A Review on CFD Analysis of Drag Reduction of a Generic Sedan and Hatchback

Shashi Kant[1],Desh Deepak Srivastava[2],Rishabh Shanker[3],Raj Kunwar Singh[4],Aviral Sachan[5]

[1]M. Tech., Mechanical Engineering, Noida Institute of Engineering & Technology, Greater Noida, Uttar Pradesh, India

[2,3,4,5]B.Tech., Mechanical Engineering, IMS Engineering College, Ghaziabad, Uttar Pradesh, India

---***---

Abstract - *In today's world, Automobile industries are working on increasing the fuel efficiency of vehicle alongwith maintaining the stability at high speed. Advanced tools like Ansys-(Fluent) may be used for analysis and hence reducing the drag force , increasing the stability and fuel efficiency. This review attempts to compare the effect of drag forces on sedan car on applying different types of spoilers, vortex generators and also compare the sedan's car drag force with that of hatchback type car.*

***Key Words*: Fluent, Drag Coefficient, Spoilers, Aerodynamics, Sedan & Hatchback**

1. INTRODUCTION

These days car designers of an automobile industry which designs the car have huge stress to make a car fulfill the aspects such as performance , fuel economical, aerodynamic shape etc. Because of this the manufacturers are looking for new ways and discovering new technologies to reduce fuel consumption and improve vehicle efficiency.[1]Several methods are used to reduce aerodynamic drag for controlling the flow separation at the rear end.These technique, includes modifying the shape of the vehicle or attaching add-on devices to reduce the aerodynamic drag, it is the easiest to implement as it appears but unfortunately it is only dedicated for specific application [2].As price of fuel increased from 2002 the decreasing sales of automobiles severely damaged the industry all over the world.[3]

Drag forces is determined by the airflow over a vehicle, which in turn affects the car's performance and design of efficiency testing equipment has been done to measure both the vertical and horizontal components of air resistance on a model car. Down force, the vertical component is simply negative lift.[4]

The performance and fuel efficiency of aircraft, road vehicles or racing cars are affected by the drag force as it works opposite to the direction of movement. About 30% to 40% of the total fuel energy is lost to overcome road resistance, about 10% to 20% for operating electrical appliances and 50% to 60% to overcome the drag force. So reducing of aerodynamic drag has become the primary concern in vehicle aerodynamics great efforts in research have been employed for better fuel economy and performance of aircraft and road vehicles due to market competition.[5]

Study of flow around solid objects of various shapes is called an external aerodynamics. Examples of external aerodynamics are evaluating the lift and drag on an airplane, the shock waves that form in front of the nose of a rocket, or the flow of air over a wind turbine blade are examples of external aerodynamics. On the other hand, internal aerodynamics is the study of flow through passages in solid objects. Examples of internal aerodynamics comprises the study of the airflow through a jet engine or through an air conditioning pipe and other internal flow conditions.[6]

Most of the finest sedan cars were manufactured in 20th century. From highest speed Hennessey Venom GT reaching up to 270.49 mph, Bugatti Veyron to the luxurious Rolls Royce phantom and much more. Depending on the customer's choice , personal cars ranging from hatch backs, sedans & SUV have seen major changes in their design and ergonomics. Change of aerodynamics in car is gradually seen from initial designers to the manufacturers' to obtain more power under the hood. This in turn will provide more stability; better performance, better grip and most prominently increase the comfort of the car. To have the best output performance people seem to have sportier look. This certainly does mean that the cars are equipped with more additional parts such as front and rear spoilers, diffusers, VGs (vortex generators) ,air dams on the surface of the cars. Rear spoilers are used most widely in passenger cars. This aids in greater drag reduction and in the same occasion increases the stability of the car.[7]

2. ABOUT CFD

Science of predicting fluid flow, heat and mass transfer ,chemical reactions, and related phenomenon is called as Computational Fluid Dynaimcs(CFD) . The equations used ensure the conservation of mass, momentum , energy, etc. CFD is used in the following stages of the design process:

- Redesign
- Conceptual studies of new designs
- Detailed product development
- Troubleshooting

By reducing the total effort and cost required for experimentation and data acquisition CFD Analysis complements testing and experimentation. CFD is being used im following areas and they are:

- Automobile
- Aerospace
- Food Processing
- HVAC
- Marine
- Electronics

3. ADVANTAGES OF CFD

CFD has changed from high level mathematics to an essential tool in almost every branch of fluid mechanics as the availability of high speed computer has become so easy. Now CFD results can give more confident, consistent and more reliable results. Some of the advantages are listed below

a) As compared to laboratory testing evaluation of geometric changes and answer "What if" questions can be solved in much less time and cost.
b) It has become almost mandatory in simulating conditions, where it is not possible to take detailed measurements such as high temperature or dangerous environment like in an oven.
c) It provides a detailed understanding of mass and heat transfer, particulate separation , flow distribution etc. So this provides a much better and deeper understanding of what is happening in a particular process or system to plant managers.[8]

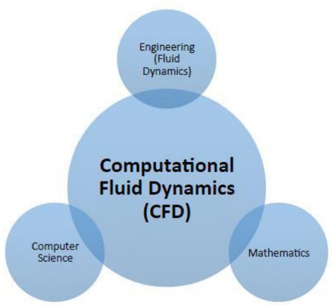


Figure 1: The different disciplines contained within computational fluid dynamics[9]

For the automotive and aerospace industries external aerodynamics simulation using CFD are well established tools in the development process. The technology of CFD simulation helps engineers to understand the physical phenomena taking place around the design and helps the engineers to optimize the performance with respect to certain criteria. In the automotive domain for example the aerodynamic forces i.e. lift, drag have strong impact on the vehicle fuel efficiency and handling behaviour. CFD can be leveraged to analyse this forces and determine an optimum design..[10].Navier–Stokes equations are the fundamental basis for almost all CFD problems.[11]

4. LITERATURE REVIEW

Fluid Mechanics has a branch called "Aerodynamics" which is concerned with the forces generated on a body in a flow and thus the aerodynamics usually involves a lot of calculation in various properties of the flow such as velocity, pressure, temperature, density and even time [12]. The point of difference in between the aerodynamics of passenger car and aerodynamics of race car is that race cars aim to increase the downforce while the passenger cars aim to decrease the drag.[13]

The first car in wind tunnel was tested by the pioneers of aerodynamics in cars named Edmund Rumpler and Viennese .Drag of about 1/3rd of the contemporary vechiles was found by them in car Trophenwagen. In the same period Paul Jaray, an Austo-Hungarian designer well know for his aerodynamic and streamline design of cars. He innovated the smooth surfaces of the headlamps , cambered windshields and body of the car. Big car manufacturing players like BMW, Mercedes, Audi, Diamler-Benz (Dimitris, 2007)copied much of his work.Streamline shape of cars generated a high drag cofficient of around 0.4 so streamline shape was never a hit. Some of the streamline designs still in use are like Porsche 911, Vokswagen Beetle.

Kammback cars became prominent in early 1970's due to crisis for petrol and inorder to increase efficieny. Concept of aerodynamics in cars, which was the use of airfoils was brought by Wunibald Kamm an aerodynaist from Germany. He showed that the air foils with slight truncated tailing edge have slightly lesser drag coefficient compared to completely air foil shaped cars. Drastic change in the automobile shapes from brick designs to rain drop and streamline shapes was seen in post world war 2.

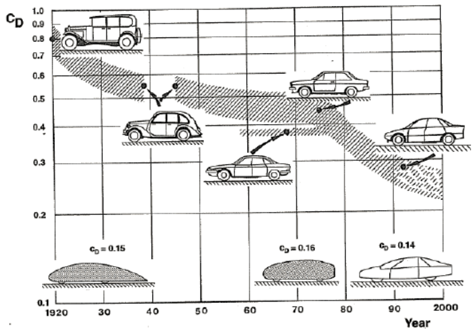


Figure 2 Coefficient of drag value of cars changing over decade

As explained by McNulty and Tong (2006) drag coefficients are used to determine how easily an object moves through the air, since a coefficient is dimensionless and relates back to the input parameters in the equation different shapes of objects can be directly compared. When the small car and a truck are compared it is obvious that the drag force on the truck will be much higher as the truck needs to push more air out of it way due its larger size. The aerodynamic efficiency of both the truck and the small car can be directly evaluated by using the drag coefficient (C_D) as a standard. Other coefficients that are also calculated include the side force coefficient (C_y) and the lift coefficient (C_L).

The acceleration of the car is least effected by the drag apart from at high speeds (Stone & Ball). Reducing the aerodynamic drag on the vehicle will benefit the fuel economy of the vehicle most at higher speeds. Theoretical increase in the fuel economy of approximately 6.7% at 50 Kmph is observed on reducing the Cd of a car from 0.45 to 0.33 as shown in the figure below(Stone & Ball).[14]

EFFECT OF REDUCING AERODYNAMIC DRAG FROM $C_d = 0.45$ TO $C_d = 0.33$ FOR CONSTANT SPEED FUEL CONSUMPTION ($A = 2.25\ m^2$, $\rho = 1.2\ kg/m^3$, $R = 225\ N$)

Speed (km/h)	50	80	120	160
Reduction in fuel consumption	6.7%	15.2%	20.0%	22.5%

Figure 3. Effect of reducing the drag on a car

4.1 Bernoulli's Equation

Daniel Bernoulli's equation defines the physical law upon which most aerodynamic concept exists. This equation is absolutely fundamentals to the study of airflows, and any attempt to improve the flow field around a vehicle is governed by the natural relationship between the fluid (air), speed and pressure. The Bernoulli's equation, which is can be obtained by integrating *F = ma* (Munson, Young, Okiishi, 2006), is derived using the assumptions that (1) the air density does not change with pressure, (2) viscous effects are assumed negligible, (3) the flow is assumed to be steady, (4) the flow is assumed to be compressible and (4) the equation is applicable along a streamline (Munson, Young, Okiishi, 2006). Therefore, the formula can be applied along any point on a streamline where the relation between the local static pressure (p), density (ρ), and the velocity (v) is:

$$p + \frac{1}{2}\rho v^2 + \gamma z = constant\ along\ streamline\ (Munson, 2006)$$

Or

$$\frac{p}{\rho} + \frac{1}{2}v^2 = constant\,(Katz, 1995)$$

if it does not take into account any height term.

From the equation, this indicates that an increase in pressure will cause a decrease in velocity and vice versa.

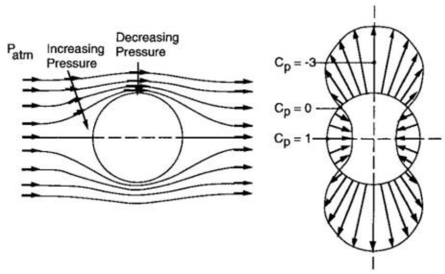


Figure 4: Pressure and velocity gradient in the air flow over body

This moment of the air flow near the body creates a velocity distribution which in turn creates the aerodynamics loads acting on the vehicle. These loads, in general, can be divided into two (2) major contributions. The first force is the pressure, which acts normally (perpendicular) to the surface and contributes to both lift and drag meaning that "the vehicle downforce is really the added effect of the pressure distribution". (Katz, 1995) whereas the second is the shear (skin friction) force, resulting from the viscous boundary layer, which acts tangentially to the surface and contributes to drag. [15]

4.2 Drag & lift Coefficient

It is a dimensionless quantity that describes a vehicles aerodynamic resistance and is a useful tool when comparing different vehicle shapes regardless of size and speed . The drag coefficient can be expressed as in equation given below

.The drag coefficient can be divided into two components a friction and form component [16]

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

Where ρ = air density , U = freestream velocity , F_D = drag force , A = frontal area

When air streams around a body there will be pressure difference between the upper and lower part , if no separation occurs in the flow field the air on the upper surface will travel a longer path to reach the end of a vehicle . This difference in travel length will create a diference in the speed of the fluid , longer way to travel will give a higher speed , and lower pressure . On a vehicle the pressure will be lower on top of the vehicle and higher underneath it , this gives the lifting force (F_L). The lift coefficient is a dimensionless coefficient that describes the lift generate on the body and is expressed as:-

$$C_L = \frac{F_L}{\frac{1\rho U^2 A}{2}}$$

4.3 Spoiler Concept

An automobile rear boot has an aerodynamic device attatched to it called a "spoiler" ,whose intended design function is to 'spoil' unfavourable air movement across a body of a vehicle of some kind in motion. Usually sedan type cars such as NASCAR stock carsuses spoilers. These aerodynamic aids produce down force by creating a "dam" at the rear lip of the trunk. An improved vehicle stability is created by decreasing lift or decreasing drag that may cause unpredictable handling in a car at high speed. Spoilers are often fitted to race and high performance sports car, although they have become common on passenger vehicles as well.

Rear spoilers provide the following advantages.

- At very high speed in offers stability.
- Capability of the tyre to produce the required force is increased.
- Braking performance increases
- Better traction generating fuel efficiency

Figure 5 Different types of spoilers available in market.

In Kelbessa Kenea Deressa research work the spoiler base curve was modelled in sketcher module of CATIA by taking the vehicle roof surface as reference. The dimensions of spoiler base curve were plotted according to the geometry of the vehicle and by taking reference from some journals. These curves were plotted over the roof surface of Bolero vehicle using splines and providing sufficient tension to the curves.[17]. By increasing the pressure in the base area and reducing the base area the drag at the SUV base can be reduced.[18]. In Formula 1 season of 2010 McLaren Engineers developed an ingenious device, called RW80, for the MP4/25 car. Estimation was done that with this device, the system would be able to increase the car's top speed by 6 to 10 Kmph . This device got its nickname "F-Duct".[19]

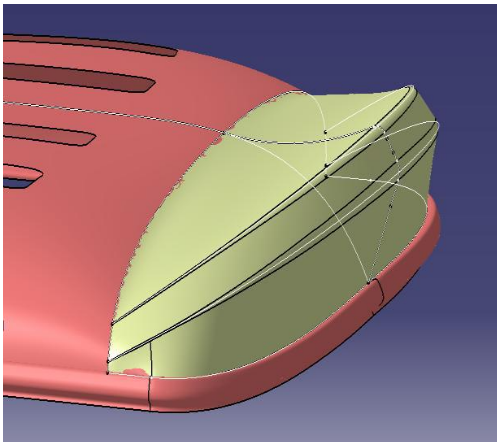


Figure: 6 Surface modelling of spoiler

The lift coefficient can be altered drastically in case of "wings" based only on its orientation with respect to the oncoming flow. Wing orientation is described by an angle called as the *angle of attack* (alpha) and more specifically, it describes the angle between the chord line of a wing and the direction of the oncoming flow. A pictorial representation can be found in fig. 7 below[20] .The type of drag produced in a wake region is determined by the angle of attack and the shape of the body. As an instance, an airfoil is considered as a body with a small angle of attack by the fluid flowing over it. This means that it has attached boundary layers, which produce less pressure drag on airfoil.[21]. The control of the 3D behavior exhibited behind the car due to wake is very difficult to control since it offers unsteadiness and sensititvity to the car geometry.[22]

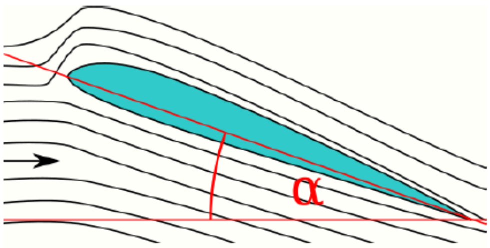
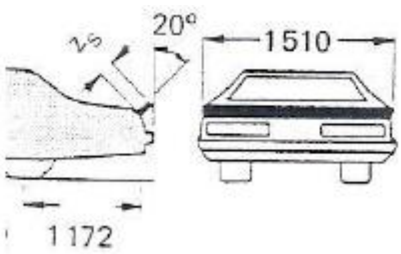


Figure 7. Diagram describing angle of attack.

The aerodynamic fairings(a structure used to reduce drag and improve appearance) have notable impact on aerodynamic drag. 17% of drag can be reduced by external device front fairing alone can. More drag reduction up to 26% may be possible using various combinations of aerodynamic fairings in different parts of the truck body.[16] Pressure drag is the most dominant drag in case of heavy tractor-trailer combinations. The reason being large area of the front portion of the truck available for resistance for the in-coming air. Also large wakes originating from the blunt back end of the trailer. The drag which accounts for the resistance from side and top portion of the trailers is the frictional drag. But contribution of frictional drag in the overall drag is minimum.[24]



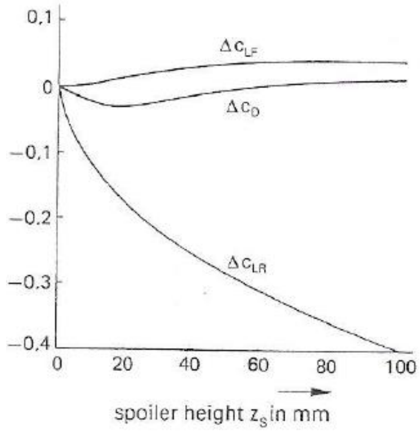


Figure 8 Variatiom of pressure coefficient along (a)Angle of application (b) with spoiler height

5. METHODOLOGY

CFD analysis basically involves three major tasks called Pre-Processing, Solving and Post Processing.

Pre-Processing: We need to do some processes before the numerical solution which is called as pre-processing. It includes defining the problem, creating its 3D model, meshing, and applying physical operating condition called boundary conditions. A fluid volume which encompasses a vehicle is created in order to simulate the air flow around the vehicle. This is done by creating an enclosure around the vehicle and subtracting the vehicle body by using Boolean command in ANSYS. This enclosure acts as the air domain. To reduce the overall computational cost and time, the vehicle was considered symmetric laterally.[25]

V.Vasudevan in his work created the models of vehicle and two different spoilers which was further 3D printed using the software called SolidWorks to CAD format for numerical analysis. Then after, this model has been analysed for drag coefficient and forces under the AUTODESK Flow design module and values of drag coefficient, lift coefficient. [26]

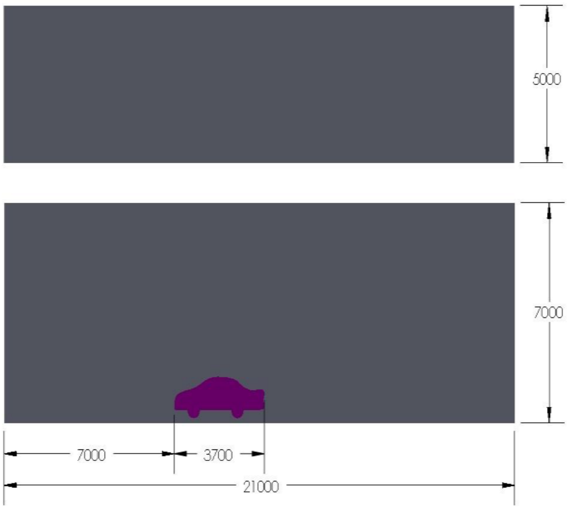


Figure 9. Enclosure Dimensions

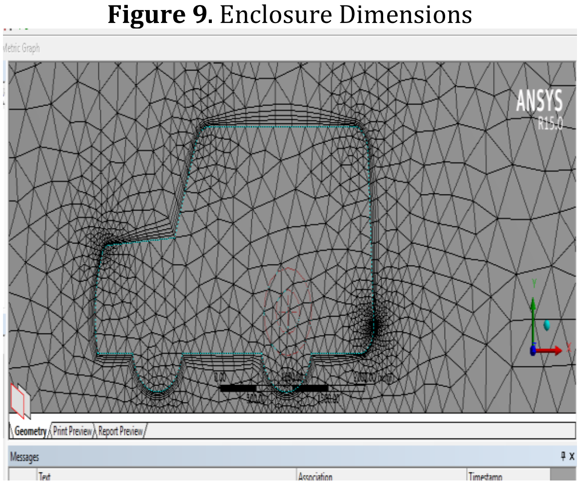


Figure 10. Inflation Meshing

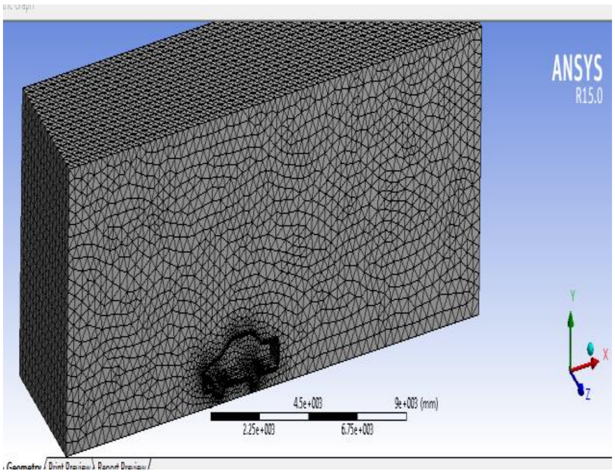


Figure 11. Meshing

Satyan Chandra, Allison Lee, Steven Gorrell completed meshing in three steps: capsuling the car using the Star CCM+ Surface Wrapper tool; creating a surface mesh; and creating a volume mesh that included boundary layers.[27]

R. B. Sharma, Ram Bansal created tetrahedron mesh on cars surface and a surface mesh of 1.5mm size is created on the vehicle surface.[28]

Processing: Processing involves solving mathematical equations of fluid flow until inacceptable convergence is achieved. Usually it requires the computer to solve many thousands of equations and might take few hrs. to few days.

Post-processing: 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. Vector plots
2. Line and shaded contour plots
3. 2D and 3D surface plots
4. XY plots and graphs of results
5. Domain geometry and Grid display
6. Particle tracking

5.1 Boundary Conditions

Adarsh P with his group used the RANS k-ϵ turbulence model for the analysis. The air was assumed to be incompressible and steady ,as the Mach number under consideration was less than 0.3. Standard wall functions are assumed during this analysis.[29]

5.2 Velocity Inlet Boundary Conditions

It is used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets. The static properties of the flow are not constant, so they will rise to whatever value is necessary to get the required velocity distribution. Sharath Kumar S N in his research solved the problem using ANSYS Fluent for both sedan and hatchback car models at 4 different velocities 10m/s, 15 m/s, 20m/s, 25 m/s . At first he went for 7000 iterations at a particular velocity. Since there were no appreciable changes in the residue values above 2000 iterations, all the other flow problems were made to run for 2000 iterations, thereby reducing computational time. Fig. 12 shows the convergence plot for Sedan car model at 20 m/s. Convergence absolute criteria for scaled residues was set to 10^{-6}. As the scaled residues for all the parameters are less than 10^{-6} it can be said that solution is converged.[30]

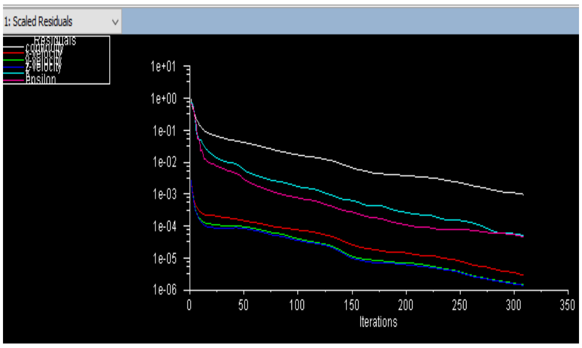


Figure. 12: Convergence Plot for Sedan Car Model

Specification of a static (gauge) pressure is required for the pressure outlet boundary condition. The value of static pressure specified is used only while the flow is subsonic.The specified pressure is no longer used if the flow becomes locally supersonic. The default values of pressure outlet boundary conditions in FLUENT are given in Table 1. Also on the two walls of the domain under consideration no slip boundary conditions are specified and the body is considered to be stationary.

PROPERTIES	VALUES
Gauge Pressure	0
Back Flow Direction Specification	Normal to Boundary
Turbulence Specification Method	k-ϵ method
Back Flow Turbulent Kinetic Energy	1
Back Flow Turbulent Dissipation Rate	1

Table 1. Outlet Boundary Conditions

Figure 13 and 15 shows the velocity contours for the sedan car and jeep without spoilers Maximum Coefficient of drag Value obtained for sedan car was0.5225 and for jeep Coefficient of drag value was about 0.72.

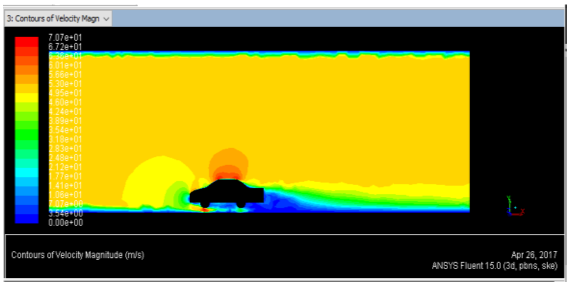


Figure 13. Velocity contours for sedan car without spoilers

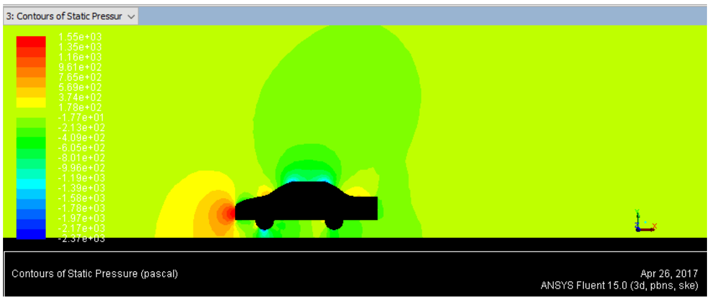


Figure 14. Pressure contours for sedan car without spoilers

In fig 14 two positive pressure areas can be found: one is at the front of the vehicle body, and the other is between the hood and the windshield. At the same time, high negative pressure areas can be found at the front and rear end of the roof and a small area at the front end of the hood; as the pressure coefficient increases to more than definite value the flow accelerates over the bonnet and due to this acceleration the pressure drops and becomes negative.[31]

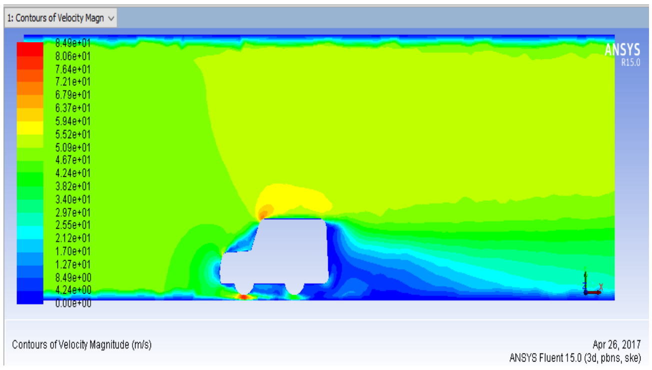


Figure 15. Velocity contours for jeep without spoiler

Naveen Kumar, K. Lalit Narayan did the analysis on the sedan cars by applying 2 different types of spoilers and compared the results with the car without the spoilers at the inlet velocity of 70km/hr whose results are as follows:

S. No.	Name	Velocity(Km/hr)	Drag Force (N)	Lift Force (N)	C_D	C_L
1	Car Without Spoiler	70	153.86	51.109	0.329	0.1183
2	Car with Spoiler 1	70	147.431	49.646	0.3326	0.112
3	Car With Spoiler 2	70	144.062	49.867	0.325	0.1125

Table 2. Comparison of Data

Advantages of using spoiler2:

- Increases tires capability to produce cornering force
- Stabilizes vehicles at high speed
- Improves braking performance
- Gives better traction

6. LIMITATION OF USING SPOILER

1) Spoilers tend to be quite expensive.
2) It is difficult to install and it can involve some permanent changes in car .
3) Spoilers are usually made of fiberglass, ABS plastic, silicon, or carbon fiber which can't take beating so they suffer the lack of durability.
4) Locating the exact lip spoiler for specific car is rarely an easy task. Even if one is lucky enough to find a lip spoiler that they may think is going to fit, there is always a chance that something might go wrong.

7.SCOPE OF FUTURE WORK

This work can performed on different car models, building shapes, aero plane models etc, with different helical angles 30^0, 60^0, 90^0, 180^0, 360^0. This can be performed on the building shapes with square, taper square, helical, pentagon, hexagon etc. with different velocities like 5m/s, 10m/s, 12m/s, 30m/s. And at different angles like 12^0, 25^0, 30^0, 45^0, 60^0 etc, to the flow direction And Also by locating pressure ports at different heights on model surfaces.

Although this review focussed on the Drag force reduction using spoilers but their can be more things on which one can work upon. The drag force reduction using other add on devices like front spoilers , diffuser, vortex generators etc can be also done . Jaspinder Singh1 in his study told that the with an optimized body shape and without any additional devices an aerodynamic drag can be reduced .[32]

8. CONCLUSION

This review mainly focussed on the comparison of drag forces in sedan and jeep and was concluded that jeep has much larger drag value than sedan .Different types of car in sedan class use rear trunk spoilers so review also focusses on the application of different types of spoilers and comparing the drag forces on application of different spoilers. By talking about the values of drag and velocities of models like opel astra, Honda city, Hyundai Santro, Hyundai Accent, G Radha Krishna concluded that Honda city model is with good aero dynamical shape and has the least aerodynamic drag.[33] The coefficient of drag exhibits negligible variation with respect to velocity, while it has a considerable variation with the change in the shape of the body.

REFERENCES

[1] Shashank Arya, Pankaj Goud, Shubham Mathur, Vishal Sharma, Sourabh Tripathi, Vishal Shukla, International Research Journal of Engineering and Technology (IRJET) Volume: 03 Issue: 11 Nov -2016

[2] Mohd Nizam Sudin, Mohd Azman Abdullah, Shamsul Anuar Shamsuddin, Faiz Redza Ramli, Musthafah Mohd Tahir," Review of Research on Vehicle Aerodynamic Drag Reduction Methods", International Jouranal of Mechanical & Mechatronics Engineering IJMME-IJENS VOL:14 NO:02.

[3] Ram Bansal and RB Sharma, Journal of Aerodynamics Volume 2014 (2014).

[4] Bhagirathsinh H Zala, Harilal S Sorathia, Dinesh L. Suthar, Vashant K. Pipalia International Journal of Advanced Engineering Technology Vol. VI/Issue I/Jan.-March,2015

[5] P Ramya,A Hemanth Kumar, Jaswanth Moturi , N.Ramanaiah ,International Journal of Engineering Trends and Technology (IJETT) – Volume 29 Number 4 - November 2015

[6] P.N.Selvaraju, Dr.K.M.Parammasivam, Shankar, Dr.G.Devaradjane ,Journal of Chemical and Pharmaceutical Sciences ,Issue 7: 2015

[7] Dibyajyoti Laha, Computational aerodynamic analysis of a rear spoiler on a car in two dimensions, Anglia Ruskin University

[8] N. Bhagat ,Shashi Kant and Amit Tiwari International Journal of Mechanical Engineering and Technology (IJMET) Volume 7, Issue 2, March-April 2016

[9] V. Naveen Kumar, K. Lalit Narayan, L. N. V. Narasimha Rao and Y. Sri Ram, International Journal of Advances in Scientific Research VOL 01 ISSUE 08,2015

[10] Gavin Dias, Nisha R. Tiwari, Joju John Varghese, Graham Koyeerath IOSR Journal of Mechanical and Civil Engineering Volume 13, Issue 3 2016

[11] Ms.SujataShenkar, Mr. Sanjay D. Nikhade Mr.Sagarkumar Banerjee, Mr.AmarjeetKunal, Mr. Santosh Mote, Mr. Mustufa Sadikot, International Journal of Engineering Science Invention Volume 5 Issue 11 November 2016

[12] Raguvaran A/L Jayahkudy, An Aerodynamic study on MPV spoiler, University Teknilal, Malaysia Melaka

[13] Nathan Pandian, , A review of engineering in everyday life, Volume XVII Issue III, April 22, 2017

[14] Johan Basson, Research Project, University of Southern Queensland Faculty of Health, Engineering & Sciences, October 2013

[15] Ridhwan Bin Che Zake, Research Project, University Malaysia Pahang, November 2008

[16] Johan Levin , Rikard Rigdal, Division Of Vehicle Engineering and Autonomous Systems, Chalmers University Of Technology, Goteborg, Sweden 2011

[17] Kelbessa Kenea Deressa, Kiran Kumar Sureddy, International Research Journal of Engineering and Technology (IRJET) Volume: 03 Issue: 06 June-2016

[18] Pramod Nari Krishnani, Thesis, B.S., Mumbai University, Mumbai, India 2006

[19] Vishal Vinod Raul, Research Work, Wichita State university , December 2013

[20] T. Ragavan , S. Palanikumar, D. Anastraj, R. Arulalagan, Journal of Basic and Applied Engineering Research Volume 1, Number 4; October, 2014

[21] Datta Kisan Wakshe, G.N. Thokal International Journal of Advanced Research in Education & Technology (IJARET) Vol. 3, Issue 1 (Jan. - Mar. 2016)

[22] Laurent Dumas Universit´e Pierre et Marie Curie Laboratoire Jacques-Louis Lions4, place Jussieu, 75230 Paris Cedex 05

[23] Sanat M. Gameti,Prof. H. P. Patel,Musa Jacob Khatri,Prof. P. D. Patel, IJSRD - International Journal for Scientific Research & Development| Vol. 3, Issue 12, 2016

[24] Rishabh S. Singh ,International Journal of Latest Trends in Engineering and Technology (IJLTET) Vol. 6 Issue 2 November 2015

[25] Akshay Parab Ammar Sakarwala Bhushan Paste Vaibhav Patil Amol Mangrulkar International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE),Volume: 1 Issue: 4

[26] G Ganesh, V Vasudevan, International Journal of Scientific & Engineering Research, Volume 6, Issue 6, June-2015

[27] Satyan Chandra, Allison Lee, Steven Gorrell and C. Greg Jensen Brigham Young University, Computer-Aided Design & Applications, PACE (1), 2011

[28] R. B. Sharma, Ram Bansal, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) Volume 7, Issue 5 (Jul. - Aug. 2013)

[29] ADARSH P, ANDREW GEORGE CHERIAN, ARUN K S, D ROSHAN KUMAR International Journal of Mechanical And Production Engineering, Volume- 2, Issue- 4, April-2014

[30] Sharath Kumar S N, Dr. C. K. Umesh IJRMET Vol. 6, Issue 1, Nov 2015-Apri l 2016

[31] Xu-xia Hu, Eric T.T. Wong International Scholarly and Scientific Research & Innovation 5(9) 2011

[32] Jaspinder Singh, Jagjit Singh Randhawa International Journal of Science and Research (IJSR) Volume 3 Issue 6, June 2014

[33] G. Radha Krishna, K.Vishnu Vardhan, K.Ravi Teja, B.Sai Krishna, I.Praveen International Journal of Engineering Science and Computing, May 2016

BIOGRAPHIES

Shashi Kant received his graduate degree B.Tech (Mechanical Engineering) from UCER,Greater Noida (UPTU Lucknow) in the Year 2014 and pursuing post graduate degree M.Tech (Computer Aided Design) from Noida Institute of Engineering and Technology, Greater Noida (Dr. A.P.J. Abdul Kalam Technical University, Lucknow). He is certified Solidworks Professional and Associate by Dassault Systemes.

He is AutoCAD certified professional by AutoDesk. He has 2.5 years experience of service in LeLogix Design Solutions Pvt. Ltd. Greater Noida as Senior Design Engineer. He has taught a number of designing softwares of Mechanical and Civil Engineering at LeLogix CAD Training Centre, Greater Noida. He has published four research papers in International Journals with good impact factor. He has very good experience on FEA Software (Ansys).

Desh Deepak Srivastava is currently pursuing his B.Tech (Mechanical Engineering) from IMS Engineering College, Ghaziabad batch 2013-2017. He has a good knowledge of the several designing software of Mechanical Engineering. He is also a member of SAE India collegiate club . He has experience of internship in some reputed companies.

Rishabh Shanker is currently pursuing his Btech in Mechanical Engineering from IMS Engineering College , Ghaziabad batch 2013-2017 . He is a member of SAE India Collegiate club and he is well equipped with several designing softwares related to Mechanical Engineering.