International Journal of Research
Available at
https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 10
September 2017

Aerodynamic Analysis on a Car Profile

M. Mahesh Babu[1], M. Rajkumar Reddy[2] & K. Krishna[3]
[1]Assistant Professor, Department of Mechanical Engineering, Anurag Group of Institutions, Hyderabad.

Abstract:

Now a days automobile industry is facing one of the major problem i.e. fuel crunch. In order to utilize the fuel effectively, the power losses due to various factors such as friction loss, aerodynamic effects, has to be reduced. This project mainly deals with engine losses due to aerodynamic effect of car profiles. TATA INDICA car has been selected for Aerodynamic Analysis. A scale model of 1:16 car was made and tests in the wind tunnel are conducted. The coefficient of drag was calculated for various wind velocities. .

Keywords

TATA INDICA; CATIA; Aerodynamics

Introduction

The Engineering problems requiring knowledge on drag force are varied and many. The determination of wind pressures on buildings, on bridges, determination of efficient body shape of automobiles and trains, which would result in minimum fluid resistance are some of the important problems which would require a thorough understanding of phenomenon of drag. The principle of drag and lift are now a day is being used in the design of propellers, turbines and centrifugal pumps in addition to hydrofoils.

The analysis of problems involving these phenomena, where the objects moves through the stationary fluid, a force is exerted by the fluid on the body or the fluid passes around the stationary objects, force is exerted by the fluid on the body . It is, thus possible to test aero planes, automobile models in wind tunnels, submarines and torpedo models in water tunnels to predict the behavior of their prototype.

Aerodynamics is the science of how air flows around and inside objects. More generally, it can be labeled "Fluid Dynamics" because air is really just a very thin type of fluid. Above slow speeds, the air flow around and through a vehicle begins to have a more pronounced effect on the acceleration, top speed, fuel efficiency and handling.

Therefore, to build the best possible car we need to understand and optimize how the air flows around

and through the body, its openings and its aerodynamic devices.

For a moving vehicle the flow process is subjected to fall into the following three categories.

1. Flow of air around the vehicle.
2. Flow of air through the vehicle's body
3. Flow processes within the vehicle's machinery.

The first two flows are closely related and it must be considered together. On the other hand, third one is within the engine and transmissions are not directly connected with the first one.

Automobile vehicles are influenced by the aerodynamic drag and its non-dimensional number CD the drag coefficient has almost place a important role for the entire discipline.

Performance, fuel economy, emission and top speeds are important factors of a vehicle; they are all influenced by drag.

The flow around a vehicle is responsible for its directional stability as well as straight line stability. Dynamic passive steering and response to cross-wind depends on external flow field. Further the outer flow should be turned to prevent the droplets of rainwater from accumulating on windows and outside mirror to keep the headlights free from dirt to reduce wind noise, to prevent the wind shield wires from lifting off and to cool the engine's oil an muffler brakes etc.

The main characteristics of the flow around a vehicle can be made visible by using a wind tunnel. The aerodynamic drag FD as well as the other components of the resulting air force and moments increases with the resulting air force and moments increases with the square of vehicle's speed V.

$$FD \propto V^2$$

With a medium sized car aerodynamic drag typically accounts for 70% to 80% of the total resistance to motion at 100 kmph. Hence reducing aerodynamic drag contributes significantly to fuel economy of car.

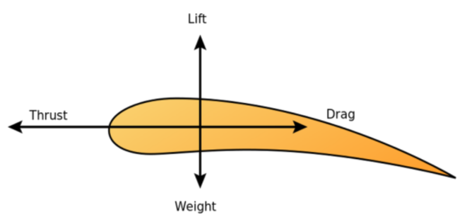


Fig 1. Aerodynamic forces

Aerodynamic force is exerted on a body by the air (or some other gas) in which the body is immersed, and is due to relative motion between the body and the gas. Aerodynamic force arises from two causes

•The normal force due to the pressure on the surface of the body

•The shear force due to the viscosity of the gas, also known as skin friction

Pressure acts locally, normal to the surface, and shear force acts locally, parallel to the surface. The net aerodynamic force over the body is due to the pressure and shear forces integrated over the total exposed area of the body.

When an aerofoil (or a wing is moving relative to the air it generates an aerodynamic force, in a rearward direction at an angle with the direction of relative motion. This aerodynamic force is commonly resolved into four components

DRAG FORCE: Is the force component parallel to the direction of relative motion,

LIFT FORCE: Is the force component perpendicular to the direction of relative motion.

In addition to these two forces, the body may experience an aerodynamic moments. The value of which depends on the point chosen for calculation.

THRUST FORCE: The force created by the Propeller or a jet engine is called thrust force and it is also an aerodynamic force (since it also acts on the surrounding air

The aerodynamic force on a powered airplane is commonly represented by three vectors thrust, lift and drag.

WEIGHT FORCE: The force acting on an aircraft during flight is its weight force Weight is a body force and is not an aerodynamic force.

DRAG:

The component of force acting on the car parallel to direction of flow is called Drag. The non-dimensional number CD of the drag coefficient has become very important factor for the entire discipline.

Performance, fuel economy, emissions and top speed are important attributes of a vehicle, they represent decisive sales arguments and they all are influenced by drag.

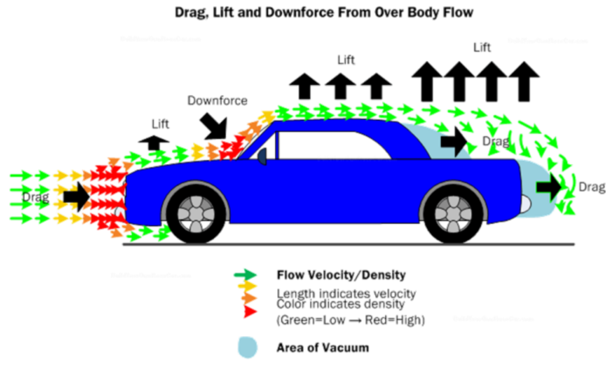


Fig 2. Forces over a body of automobile

The complete expression for drag force is

$$FD=CD\rho V^2/2$$

Where

CD is the non-dimensional Drag coefficient.

A is the largest projected area or frontal area of the vehicle

ρ is density of the ambient air.

V is the velocity of air.

The drag of a vehicle is therefore determined by its size, which is pretty well defined by the frontal area and its shape.

No matter how slowly a car is going, it takes some energy to move the car through the air. This energy is used to overcome a force called Drag.

Drag, in vehicle aerodynamics, is comprised primarily of three forces:

1. Frontal pressure, or the effect created by a vehicle body pushing air out of the way.

2. Rear vacuum, or the effect created by air not being able to fill the hole left by the vehicle body.

3. Boundary layer or the effect created of friction by slow moving air at the surface of the vehicle body.

Between these three forces, we can describe most of the interactions of the airflow with a vehicle body.

International Journal of Research
Available at
https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 10
September 2017

PRESSURE DRAG AND FRICTION DRAG:

When the flow occurs past a flat surface at zero incidence (flat surface held parallel to the flow direction), the fluid exerts a drag force on the surface as consequence of viscous action. The resultant force of friction acts in the downstream direction and is usually known as the friction drag or skin friction drag. But when flow occurs past a vertical surface, drag force results on accounts of difference of pressure over the body , it is also known as the pressure drag. Since the magnitude of this drag force depends upon the form or shape of the body, it is also sometimes called as the form drag.

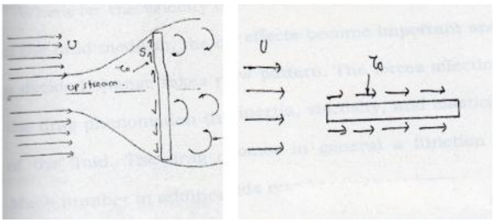


Fig 3. Pressure and friction drag

TOTAL DRAG:

It is usually the total drag summation of pressure and friction drags. It gives the magnitude of resistance to flow.

PROFILE DRAG:

The profile drag which depends only up on a shape or profile and orientation of the airfoil is called the profile drag. The drag which would be developed if the airfoil is infinitely long causing the flow to be truly two dimensional.

INDUCED DRAG:

The induced drag which depends up on airfoil plan form and is induced by the lift force is known as induced drag. The drag produced by end effects due to finite length of the air foil. For objects which exhibit no lift force the induced drag, will obviously be zero and the profile drag will be equal to the drag.

EFFECT OF COMPRESSIBILITY ON DRAG:

Whenever the velocity if the fluid approaches the velocity of sound in the fluid medium, the compressibility effects become important and a decided change takes place in the flow pattern. The forces affecting the drag phenomenon then are due to inertia, viscosity, and elasticity of the fluid. The drag coefficient becomes in general a function of Mach number in addition to the Reynolds number thus we have.

$$CD=f(R_e, M)$$

At very high Reynolds numbers, the viscous forces are little importance as compared to the elastic forces and may thus be ignored, Under these conditions of flow the drag coefficient will be dependent only the Mach.

STREAMLINED BODY:

It the shape of the body is such that streamline separation occurs towards the rear most part of the body, it results in small wake. The drag is then due to existence of small wake gives rise to a small pressure drag. The friction drag thus makes a major contribution to the total drag. Such a body is called streamlined body. The friction drag of such bodies is many times greater than the pressure drag.

BLUFF BODY:

If the body shape is such that the flow is separated much ahead of its rear end resulting in a large wake, the pressure drag is much greater than the friction drag. Body of such a shape in which the pressure drag is large as compared to the friction drag is called a bluff body.

ANALYSIS OF DRAG FORCE

POSSIBLE APPROACHES:

The objectives of analyzing aerodynamic drag are to establish a relationship between CD and V. As already emphasized, this task is made extremely difficult by the interaction of individual flow fields around a vehicle. Drag can be considered from three different points of view. We can

- Examine the physical mechanisms generating drag.
- Allocate fractions of drag to locate origins.
- Investigate the effect of drag on the surrounding flow field.

All three results can yield correct results however; errors are often made when mixing elements at the different categories together.

FACTORS FOR REDUCING DRAG FORCE:

1. ROOF:

Arching the roof in the longitudinal direction can reduce the drag coefficient. However if the curvature is too great CD again can increase. The favorable effect of arching depends on sufficiently large bent radii at the junctions between wind shield, roof and rear window, so that the negative pressure peaks at these locations are not large and the corresponding pressure gradients are reasonably small.

2. UNDER BODY:

The underside of most vehicles still resembles a rough surface. A smooth under body reduces car's drag significantly. Carefully adjusted trim panels could prevent adequate cooling of the brake oil sump and exhaust system.

3. WHEELS AND WHEEL HOSING:

The contribution of wheels to the drag coefficient of car is very high. With streamlined cars it can account for as much as 50% of the total drag. This fact was already discovered by W .Klemperer. The local flow approaches them under yaw.

In the early 1920s the cause of disproportionate contribution of the wheels to overall drag has three reasons.

- Wheels are not stream lined.
- They rotate within wheel housings.

The smaller the volume of wheel housing relative to the wheel's volume, the smaller will be the lift of a wheel.

4. ATTACHEMENTS:

Attachments such as outside mirrors and antennas have high drag coefficients if their drag is related to their individual frontal areas. However their frontal areas small compared to the overall drag is also small but not negligible. Their contribution to wind noise and dirt deposition is significant.

5. FRONT SPOILER:

Spoilers are nothing but projections on the surface of the car. Three positive effects can be achieved with a properly designed form underbody spoiler.

- Reduced drag
- Reduced lift on the frontal axle.
- Increased volumetric flow of cooling air.

6. REAR SPOILER:

A rear spoiler can also have three effects.

- Reduced drag
- Reduced rear axle lift
- Reduced dirt on the rear surface

7. FRONT END:

The front end of the car can be roughly approximated as square block. The streamlines around this block are schematically in the fig. a stagnation point is formed on the vertical front face. Because of the close proximity of the road the air tends to flow over and around the vehicle rather than under it. The streamline near the front end are therefore directed upward it. The flow is significantly deflected at the intersections between the front face and hood and fenders without special measures, this flow pattern will cause separation, with result that the pressure distributions near the edges of the fore body will deviate from those for ideal flow.

DIRECTIONAL STABILITY

HISTORY:

Up to early 1930s, the condition of roads and the technical state-of art of motor vehicles allowed only relatively low driving speeds. Therefore, the interdependence of aerodynamics and directional stability did not have any noteworthy significance to automobiles.

At, the same time, a new situation arose in road traffic; three important changes contributed to making the effects of aerodynamic forces on the driving behavior are noticeable.

1. First, improved roads and highways allowed higher speeds.

2.Encouraged by aerodynamicists who were striving to reduce the drag force, and by designers who were searching for new, more dynamic styling to carriage related "monumental" shapes, flowing lines and softly lined curves became popular.

3. Finally, quite a number of vehicles with rear engines were introduced to the market at that time. This concept reflected the new styling rends towards "teardrop" shapes.

The flow around a vehicle not only leads to drag but also causes other aerodynamic forces and moments, which affect driving stability. At high road speeds, their influence on driving comfort can be felt and, in extreme cases, safety is affected.

The airflow pattern resulting from the forward motion of the vehicle produces a lift and a pitching moment. This results in changed road loads of the wheels and affects the road gripping ability of the types. The alternative play of these forces and moments on the vehicle influences the directional stability in straight ahead as well as its inherent steering behavior during directional changes.

OTHER AERODYNAMIC EFFECTS:

Apart from drag force the shape influences the key aerodynamic effects

1. Pitching moment
2. Yawing moment
3. Rolling moment

PITCHING MOMENT:

Pitching is said to occur when a moment is obtained with reference to transverse axis of the vehicle. The pitching moment occurs due to lift. When the stagnation point on the front edge is high this moment can be observed. A smooth and rounded front edge is high this

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 10
September 2017

moment can be observed. A smooth and rounded front edge could decrease the pitching moment.

YAWING MOMENT:

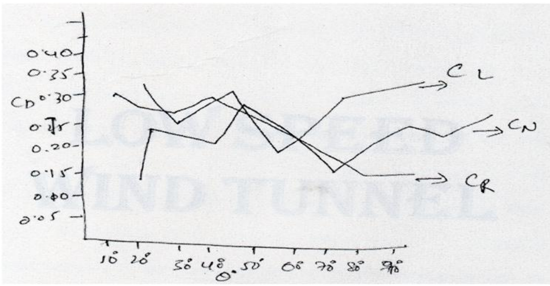
Yawing is said to occur when a moment is obtained with reference to the vertical axis of the vehicle. This moment results in directional instability. Rounded edges could prevent this moment.

ROLLING MOMENT:

Rolling is said to occur when the moment is obtained with reference to the longitudinal axis of the vehicle. This results in tilting of vehicles. Rounding of bottom edges could result in rolling moment. Sharp edges at bottom could reduce the rolling moment.

The trends of these three moments can be observed from the graph below

.



CL … Coefficient of pitching
CN … Coefficient of yawing
CR … Coefficient of rolling

CAR MODEL PREPARATION

TATA INDICA:
TECHNICAL SPECIFICATION:
Dimensions in mm:

- Length : 3675
- Width : 1665
- Height : 1485
- Wheel base : 2400
- Frontal base : 1.16
- Ground area : 170
- Maximum power in bhp :53.5@5500rpm
- Maximum torque in kgm : 85 @ 2500 rpm

PREPARATION OF PROTOTYPE MODEL:

A model of TATA INDICA car is made of wood to the scale of 1:16 and the surface is finished by painting.

Fig 4. Car models

PROCEDURE:

Drag forces are measured for TATA INDICA for this a wooden model has been made to the scale 1:16. This scale is made to suit the test section of the wind tunnel. The maximum size of test section is 300 mm×300 mm.

Clamps are provided to hold the wooden model in the test section. The wind tunnel is of suction type and the car model is placed in the directional opposite to the direction of flow of wind.

In order to measure the pressure distribution over the car, nodal points are made along the transverse section of the car model. Pressure taps are connected to the nodal points and corresponding piezometers.

OBSERVATIONS:

Manometer readings (in cm. of water)

S.NO	M0	M10	M 0	M20	M0
1	65.5	65.3	65.5	63.4	65.5
2	65.5	65.3	65.5	63	65.5
3	65.5	65.5	65.5	65	65.5
4	65.8	66	65.5	67.4	65.5
5	65.7	65.7	65.5	66	65.5
6	65.5	65.5	65.5	65.5	65.5
7	66	66	65.5	66	66
8	66	66.5	65.5	68.5	66
9	65.5	66.5	65.5	70	65.5
10	65.5	66.5	65.5	70	65.5
11	65.5	68.5	65.5	69.5	65.5

1.Observation table

M 0 = Atmospheric pressure reading
M 10 = Pressure reading @ 10 m / sec
M 20 = Pressure reading @ 20 m / sec
M 30 = Pressure reading @ 30 m / sec

. MODEL CALCULATIONS:

When the speed of wind in the wind tunnel is 20 m / sec i.e. 72 kmph Scale model is 1:16

Net area of the pressure diagram A=4.54 cm2

Length of the pressure distribution on the model, L = 22.96 cm

Coefficient of DRAG,

CD = A / L
= 4.54 / 22.96
= 0.197

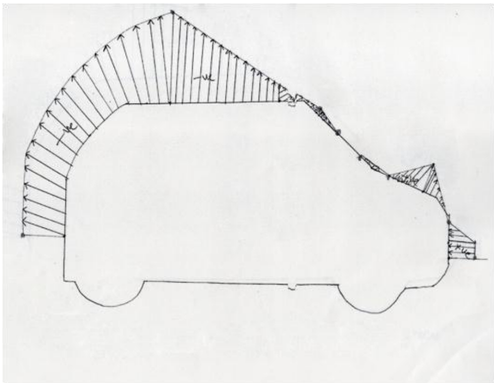
Width = 1665 mm
Height = 1485 mm
Model width = (1 / 16) 1665
= 104.06 mm
= 0.104 m
Model frontal area
= 0.073 m^2
Full scale prototype frontal area = 0.073 × 16
= 1.168 m^2
Modal area = $(1 / 16)^2$ × 1.168 m
= 0.00456 m^2
Drag force on prototype
FD = CD ρ AV^2 / 2
= 0.197 1.25 × 1.16 8 × 20^2 / 2
= 57.52 N

Corresponding power required to overcome aerodynamic drag is

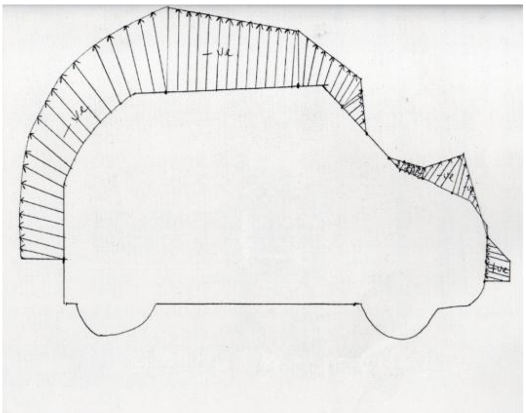
P = FD V
= 57.52× 20 = 1150.48 W
= 1.150 KW

Calculation table

S. NO	Δ P / (ρV2 / 2) = (M_0 – M_{10}) / M_0	(M_0 – M_{20}) / M_0	(M_0 – M_{30}) / M_0
1	+ 30 .0	+ 32.0	+22.9
2	+ 30 .0	+ 38.1	+22.9
3	0	+ 7.6	0
4	- 30 .0	- 29.0	- 30.5
5	0	- 7.6	- 7.6
6	0	0	0
7	0	- 7.6	0
8	- 7.5	- 45.8	- 30.3
9	- 15.2	- 68.7	- 45.8
10	- 45.8	- 68.7	- 68.7
11	- 45.8	- 61.0	- 45.8



- Pressure distribution over a Car Model at a velocity 10 m/s



- Pressure distribution over a car model at a velocity of 20 m/s

RESULTS:

- The experimental was conducted and results are follows.
- FOR TATA INDICA:

- The drag force on the full scale vehicle at 72 kmph is 57.52 N
- The corresponding power required to overcome aerodynamic drag is 1.150 kW
- The Reynolds number above which the coefficient is constant is 3.0613 × 105

RESULT TABLE:

S.NO	velocity (M / sec)	Cd	Fd (N)	Power Lost (Pd) Kw)	Reynols Number (×105)
1	10	0.186	13.60	0.136	1.5306
2	20	0.197	57.52	1.150	3.0613
3	30	0.21	137.9	4.139	4.5920

2.results table

CONCLUSIONS:

•An attempt is made to draw the pressure distribution diagram of scale model 1:16 for different wind velocities and measure the area of the pressure distribution by using graph sheet.

•Computed the coefficient of drag for the model by using area of the pressure distribution of the model.

•Computed drag force exerted by the fluid on model for different wind velocities.

•The major attributes of a vehicle such as performance, fuel economy, top speeds which represents decisive sales arguments are all influenced by drag.

•The results showed that the drag forces effects the power requirement and there by more fuel consumption.

•These effects can by reduced by implementing the suggested changes in the design.

LIMITATIONS:

•Drag force is calculated for the scaled model not for the actual cars

•It is important that Wind tunnel air stream be aligned precisely relative to the test vehicle.

•Maximum size of Wind tunnel test section is limited to 30 m / s

•Maximum size of Wind tunnel test section is limited to 300mm × 300mm

SCOPE FOR FUTURE DEVELOPMENT:

The surface models generated in CATIA can be used for knowing the drag and the lift forces on the car. Various parameters can be altered and the variation of drag and lift forces can be studied.

INTRODUCTION OF CFD:

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing.

Traditionally, aerodynamic forces were measured in wind tunnels using scale models. Testing of this type become expensive and time consuming since one-of –kind prototypes must be manufactured. Today, wind tunnel testing is supplemented by CFD. Widely used in the past in the aerospace industry, CFD has more recently been recognized in motor sports as a valuable tool. CFD, the Virtual wind tunnel, allows engineers to simulate the aerodynamic forces on wings and complete car bodies using computer work stations. This allows engineers to compare different designs without expense of wind tunnel testing.

The prediction of external aerodynamics has been one of the most challenging automotive CFD applications. Complex shapes and physics require sophisticated geometry and meshing tool, physical models, and substantial computer and developed a methodology that produces wind tunnel verifiable results.

CFD analysis of the under hood thermal field has the potential to reduce the expense of the prototype construction and testing. Problems can be identified early and remedied, significantly reducing the design time and cost, and increasing the reliability of the vehicle.

Vehicle under hood regions pose a significant challenge for CFD because of their geometric complexity, however, and mesh generation time is prohibitive with conventional body-fitted hexahedral grids. Furthermore, the simultaneous solution of the external and under hood flows leads to large computational models, so the turnaround time for calculations is also critical.

REFERENCES

1. Aerodynamics of road vehicles wolf - Henrich Hucho
2. Fundamentals of aerodynamics - John D. Anderson. Jr
3. Applied fluid mechanics - D.N.Roy
4. Fluid mechanics - Dr .A.K. Jain
5. Fluid mechanics - Mohanty
6. Fluid mechanics - K.L.Kumar