Drag Analysis of Vehicle Using CFD

'Rohan Easwaran', 'Rohit Hankare', 'Raghuvendra Singh '
Department of Mechanical Engineering, Lokmanya Tilak College Of Engineering, Navi Mumbai
Department of Mechanical Engineering, Lokmanya Tilak College Of Engineering, Navi Mumbai
Department of Mechanical Engineering, Lokmanya Tilak College Of Engineering, Navi Mumbai
Under the guidance of
Dr. Vivek Yakkundi
Principal, Lokmanya Tilak College Of Engineering, Navi Mumbai

***Abstract-** To save energy and to protect our environment, the reduction of consumption of fuel is a primary concern for all the modern car manufacturers. Reduction of drag is essential for reducing the overall consumption of fuel. Designing a vehicle with minimal drag resistance provides huge economical and performance advantages. Reduced resistance to forward motion allows one to gain higher speeds for the same power output, or lower power output for the same speed. The shape is an important factor for drag reduction. Designing an efficient body for a car that will offer a much lower resistance to the forward motion which will fulfil the most important functional requirement today, which is low fuel consumption. The resistance, termed as the drag force (or the drag coefficient in non-dimensional terms), is a strong function dependent on the shape of the car. This suggests the importance of how the fluid (air) particles move around the car and how fast they move along their path. The main intention behind this project is to compute the Drag co-efficient, Drag force and moments on three different vehicle profiles and compare them using Ansys Fluent 16 and CFD software (computational fluid dynamics) to determine the best efficient design which possess low coefficient of drag.*

***Keywords** - Drag Resistance, Efficient Body, Low Fuel Consumption, Ansys Fluent*

1.INTRODUCTION

Since the 21st century in stride, virtually all manufacturers have adopted some form of computer aided design, Engineering, Manufacturing and Analysis. A common belief to stay ahead of their competition is by continuously introducing new products that are differentiated by the latest Technology revolution, innovative designs, higher functionality and superior quality. The best technology to integrate expertise knowledge and companies unique practice is the CAD/CAM and analysis software packages; a comprehensive solution to meet the present day demand.

In general term CAD/CAM means computer assistance whilst a designer converts his ideas and knowledge into a graphical model and graphical model into physical model. After that he analyses the physical model to meet the Environmental conditions.

The automotive industry is a large user of commercial CFD packages. The advantage of CFD results in better designs and reduced time for the automotive manufacturers. CFD is not only used to improve the aerodynamics of vehicles, but also for the optimization of domains such as engine cooling, brake cooling, airbags, lighting and fuel system. During the development of new vehicles, understanding the flow phenomena and how aerodynamic forces are influenced by changes in body shape are very important. A large variety of complex flow properties such as three-dimensional turbulent boundary layer on the body surfaces, longitudinal vertices induced by three-dimensional separation, recirculation flows caused by separation and the ground plane boundary layer and their interaction are important to be understood. However, using CFD is a good way for designers to obtain results in a shorter time.

The job of the aerodynamics engineer is to create a body shape that maximizes the force in direction of the ground, the down force, and minimizes the force that opposes the movement, the drag force.

The CAD software Solid-works is used to design the outer body shape by finding out the co-ordinates of the body. The mesh generation and analysis to find the Co-efficient of Drag is done by using the Ansys Fluent Software.

Drag Force analysis of car comprises of following steps:

1) Geometric modelling in 2D by importing the image of car in Solid-works and finding out the co-ordinates of the car.
2) Entering the dimensions of the vehicle.
3) The co-ordinates of car are used to draw the image of car and then analyse the car in Ansys Fluent to find out the C_D by giving boundary conditions i.e. Pressure, velocity, Density of air and Atmospheric temperature.
4) Calculate the Drag Co-efficient by using the values obtained from analysis.

One of the objectives of this project is to conceive a body which is optimized to have good performance. The project is focused on the outer body shape of the car. The CFD tools have been used to assess the quality of the design. The main purpose behind this project is to analysis the shape of simple car in 2D and find out the drag co-efficient which will help to modify the car shape or can help in design of new car aerodynamically.

2.RESEARCH CONDUCTED

A survey on the topic was undertaken and it was found that a large amount of research has been done on the topic. After an extensive research three research papers were found to have produced the best result. A short summary on each research conducted in provided

- **SIMULATION OF AERODYNAMIC BEHAVIOUR OF A ROAD VEHICLE IN TURBULENT FLOW-**Ahmed Al-Saadi

The study concentrated on various aerodynamic drag reduction techniques to reduce the aerodynamic drag coefficient and increase the stability of a three-dimensional full-size road vehicle. There were ample modifications which were used in this research paper, which can be used either individually or in combination. Computational Fluid Dynamics (CFD) analysis based on steady state Reynolds-Averaged Navier-Stokes (RANS) turbulence modelling , Realizable k –ω, Standard k- ω, Shear Stress Transport k-ω Stress Model (RSM), were used in this study. The simple geometry of wagon model was achieved by modifications of the front part using Computational Fluid Dynamics (CFD). The k turbulence model was used to calculate the drag coefficient. It is assumed that the bottom part is on a flat surface. In this study four different LES models, QR,VMS,SIGMA and WALE were used. The SIGMA, QR and VMS models were used for the first time to resolve the flow around simplified vehicle models. The Reynolds number of 7.68×105 based on the height of the body was used in this case. It was found that coarse grids are useful in LES simulations. The drag coefficients and the body-fitted methods were within 3% and 3 7% of the experimental measurement, respectively. The drag coefficients of Sports Utility Vehicles (SUVs) were higher than saloon cars because of the size and the rear part design of this kind of vehicles. However, the flow field analysis around SUVs was difficult because of the low pressure contour behind the car. ANSYS 16.0 was used to generate mesh with varying levels of refinement. The CFD simulations were performed with the ANSYS Fluent 16.0 software. The best modification for the SUV with velocity less than 120 km/h (Re = 1.12×107) is a spare tire because the minimum drag coefficient and low cost. The drag coefficient of the baseline model is 0.4 while 0.372 for the model with spare tire. Drag coefficient decreases with increasing of Reynolds number except spare tire which decreases then slightly increases again. It was found that good agreement has been achieved between the calculated drag coefficient for the baseline models and the experimental data for all types of turbulence models.

- **CFD ANALYSIS OF DRAG REDUCTION FOR A GENERIC SUV -** Pramod Nari Krishnani

The flow in the near wake of the blunt bodies of road vehicles like SUVs plays an important role in determining the pressure forces acting on the surface of the body. To better understand the wake profiles and stagnation pressure gradient of the vehicle, this paper performed numerical analysis on CAD model of a Generic SUV which was previously tested in the wind tunnel. Commercial software package ofANSYS® GAMBIT, T-grid and FLUENT® was used for multi cell meshing and solving of the governing equations. The pressure coefficient ‘Cp’plots at the symmetry plane of the model was compared with the experimental results from the wind tunnel tests to validate the simulation. Results and conclusions were presented from the simulations of the CAD model using upper and lower flat boat tail plates with gradual increment in the angle of inclination.

- **Aerodynamic Drag Reduction for A Generic Sport Utility Vehicle Using Rear Suction-** Abdellah Ait Moussa

  In this paper, an investigation into the effect of adding a suction slit in the rear of a generic model of an SUV on its overall aerodynamic performance was conducted. The author introduced a robust method to identifying the size, location and boundary suction inlet velocity for maximum reduction in aerodynamic drag. Proper design of the opening, the actual geometry of the vehicle rather than simple models since small changes in geometrical details must be taken into account that can lead to large changes in the aerodynamic flow around the vehicle.

3. METHODOLOGY AND FINDINGS

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

(i) 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 codesoftware 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.

The finite difference and element method are usually suitable for stress and structure analysis and does not suite the requirements of the CFD process. The finite volume method is the most suitable method for the CFD process. As the name implies, finite volume method is the numerical algorithm calculation process involving the use of finite volume cells.

The steps involved in this solving process are usually carried out in the following sequence:

1. Formal integration of the governing equations of fluid flow over all the control volumes or finite volumes of the solution domain.
2. The conversion of the integral forms of the equations into a system of algebraic equations.
3. Calculations of the algebraic equations by an iterative 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

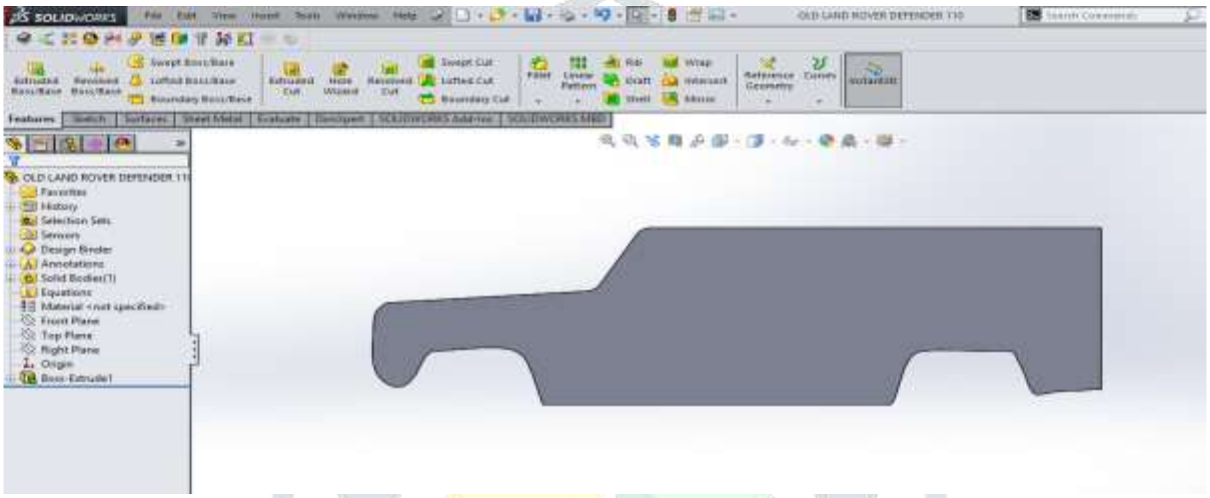


Figure 8.3 Original model of Land rover defender 110

The generic model of Land Rover Defender 110 is shown in the fig above with relevant dimension

Length = 4685 mm

Width = 1992 mm

Height = 2071 mm

Center Distance between two tyres along the length of the car = 2794 mm

Distance between two tyres along the Width of the car = 1790 mm

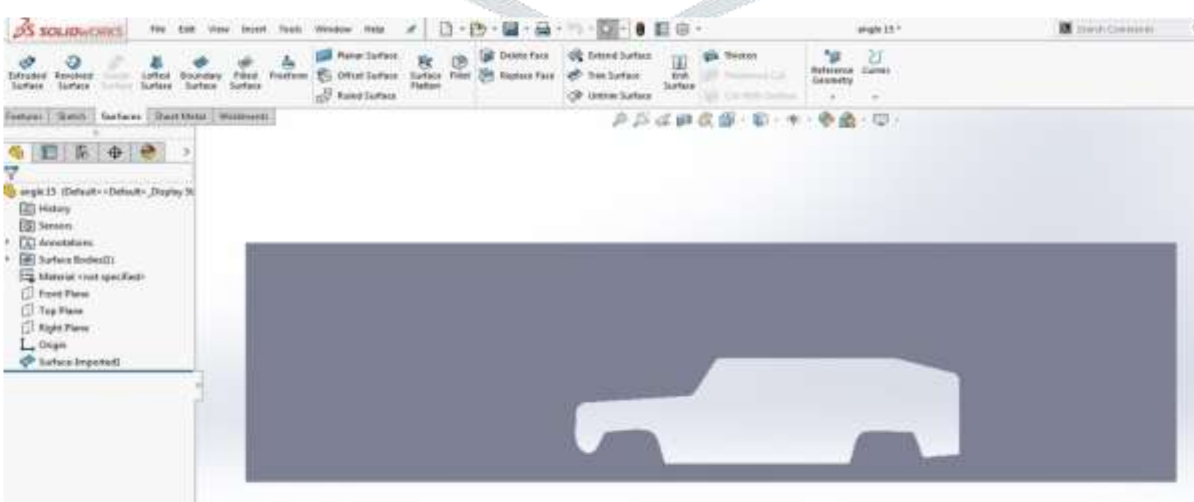


Figure 8.4 Land rover defender 110 with backlight angle 15 degree

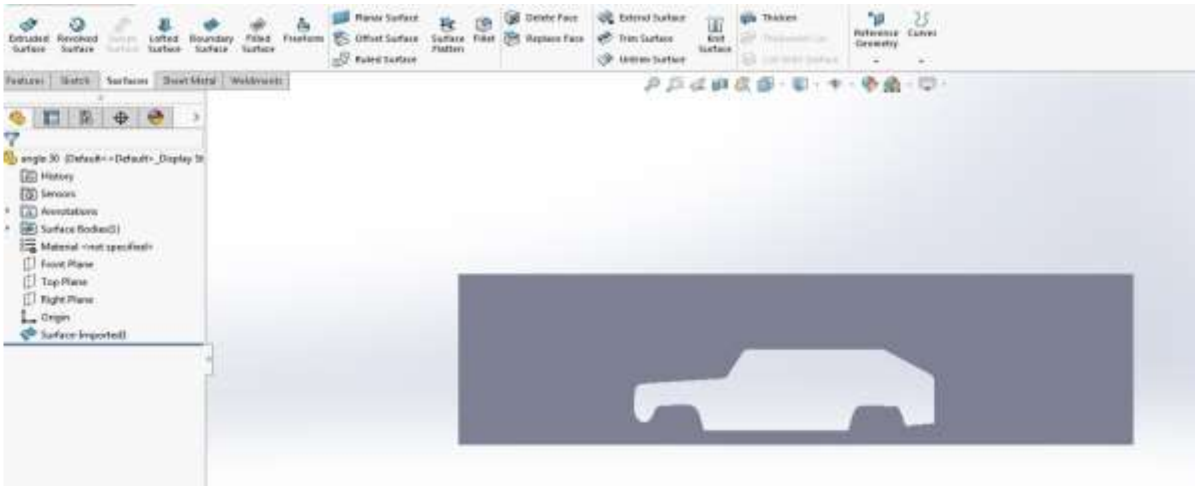
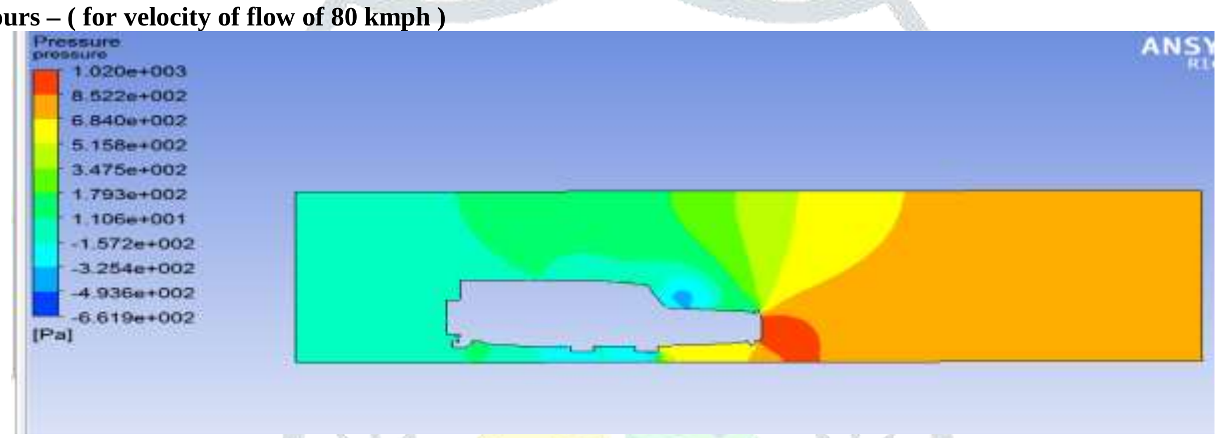


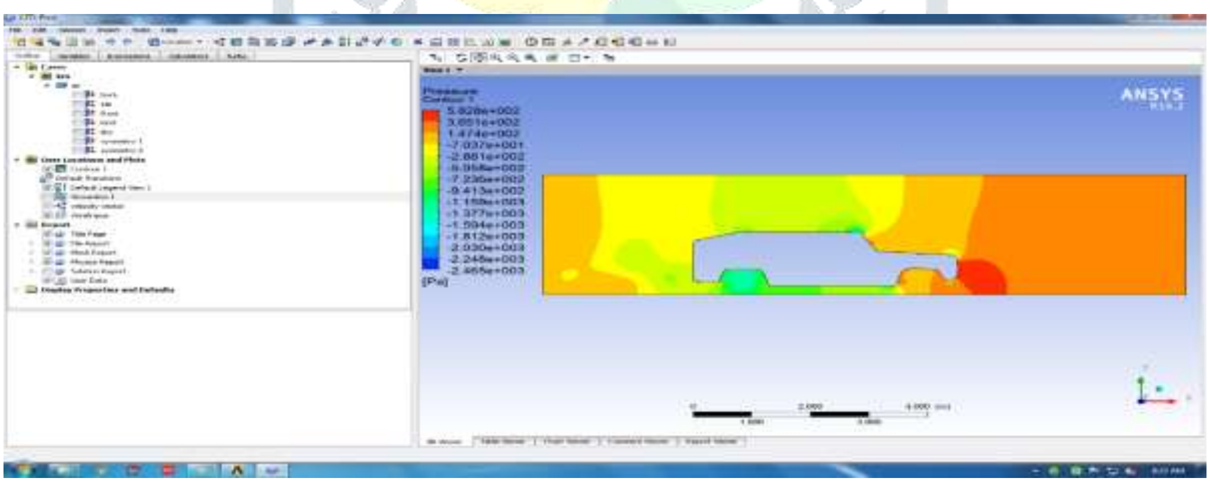
Figure 8.5 Land rover defender 110 with backlight angle 30 degree

4. RESULTS

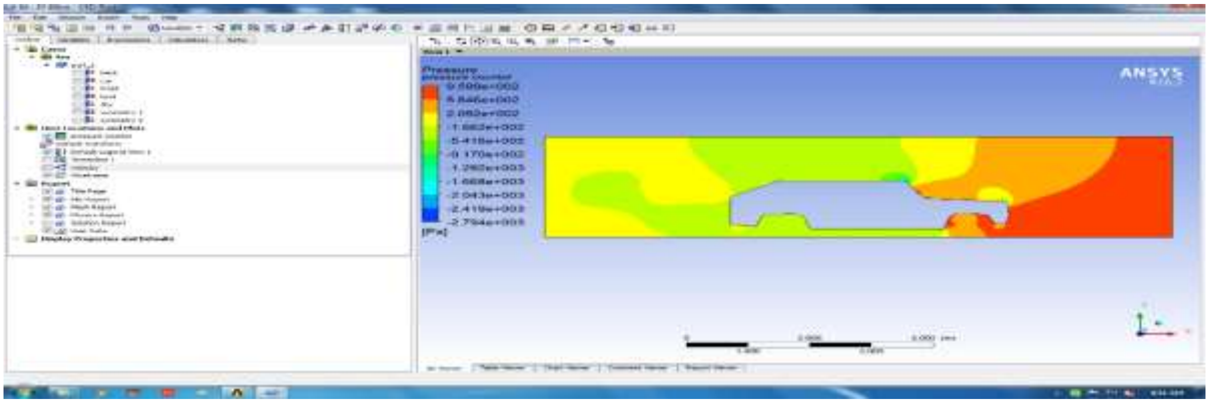
Pressure contours – (for velocity of flow of 80 kmph)



Original model of Land rover defender 110

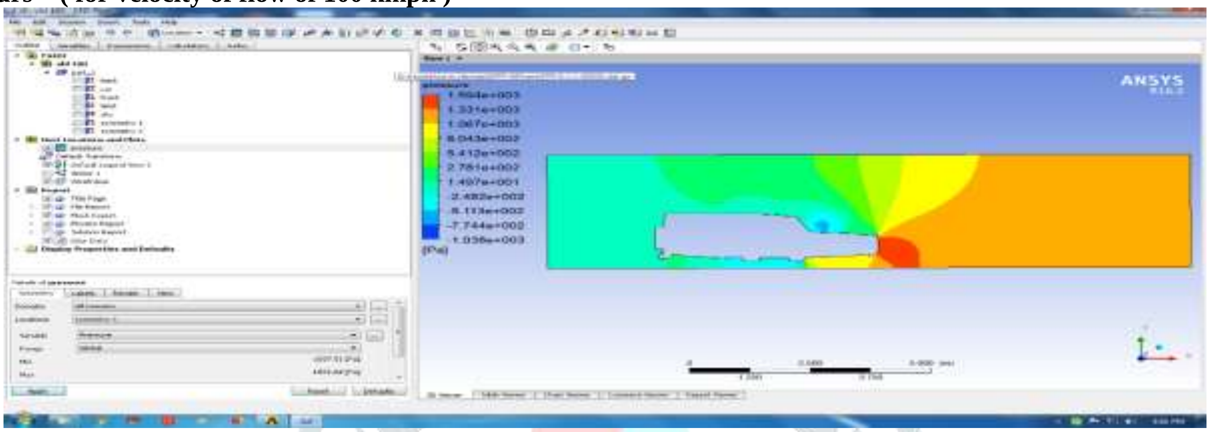


Land rover defender 110 with backlight angle 15 degree

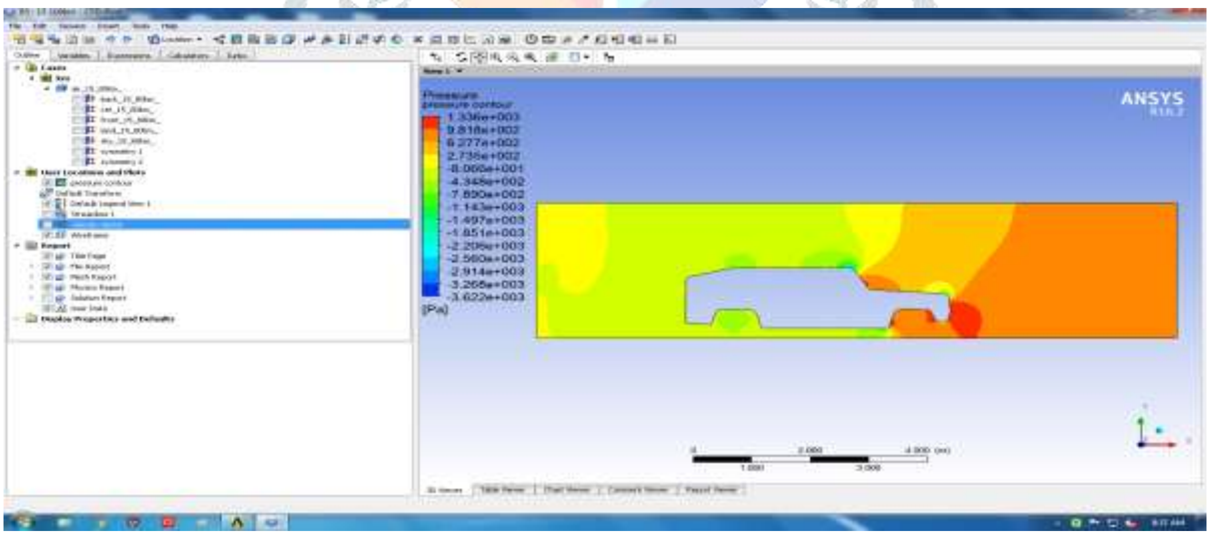


Land rover defender 110 with backlight angle 30 degree

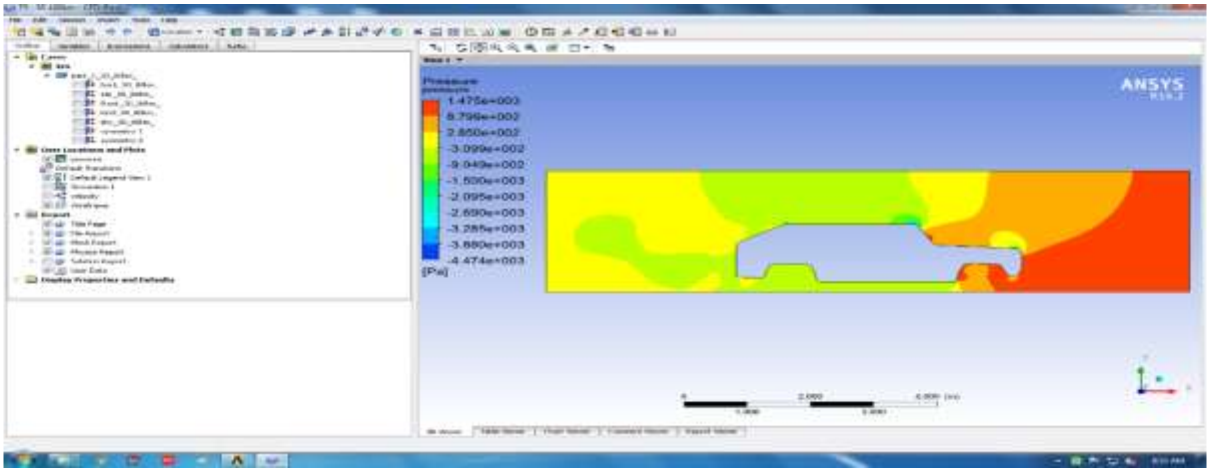
Pressure contours – (for velocity of flow of 100 kmph)



Original model of Land rover defender 110

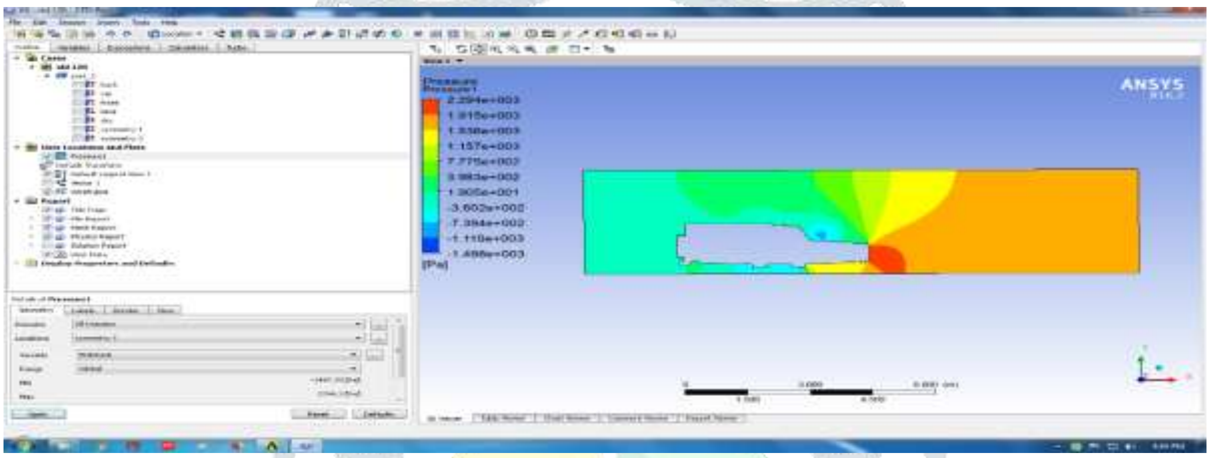


Land rover defender 110 with backlight angle 15 degree

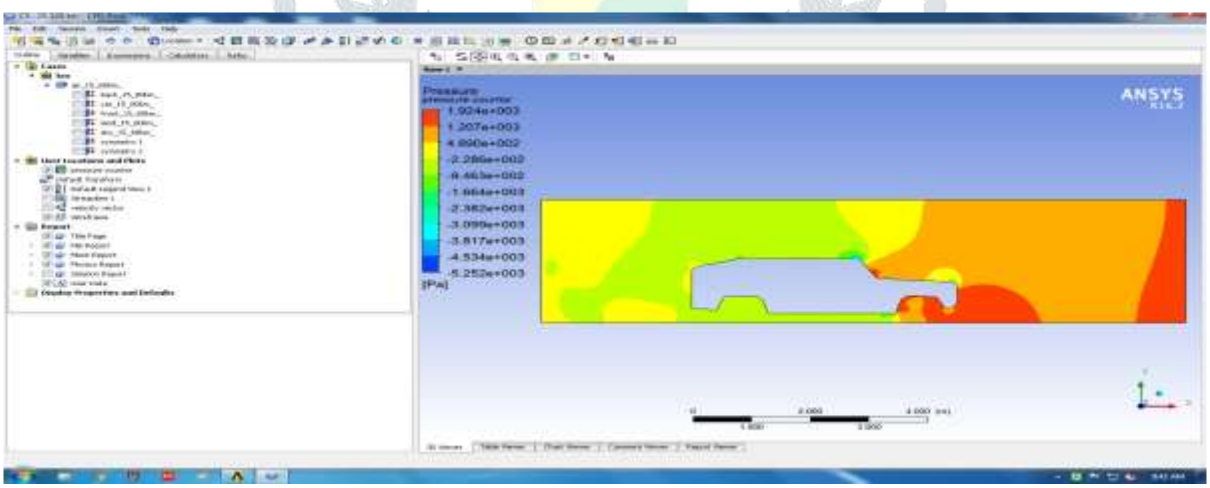


Land rover defender 110 with backlight angle 30 degree

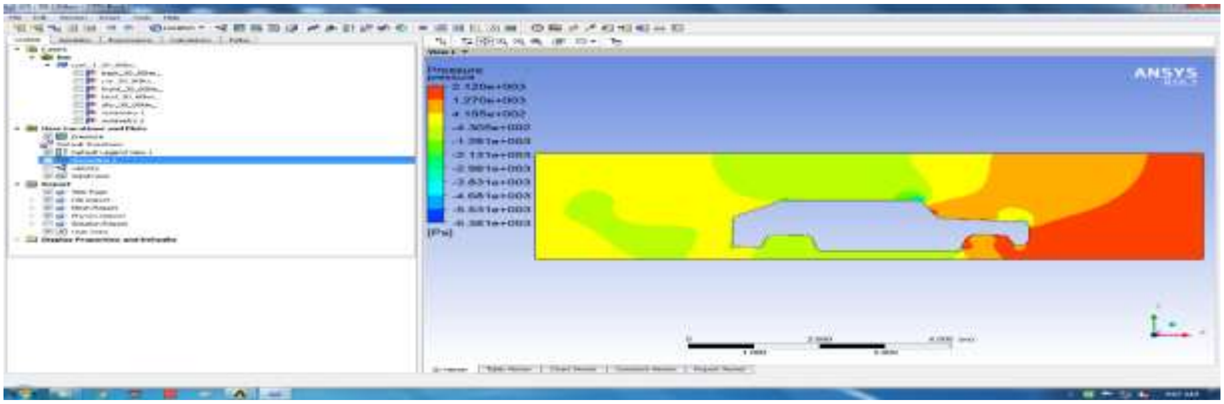
Pressure contours – (for velocity of flow of 120 kmph)



Original model of Land rover defender 110

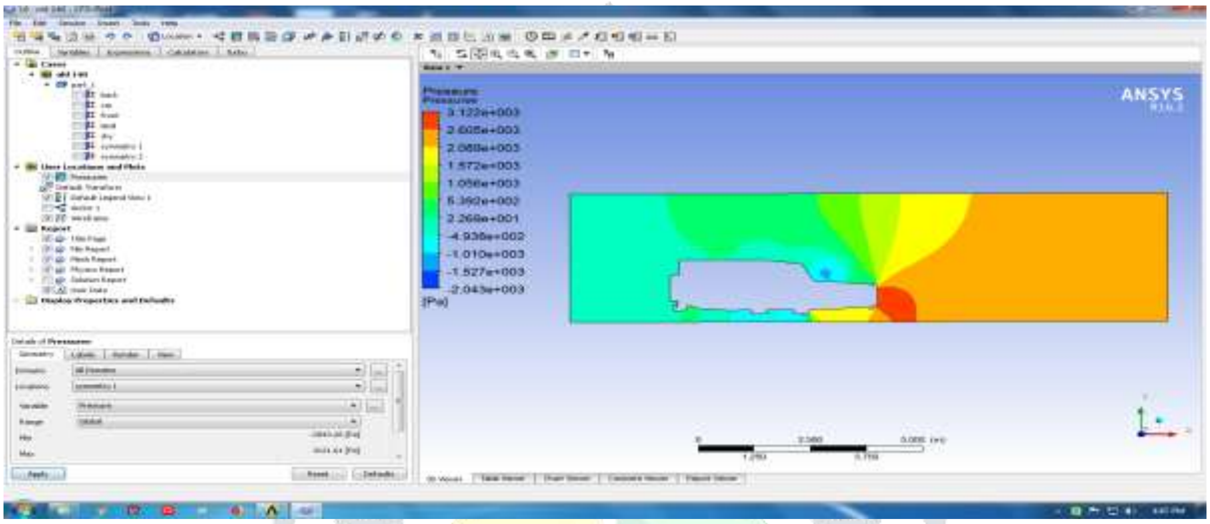


Land rover defender 110 with backlight angle 15 degree

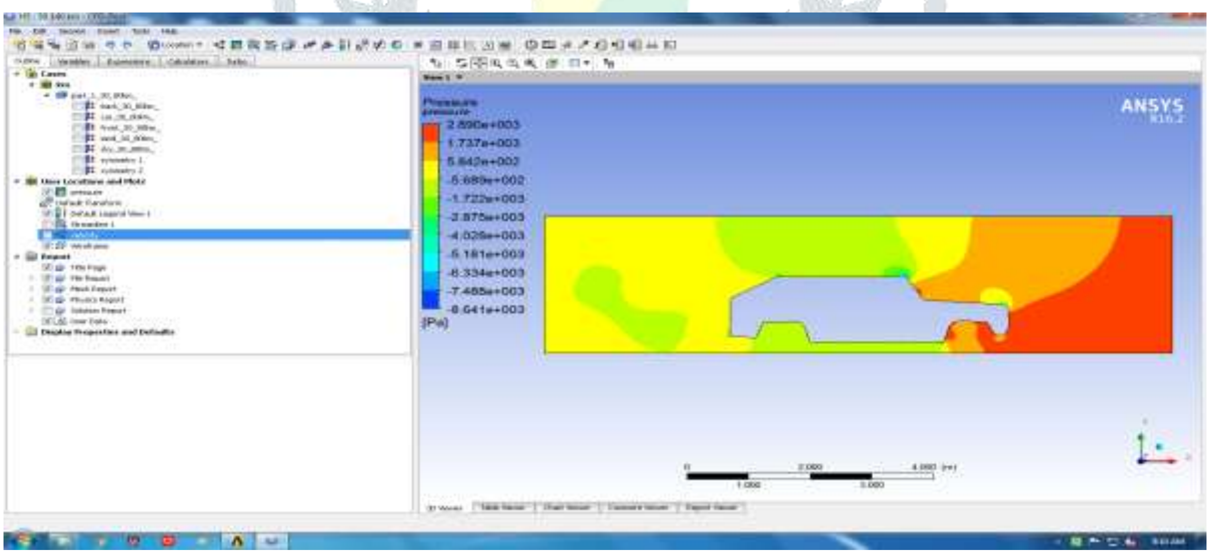


Land rover defender 110 with backlight angle 30 degree

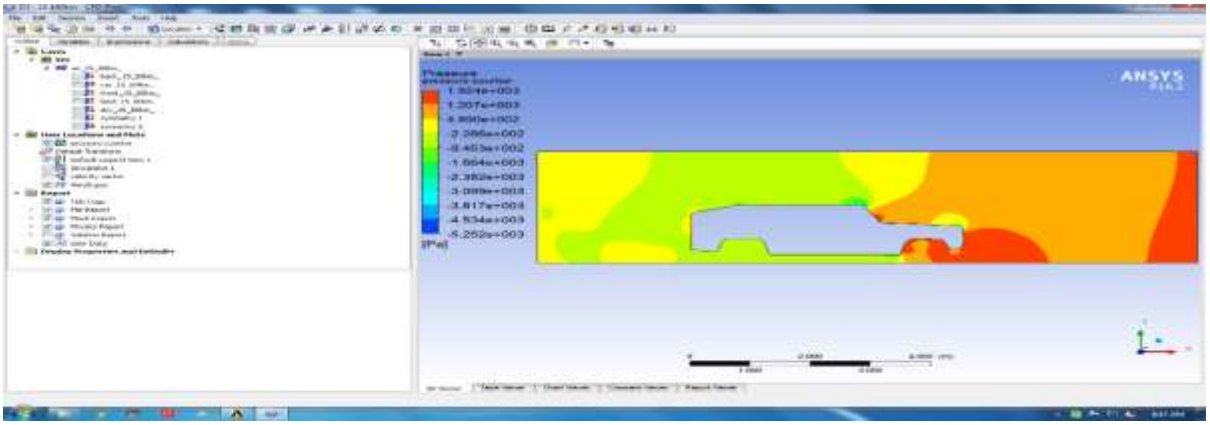
Pressure contours – (for velocity of flow of 140 kmph)



Original model of Land rover defender 110

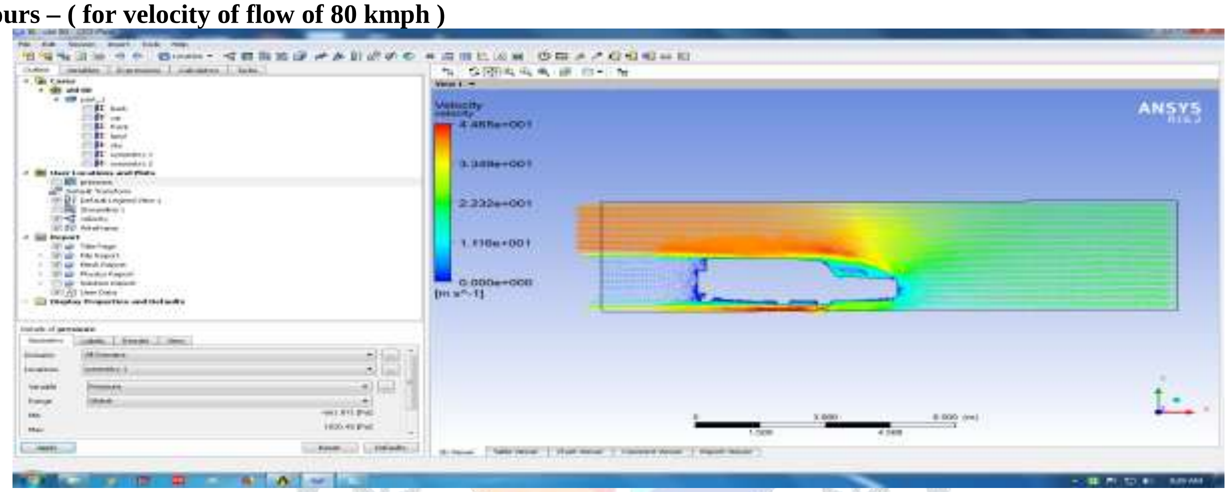


Land rover defender 110 with backlight angle 15 degree

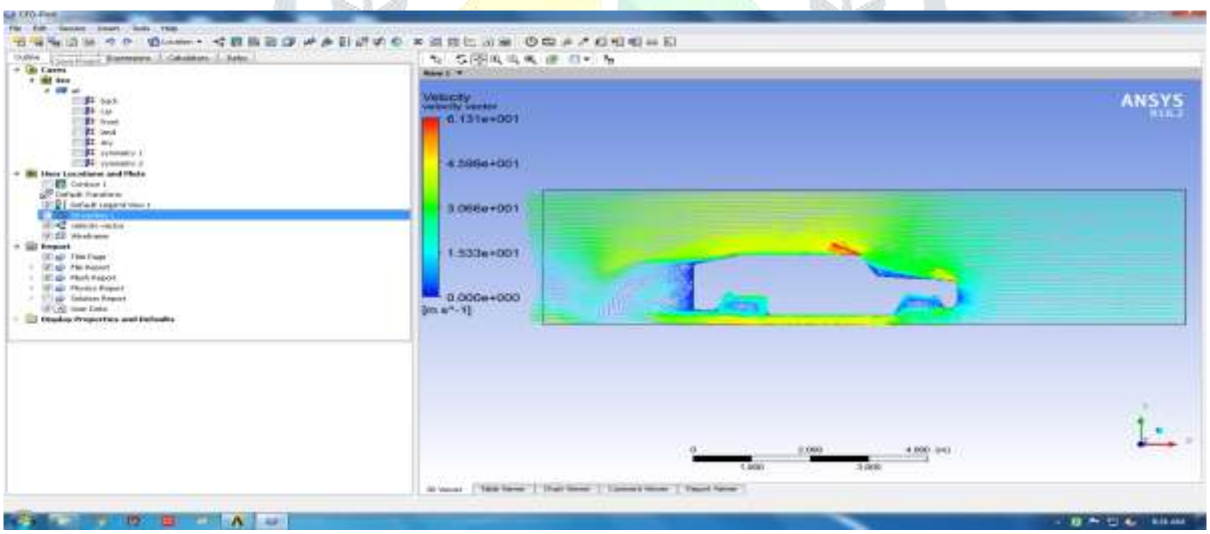


Land rover defender 110 with backlight angle 30 degree

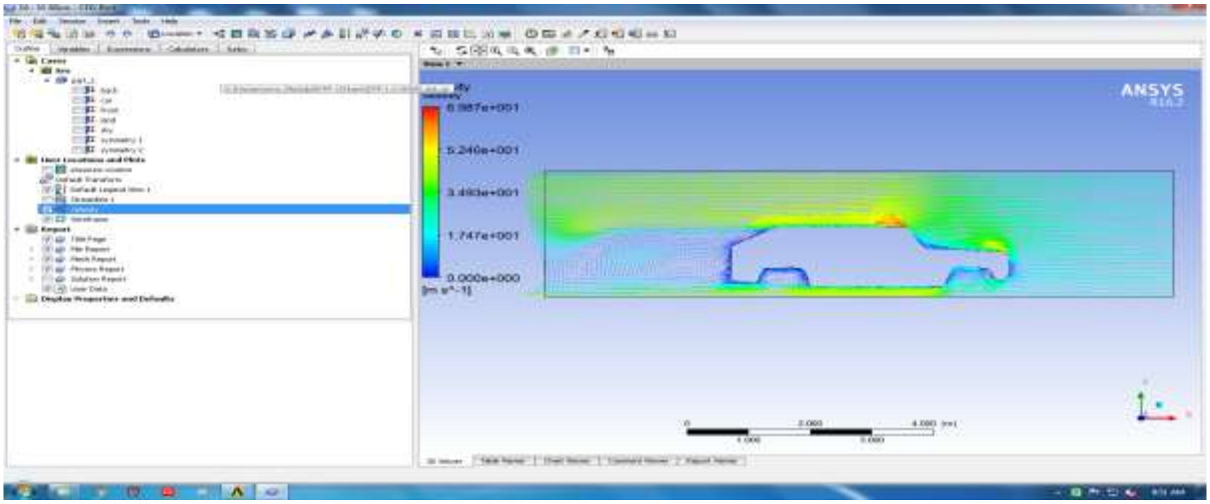
Velocity contours – (for velocity of flow of 80 kmph)



Original model of Land rover defender 110

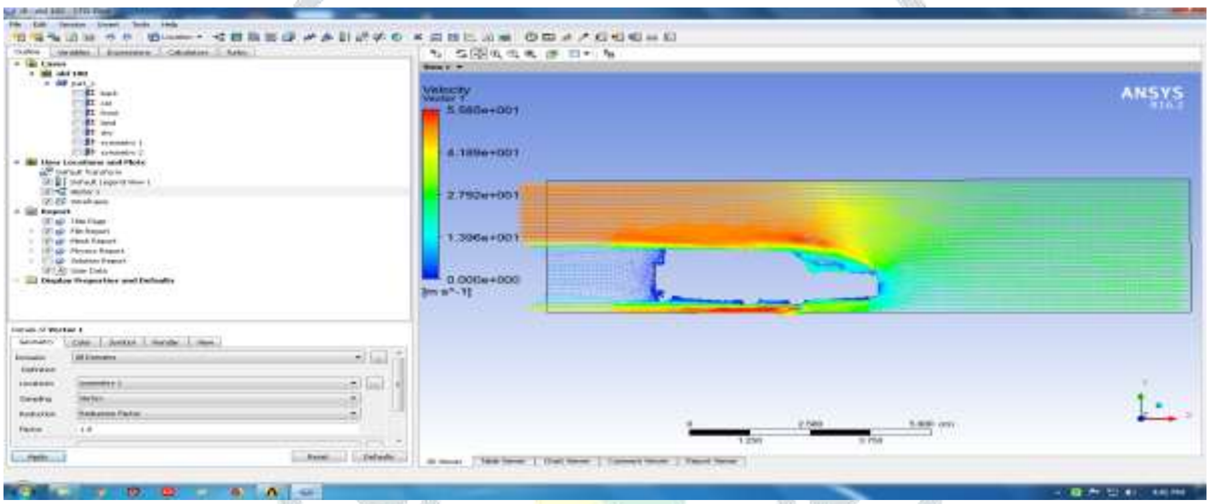


Land rover defender 110 with backlight angle 15 degree

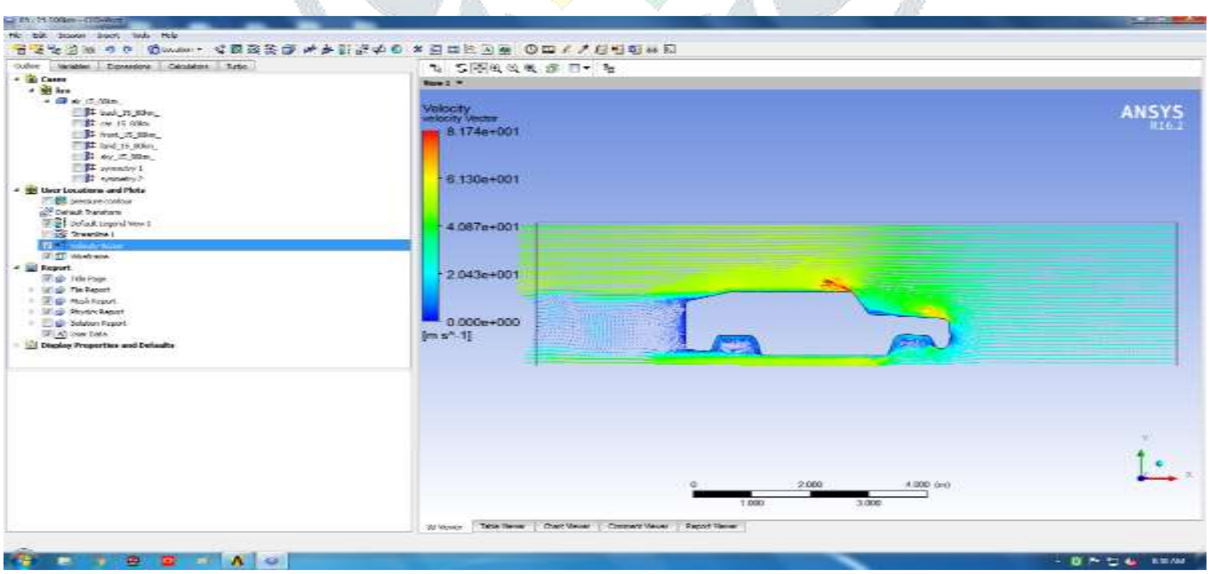


Land rover defender 110 with backlight angle 30 degree

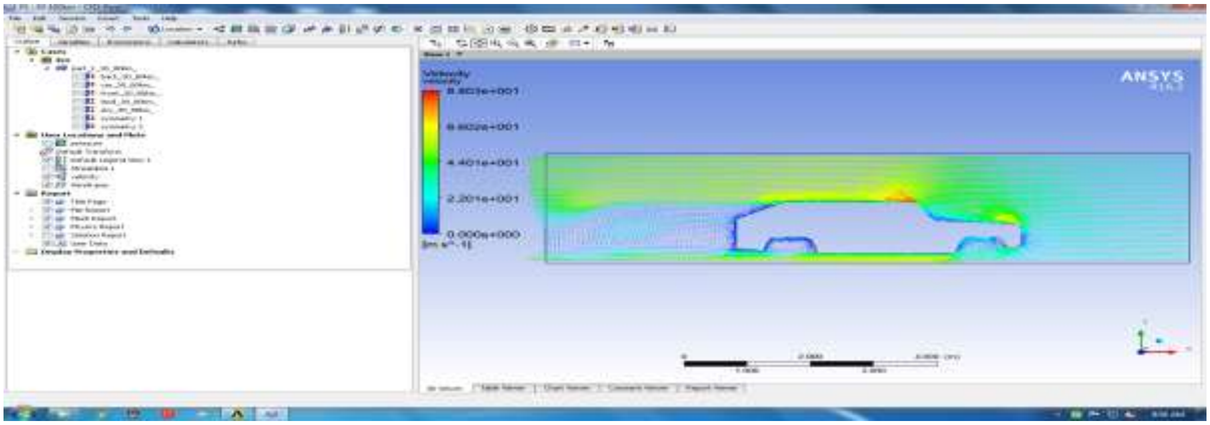
Velocity contours – (for velocity of flow of 100 kmph)



Original model of Land rover defender 110

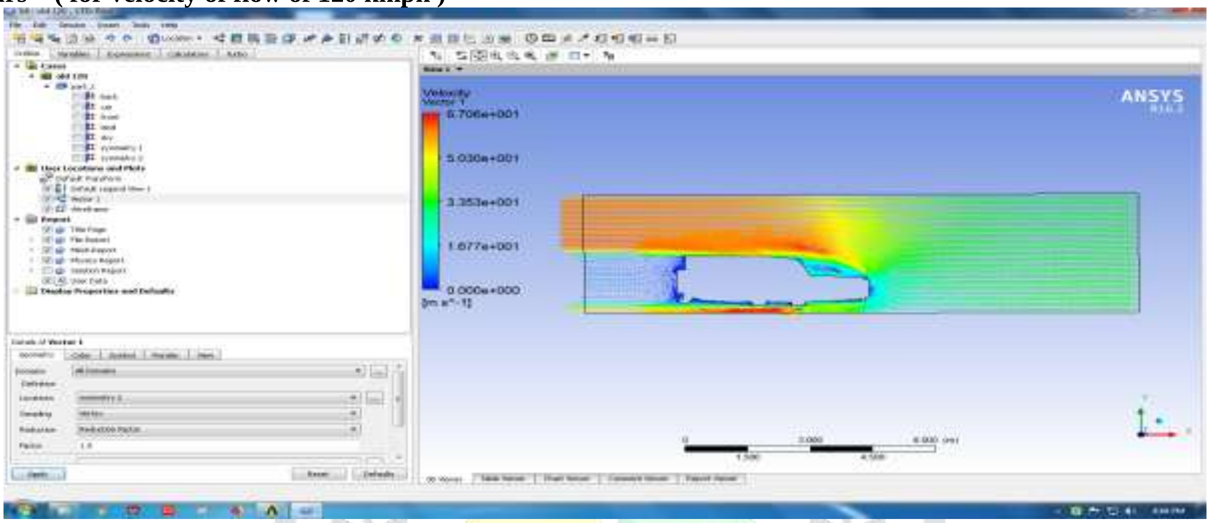


Land rover defender 110 with backlight angle 15 degree

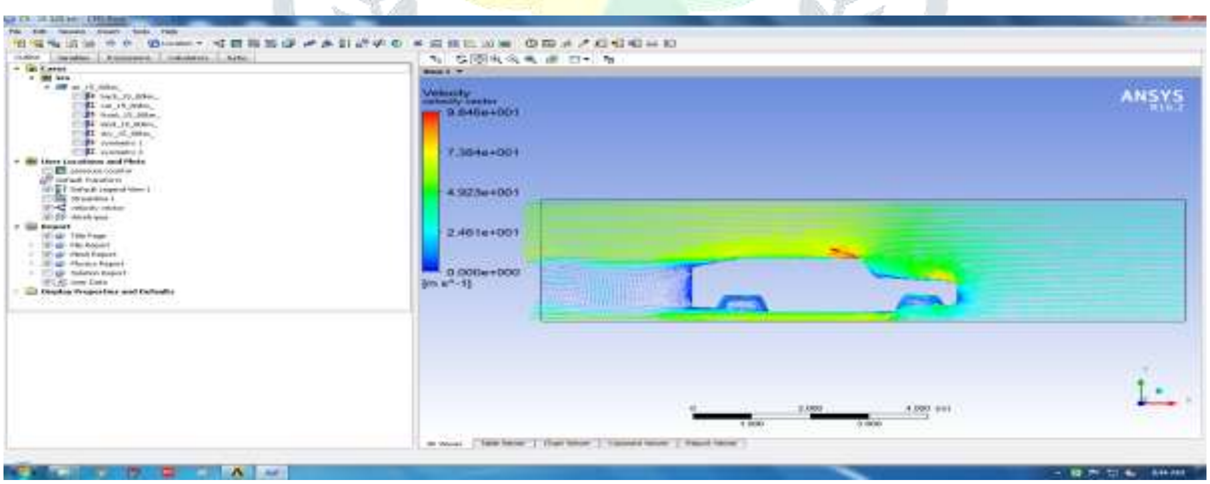


Land rover defender 110 with backlight angle 30 degree

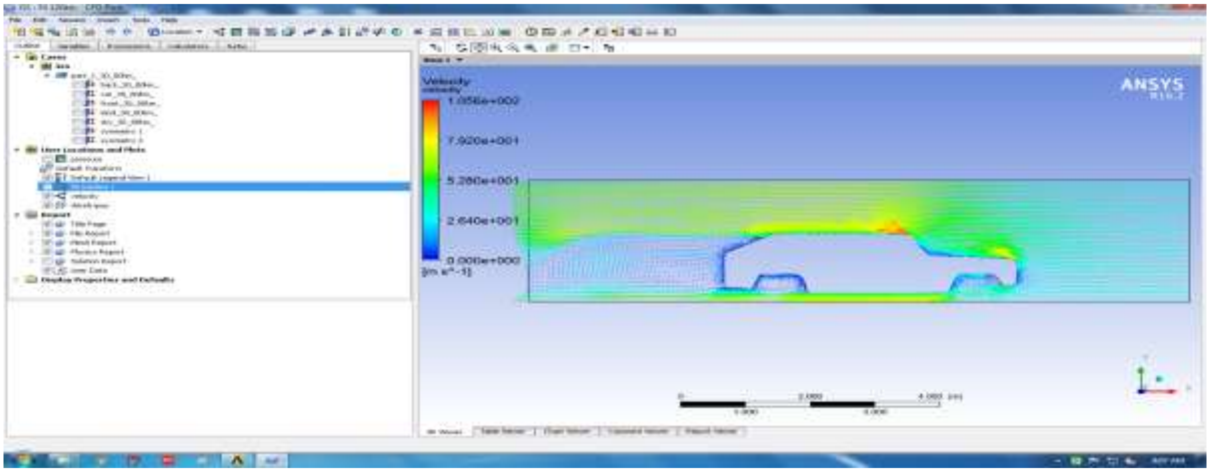
Velocity contours – (for velocity of flow of 120 kmph)



Original model of Land rover defender 110

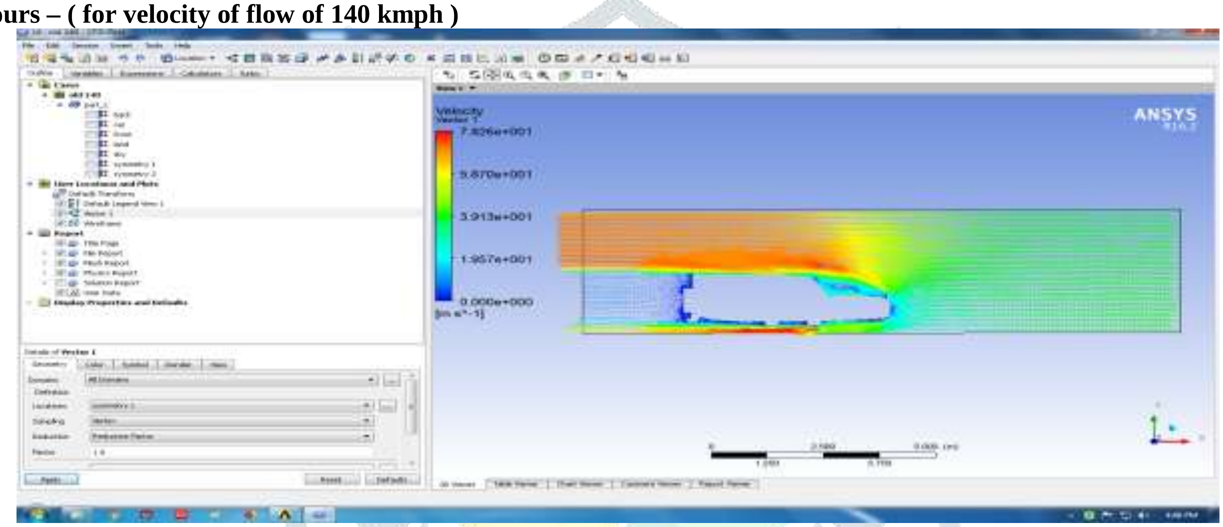


Land rover defender 110 with backlight angle 15 degree

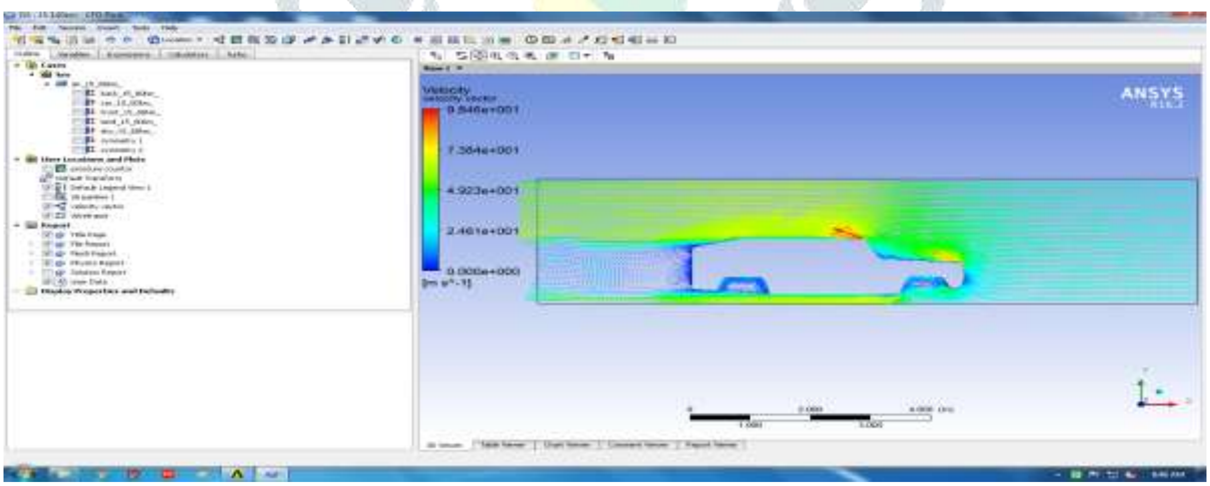


Land rover defender 110 with backlight angle 30 degree

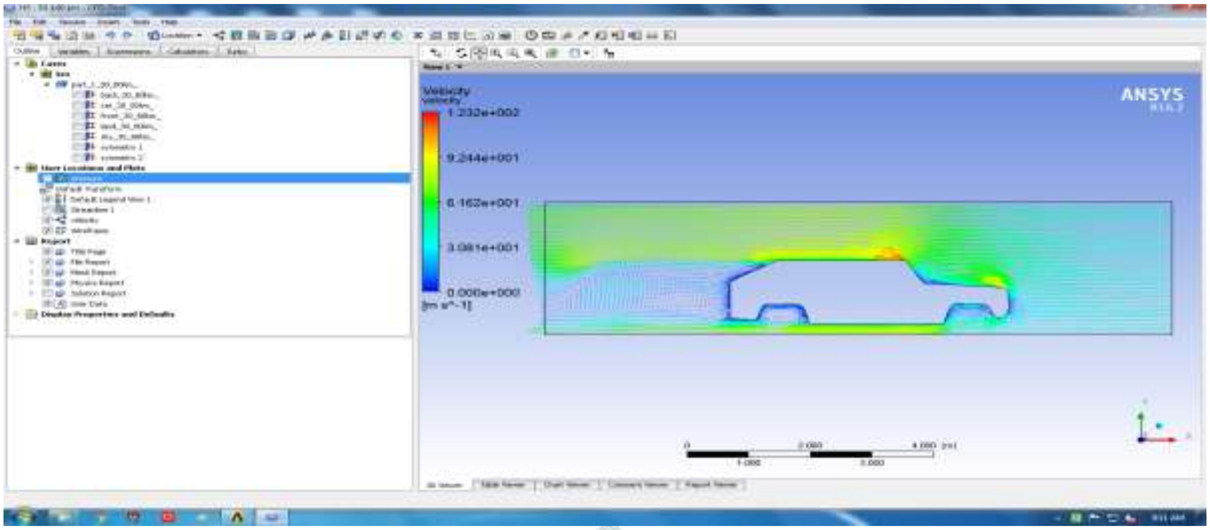
Velocity contours – (for velocity of flow of 140 kmph)



Original model of Land rover defender 110



Land rover defender 110 with backlight angle 15 degree



Land rover defender 110 with backlight angle 30 degree

CONCLUSIONS

Velocity of car (Km/Hr)	Pressure drag co-efficient (land rover defender old)	Pressure drag co-efficient (fast back) 15	Pressure drag co-efficient (square back) 30
80	2.4426721	1.99378	2.3513031
100	2.4413623	1.982857	2.3560686
120	2.447794	1.972242	2.2584223
140	2.44672	1.9691887	2.2416689

Velocity of car (Km/Hr)	Viscous drag co-efficient (land rover defender old)	Viscous drag co-efficient (fast back) 15	Viscous drag co-efficient (square back) 30
80	0.01297131	0.028517546	0.0341801
100	0.01257276	0.027649024	0.03338895
120	0.012256214	0.026974672	0.03225341
140	0.012001814	0.026426099	0.031462962

Velocity of car (Km/Hr)	total drag co-efficient (land rover defender old)	total drag co-efficient (fast back) 15	total drag co-efficient (square back) 30
80	2.4556434	2.0223008	2.3854832
100	2.453935	2.0105063	2.3894575
120	2.4600503	1.999217	2.2906757
140	2.458721	1.9956148	2.2731318

RESULTS FOUND

FROM THE RESEARCH CONDUCTED, IT WAS FOUND THAT A CHANGE IN THE EXISTING ANGLE OF THE SUV WOULD LEAD TO A DRASTIC IMPROVEMENT IN THE DRAG REDUCTION. THE DIFFERENT ANGLES PROVIDED RESULTED IN A CHANGE IN THE DRAG FORCE WHICH CAN BE SEEN IN THE EXPERIMENTAL RESULTS ABOVE.ANGLE 30 IS CRITICAL AT WHICH DRAG IS MAXIMUM AND IT IS FOUND THAT THE DRAG VALUE IS MINIMUM AT AN ANGLE 15 DEGREE .

I. REFERENCES

[1] BEWLEY T, LIU S. OPTIMAL AND ROBUST CONTROL AND ESTIMATION OF LINEAR PATHS TO TRANSITION. JOURNAL OF FLUID MECHANICS 1998; 365:305–349.

[2] CATHALIFAUD P, LUCHINI P. OPTIMAL CONTROL BY BLOWING AND SUCTION AT THE WALL OF ALGEBRAICALLY GROWING BOUNDARY LAYER DISTURBANCES. IN PROCEEDINGS OF THE IUTAM LAMINAR-TURBULENT SYMPOSIUM, V. SEDONA, AZ, U.S.A., SARIC W, FASEL H (EDS). 2002; 307–312

[3] FOURNIER G, BOURGOIS S, PELLERIN S, TA PHUOC L, TENSI J, EL JABI R. WALL SUCTION INFLUENCE ON THE FLOW AROUND A CYLINDER IN LAMINAR WAKE CONFIGURATION BY LARGE EDDY SIMULATION AND EXPERIMENTAL APPROACHES. 39E COLLOQUE D'A´ERODYNAMIQUE APPLIQUEE, CONTROLE DES ECOULEMENTS, MARS, PARIS, 2004; 22–24.

[4] R. K. ROY, DESIGN OF EXPERIMENTS USING TAGUCHI APPROACH: 16 STEPS TO PRODUCT AND PROCESS IMPROVEMENT, JOHN WILEY & SONS, NEW YORK, NY, USA, 2001.

[5] P. J. ROSS, TAGUCHI TECHNIQUES FOR QUALITY ENGINEERING, MC-GRAW HILL, NEW YORK, NY, USA, 1988.

[6] R. S. RAO, C. G. KUMAR, R. S. PRAKASHAM, AND P. J. HOBBS, "THE TAGUCHI METHODOLOGY AS A STATISTICAL TOOL FOR BIOTECHNOLOGICAL APPLICATIONS: A CRITICAL APPRAISAL," BIOTECHNOLOGY JOURNAL, VOL. 3, NO. 4, PP. 510–523, 2008.

[7] W. CUI, X. LI, S. ZHOU, AND J. WENG, "INVESTIGATION ON PROCESS PARAMETERS OF ELECTROSPINNING SYSTEM THROUGH ORTHOGONAL EXPERIMENTAL DESIGN," JOURNAL OF APPLIED POLYMER SCIENCE, VOL. 103, NO. 5, PP. 3105– 3112, 2007.

[8] S. CHEN, X. HONG, AND C. J. HARRIS, "SPARSE KERNEL REGRESSION MODELING USING COMBINED LOCALLY REGULARIZED ORTHOGONAL LEAST SQUARES AND D-OPTIMALITY EXPERIMENTAL DESIGN," IEEE TRANSACTIONS ON AUTOMATIC CONTROL, VOL. 48, NO. 6, PP. 1029–1036, 2003.

[9] W. ZHOU, X. ZHANG, M. XIE, Y. CHEN, Y. LI, AND G. DUAN, "INFRARED-ASSISTED EXTRACTION OF ADENOSINE FROM RADIX ISATIDIS USING ORTHOGONAL EXPERIMENTAL DESIGN AND LC," CHROMATOGRAPHIA, VOL. 72, NO. 7-8, PP. 719– 724, 2010.

[10] GHIASI, H., PASINI, D., AND LESSARD, L., CONSTRAINED GLOBALIZED NELDER-MEAD METHOD FOR SIMULTANEOUS STRUCTURAL AND MANUFACTURING OPTIMIZATION OF A COMPOSITE BRACKET. J.COMPOS MATER 42(7):717736, 2008 [22] LANFRIT, M., BEST PRACTICE GUIDELINES FOR HANDLING AUTOMOTIVE EXTERNAL AERODYNAMICS WITH FLUENT, VERSION 1.2, HTTP://WWW.FLUENTUSERS.COM, 2005.

[11] HUCHO, W.H., AERODYNAMICS OF ROAD VEHICLES, 4TH EDITION, SAE INTERNATIONAL, 1998

[12] SOVRAN, G., ET AL. (ED), AERODYNAMIC DRAG MECHANISMS OF BLUFF BODIES AND ROAD VEHICLES, PLENUM PRESS, NEW YORK, 1978

[13] HUCHO, W.H., SOVRAN, G., AERODYNAMICS OF ROAD VEHICLES, VOL. 25: 485-537,