Design And CFD Analysis of Car With And Without Spoiler

Nethala G Priyanka[1], G Swetha[2]
[1]Dept of Mechanical Engineering
[2]Assistant Professor, Dept of Mechanical Engineering
[1, 2] Methodist College Of Engineering And Technology
King Koti Road, Abids, Hyderabad, Telangana, 500001

Abstract- *Aerodynamic characteristics of a racing car are of significant interest in reducing car-racing accidents due to wind loading and in reducing the fuel consumption. Sports cars are most commonly seen with spoilers, such as Ford Mustang, Subaru Impreza, and Chevrolet Corvette. Even though these vehicles typically have a more rigid chassis and a stiffer suspension to aid in high-speed manoeuvrability, a spoiler can still be beneficial.*

One of the design goals of a spoiler is to reduce drag and increase fuel efficiency. Many vehicles have a fairly steep downward angle going from the rear edge of the roof down to the trunk or tail of the car. Air flowing across the roof tumbles over this edge at higher speeds, causing flow separation. The flow of air becomes turbulent and a low pressure zone is created, thus increases drag. Adding a spoiler at the very rear of the vehicle makes the air slice longer, gentler slope from the roof to the spoiler, which helps to reduce the flow separation. Reducing flow separation decreases drag, which increases fuel economy; it also helps keep the rear window clear because the air flows smoothly through the rear window.

The limitations of conventional wind tunnel experiment and rapid developments in computer hardware, considerable efforts have been invested in the last decade to study vehicle aerodynamics computationally. This project will present a numerical simulation of flow around racing car with spoiler positioned at the rear end using commercial fluid dynamic software ANSYS FLUENT. The thesis will focus on CFD-based lift and drag prediction on the car body after the spoiler is mounted at the rear edge of the vehicle. A 3D computer model of 4-door Passenger car (which will be designed with commercial software Solid Works) will be used as the base model. Spoiler will be positioned at the rear end vehicle and the simulation will be run in order to determine the aerodynamic effects of spoiler.

I. INTRODUCTION

1.1 WHAT IS AERODYNAMICS?

Aerodynamics is the way objects move through air. The rules of aerodynamics explain how an airplane is able to fly. Anything that moves through air is affected by aerodynamics, from a rocket blasting off, to a kite flying. In Airplanes the aerodynamic characteristics plays a important role to raise off the ground by creating lift from its wings used in reverse to apply force that presses the race car against the surface of the track. This effect is referred as aerodynamic grip Since they are surrounded by air, even cars are affected by aerodynamics. “Aerodynamics” is a branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with a moving object. Aerodynamics is also a subfield gas dynamics, with much theory shared with fluid dynamics. Aerodynamics has often used synonymously with gas dynamics, with the difference being that gas dynamics applies to compressible flows. Understanding the motion of air (often called a flow field) around an object enables the calculation of forces and moments acting on the object. Typical properties calculated for a flow field include velocity, pressure, density and temperature as a function of position and time. By defining a control volume around the flow field, equations for the conservation of mass, momentum, and energy can be defined and used to solve for the properties. The use of aerodynamics through mathematical analysis, empirical approximation and wind tunnel experimentation form the scientific basis.

Aerodynamics can be divided into two sub-categories as external and internal aerodynamics. External aerodynamics is basically the study of flow around solid objects of various shapes. Evaluating the lift and drag on an airplane, the flow of air over a wind turbine blade or the shock waves that form in front of the nose of a rocket are examples of external aerodynamics. Internal aerodynamics on the other hand is the study of flow through passages in solid objects. For instance, internal aerodynamics encompasses the study of the airflow through a jet engine or through an air conditioning pipe. This thesis concentrates more on the external category of the aerodynamics related to vehicle with the domain geometry and grid display,vector plots, line and shaded contour plots, 2D and 3D surface plots, particle tracking and lastly XY plots and graphs of results

1.2 SCOPE OF AERODYNAMICS

The regulation of greenhouse gases to control global warming and rapidly increasing fuel prices have given tremendous pressure on the design engineers to enhance the current designs of the automobile using minimal changes in the shapes. To fulfil the above requirements, design engineers have been using the concepts of aerodynamics to enhance the efficiency of automobiles.

Although aerodynamics depends on so many factors, this thesis concentrates on external devices, which affect the flow around the automobile body to reduce the resistance of the vehicle in normal working conditions.

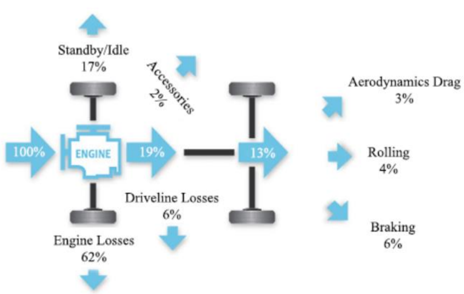


Figure 1.1 **Fuel energy usages at urban driving**

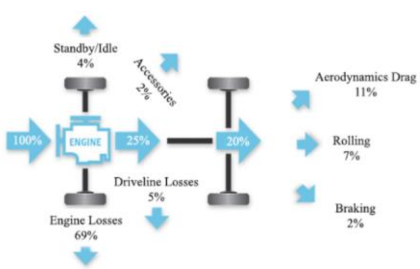


Figure 1.2 **Fuel energy usages at highway driving**

Figure 1.1 and ***Figure 1.2*** show the description of the fuel energy used in a modern vehicle at urban driving and highway driving respectively. The shape of the vehicle uses about 3% of fuel to overcome the resistance in urban driving, while it takes 11% of fuel for the highway driving. This considerable high value of fuel usage in highway driving attracts several design engineers to enhance the aerodynamics of the vehicle using minimal design changes. This brings the idea of using external devices, which could be attached to the present vehicle without changing the body. This project based on the design, developments and numeral calculation of the effects of external device, which will be spoiler that mounted at the rear side of the vehicle to make the present vehicles more aerodynamically attractive.

1.3 EXTERNAL FLOW PHENOMENA OF AUTOMOBILE

Figure 1.3 shows the streamline of an external flow around a stationary vehicle. When the vehicle is moving at a certain velocity, the viscous effects in the fluid are restricted to a thin layer called boundary layer. Outside the boundary layer is the in viscid flow. This fluid flow imposes pressure force on the boundary layer. When the air reaches the rear part of the vehicle, the fluid gets detached. Within the boundary layer, the movement of the fluid is totally governed by the viscous effects of the fluid.

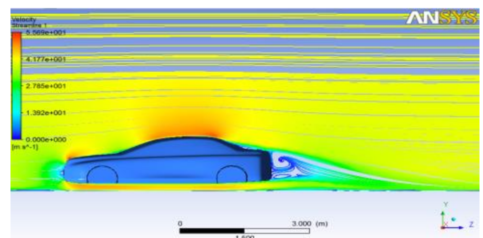


Figure 1.3 **Streamline of external flows around a stationary vehicle**

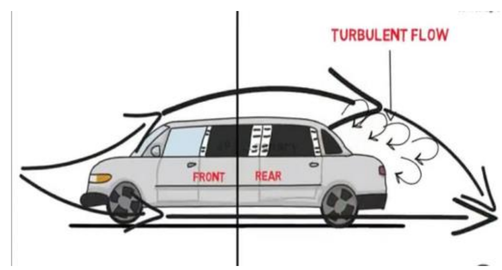


Figure 1.4 Difference in stream line flow i,e.laminar flow at front and turbulent flow at back

1.3.1 FLOW SEPARATION

During the flow over the surface of the vehicle, there are some points when the change in velocity comes to stall and the fluid starts flowing in reverse direction. This phenomenon is called 'Separation' of the fluid flow. This usually occurs at the rear part of the vehicle. This separation is mostly dependent on the pressure distribution, which is imposed by the outer layer of the flow. This separation causes the flow to change its behaviour behind the vehicle and thereby affects the flow field around the vehicle. This phenomenon is the major factor to be considered while studying the wake of the vehicle. Flow separation is bad because it leads to a larger wake and less pressure on the rear surface which reducing pressure recovery. To avoid bad flow separation, the transitions of the airflows from roof to the rear window need to be smoothed. The bad separation also can create more drag. The aerodynamic will be more effective if the flows working in clean air (laminar flow). By improving the aerodynamic of the car can reduce the boundary layer thickness thus avoids worst flow separations.

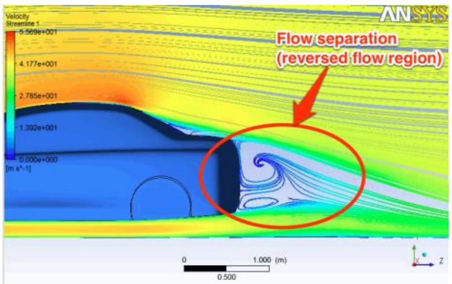


Figure 1.4 **Flow Separation at the rear of vehicle**

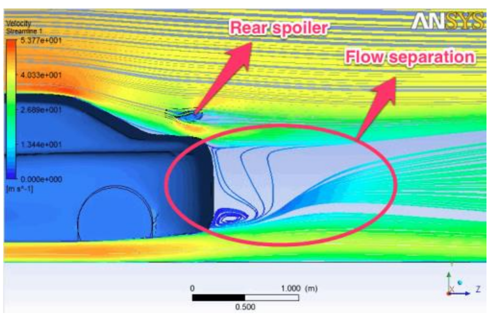


Figure 1.5 **Flow Separation at the rear of vehicle with rear spoiler**

1.4.3 FRICTION DRAG

Every material or wall has a distinct friction, which resists the flow of fluids. Due to molecular friction, a stress acts on every surface of the vehicle. The integration of the corresponding force component in the free stream direction leads to a friction drag. If the separation does not occur, then friction drag is one of the main reasons to cause overall drag.

1.4.4 PRESSURE DRAG

Behind the vehicles, there is a steep pressure gradient, which leads to the separation of the flow separation in viscous flow. The front part of the flow field shows high-pressure value, whereas on the rear part flow separates leading to a high suction in the area. As we integrate the force component created by such high change in pressure, the resultant is called as 'Pressure Drag'. This factor is affected by the height of the vehicle as well as the separation of the flow field.

1.4 FORCES AND MOMENT ON VEHICLE

When the vehicle is moving at a considerable speed, there are several forces are applied to vehicle in different directions. ***Figure 1.5*** shows the details sketch view of the various forces acting on the vehicle body. As shown in the free body diagram below, there are six forces acting on the vehicle:

- Rolling Resistance
- Drag
- Lift
- Gravity
- Normal
- Motor

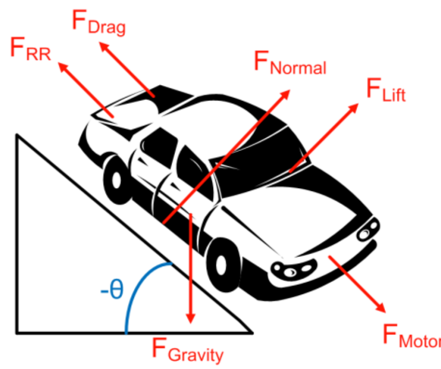


Figure 1.6 **Forces on Vehicle Body**

Rolling resistance force is due the tires deforming when contacting the surface of a road and varies depending on the surface being driven on. The normal force is the force exerted by the road on the vehicle's tires. Because the vehicle is not moving up or down (relative to the road), the magnitude of the normal forces equals the magnitude of the force due to gravity in the direction normal to the road.

Lift force acting on the vehicle body vertically. This force causes the vehicle to get lifted in air as applied in the positive direction, whereas it can result in excessive wheel down force if it is applied in negative direction. Engineers try to keep this value to a required limit to avoid excess down force or lift. The formula usually used to define this force is written as:

$$C_L = \frac{L}{{}^1\!/_2\,\rho V^2 A}$$

Where;

L	: Lift force
C_L	: Lift coefficient
A	: Frontal area of the vehicle
ρ	: Air density
V	: Vehicle velocity

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.

II. LITERATURE REVIEW

P.N.SELVARAJU, Dr.K.M.PARAMMASIVAM

ETAL: - In this mainly concentrating on without changing the outer profile of the car body by the adoption of vortex generator as add on device to reduce the drag and lift force of the sedan type car body. The approximate outer profile of the sedan type car body (Hyundai Elantra) is generated by using 3D modeling software. Then it is imported for meshing and analyzed for aerodynamic forces with and without vortex generator. Therefore without modifying the car profile the reduction of 6% of drag and negative lift force can be attained by adopting vortex generator as an aerodynamic add on device.

ShobitSengarEt al.(2014) determined forces acting on three different segments of vehicle, the Hindustan Ambassador, BMW and an F1 car, by testing their models in wind tunnel. A comparison is done between 3models for best aerodynamic features. The scaled models are tested under different wind conditions. It was found that F1 car is more aerodynamic amongst the 3 followed by the ambassador and

BMW. The former 2 has this result due to his low slung body which results in lower ground clearance. Also the lining of the coupe help in channelizing the air when the vehicle is in motion which leads the air to rear end were spoilers are provided which provides additional stability at high speeds.

.

Ahmed.H.AndChako, 2012 conducted optimization exercise for aerodynamic shapes with computational methods.They found that additional spoiler increase drag by 5% and reduces lift by 7%

III. GENERIC MODELS

3.1 VEHICLE GENERIC MODELS AND DIMENSIONS

The Generic Model of the BMW car is shown in Figure 3.1 and Figure 3.2 below with relevant dimensions. The length of the model is2037mm, the width of the model is 129mm, and the height of the model is 1469mm.

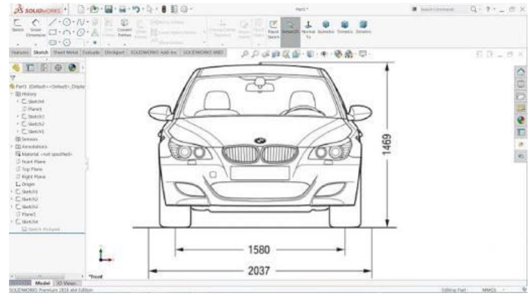


Figure 3.0**Front View of Basic Car Model**

3.2 SPOILER GENERIC MODELS AND DIMENSIONS

In the numerical analyse, a spoiler has been used. The spoiler was a “dual rear wing” style spoiler, which was mounted with different angles i,e. (0°,15°,30°,-15°,-30°)above the surface of the vehicles rear-end. The generic model of the spoiler is shown in below figure with relevant dimensions.

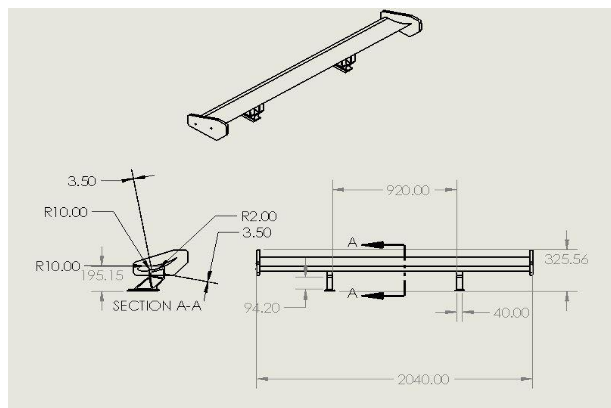


Fig:3.1 Line diagram of spoiler with dimension.

IV. NUMERICAL SIMULATION

Numerical simulations have been performed on the vehicle (with/without spoiler) 3D CAD models using the CFD techniques. The software used for the numerical analysis was ANSYS FLUENT.

4.1 CAD MODEL

The passenger vehicle BMW car and spoilers have been 3D printed using the software called SolidWorks to CAD format for numerical analysis.

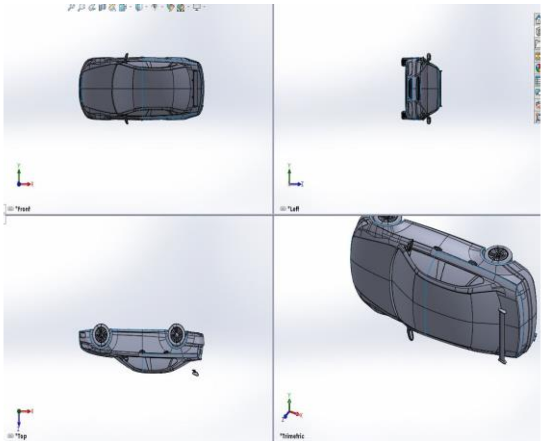


Figure:4.1 Vehicle 3D Basic Car Model in all views

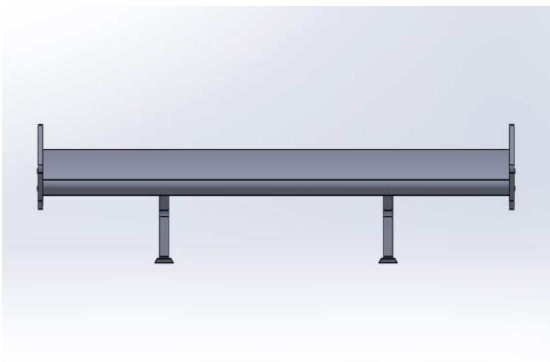


Figure:4.2 Vehicle 3D spoiler model

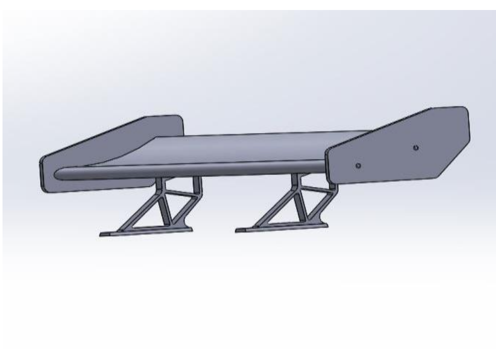


Figure :4.3 spoiler generation

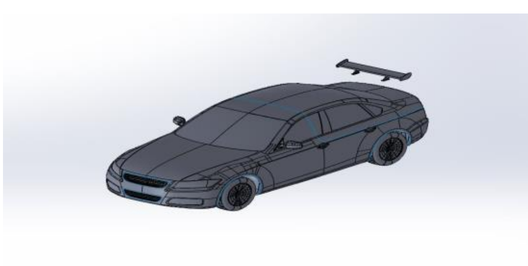


Figure:4.4 Assembly of spoiler on rear end of vehicle

Spoilers is mounted in a rear portion of a vehicle with using “assembly” functionality of Solid Works. These assembled models are now ready to use for numeral analysis.

V. RESULTS

5.1 Car Without spoiler and With spoiler at different angles in ANSYS FLUENT

Pressure profiles for car at 150km/hr in all angels

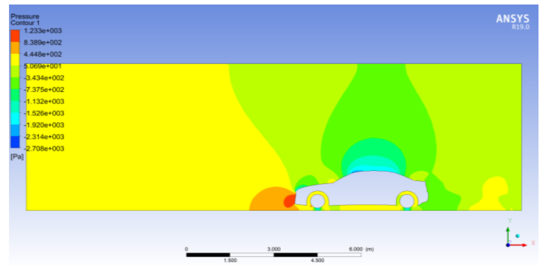


Figure:5.1 Car without spoiler at 150km/hr

Figure :5.2 Car with spoiler at angle 0 degree

Figure :5.3 Car with spoiler at angle 15 degrees

Figure:5.4 Car with spoiler at angle 30 degrees

Figure: 5.5 Car with spoiler at angle -15 degrees

Figure: 5.6 Car with spoiler at angle -30 degrees

The Different Pressure Profiles of BMW car are illustrated in above figure. As the car design becomes more aerodynamic, its adaptability to pressure is greatly increased

5.2 Comparison of velocity in all cases at 150km/hr

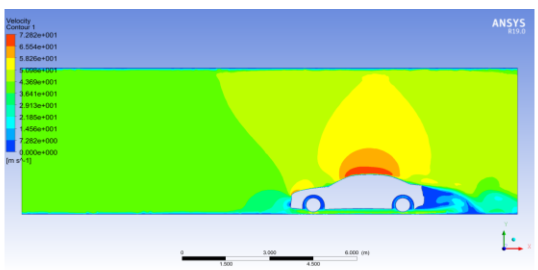


Figure:5.7 Car without spoiler

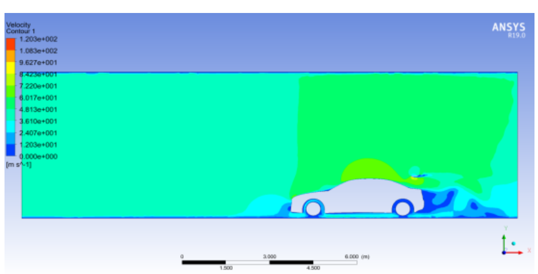


Figure: 5.8 Car with spoiler at angle 0 degree

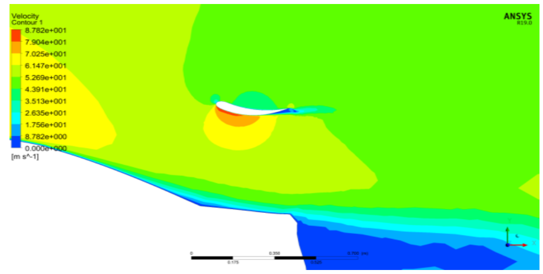


Figure:5.9.At 15 degrees

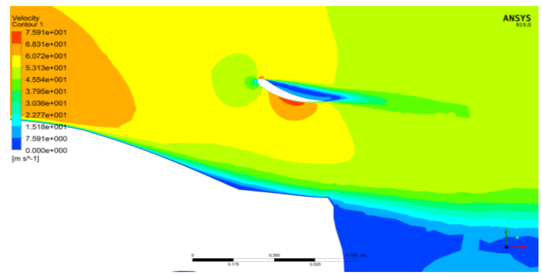


Figure: 6.0 At 30 degrees

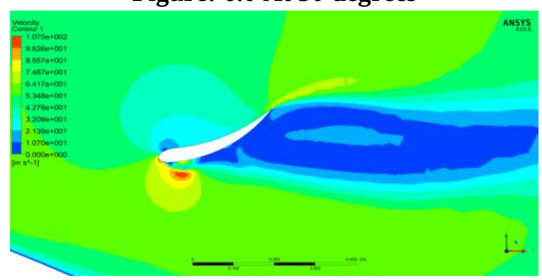


Figure :6.1. At -15 degrees

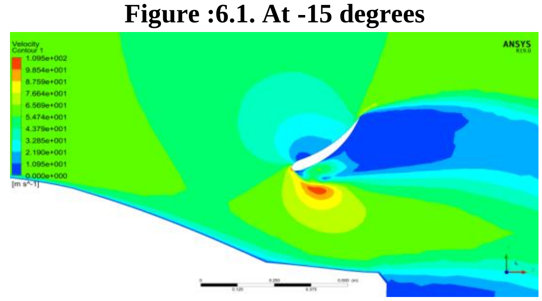


Figure:6.2 At -30 degrees

The velocity profile of air has changed according to the surface modifications

5.3 Comparison of Turbulence energy in all cases at 150km/hr

Figure: 6.3. At angle 15 degrees

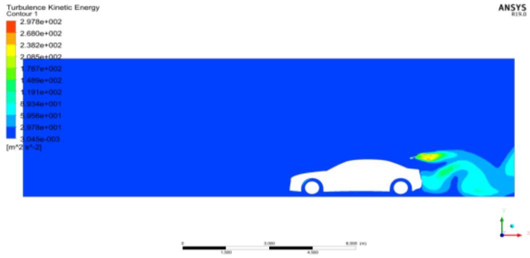


Figure: 6.4.At angle 30 degrees

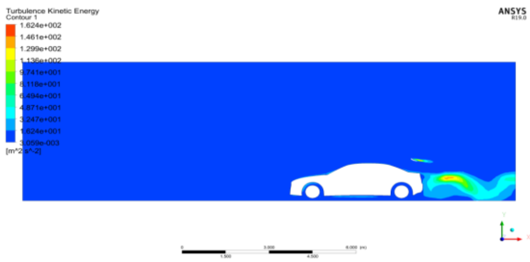


Figure :6.5 At angle -15 degrees

Figure:6.6 At angle -30 degrees

Hence it was shown the turbulence energy is very less at angle -15 degrees when compared to all other angles

6.5 Comparison of Velocity vector in all cases

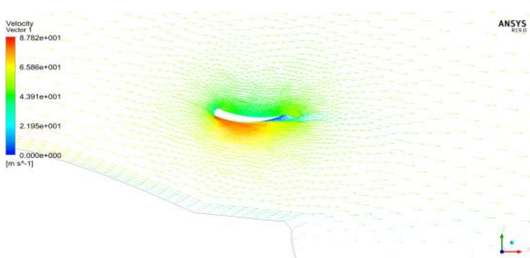


Figure:6.7 At angle 15 degrees

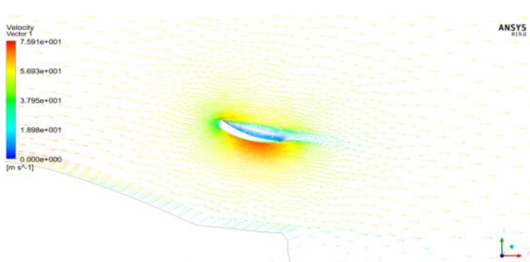


Figure:6.8 At angle 30 degrees

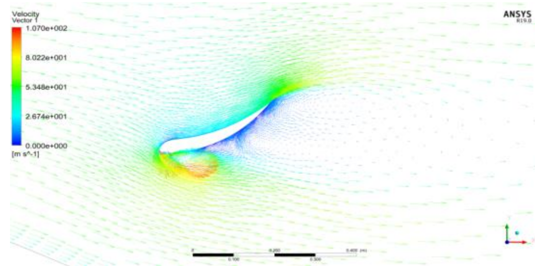


Figure:6.9 At angle -15 degrees

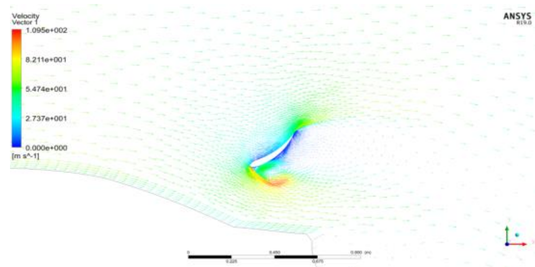


Figure: 7.0 At angel -30 degrees

When a Car is moving in normal conditions it doesn't require any spoiler on its read end but when it is moving in high speed conditions due to pressure differences there will be drag and lift forces due to this the vehicle tend to loose traction of contact with road and causes accidents and also power required will be high so automatically more fuel consumption is goi to take place so to avoid it spoiler at rear end is fixed at different angels as you seen in above images we can notice pressure, velocity, turbulence in all conditions from them we got drag and lift force by keeping them in formula we got how much fuel is consumed at different angles in different speed conditions is observed

5.4 FORMULA

$D=C_d*S*(1/2)\rho U^2$

D=Net drag coefficient

C_d=Drag coefficient

S=Cross sectional area o the entire body=11.9114mt

P=Dynamic pressure of air=1.225kg/mt

U=Velocity

Relation between changes in Drag and changes in Fuel consumption

Power=D*U=+RR*U+AuxP

RR=Rolling resistance =90.30N

AuxP=Auxillary power consumption

Fuel consumption=FC=(bsfc)*Power

Bsfc=break fuel consumption for BMW car =(190.8g/kw.h)

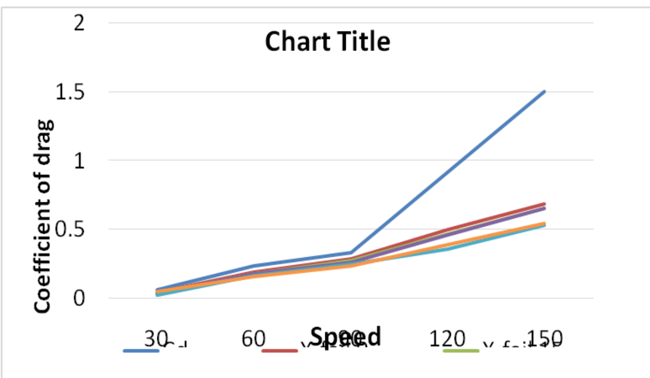
VI. RESULTS AND DISCUSSION

6.1 SIMULATION RESULTS

Speed km/hr	With out spoiler	X foil 0°	X foil 15°	X foil 30°	X foil -15°	X foil -30°
30	0.0582	0.0413	0.0314	0.031	0.0211	0.0484
60	0.234	0.1892	0.1744	0.172	0.1647	0.1547
90	0.329	0.2844	0.2684	0.2547	0.2455	0.2347
120	0.915	0.4956	0.4677	0.4599	0.3544	0.3847
150	1.5	0.682	0.6547	0.6527	0.5314	0.5413

In above table drag values for different speed conditions i,e.30lm/hr-150km/hr is taken at different spoiler angels and it was observed when a car moving at high speed condition i,e at 150km/hr spoiler at angle -15° will have less drag force when compared to all other angel.

6.1.2 Speed vs Coefficient of drag



6.8 Lift and drag values of car at 150km/hr

parameter	C_d	C_l
With out spoiler	1.5	0.549
With spoiler at 0°	0.682	0.554
With spoiler at 15°	0.6547	0.551
With spoiler at 30°	0.5527	0.52
With spoiler at -15°	0.5314	0.535
With spoiler at angle -30°	0.5413	0.531

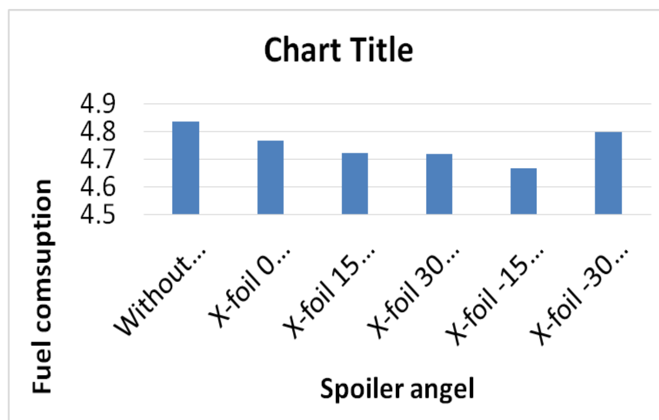
Coefficient of drag and lift values is reduced at angle of -15degrees when compared with all other angels and without spoiler

6.3 Fuel consumption due to Drag at 150km/hr

parameter	Fuel consumption(%)
With out spoiler	4.83364
With spoiler at 0°	4.76575
With spoiler at 15°	4.71974
With spoiler at 30°	04.71779
With spoiler at -15°	4.66072
With spoiler at	04.79627

Fuel consumption at angle -15 degrees is less when compared with all other angels and without spoiler at high speed i,e. 150km/hr

6.3.1 Spoiler vs Fuel consumption

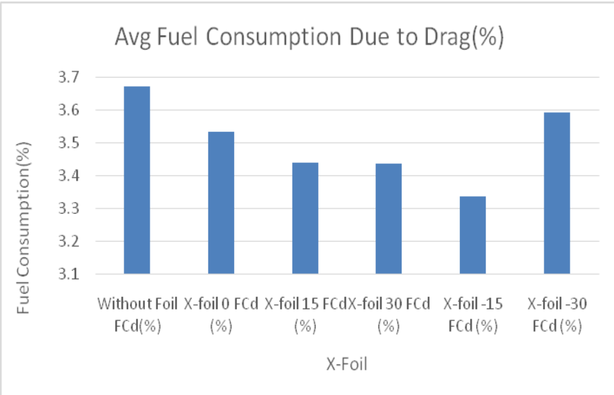


6.4 Average fuel consumption due to drag from 30km/hr-150km/hr

parameter	Fuel consumption(%)
With out spoiler	3.67018
With spoiler at 0°	3.531502
With spoiler at 15°	3.4393844
With spoiler at 30°	3.43558355
With spoiler at -15°	3.33404715
With spoiler at angle -30°	0.3592545

It is observed that the average fuel consumption due to drag reduced at angle -15 degrees

6.4.1 Average fuel consumption due to drag from 30-150km/hr



VII. CONCLUSION

- Based on the study of the aerodynamic flow around a car spoiler at rear end of car at angle -15degrees in high speed condition i.e., 150kmph stood best among all other angels. The result obtained in terms of drag is about 0.53
- Aerodynamics drag for car body of profiles are successfully simulated with ANSYS FLUENT Software .The analysis shows aerodynamic drag in terms of drag coefficient. The computational analysis shows that there is possibility of improving the aerodynamic performance of car by modifications in exterior design of car body. The modifications are helpful in reducing C_d which also improves in efficiency of fuel consumption .By these modification C_d is reduced by 64.57%

VIII. FUTURE WORKS

Some passenger cars have already included spoiler at rear end in order to maximize the efficiency of it by eliminating the side effects in low speeds and increase the advantages on high speed.

Future work would include trying to optimize the current additional components this could include the analyze of different types of spoiler at the rear end of car for the less drag and lift forces to increase the advantages on high speed conditions.

REFERENCES

[1] P.N.Selvaraju, Dr.K.M.Parammasivam, Shankar, Dr.G.Devaradjane: Analysis of Drag and Lift performance in sedan car model using CFD, Journal of Chemical and Pharmaceutical Sciences.

[2] Angelis, W., D. Drikakis, F. Durst., W. Khier.: Numerical and experimental study of the flow over a two dimensional car model, Journal of wind engineering and industrial engineering. 62, 57-69 (1996)

[3] Baker, C. J., and N. D Humphreys.: Assessment of the adequacy of various wind tunnel techniques to obtain aerodynamic data for ground vehicles in crosswinds". Journal of Wind Engineering and Industrial Aerodynamics, 60, 49–68 (1996)

[4] Cogotti, A.: Evolution of performance of an automotive wind tunnel, Journal of Wind Engineering and Industrial Aerodynamics 96, 667–700 (2008)

[5] Dr.Ing. Thomas Schultz.: Progress in CFD validation in aerodynamics development, auto - technology Dissertation University Stuttgart 3, 28-33 (2009)