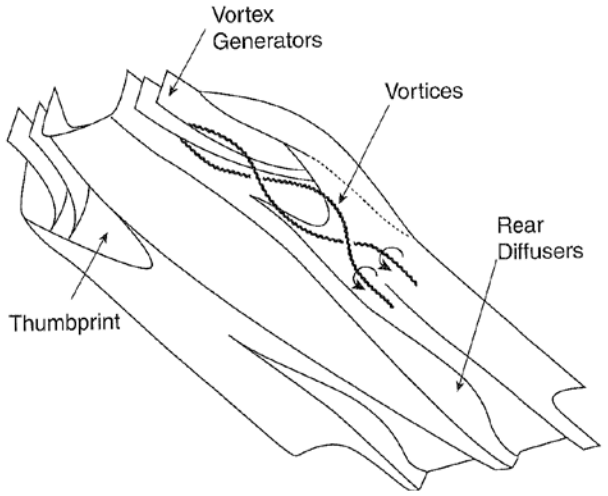
Vortex Generators

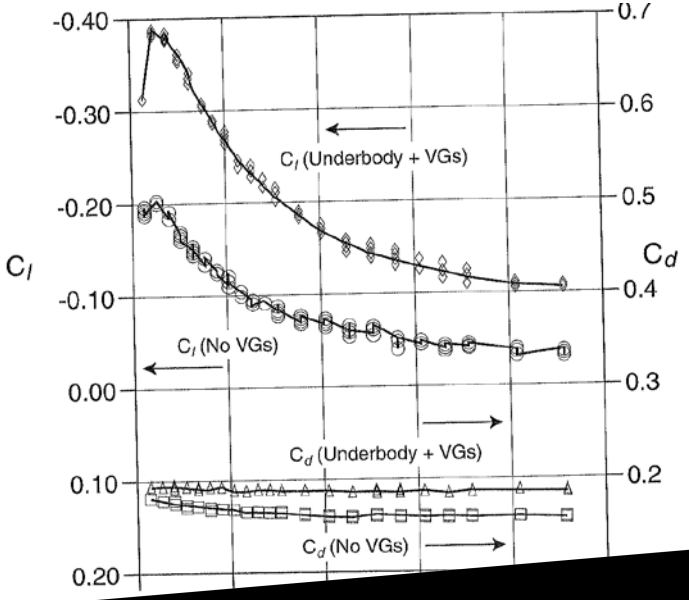
- When the car is out of ground effect the vortices roll round each other as shown reducing their suction effect.

- However when in ground effect the vortices are forced to unroll and squeeze through the narrow gap.

- This increases the vortex strength and reduces the distance from the body, increasing suction.

- At very low ground clearance the flow is blocked and hence the vortices break down.





Aerodynamics
Other Devices

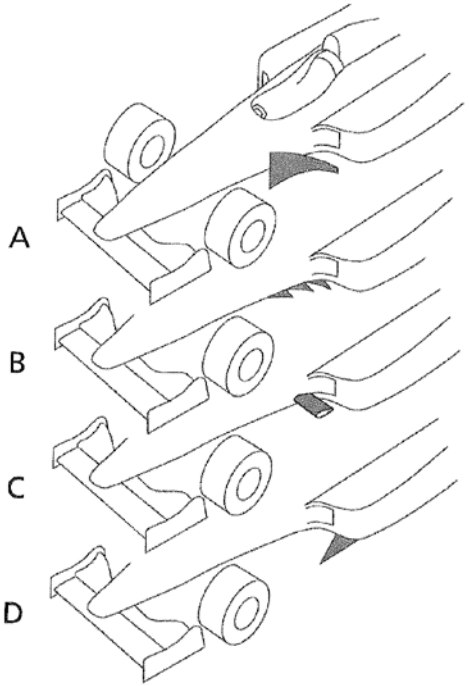
Vortex Generators

• As mentioned previously devices ahead of the side pod are often used to also aid the under body flow by means of creating vortices.

• These have taken the form of saw-tooth or small wings.

• Many of these devices use the principle of wing tip vortices to either:

- Seal the sides of the under body
- Increase the air flow under the vehicle
- Increase the velocity of the air flow under the body by feeding energy from the free stream.





Aerodynamics
Other Appendages

Dive Planes

- Dive planes are flat plates attached to the front of a vehicle which have an inclination to the horizontal.

- These devices,

    - Help divert air flow away from the front wheels.

    - Produce downforce due to their inclination to flow.







Aerodynamics

Other Devices

NACA Ducts

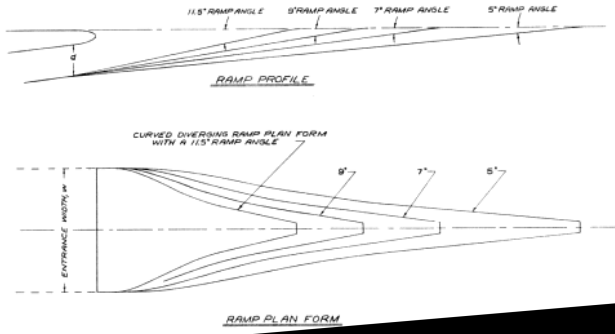


• Submerged NACA ducts are a type of air intake which were developed by NACA (predecessor to NASA)

• The purpose of a NACA duct is to increase the flow rate of air through it by reducing the thickness of the boundary layer inside it.

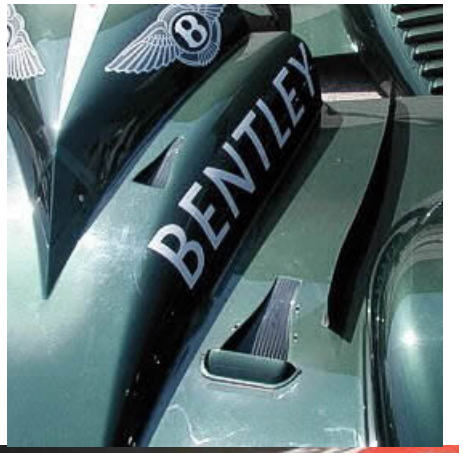


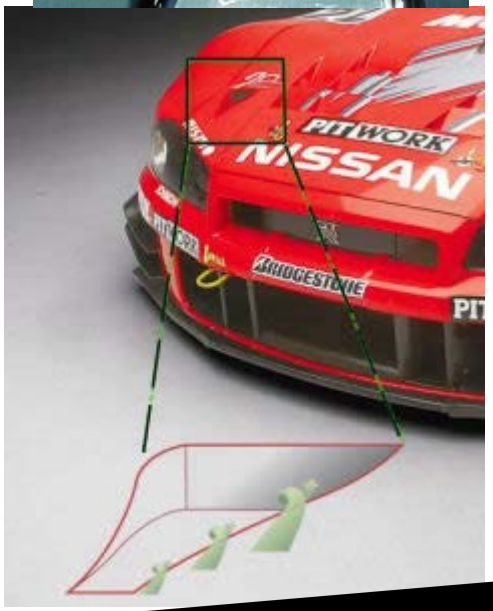
•The reason why the duct is narrow, then suddenly widens is to prevent separation and turbulence (and drag) whilst inducing counter rotating vortices.



Aerodynamics
Other Devices

NACA Ducts





- This design works because the combination of the gentle ramp angle and the curvature profile of the walls creates counter-rotating vortices at the sides of the duct.

- These vortices pull momentum energy from the free stream to re-energise the boundary layer within the duct and reduce its thickness. This also helps draw in more air.

- However the NACA duct avoids the form drag and flow separation that can occur with protruding inlet designs.

Aerodynamics
Other Devices

NACA Ducts

•If you are installing a NACA duct

-Design is very important. Use a predefined shape.

-The duct needs to be installed in an area of relative high pressure.

- Edges on the slopes have to be sharp, to create vortices.

- NACA duct is only useful in applications where you don't really need lots of air.

- NACA ducts only really work under a positive pressure gradient (where the air sees the car increasing, not decreasing, in size.

Aerodynamics
Other Devices

Louvers

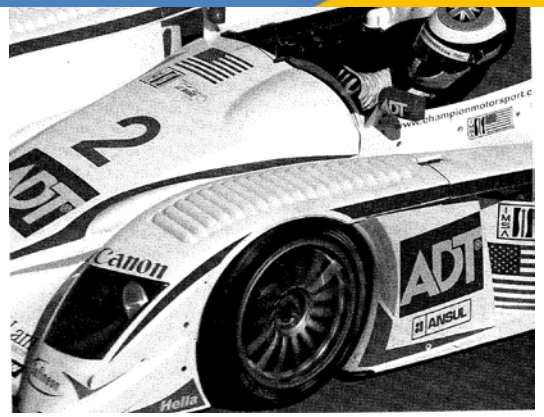
- Louvers are ventilation slots which resemble cooling exits, usually found in front fenders of vehicles.

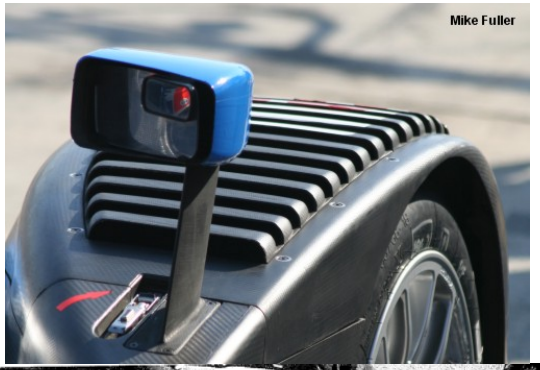
- Also sometimes called gills or slots.

- When properly placed they can create downforce.

- Near a surface with convex curvature, such as above a wheel arch, low pressure results which contributes to lift.

- Louvers may be placed in such areas to let higher pressure bleed into these areas (from below) and spoil high speed laminar flow.







Aerodynamics

Other Devices

Louvers

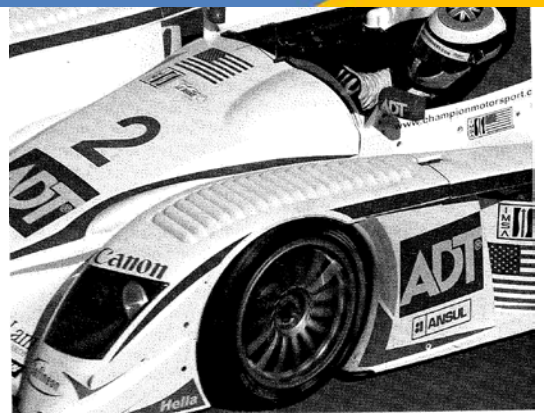
- Louvers may also be used to improve under body flow in some cases.

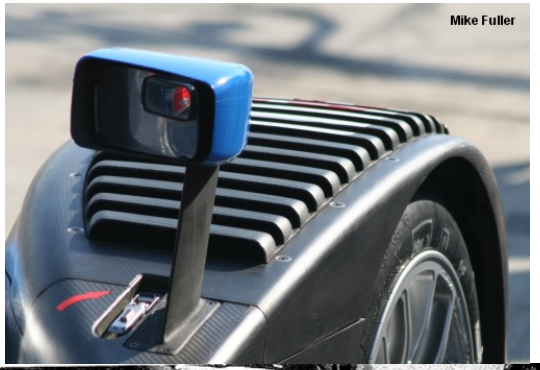
- A louver could be used to bleed high pressure from ahead of the front wheels to help increase the flow under the vehicle.

- This leads to a further increase in downforce.

- A mutation of this idea can be seen in the lowest picture.

- This cowl flap is an adjustable opening which reduces the lift produced by the curved wheel arch.







Aerodynamics
Other Devices

Louvers

- Louvers are also used for cooling.

- High speed flow on upper surfaces creates negative pressure and hence lift.

- Cooling outlets benefit from exhausting into low pressure regions

- The extra flow into these regions helps reduce this negative pressure and hence lift.



Aerodynamics
Other Devices

Other Appendages!

• Other appendages to try and improve aerodynamic performance have also been used on race cars.

• Before 2009 F1 cars could use lot of different appendages to try and condition flow by means of specific turning vanes or VGs.

• Below are some examples of how such devices were used to condition the flow around the cockpit.







Aerodynamics
Other Devices

Other Appendages!

- Flow conditioners also appeared near the air box.

- These aimed to smooth and direct the ‘dirty’ air from around the cockpit to the rear wing.

- Shark fins also have been employed to help improve yaw stability of the vehicle,

- Flip ups are also used to direct flow above and around wheels to help reduce lift and drag.







Aerodynamics
Internal Flow

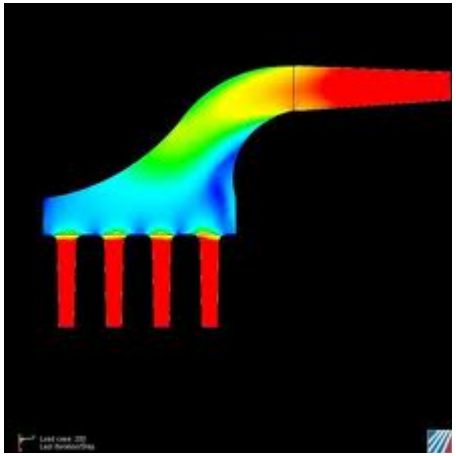
Internal Flow



• Aerodynamic analysis is not confined to the external aerodynamics of a vehicle.

• Aerodynamics obviously have a large influence on;

- Cooling system shape
- Air induction systems
- Intake manifold
- Exhaust manifold
- Cylinder head design

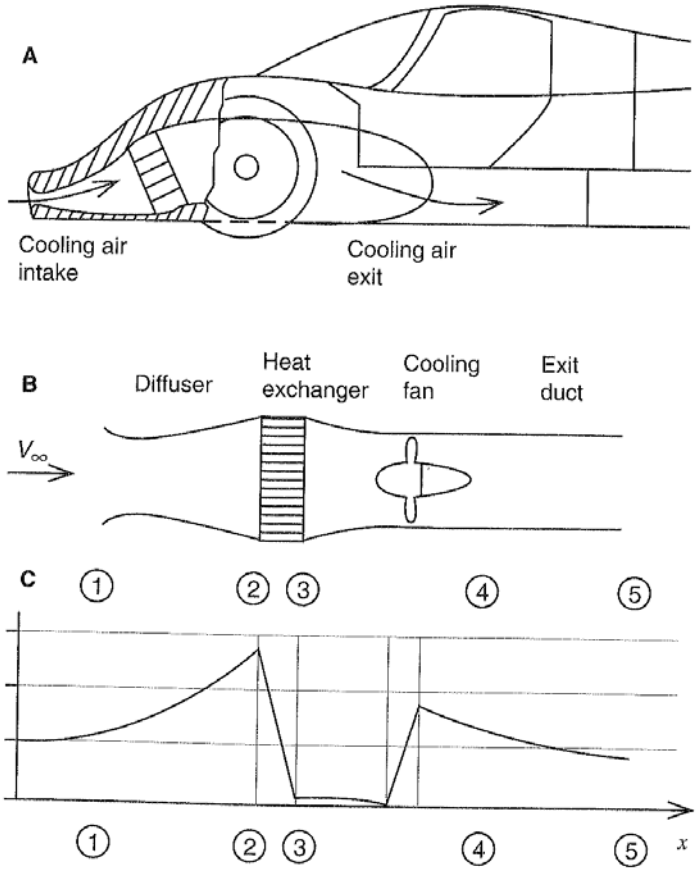


• Flow in these areas often experience viscous flow effects (turbulence) and separation as well as heat transfer.

Aerodynamics
Internal Flow

Cooling System

- Most common internal flow system on race cars is the cooling system.
- Transfers heat rejected from radiators to external flow.
- General layout of a cooling system shown in B.
- Air enters through nose intake.
- Air slowed by short diffuser and passes through radiator.
- Air ejected into low pressure behind wheel to reduce drag.



Aerodynamics
Internal Flow

Cooling System

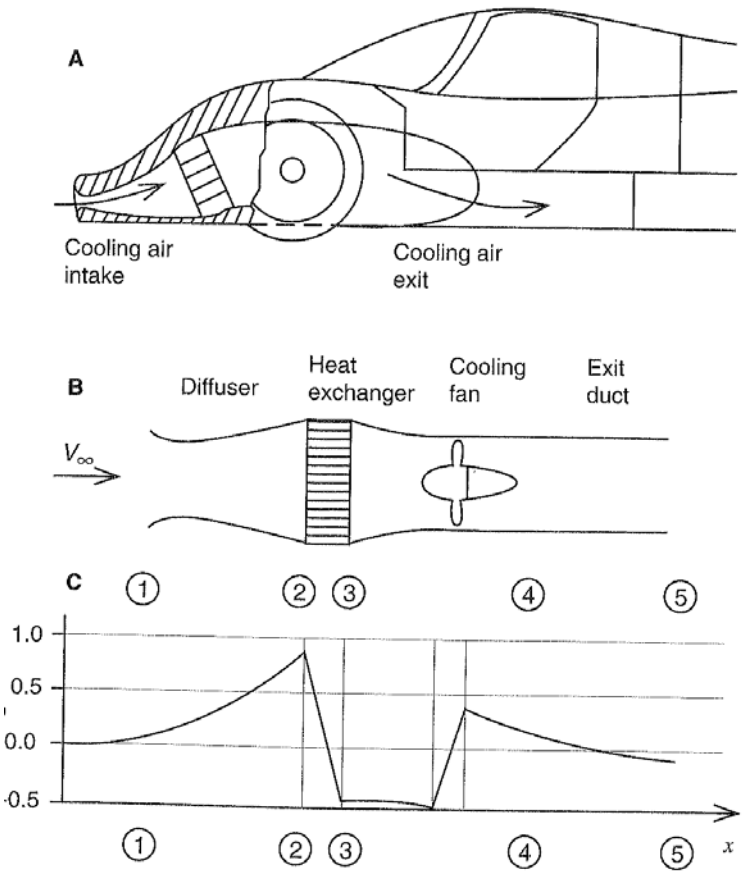
- Pressure co-efficient shown in C

- Diffuser slows air and hence pressure increases.

- As flow passes through radiator pressure drops due to heating effect and friction

- Most systems then have a fan which helps create a negative pressure at the exit to drive flow.

- This negative pressure may be produced by placing the exits in an area of sufficient low pressure. Regulations normally require this.



Aerodynamics
Internal Flow

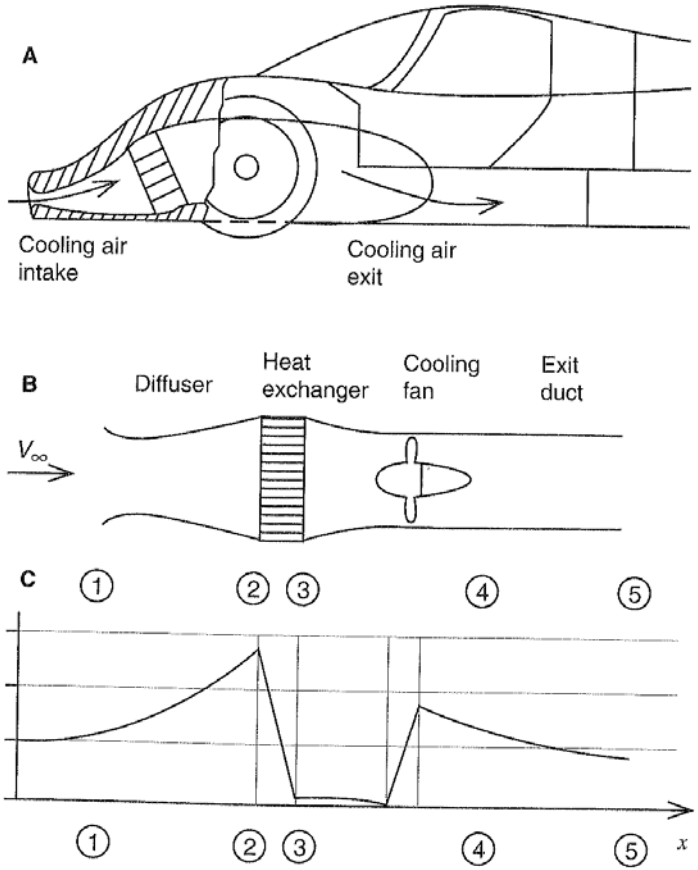
Cooling System

• Many race car regulations do not allow moving aero devices such as fans.

• In this case the cooling flow must be driven by negative pressure at the exit

•Exits must be placed in an area of sufficient low pressure to create a large enough pressure differential.

• Main disadvantage is pressure differential is a function of vehicle speed.



Aerodynamics
Internal Flow

Cooling System - Drag

• Cooling systems with internal flow, usually result in an increase in drag.

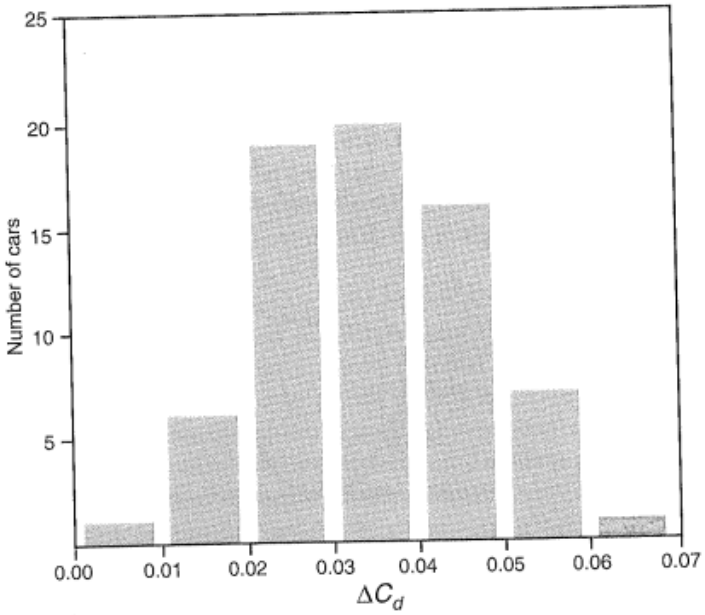
• Graph shows increase due to cooling system for a variety of passenger cars.

• Main points to note.

  • A poorly designed cooling system may increase drag by up to 0.07!

  • But a good system may achieve a drag increase of close to zero.

  • Average is around 0.035...



Aerodynamics
Internal Flow

Cooling System – Minimising Drag

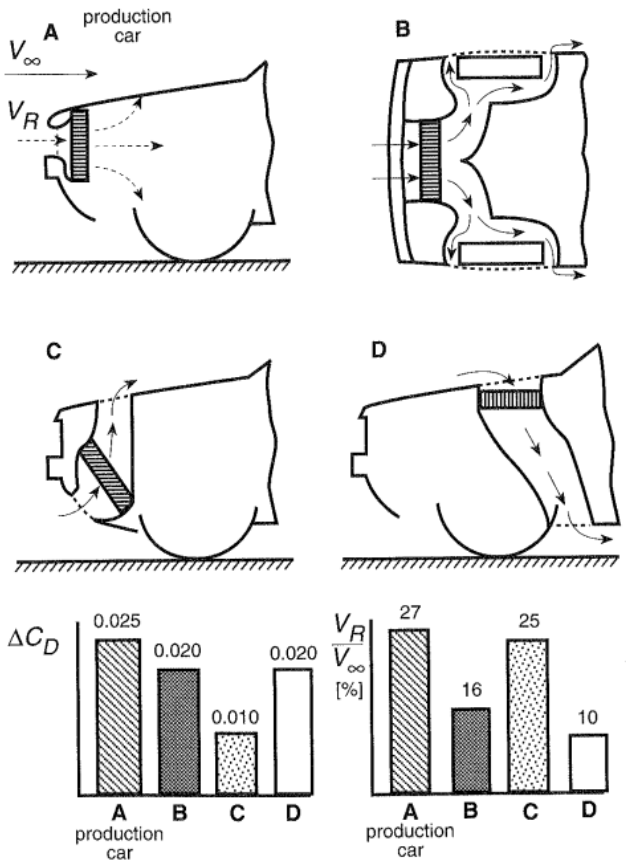
- Negligible increases in drag (or even a total reduction of drag) can be achieved through careful design.

- Use the momentum of the heated air to keep flow attached or provide pressure to low pressure regions.

- Diagram shows drag increments of various configs.

- Note positioning of exit or inlet ducts perpendicular free stream doesn't guarantee lower drag.

- Important to match cooling system design with external aerodynamics .

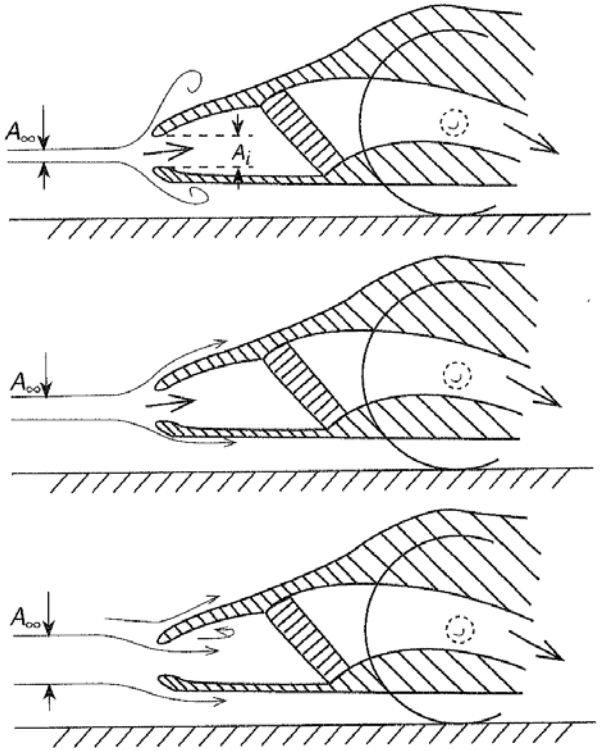


Aerodynamics
Internal Flow

Cooling Systems –Speed

- Case study of system design.
- Intake area designed for specific speed and not optimal for higher or lower speeds.
- At higher speed only fraction of incoming streamlines can enter and external spillage results.
- At lower speeds internal separation may occur.

Flow condition with fixed inlet but at varying speed	Inlet size at constant speed
High speed	Too large
Result: spillage, external separation	
Design speed	Matched
Result: desirable smooth flow	
Low speed	Too small
Result: internal separation	



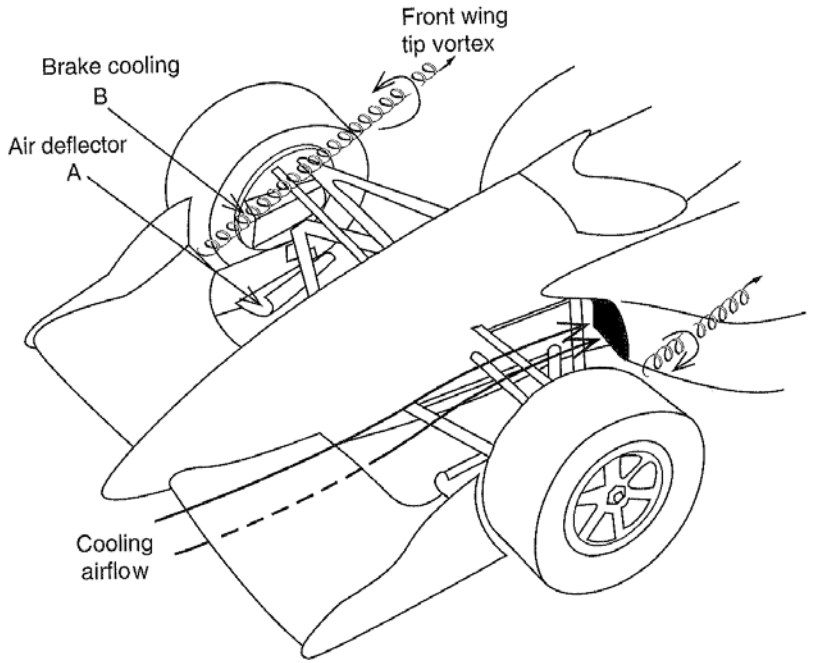
Aerodynamics
Internal Flow

Front wings and cooling.

•Front wing designs must often take into account cooling requirements.

• Front wings make the air flow do work and often leave vortices and wake downstream.

• Wings often designed to minimise dirty air to cooling intakes by reducing frontal area in area in front of intakes.



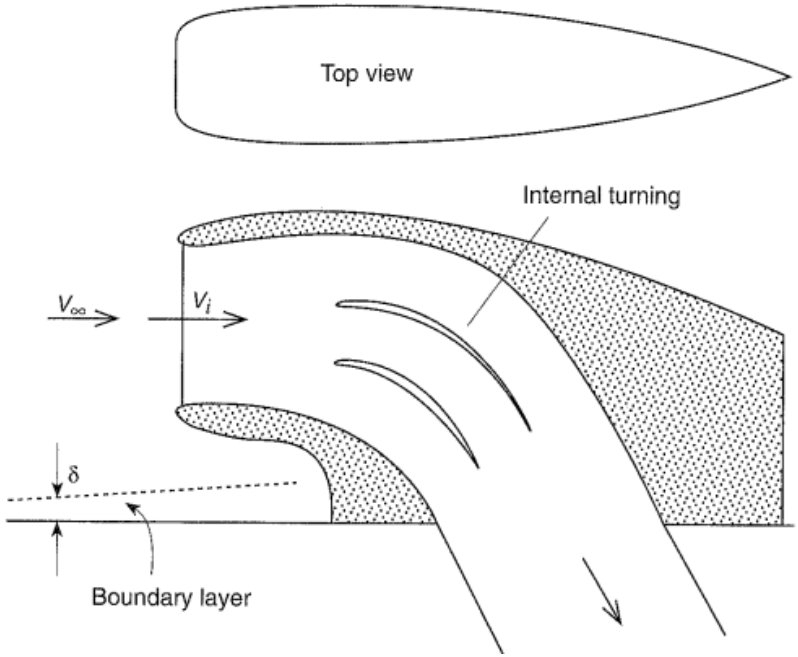
Aerodynamics
Internal Flow

Airscoops

- Most common intake locations are at front of vehicle near stagnation.

- Some intakes such as brake ducts and air induction mounted in less desirable locations

- Typical airscoop
  - Streamlined top view
  - Elevated above boundary layer
  - Internal diffuser to decelerate flow.
  - Avoid internal separation by guide vanes
  - Drag increments ~ 0.12-0.3





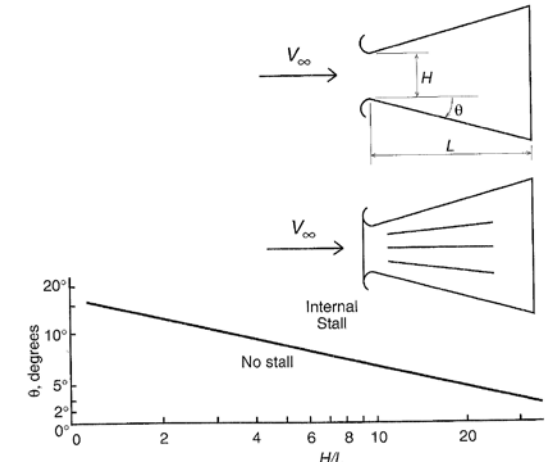
Aerodynamics
Internal Flow

Internal Diffusers

- Performance of internal intake diffusers is often sensitive to;
  - External flow conditions
  - Internal geometry

- Large bodies of data available on diffuser design

- The length to width ratio of a diffuser should fall below the no stall line on the graph shown.



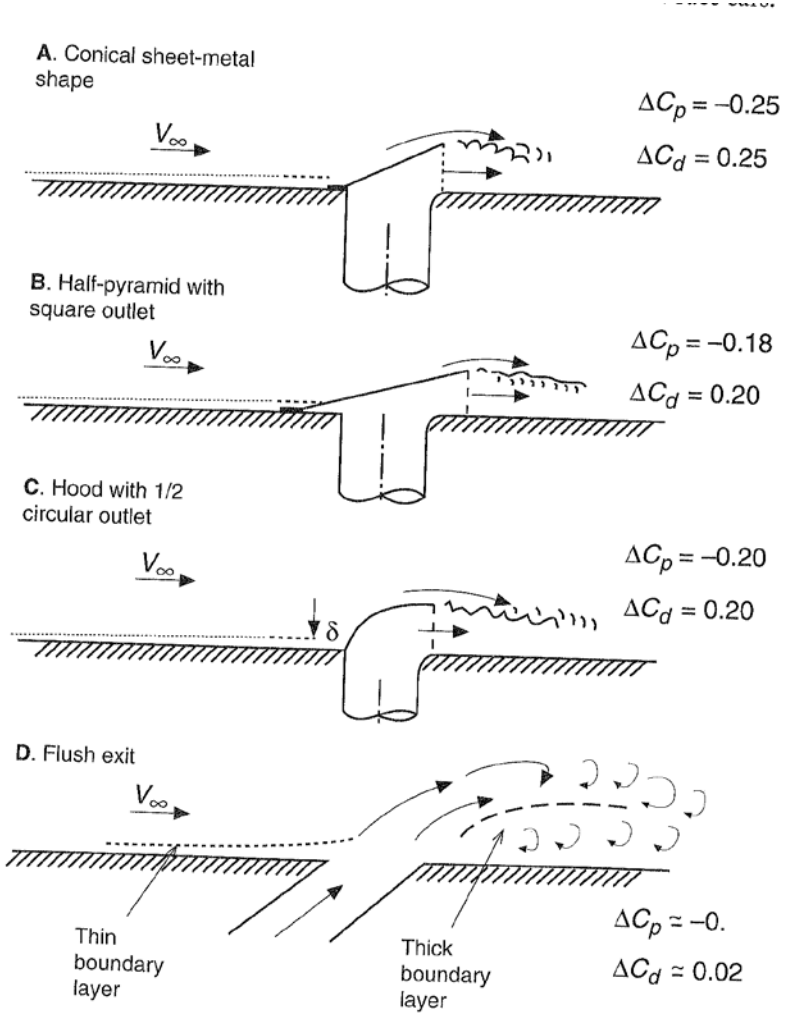
- Interesting to note that longer diffusers can have a larger ramp angle but angles above 15 degrees cannot be used (according to diagram!!)

- Splitter vanes can help improve performance of splitter operating near stall line.

Aerodynamics
Internal Flow

Exit Design

- Performance data for different forms of exits is also available.
- Exits should normally be placed in areas of low pressure to increase pressure drop.
- Increment in pressure drop due to such devices along with corresponding drag increment shown.
- Most of these increase BL thickness downstream
- This increases likelihood of separation of downstream flow.

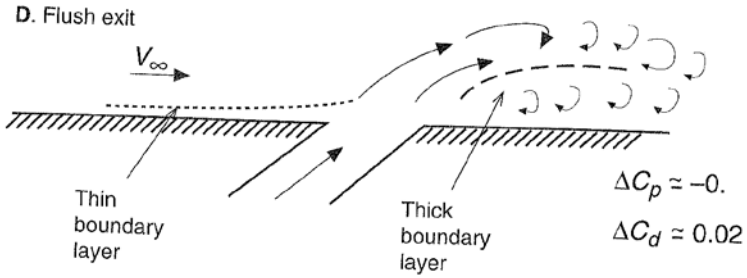


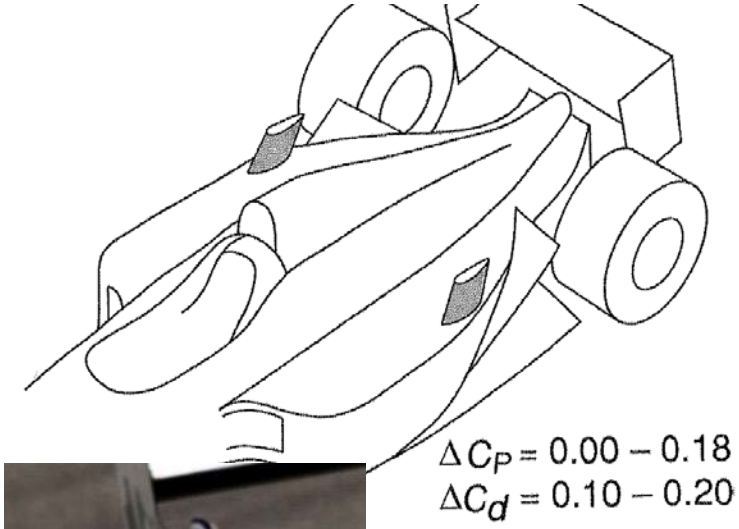
Aerodynamics
Internal Flow

F1 Chimneys

- F1 cars used exhaust chimneys which are a mutation of the previous exit designs.
- Used on variety of race cars
- In this case design D is extended into free flow
- Raised exit helps minimise boundary layer thickness and separation.
- Chimneys can also be used to help direct hot exhaust gas flow away from rear wing

(hot gas = low density = low downforce)







Aerodynamics
Internal Flow

Using Cooling Flow (A)

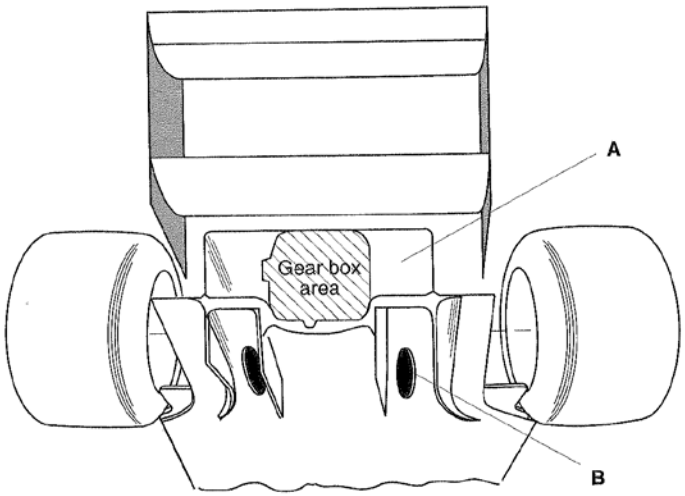
• Cooling flow can also be used to help prevent separation in some areas behind vehicles.

• Momentum of cooling flow is exited at points where separation is possible (A)

• Extra momentum helps re-energise boundary layer.

• Negative pressure region also helps draw air through cooling system.

• Implemented on sedans where measurable drag benefits observed.

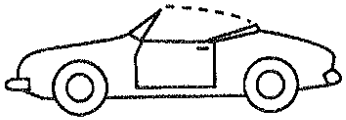
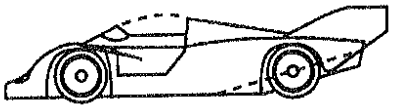




Aerodynamics
Other

Other factors

- Other factors that can increase drag due to internal recirculation.
    - Open and closed windows
    - Open and closed cockp
- Sports prototype increment smaller than convertible due to careful cockpit shaping.

			ΔC_D	ΔC_L	Ref.
Effect of open window	Sedan		0.067	N/A	2.3
	Group B race car		0.025	0.005	
Effect of top	Convertible sports car		0.09	N/A	2.6, P. 194
	Group C race car		0.05	0.05-0.10	1/5 scale experiment
Effect of venturi			0.05	0.6	

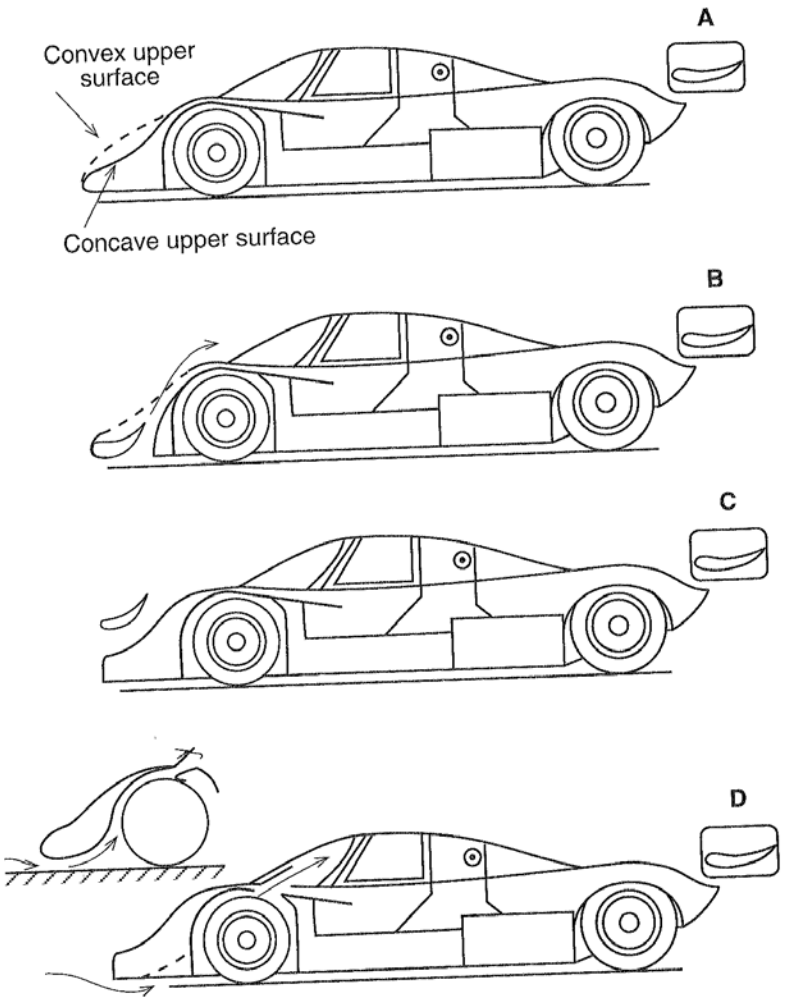
Aerodynamics
Other

Hard to recognise wings

- Some wings are also hard to recognise as flow is ‘almost’ internal!

- Some examples are shown on right in B,C, and D

- Top diagram shows concave surface creates stagnation pressure and horizontal surface on which it can act.



Aerodynamics
Other

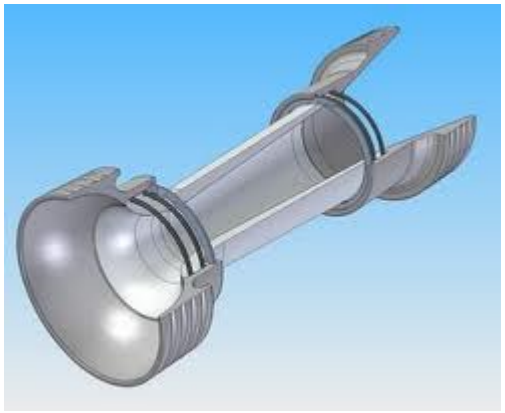
Restrictors

• Restrictors are used to limit the power of an engine by reducing the amount of air that can be drawn into the cylinder.

• Effectively just a venturi which increases speed of flow.

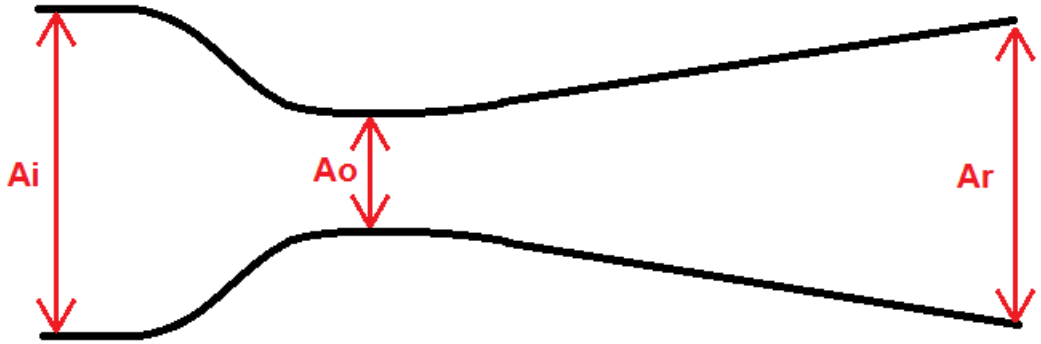
• When speed of flow reaches speed of sound restrictor ‘chokes’ as air becomes compressible and shock wave forms.

• As soon as speed of sound is reached this is the maximum flow rate.



Aerodynamics
Other

Restrictors



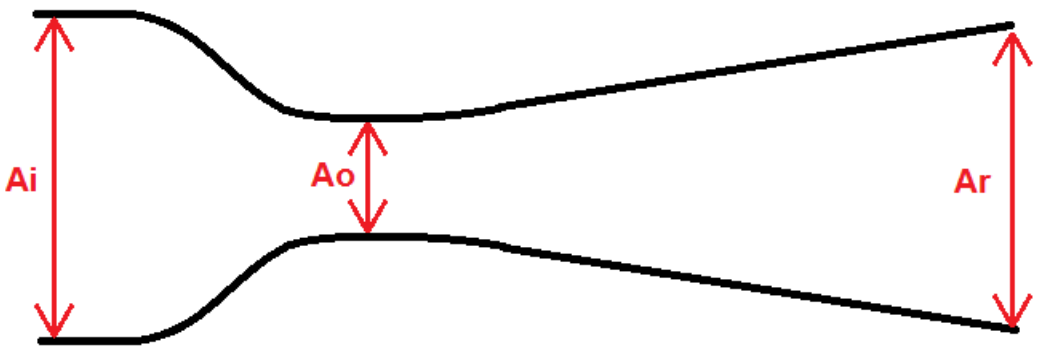
- Flow Rate

$$Q_i = v_i A_i$$
$$Q_r = v_r A_r$$
$$Q_o = v_o A_o$$

- Max flow rate when velocity at restriction reaches speed of sound (340.29 m/s)
- No more air can be forced through restriction after this.

Aerodynamics
Other

Restrictors



- Volume of air drawn by engine

$$V_{air/\sec} = \frac{RPM}{60} \times \frac{Total\ Capacity_{(cc)}}{2} \times 1 \times 10^{-6}$$

$$Max\ V_{air/\sec} = v_s \times A_o$$

$$v_s \times A_o = \frac{RPM_{\max} \times Total\ Capacity_{(cc)} \times 1 \times 10^{-6}}{120}$$

$$RPM_{\max} = \frac{120 \times v_s \times A_o}{Total\ Capacity_{(cc)} \times 1 \times 10^{-6}}$$

Aerodynamics
Other

Restrictors - Questions

• Calculate the chocking RPM for a 2000cc 4 stroke race engine which must run through a restrictor of radius 12mm.

$$RPM_{\max} = \frac{120 \times v_s \times A_r}{Total\ Capacity_{(cc)} \times 1 \times 10^{-6}} = \frac{120 \times v_s \times \pi \times (12 \times 10^{-3})^2}{2000 \times 1 \times 10^{-6}} = 9237 RPM$$

• Calculate the restrictor size (diameter) required to limit the engine speed of a 1800 cc 4 stroke race engine to 13000 RPM.

$$A_r = \frac{RPM_{\max} \times Total\ Capacity_{(cc)} \times 1 \times 10^{-6}}{v_s \times 120} = \frac{13000 \times 1800 \times 1 \times 10^{-6}}{340.29 \times 120}$$

$$A_r = 573.04 \times 10^{-6} m^2$$

$$r = \sqrt{\frac{A_r}{\pi}} = \sqrt{\frac{573.04 \times 10^{-6}}{\pi}} = 13.51 mm$$

Aerodynamics

Other