## DELHI TECHNOLOGICAL UNIVERSITY, DELHI



## DRAG REDUCTION AND COMPUTATIONAL FLUID DYNAMICS

**Submitted By:**

**Abhinav (2K20/CE/06)**

DELHI TECHNOLOGICAL UNIVERSITY (FORMERLY Delhi College of Engineering) Bawana Road, Delhi-110042

**CANDIDATE’S DECLERATION**

## I, (Abhinav, 2K20/CE/06) student of B. Tech. hereby declare that the project Dissertation titled “DRAG REDUCTION AND COMPUTATIONAL FLUID DYNAMICS” which is submitted by us to the Department of Civil Engineering, Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of the degree of Bachelor of Technology, is original and not copied from any source without proper citation. This work has not previously formed the basis for the award of any Degree, Diploma Associateship, Fellowship or other similar title or recognition.

Place: Delhi Abhinav (2K20/A3/68)

Date:

**FLUID MECHANICS**

DELHI TECHNOLOGICAL UNIVERSITY (FORMERLY Delhi College of Engineering) Bawana Road, Delhi-110042

**CERTIFICATE**

## I hereby certify that the project Dissertation titled “DRAG REDUCTION AND COMPUTATIONAL FLUID DYNAMICS” which is submitted by Abhinav(2K20/CE/06), Delhi Technological University, Delhi in complete fulfilment of the requirement for the award of the degree of the Bachelor of Technology, is a record of the project work carried out by the students under my supervision. To the best of my knowledge this work has not been submitted in part or full for any Degree or Diploma to this University or elsewhere.

Place: Delhi Dr. T Vijay Kumar

(Professor’s)

Date:

**ABSTRACT**

Aerodynamics is the most important factor when it comes to resistive forces acting  
on the vehicle. It comes into the picture when a vehicle is moving in a fluid medium.   
There are numerous factors such as lift, side force and drag which are responsible for this resistance. Reducing the aerodynamic drag will not only open the doors for higher top speed but will also reduce the overall fuel consumption of the vehicle and increase comfortability. These above factors are very vital when it comes to passenger cars. These factors also determine the popularity and set the base for marketing strategies for a particular passenger car. Hence, various researchers are constantly trying to optimize car design features due to above mentioned reasons. In the following project you will find various researches which have already been done in order to reduce the drag in various segments of vehicles.

Today aerodynamics of automobile engineering plays a vital in both industry and as well as in research labs. The performance of any automobile vehicle purely depends on its internal and external aerodynamics. To accomplish better performance in the field of automobile engines all the aerodynamics factors must satisfy the design requirements. This review article emphasis more on two broad categories one is heavy good vehicles and second one is lighter vehicles. Both the classification required different aerodynamic shapes and design requirement. Therefore, further this project focuses into more on reduction of drag on vehicle, vehicle aerodynamic instabilities and its possible solutions, design optimization of vehicles which involve finding out the areas that need improvement in the existing designs and looking at some new concept designs. Further to the upcoming researchers this paper also helps in providing the information on recent trends in aerodynamic performance developments of automobile vehicles.

**Therefore, this project focuses on aerodynamics, drag reduction in automobile vehicles and the growing use of computational fluid dynamics in various industries and how it can be a game changer.**

**ACKNOWLEDGEMENT**

In performing our major project, we had to take the help and guideline of some respected persons, who deserve our greatest gratitude. The completion of this assignment gives us much pleasure. We would like to show our gratitude Dr. T Vijay Kumar, mentor for major project. Giving us a good guideline for report throughout numerous consultations. We would also like to extend our deepest gratitude to all those who have directly and indirectly guided us in writing this assignment.

Many people, our classmates and team members itself, have made valuable comment suggestions on this proposal which gave us an inspiration to improve our assignment. We thank all the people for their help directly and indirectly to complete our assignment.

In addition, we would like to thank Department of Civil Engineering, Delhi Technological University for giving us the opportunity to work on this topic.

## INDRODUCTION

* **AERODYNAMICS**

Aerodynamics is the study of the motion of air, particularly when affected by a solid object, such as an airplane wing. It is a sub-field of fluid dynamics. the use of aerodynamics through mathematical analysis, empirical approximations, wind tunnel experimentation, and computer simulations has formed a rational basis for the development of heavier-than-air flight and a number of other technologies. Recent work in aerodynamics has focused on issues related to compressible flow, turbulence, and boundary layers and has become increasingly computational in nature.

* **DRAG**

In fluid dynamics, drag sometimes called air resistance, a type of friction, or fluid resistance, another type of friction or fluid friction is a force acting opposite to the relative motion of any object moving with respect to a surrounding fluid. This can exist between two fluid layers or surfaces or a fluid and a solid surface. Unlike other resistive forces, such as dry friction, which are nearly independent of velocity, drag force depends on velocity. Drag force is proportional to the velocity for low-speed flow and the squared velocity for high speed flow, where the distinction between low and high speed is measured by the Reynolds number. Even though the ultimate cause of a drag is viscous friction, the turbulent drag is independent of viscosity.

* **TYPES OF DRAG**

1. Form drag or pressure drag due to the size and shape of a body.
2. Skin friction drag or viscous drag due to the friction between the fluid and a surface which may be the outside of an object or inside such as the bore of a pipe.
3. Lift-induced drag appears with wings or a lifting body in aviation and with semi-planing or planing hulls for watercraft.
4. Wave drag (aerodynamics) is caused by the presence of shockwaves and first appears at subsonic aircraft speeds when local flow velocities become supersonic.
5. Wave resistance (ship hydrodynamics) or wave drag occurs when a solid object is moving along a fluid boundary and making surface waves.
6. base drag, (aerodynamics) a pressure drag due to flow separation at the base of a projectile or termination of an aircraft fuselage with a flat area.
7. Boat-tail drag on an aircraft is caused by the angle with which the rear fuselage, or engine nacelle, narrows to the engine exhaust diameter.

* **FACTORS ON WHICH DRAG DEPENDS**

1. The pressure distribution acting on a body's surface exerts normal forces on the body. Those forces can be summed and the component of that force that acts downstream represents the drag force, due to pressure distribution acting on the body.
2. The viscosity of the fluid has a major effect on drag. In the absence of viscosity, the pressure forces acting to retard the vehicle are canceled by a pressure force further aft that acts to push the vehicle forward.
3. The friction drag force, which is a tangential force on the aircraft surface, depends substantially on boundary layer configuration and viscosity. The net friction drag, is calculated as the downstream projection of the viscous forces evaluated over the body's surface.

## DRAG REDUCTION

* **WHY THERE IS THE NEED OF DRAG REDUCTION**

There are different types of forces acting on a vehicle when.It is in motion such as drag force and down force. Drag force being the more prominent one is more responsible for increased fuel consumption and lower top speed of a vehicle. There are various types of drag forces acting on a vehicle namely: Parasitic drag, lift, induced drag and wave drag. Parasitic drag is further sub divided into form, skin friction and interference drags. These individual drags are very difficult to calculate and hence most people are concerned in finding the overall drag coefficient of a vehicle. Due to the non-streamline body shape of heavy vehicles, aerodynamic drag is larger compared to smaller vehicles; hence reduction of aerodynamic drag on commercial vehicles is to be achieved to increase fuel efficiency.

Reducing the aerodynamic drag will not only open the doors for higher top speed but will also reduce the overall fuel consumption of the vehicle and increase comfortability. These above factors are very vital when it comes to passenger cars. These factors also determine the popularity and set the base for marketing strategies for a particular passenger car.

Aerodynamic drag force is the force acting on the vehicle body resisting its forward motion. This force is an important force to be considered while designing the external body of the vehicle, since it covers about 65% of the total force acting on the complete body. The Aerodynamic drag force is calculated by the following formula.



**RECENT DEVELOPMENTS IN DRAG REDUCTION**

Tests were conducted with different wind speeds and different yawn angles, the sensor measured all 3 forces (drag, lift and side forces) simultaneously. The aerodynamic drag generated from the under-body flow of a heavy vehicle was studied using wind tunnel.

A model fitted with normal side skirts is referred to as the standard, different types of flaps at different angles and a new type of design of side skirts to smoothen the under-body flow are studied. The skirting with inner panel folded, at an angle of 60 degree, the drag reduction was huge compared to the models without a skirt. To improve the fuel efficiency of the vehicle we studied Adaptive aerodynamics.

By the usage of a rear spoiler the aerodynamic drag on a hatchback model is reduced in order to obtain higher fuel economy.

IT can be seen that the attempts to reduce drag on heavy vehicles have been done using several types of external add-on parts like skirts, front fairing, flaps,and vortex generators and so on. In 26% reduction in drag can be achieved by using external attachments which involved covering up of most of the gaps such as gap between tractor and trailer unit and also by using side skirts to cover up the space between the wheels and by using a front fairing to streamline the flow as much as possible. However this experiment was done on a scaled model so the actual size model may have slightly different values for reduction of drag. The ground clearance will also be reduced. From different types of vortex generators used the vane type of VG did perform better when the size was bigger and placed to the front end of the trailer. In cars it was seen that the attack angle of the spoiler plays an important role in either decreasing or increasing the coefficient of drag. The angle of attack is dependent on the shape of the aerofoil. The combination of different add-ons like front wing, diffusers, and rear wing all help in reduction of drag, and each part can either increase or decrease the drag.

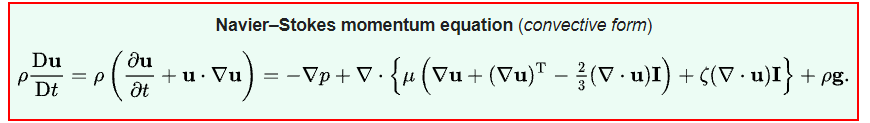
Instability caused due to the aerodynamic effects: The main reason for instability of heavy vehicles are crosswinds acting on the side of the vehicles which may cause the vehicle to roll over, and these crosswind effects will also be dependent on the surrounding elements. The rollover is very high for the vehicles with larger surface areas on the side, and is lesser for the vehicles with rounder and smother surface on the windward side. In cars the lift is the main reason for instability and difficulty in handling. The stability can be obtained by creating enough down-force. The down forces can be created by using rear diffusers, rear spoilers, and front spoiler and so on. The techniques used by race cars can also be implemented on to the passenger cars up to a certain extent.

**COMPUTATIONAL FLUID DYNAMICS(CFD)**

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is typically performed using experimental apparatus such as wind tunnels. In addition, previously performed analytical or empirical analysis of a particular problem can be used for comparison. A final validation is often performed using full-scale testing, such as flight tests.

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, weather simulation, natural science and environmental engineering, industrial system design and analysis, biological engineering, fluid flows and heat transfer, and engine and combustion analysis.

The fundamental basis of almost all CFD problems is the Navier–Stokes equations, which define many single-phase fluid flows. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows not transonic or hypersonic these equations can be linearized to yield the linearized potential equations.



**OUTLINE OF COMPUTATIONAL FLUID DYNAMIC PROCESS**

Computational Fluid Dynamic codes are structured around the numerical algorithms that can tackle fluid flow problems. All the CFD codes available in the market have three basic elements which divide the complete analysis of the numerical experiment to be performed on the specific domain or geometry.

The three basic elements are

(i) Pre-processor

(ii) Solver and

(iii) Post-Processor

**MESHING AND PRE-PROCESSING**

The pre-processing of the CFD process consists of the input of a flow problem by means of user-friendly programs or software and the subsequent transformation of this input into a form is made suitable to use by the solver. The pre-processor is the link between the user and the solver. The user activity at the pre-processing stage of the CFD process involves the following:

1) Definition of Geometry or region of Interest: This process involves several computer aided design (CAD) software like CATIA, Solidworks, Pro-E and much more. By the help of CAD software, the topology of the fluid flow region of interest is defined. This software plays a major part of the design and optimization process in research analysis.

2) Grid Generation or Meshing: Since the CFD process is a numerical approximation method using finite volume method, the given domain or region of interest needs to be divided into several structured elements. All the elements or cells are connected to each other through nodes to form the required region of flow. For this purpose, special meshing or grid generation software like GAMBIT and T-grid are used. This stage is the key element in the CFD finite volume numerical simulation and it also contributes to the accuracy of the final results.

3) Definition of Fluid properties: Every fluid domain or surface has its own distinct property. The properties of the fluid used in the CFD domain or region of interest are defined at this stage of the CFD Process. Usually the CFD code software has this facility.

4) Boundary Conditions: Every different setup of the CFD domain needs to have an initialization, which is fulfilled by the boundary conditions input. The CFD code usually has this facility to define the boundary conditions of the CFD problem, where each cells at specific boundary are given finite values.

**NUMERICAL SOLVER**

The numerical solver is the key elements of the CFD process and covers the major part of the CFD process. In the current market, the solvers usually use three distinct ways of calculating the solutions, namely, the finite difference method, finite element method and the finite volume method.

**POST PROCESSOR**

The post processor is the last phase of the CFD process which involves data visualization and results analysis of the CFD process. This phase uses the versatile data visualization tools of the CFD solver to observe the following results of the simulation:

1. Domain geometry and Grid display

2. Vector plots

3. Line and shaded contour plots

4. 2D and 3D surface plots

5. Particle tracking

6. XY plots and graphs of results

**CFD USES IN DRAG REDUCTION AND DESIGN OPTIMIZATION**

**HEAVY VEHICLES**

For drag reduction in heavy vehicles The aerodynamic drag generated from the under-body flow of a heavy vehicle was studied using wind tunnel and CFD methods. The obtained values were compared with Ahmed body details regarding streamline flow around vehicles can be found in A 15-ton truck and 40 foot trailer were the two models used fortesting. A model fitted with normal side skirts is referred to as the standard, different types of flaps at different angles and a new type of design of side skirt with additional inclined flap panels to smoothen the under body flow are studied. For the models without skirts, both models showed uniform drag coefficient. For the model with the flap inclination, at 45degree there was max drag reduction that is 5.3% and 4.7% for 15-ton truck and 40foot trailer respectively compared to the normal skirting. The skirting with inner panel folded, at an angle of 60 degree, the drag reduction was huge compared to the models without a skirt that is 5.1% and 5.0% respectively.

The CFD tests show that fuel saving can be improved by using side deflectors, controllable radiator shutter and a controllable roof deflector. The new elliptical design of rear flaps performed better than the standard designs that are commercially in use as of now, the elliptical flap resulted in symmetric pressure distribution in the wake which in turn reduces the drag.

**LIGHT VEHICLES**

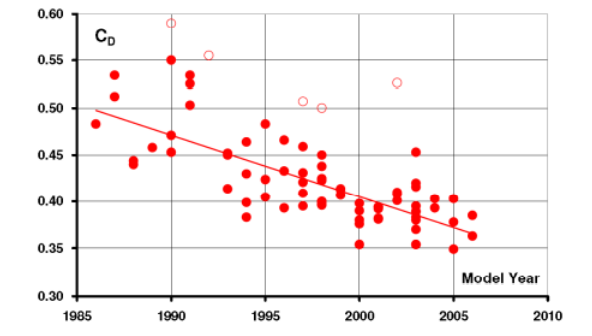
The car is tested with and without a rear spoiler and the results are discussed. The test was conducted in a subsonic suction type wind tunnel. The scaled model of the car is placed in the wind tunnel. It was found that the car with spoiler had lower pressure variation compared to car without spoiler. It was also found that the car with spoiler had lesser drag force. The coefficient of pressure will be very effective for car with spoiler at high speeds.

Using wind tunnel experimentation the aerodynamic characteristics of a race car wing is tested and later the values obtained are compared with the values obtain from CFD simulation.

**DESIGN OPTIMIZATION**

Many experiments have shown data to obtain the areas which require developments in terms of designs. The stagnation zones, high pressure areas, wake area, flow turbulence and streamline flow, and many other reaction to the air was found out. The box fish type of design inspired from box fish is a design capable of producing a lot of reduction in drag compared to the normal design of the vehicle. But using this kind of design might not be aesthetically appealing compared to the design trends for cars as of now. Improvement of the design can be done my optimizing only a certain portion of the vehicle. Proper measurements must be made before rendering a new design as it can perform lesser than the already existing designs.

**DRAG REDUCTION IN SUV USING CFD**



**Twenty year trend of drag coefficient in SUV**

The shape of the vehicle plays an important role in the drag reduction. Low drag is achieved by a shape which avoids sudden changes in the cross sectional area and has a degree of tapering towards the base of the vehicle. In practical design environment, drag reduction comes from attention to detail and it results from the accumulation of small incremental benefits in the development process. The shape changes which can affect the performance of the vehicle body . The arrows show the required direction to morph the surface to create drag reduction, although this is totally dependent on the initial shape. So the direction is susceptible to change as the original shape of the SUV could be aerodynamically friendly or filled with blunt edges. The drag at the SUV base can be reduced by increasing the pressure in the base area and reducing the base area. Tapering the body sides and roof has a significant effect, but this will compromise the loading area at the tailgate and reduce rear passenger headroom. If steps are made to make small chamfers at the rear end of the roof and the side body, there will be a significant change in the drag. The foot step of the vehicle is moved downward to decrease the ground clearance near the wheels and this makes major changes in the drag of the vehicle. Lowering the front bumper and bonnet, inclining the front windshield, rounding off the corners and sharp edges and lastly extending the front bumper are some of the ways contributing to reduce drag. The aerodynamicist usually works closely with the designers to use these ways with high level of compromises to make the vehicle more comfortable for the customers.

