

# DESIGN AND ANALYSIS OF A NEW REAR SPOILER FOR SU VEHICLE MAHINDRA BOLERO USING CFD

Kelbessa Kenea Deressa<sup>1</sup>, Kiran Kumar Sureddy<sup>2</sup>

<sup>1</sup> Head of the Department, Department of Mechanical Engineering, Bulehora University, Ethiopia

<sup>2</sup>Lecturer, Department of Mechanical Engineering, Bulehora University, Ethiopia

**Abstract-**Environmental issues and increased fuel are driving forces for the automotive manufactures to develop more fuel efficient vehicles with lower emissions. The need for fuel efficiency is a rapidly increasing trend in automotive industries in the recent years. Therefore, extensive research is undergoing for development of aerodynamically optimized vehicle designs. One of the design goals of the spoiler is to reduce drag and increase fuel efficiency. The drag coefficient is an important factor that determines the fuel efficiency of a vehicle in close proximity to the ground. The primary objective of the project is to study the effects of fluid flow and the effective drag of the vehicle over a 3D standard car (BOLERO) with attached Rear Spoiler by using Computational Fluid Dynamics (CFD) simulation. A 1:1 scale model of the actual vehicle was designed in CAD package SOLIDWORKS and CATIA V5 R20. CFD analysis was done over the scaled model keeping conditions as close as possible to the actual road conditions. For evaluation, optimization, the Reynolds-Averaged Navier-Stokes (RANS) equations with Reliable  $k-\epsilon$  turbulence model was used over commercial package ANSYS 14, FLUENT CFD Solver. The effect of aerodynamic drag is significant only at higher velocities. Therefore, the simulation was done for vehicle speed at 80kmph and the results were compared with scaled base vehicle. Various velocity, pressure, streamline contours and velocity plots were examined and analyzed at rear part of the vehicle. It was concluded that, the Co-efficient of drag ( $C_d$ ) of the vehicle with attached Rear Spoiler went down by 4.8%.

**Key Words:** Fluid flow, Computational Fluid Dynamics (CFD), Reynolds-Averaged Navier-Stokes (RANS), Aerodynamic drag, Co-efficient of drag ( $C_d$ )

## I. INTRODUCTION

A spoiler is an automotive aerodynamic device whose intended design function is to "spoil" unfavourable air movement across a body of a vehicle in motion, usually described as turbulence or drag. Spoilers on the front of a vehicle are often called air dams, because in addition to directing air flow they also reduce the amount of air

flowing underneath the vehicle which generally reduces aerodynamic lift and drag. Spoilers are often fitted to race and high-performance sports cars, although they have become common on passenger vehicles as well. Some spoilers are added to cars primarily for styling purposes and have either little aerodynamic benefit or even make the aerodynamics worse. Spoilers on the rear of a car are known as wings when the fascia that produces the drag and down-force is physically separate from the body. A few years ago when oil crisis was not a problem, cars were mainly designed for high-speed maneuver, comfort, and safety. However, with the recent impact due to the increasing oil price since 2003, the sale of automobile industry all over the world crippled. This was immediately followed by many questions raised regarding the effect of oil supply shortage on the future of this industry. Many solutions were certainly suggested and many once-considered-infeasible solutions were now given serious second thoughts. Beside the development of electronic car and fuel cell, other proposed approaches include the integration of air conditioning system with electronic devices to cut down energy consumption, the redesign of car frame and body to reduce its total weight, and the modification of car external to improve the car overall aerodynamic characteristics for better cruising conditions, greater stability of navigation, and lower energy consumption.

These subjects are also indirectly related to environmental protection and noise pollution. The noise associated to a car can be attributed to the aspiration noise and the aerodynamic noise. The aspiration noise is generated when the magnitude of the negative pressure acting on the car body surface is greater than the car sealing pressure. Most of the contemporary car designs have taken into consideration the car sealing pressure which has led to a great improvement on sealing mechanism and reduction in aspiration noise. However, the generation of aerodynamic noise outside a high-speed vehicle due to fluctuating airflows remains a serious problem and therefore demands much more attention.

## II. ANALYTICAL INVESTIGATIONS

Spoiler designed with 2 proposals at different lift angle conditions like  $28^\circ$  and  $30^\circ$  which are carried out in the CFD analysis to know the drag force reduction. These spoilers are picked out for best condition to reduce the drag force and further it fine-tuned to have better curvature

continuity with the vehicle which gives aesthetic look and smooth surface finish. Solid model of spoiler designed with mountings on vehicle from frozen spoiler surface in CFD analysis. Finally Fabricated Spoiler is made with 1.2 mm thick Mild steel sheets (which is readily available in market) by bending and welding. Fabricated spoiler fitted in the vehicle and mileage test carried out.

**2.1 MODEL DESCRIPTION**

The base model vehicle over which design modifications were done is shown in figures

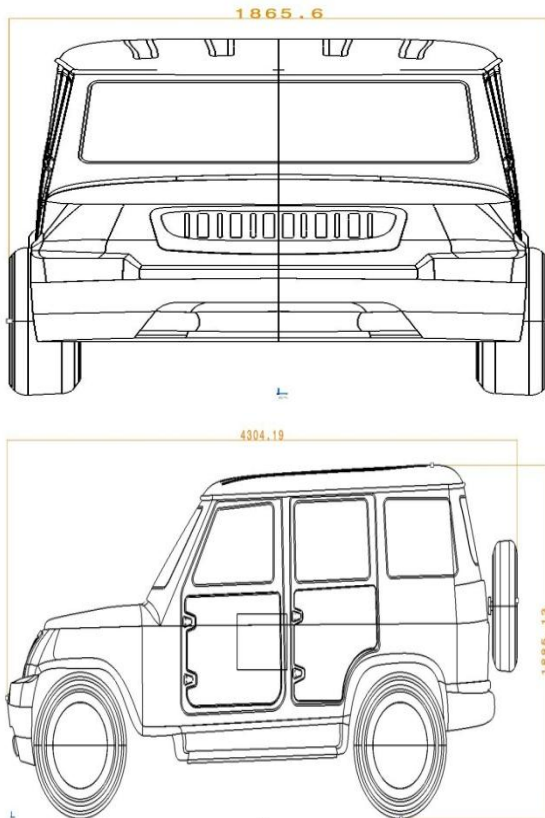


Fig: 2.1 Bolero outer dimensions

Fig2.1 shows the selected model of Bolero vehicle, which was taken from MAHINDRA & MAHINDRA COMPANY which is, designed in cad software's CATIA and SOLID WORKS. They created this model by tracing 2d Blue prints. This model may not match exactly to Mahindra Bolero but tried to match almost by 95%.This model is required to design Spoiler surface and to find current Co-efficient of drag on vehicle by performing CFD analysis.

**2.2 DESIGN OF SPOILER**

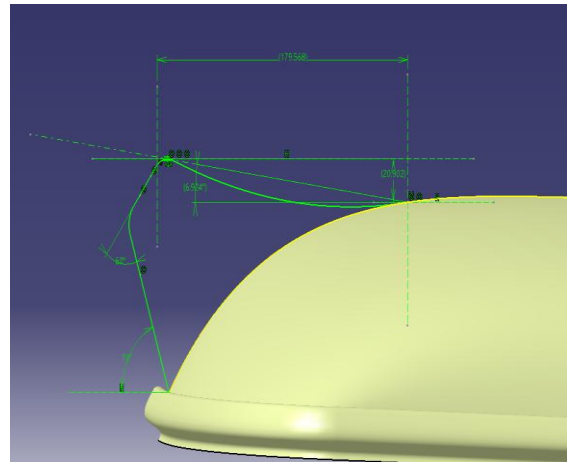


Fig: 2.2 Sketcher model of basic spoiler section

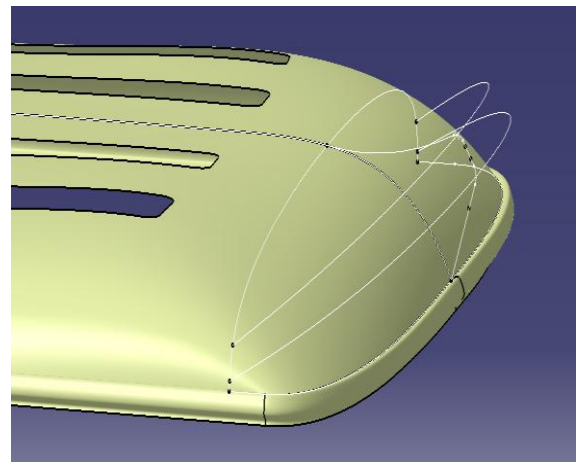
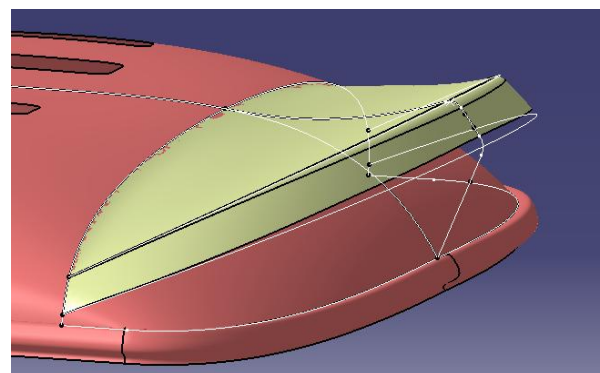


Fig: 2.3 3D Wire frame model of spoiler

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 referred journals. These curves are plotted over the roof surface of Bolero vehicle using splines and providing sufficient tension to the curves.



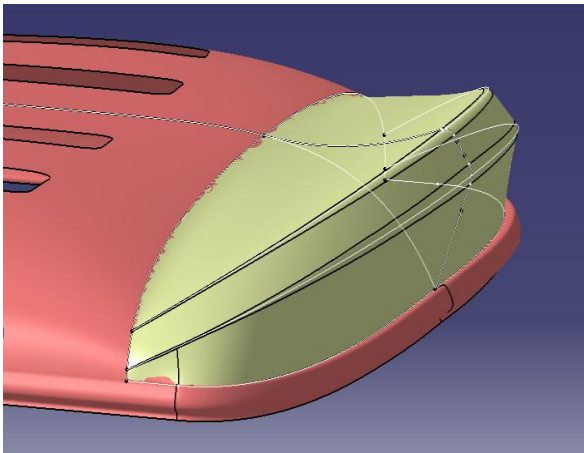
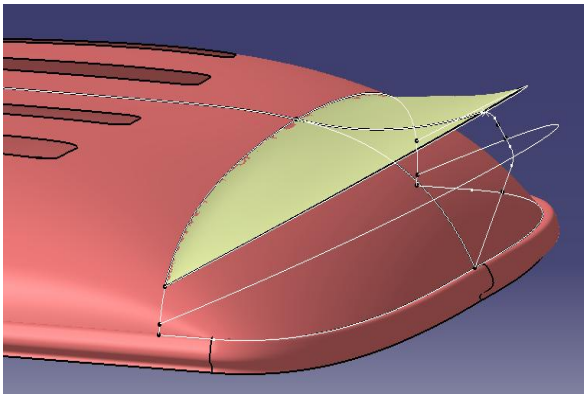


Fig: 2.4 Surface modelling of spoiler

The complete shape and view of the spoiler as shown in the above fig 2.4 is given by filling the regions of the curves in wireframe and surface design module using blend and sweep commands. Above fig 2.4 also shows that the assembly of proposed spoiler and Bolero vehicle.

The complete view of the solid model of the spoiler designed in CATIA is shown below

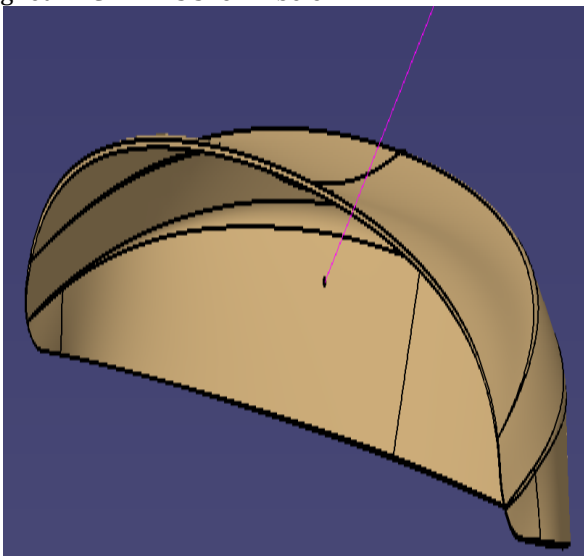
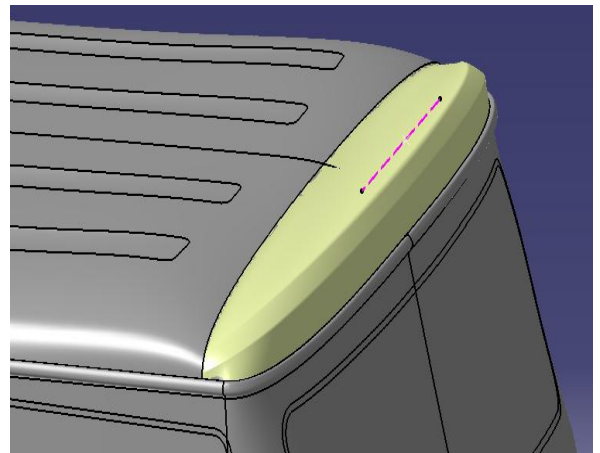


Fig: 2.5 solid modelling of Spoiler Outer & Inner

## 2.3 ANALYSIS OF REAR SPOILER

### 2.3.1 CFD ANALYSIS OF SPOILER

The ANSYS Workbench platform is the framework upon which the industry's broadest and deepest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user through even complex multi physics analyses with drag-and-drop simplicity. When setting boundary conditions for a CFD simulation it is often necessary to estimate the turbulence intensity on the inlets. To do this accurately it is good to have some form of measurements or previous experience to base the estimate on. Low-turbulence case: Flow originating from a fluid that



stands still, like external flow across cars, submarines and aircrafts. Very high-quality wind-tunnels can also reach really low turbulence levels. Typically the turbulence intensity is very low, well below 1%.

### 2.3.2 CFD ANALYSIS OF BOLERO CAR WITHOUT SPOILER

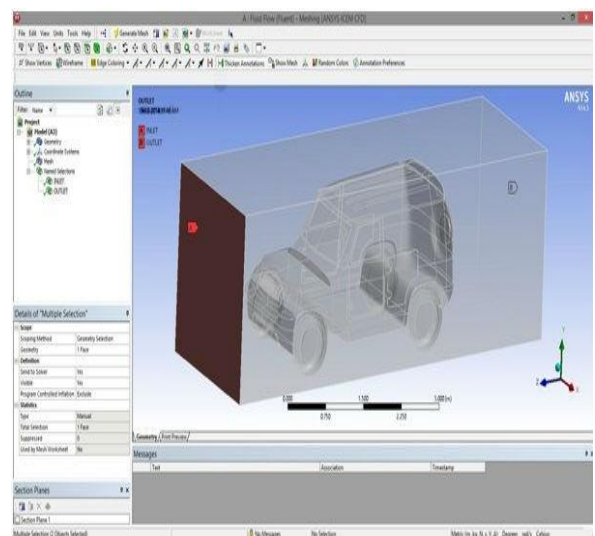


Fig: 2.6 Geometrical model with ANSYS



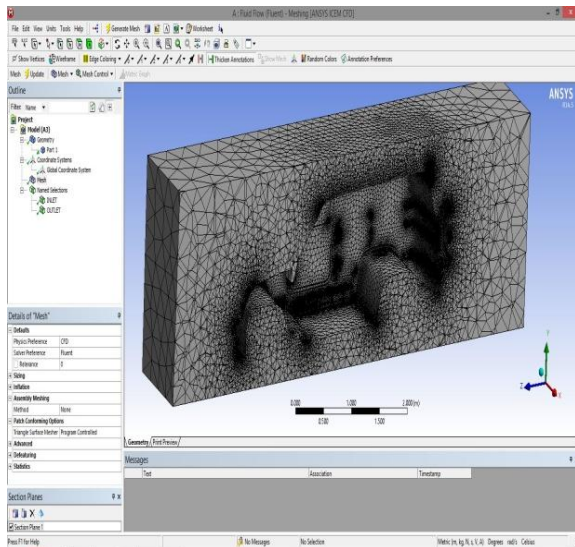


Fig: 2.7 Interface2 meshing of Solid model

To get efficient solution, computational domain was set at 2L (L=Length of the vehicle) either side of the models and 3L upstream and downstream of the models. This size of the domain was efficient enough to capture the changes in the flow field. The Bolero vehicle was meshed in commercial package ANSYS 14. In all the cases, the domain consisted of tetrahedral mesh and prismatic mesh. Prismatic mesh was incorporated in space around the vehicle, wheels and road. Prismatic mesh can capture surface forces of the vehicle accurately and improve accuracy of the calculation. The tetrahedron mesh is generated on Bolero vehicle surface and a surface mesh of 1.5mm size is created on the vehicle surface.

### 3. CFD ANALYSIS PROCESS

The below figures shows the giving of the required boundary conditions in different panels as mentioned earlier

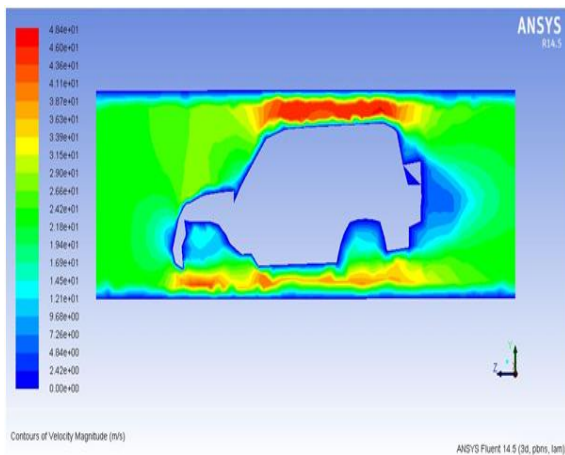


Fig: 3.1 Contours of Velocity Magnitude without spoiler

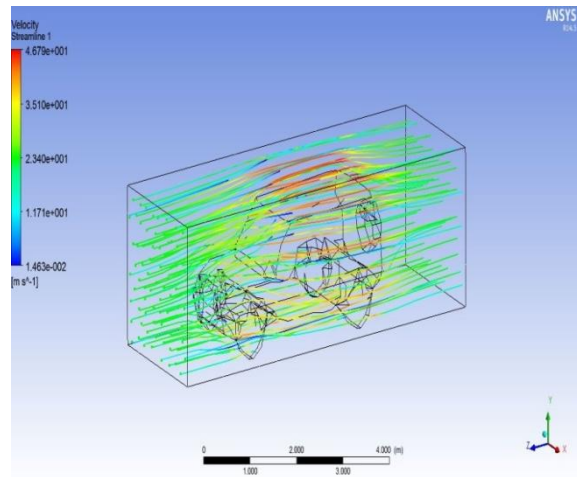
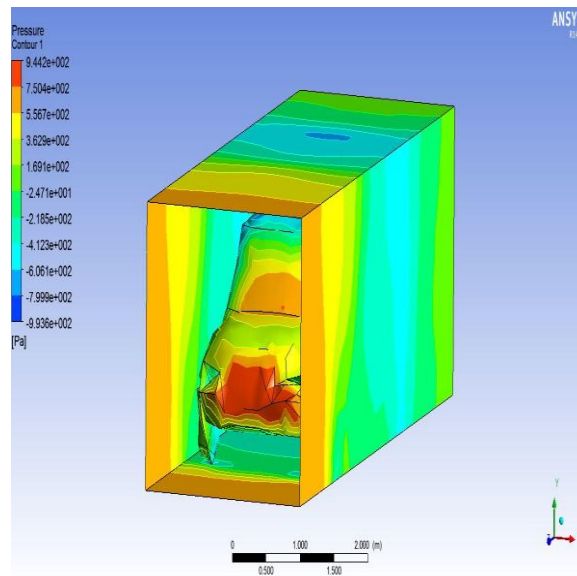


Fig: 3.2 Velocity Stream Line without spoiler

Figure 3.1 shows the total velocity distribution on the car surface, velocity inlet, velocity outlet, side wall and on the road. The distribution of velocity on the front bumper is  $2.420e+00$  m/s and at the rear boot is  $7.260e+00$  m/s are shown in it. Fig:3.2 shows the Streamlines, which are a family of curves that are instantaneously tangent to the velocity vector of the flow. This shows the direction of a mass less fluid element which will travel in any point in time. By definition, different streamlines at the same instant in a flow do not intersect, because a fluid particle cannot have two different velocities at the same point.



3.3 Pressure Contour without spoiler

The above fig shows the pressure contour formed when air was blown over the Bolero vehicle with an atmospheric pressure of 1.01325 bar and a velocity of 22 m/s, the velocity becomes zero when air strikes the vehicles front bumper which leads to the dropping of kinetic energy and increase in pressure energy of the atmospheric air. Figure

also shows the total pressure distribution on the car surface, pressure inlet and outlet, side wall and on the road. The distribution of pressure on the front bumper is  $9.4420e+002$  Pascal and at the rear side is  $-2.471e+001$  Pascal are shown in it.

### 3.2 CFD ANALYSIS OF BOLERO CAR WITH 3° SPOILER

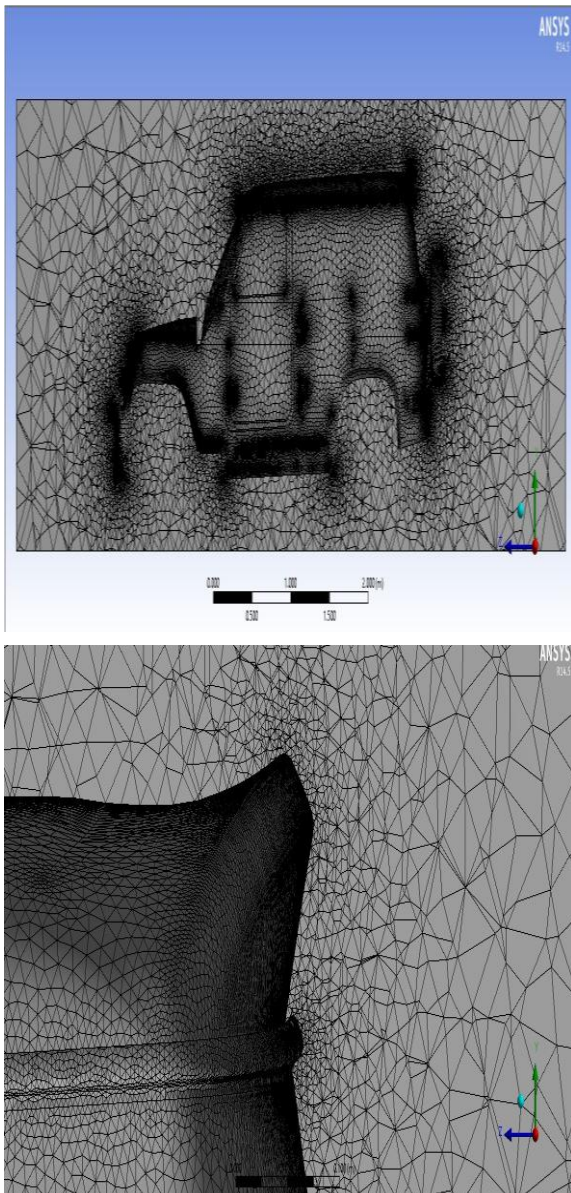
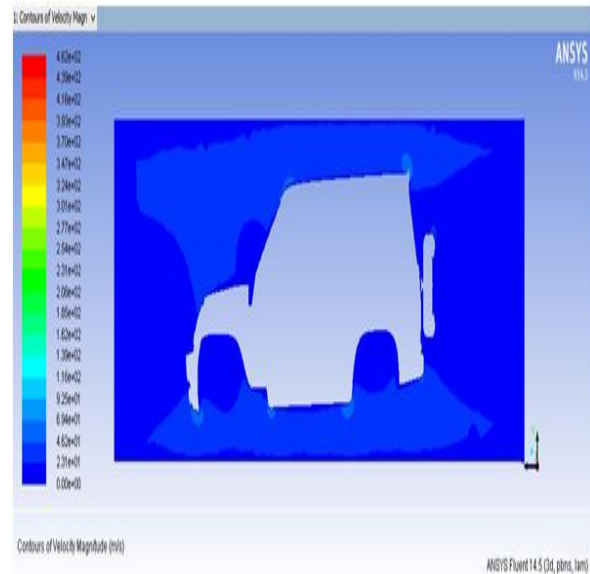


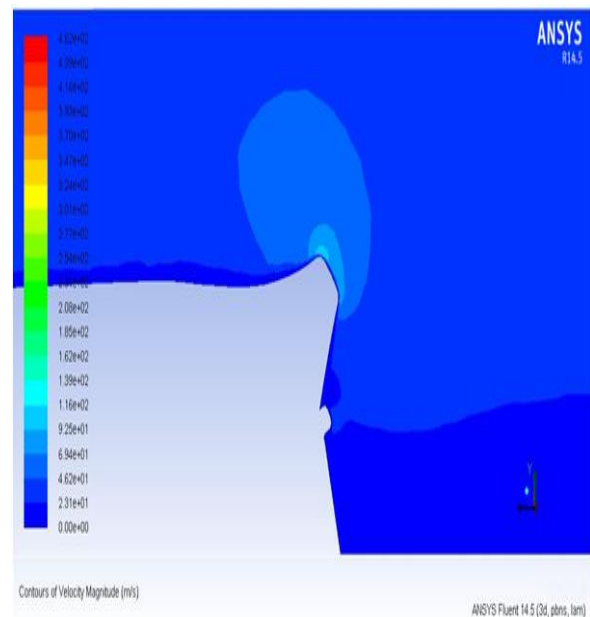
Fig: 3.4 Meshed model with 3° Spoiler

The Bolero vehicle and 3° spoilers were meshed in commercial package ANSYS 14. In all the cases, the domain consisted of tetrahedral mesh and prismatic mesh. Prismatic mesh was incorporated in space around the vehicle, wheels and road. Prismatic mesh can capture surface forces of the vehicle accurately and improve accuracy of the calculation. The tetrahedron mesh is

generated on Bolero vehicle surface and a surface mesh of 1.5mm size is created on the vehicle surface.



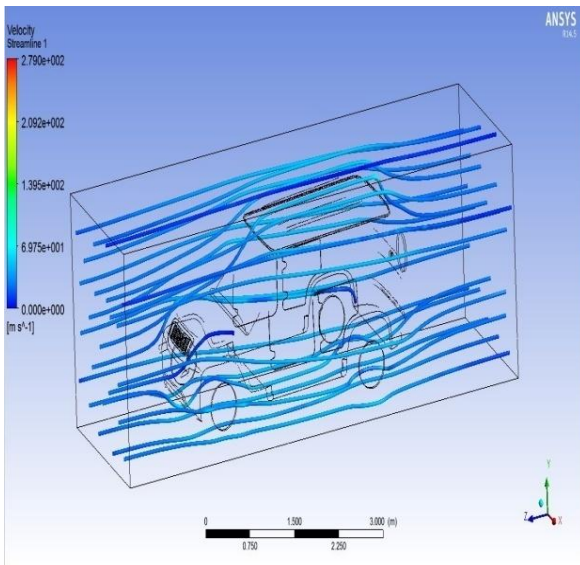
### 3.5 Contour of Velocity Magnitude -2 with 3° Spoiler



### 3.6 Contour of Velocity Magnitude -1 with 3° Spoiler

The above fig shows the total velocity distribution on the car surface, velocity inlet, velocity outlet, side wall and on the road. The distribution of velocity on the front bumper is  $2.310e+01$  m/s and at the rear boot is  $4.82e+02$  m/s are shown in it. The formation of boundary layer (turbulent) due to disturbance created by the spoiler to the uniform flow over the roof surface of the Bolero vehicle. Since the spoiler is placed to spoil the unfavourable air flow over the vehicle by diverting the air flow away from the rear wind shield.





3.7 Velocity Streamline-1 with 3° Spoiler

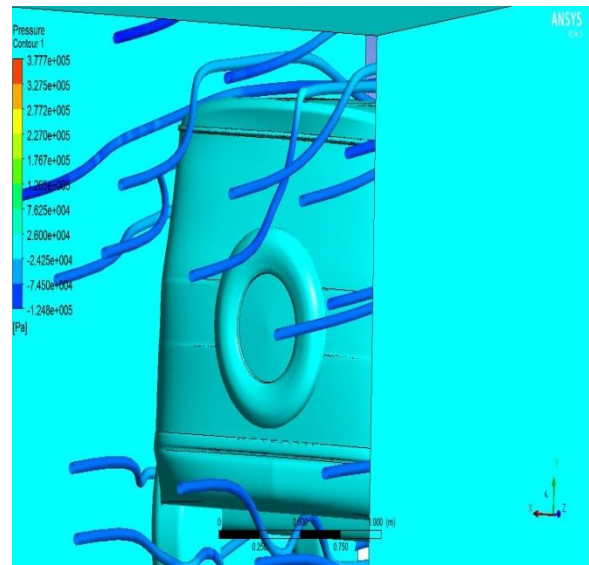
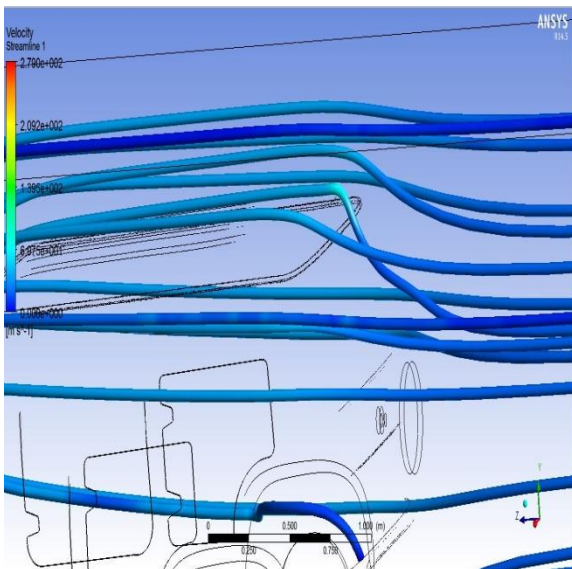


Fig: 3.9 Pressure Contour with 3° Spoiler

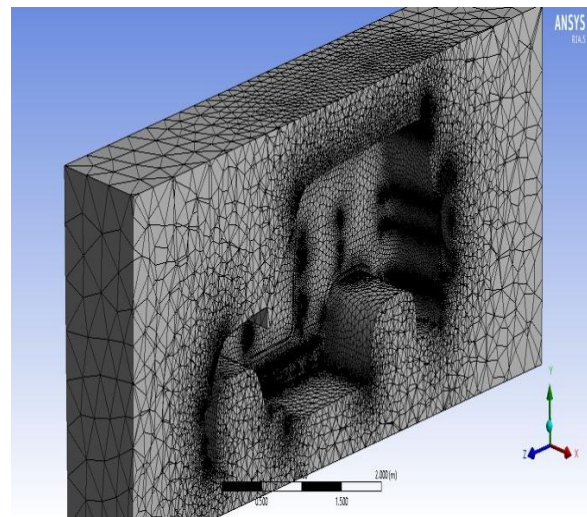
The above fig shows the pressure contour formed when air was blown over the Bolero vehicle with an atmospheric pressure of 1.01325 bar and a velocity of 22 m/s, the velocity becomes zero when air strikes the vehicles front bumper which leads to the dropping of kinetic energy and increase in pressure energy of the atmospheric air. Figure also shows the total pressure distribution on the car surface, pressure inlet and outlet, side wall and on the road.



3.8 Velocity Streamline-2 with 3° Spoiler

The above fig shows the Streamlines, which are a family of curves that are instantaneously tangent to the velocity vector of the flow. This shows the direction of a mass less fluid element which will travel in any point in time. By definition, different streamlines at the same instant in a flow do not intersect, because a fluid particle cannot have two different velocities at the same point.

### 3.3 CFD ANALYSIS OF BOLERO CAR WITH 28° SPOILER



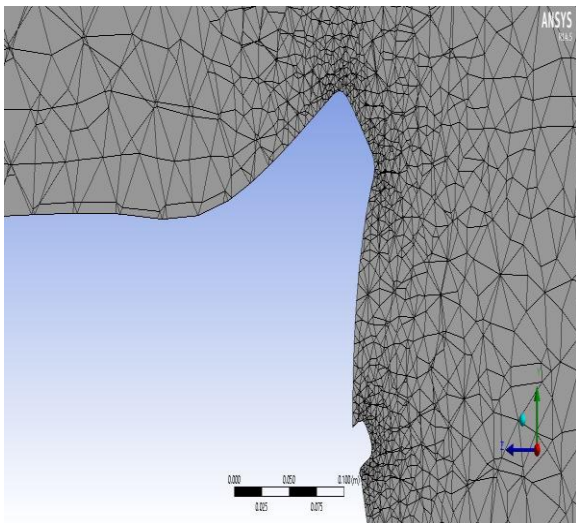


Fig: 3.10 Meshed model with 28° Spoiler

The Bolero vehicle and 28° spoilers were meshed in commercial package ANSYS 14. In all the cases, the domain consisted of tetrahedral mesh and prismatic mesh. Prismatic mesh was incorporated in space around the vehicle, wheels and road. Prismatic mesh can capture surface forces of the vehicle accurately and improve accuracy of the calculation. The tetrahedron mesh is generated on Bolero vehicle surface and a surface mesh of 1.5mm size is created on the vehicle surface.

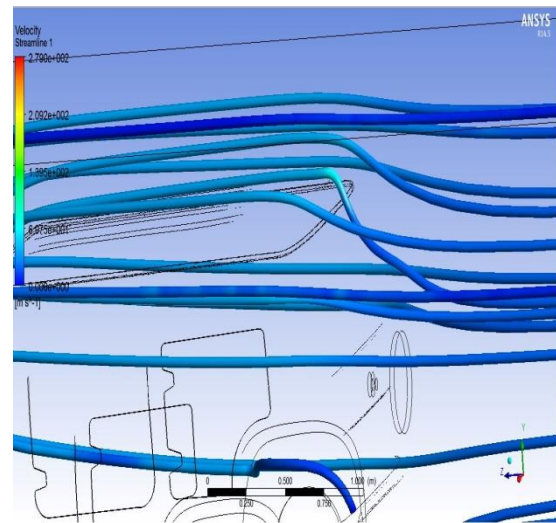


Fig: 3.12 Velocity streamline step-2 with 28° Spoiler

The total velocity distribution on the car surface, velocity inlet, velocity outlet, side wall and on the road. The distribution of velocity on the front bumper is 2.147e+001 m/s and at the rear boot is 8.573e+001 m/s are shown in it.

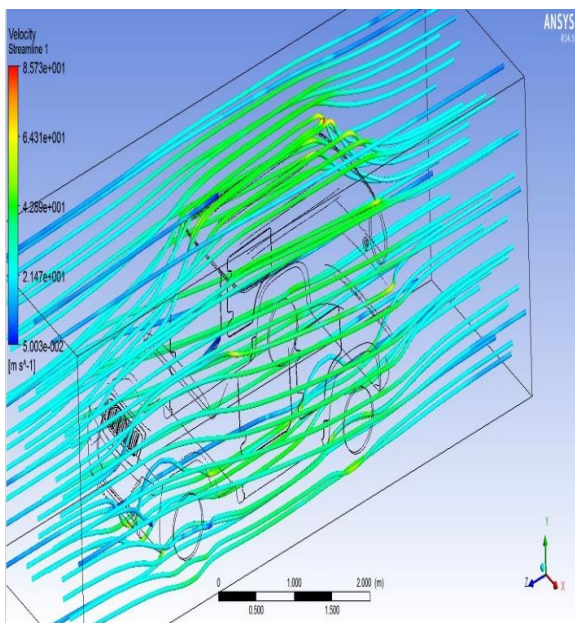


Fig: 3.11 Velocity Streamline step-1 with 28° Spoiler

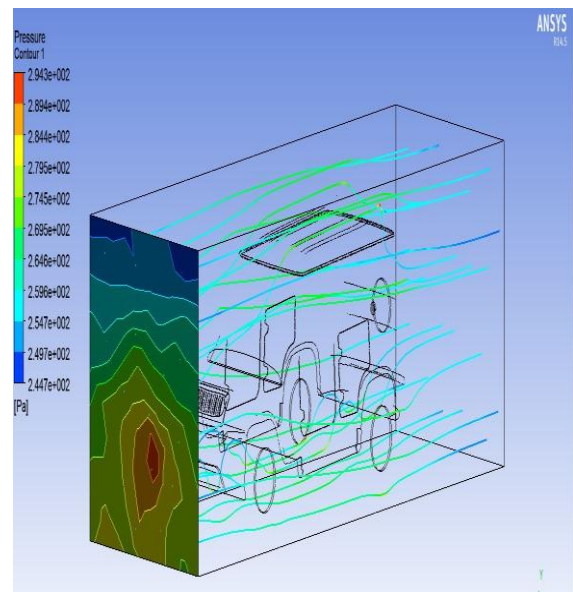


Fig: 3.13 Pressure Contour with 28° Spoiler

The pressure contour formed when air was blown over the Bolero vehicle with an atmospheric pressure of 1.01325 bar and a velocity of 22 m/s, the velocity becomes zero when air strikes the vehicles front bumper which leads to the dropping of kinetic energy and increase in pressure energy of the atmospheric air. Figure also shows the total pressure distribution on the car surface, pressure inlet and outlet, side wall and on the road.

#### 4. ANALYTICAL RESULTS

##### 4.1 $C_d$ GRAPH OF BOLERO CAR WITHOUT SPOILER

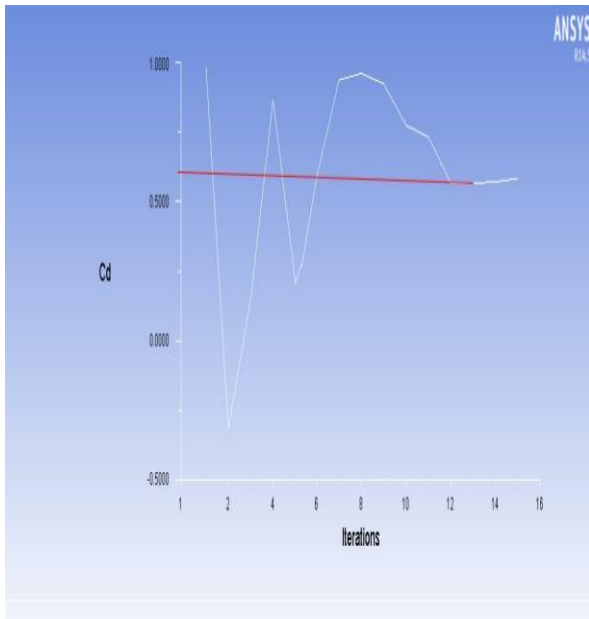


Fig 4.1 Bolero Car without Spoiler  $C_d = 0.62$

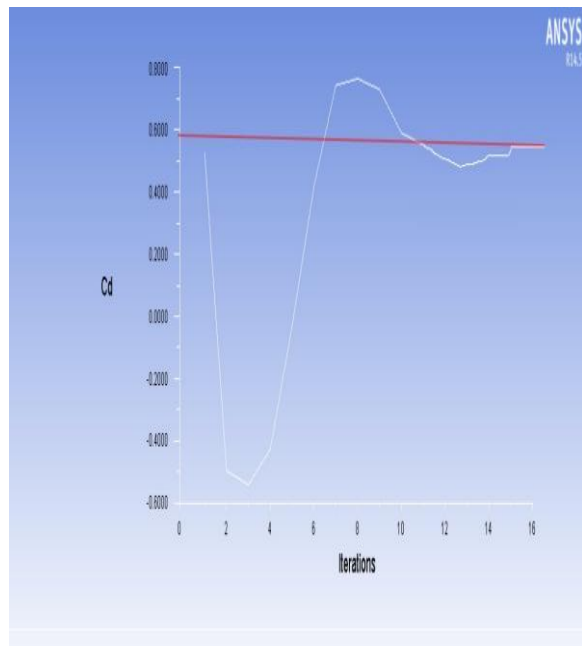


Fig 4.2 Bolero Car with 3° Spoiler  $C_d = 0.59$

When the sketched model of Bolero vehicle without spoiler was analyzed in CFD given the co-efficient of drag value as 0.62 which is matched with the actual co-efficient of drag value (0.62) given by the Mahindra & Mahindra company when vehicle was released into market. Therefore the sketched model of Bolero vehicle in CATIA is matched with actual vehicle dimensions. When the sketched model of Bolero vehicle with 3° spoiler was analyzed in CFD for

evaluating co-efficient of drag value given the co-efficient of drag value as 0.59.

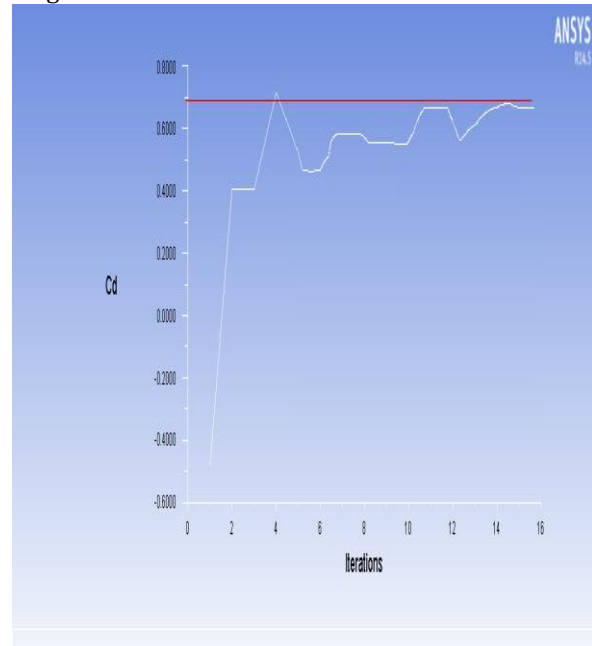


Fig 4.3 Bolero Car with 28° Spoiler  $C_d = 0.69$

When the sketched model of Bolero vehicle with 28° spoilers was analyzed in CFD for evaluating co-efficient of drag value given the co-efficient of drag value as 0.69.

#### 5. FABRICATION OF PROTO SPOILER

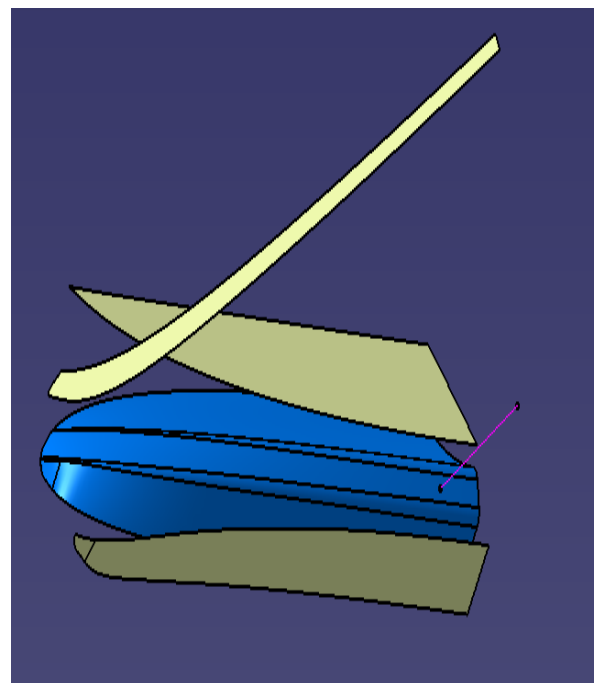


Fig: 5.1 unfolded surfaces



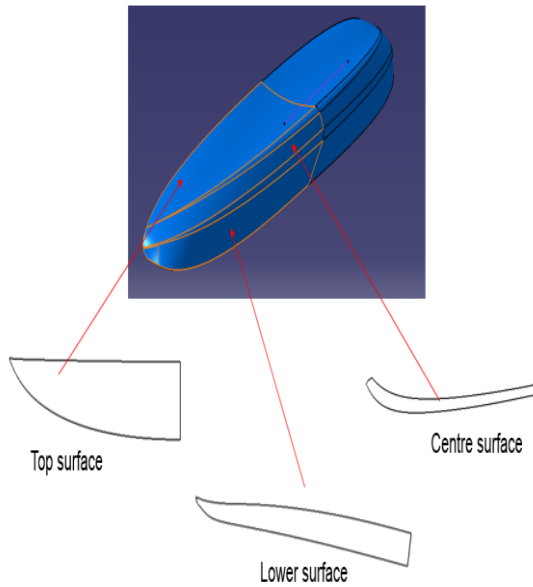


Fig: 5.2 2D curves of unfolded Surfaces for 1:1 printing on A0 Sheet

As the optimized value of  $C_d$  was obtained for  $3^\circ$  spoilers then the fabrication is done for the dimensions of  $3^\circ$  spoilers. The dimensions of  $3^\circ$  spoiler for fabrication is printed on A0 drawing sheet using drafting module in CATIA. The drafting scale chosen as 1:1 for easy understanding of exact dimensions for the fabrication.



Fig: 5.3 Fabricated Model (Welded)



Fig: 5.4 Fabricated Model A-Surface



Fig: 5.5 Fabricated Model B-Surface

2D curves generated from unfolded Spoiler surfaces using CATIA and printed on A0 sheet for tracing same curves on Mild steel sheet for cutting. Mild steel sheet cutted as per tracing with the abrasive parting wheel and then bended with hammers approximately to shape and welded (TIG welding) together.

## 6. RESULTS AND DISCUSSION

### CO-EFFICIENT OF DRAG ( $C_d$ ) VALUES

#### COMPARISON

Table 6.1 Co-efficient of drag ( $C_d$ ) Values Comparison

	Without Spoiler	With 3° Spoiler	With 28° Spoiler
Co-efficient of Drag	0.62	0.59	0.69

On getting the values of  $C_d$  for different angled spoilers in CFD analysis, the project aim for minimum co-efficient of drag is achieved by the 3° spoiler.

The following are observed after design and fabrication of the REAR SPOILER FOR MAHINDRA BOLERO

After attaching Rear Spoiler to Mahindra Bolero  $C_d$  reduced from 0.62 ( $C_d$  without Spoiler on vehicle) to 0.59 in CFD analysis done using ANSYS fluent (4.8% reduction in  $C_d$ ). The maximum streamline velocity observed for 3° spoiler vehicle in the wind tunnel is 462 m/s and similarly without spoiler the maximum velocity obtained is 48.4 m/s. The maximum pressure observed for 3° spoiler vehicle in the wind tunnel is  $3.77 \times 10^5$  pascals and similarly without spoiler the maximum pressure obtained is  $9.442 \times 10^2$  pascals. On Road Mileage test it is expected that the fuel economy of vehicle may increase from 15.5 km/L to 16.0 km/L (3.2% Increase in fuel economy) because the test results showed the reduction in drag force.

For the fabrication MS sheet of 2mm thickness was selected because the properties of these are matched with the ABS material. In CFD analysis, air deviates from Rear wind shield which results in lesser dust accumulation. The aspiration noise is reduced, which is generated when the magnitude of the negative pressure acting on the car body surface is greater than the car sealing pressure.

## 7. CONCLUSION

Computational fluid dynamics (CFD) simulations of the steady flow field around Bolero car model with and without spoiler were presented and compared the simulated data to each other. The ANSYS-14.5 Fluent with the steady model is used for the simulations of aerodynamics. In this analysis,

1. The modelled vehicle of Bolero was analyzed in CFD for determining  $C_d$  value, and then the value obtained is equal to the actual  $C_d$  value (0.62) of Bolero vehicle which was stated in Bolero catalog. So the car modelled in CAD software's is accurately matched with actual Bolero vehicle.
2. It is obtained that  $C_d$  is 0.59 when spoiler is with 3°, so that the angle of flow of air over the spoiler is kept as minimum as possible. So it is concluded that spoiler base curve have to be designed with minimum angle of elevation to minimize  $C_d$ .

3. It is observed that  $C_d$  value is increasing with more elevation in base curve when it is analyzed with 28° of elevation. For 28° spoilers  $C_d$  obtained is 0.69.

In this analysis the coefficient of drag has been reduced by 4.8% with 3° spoiler. The Spoiler objective is to reduce aerodynamic drag acting on the vehicle is achieved and thus improves the fuel efficiency of passenger car which resulted 3.2% increase of fuel economy in On Road mileage test at 80km/h speed.

Hence, the drag force can be reduced by using add on devices on vehicle and fuel economy, stability of a passenger car can be improved.

## REFERENCES

1. J.Yuvan TV, Guan Heng Yeoh and chaoqun Liu, "Computational fluid Dynamics: A practical Approach", Butter worth-Heinemann: 1<sup>st</sup> edition, Burlington, M.A, November 2007.
2. Oleg Zikanov, "Essential Computational Fluid Dynamics", John wiley & sons, Inc.Hoboken, New Jersey, March 2010.
3. R.B. SHARMA , Ram Bansal , CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction, IOSR Journal of Mechanical and Civil Engineering(IOSR-JMCE) e-ISSN: 2278-1684,P-ISSN:2320-334X, Volume 7, Issue 5 (jul-aug.2013), PP 28-35.
4. Sambit Majumder and Somnath Saha, A Method of Drag Reduction of a Vehicle by Computational Investigation and Geometric Modification, International Journal of Applied Engineering Research, ISSN 0973-4562 Volume 9, Number 6 (2014) pp. 687-699.
5. C.H.K Williamson, "Three Dimensional Vortex Dynamics in Bluff Body Wake". Experimental thermal and Fluid Science, volume12, February1996, P.150-168.
6. Marco Lanfrit, "Best practice guidelines for handling Automotive External Aerodynamics with Fluent", Fluent Deutschland Gumb H, 64295, Darmstadt, Germany, February 2015.
7. R.H. Barnard Road vehicle Aerodynamics Design an Introduction Third Edition, March Aero, 2009.
8. L.Davidson. an Introduction to Turbulence Models. Department of Thermo and Fluid Dynamics channels university of Technology, 2003.
9. Wong J (2008) Theory of ground vehicles (4thEd) Hoboken, NJ: Wiley ISBN978-0-470-17038-0.
10. Emission: Measurement, testing & Modelling (2006). Warren dale, PA: Society of Automotive Engineers

## BIOGRAPHIES



Mr. Kelbessa Kenea Deressa has completed his B.SC Mechanical Engineering (Motor Vehicle Engineering) from Adama Science and Technology University, Ethiopia. Presently working as Head of the Department in Mechanical Engineering in Bule Hora university and he has 1 Year Experience.



Mr. Sureddy Kiran Kumar is working in Bule Hora University Ethiopia. M.TECH, B.TECH from JNTUK. Research Scholar in Acharya Nagarjuna University, A.P Working experience 5 Years working in Alternative Fuels, renewable energy resources, CFD, Heat Transfer, Solar Energy, Refrigeration and Air Conditioning.