Computations Study of Turbulent Flow around a Generic Car Body (Ahmed Body)

Rakshith K N[1] Mahesh T S[2]
[1]P.G. Student [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]Government Engineering College, Hassan

Abstract— The aerodynamic performances of the Ahmed body is explored statistically using ANSA software for pre-processing and for post processing and solving ANSYS FLUENT software and in the learning the K-ε (k-epsilon) turbulent model is recycled in demand to decrease at computational price at extraordinary Reynolds number. In the following analysis the four configurations with slant angle of 100, 200, 300 & 400 are measured. The correlation between aerodynamic drag and rear diffuser angle is precised and flow apparatus are evaluated and discoursed. In the explored the completely established turbulent flow in excess of Ahmed body and evaluate the effect of slant angle, four separate cases have been solved for different slant angle and results are compared. The results understand the better lift and drag for a body.

Key words: Ahmed Body, Aerodynamic Drag & Lift, Rear Slant Angle, K-Ɛ Turbulence Model

I. Introduction

The study of a standard (generic) car model called the Ahmed model. The principal pulverised vehicle geometry was established by Ahmed and measures its aerodynamic properties like stability, comfort and fuel consumption at high cruising speeds in wind tunnel experiment. In demand to learning the performance of freshly established turbulence model for multifaceted ground vehicle geometry of an Ahmed body. The body signifies a simple type, ground vehicle geometry of a bluff body type with slant back and rear back. The Ahmed body is made of a round part a movable slant plane placed in the rear end of the body to learning the partition phenomena at different angle and the rear slant plane and wake flow ahead the Ahmed body. It has industrialized into a standard point of reference Reynolds-averaged Navier strokes equation (RANS) model. The failure in predicting the base pressure is the major reason for the large discrepancy in drag prediction between experiments and numerical simulations.

As the wake flow behind the Ahmed body is the main contributor to the force, accurate prediction of the separation process. To simulate the wake flow accurately, resolving the near wall region using accurate turbulence model is highly desirable. This paper will study the effectiveness of K-ε (K-epision) turbulence models for the modelling of the flow over the Ahmed body and shows the behaviour of K-ε turbulence models. The physical activity of three-dimensional flow from one place to another a pulverized vehicle consumes developed an issue of major significance in the automobile production. Unique recognizable technique of filtering the fuel reduced of vehicles is to decrease aerodynamic drag by enhancing the body nature. Implementation of respectable aerodynamic design above literary restrictions involves a wide-ranging accepting of the flow singularities and particularly, just how the aerodynamics is subjective by variations in body nature. To achieve adjacent to particular factors which require existed achieved by experimental analysis to apart from the time and cost experienced used for that.

II. Methodology

A. Geometrical Modeling

The Ahmed Body model with province is demonstrated with SOLIDWORKS CAD 2010 modeling tool constructed on the constraints specified in figure 1. The slant angle at the rear end is various from 10˚, 20˚, 30˚ and 40˚ with the investigational model.

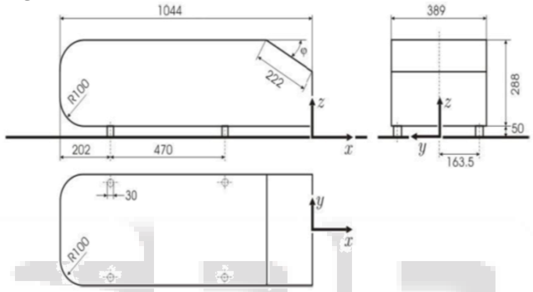


Fig. 1: Ahmed body with different slant angles (all dimensions in mm)

The Ahmed body, of length (L) 1044 mm, width (W) 389 mm, height (H) 288 mm and ground clearance is 50mm. The model is mounted on four tube-shaped supports with a diameter (f) of 30mm. The reference axis (X, Y and Z) is interrelated to the model. The dimensions of the computational domain or wind tunnel is 2×L in opposite of the ahmed body, 5×L after the geometry, 1.5×L on the edges and 2×L is the depth of the wind tunnel. The slant angle is the main variable model parameter in the experimental investigation. Initial design of the model is a design resolution and the geometry group dependent on the simplicity and the accessibility using whichever CFD modeling tools or extra design tools. The CAD model of the geometry is concentrated to ANSA software such that the numerous separations and interruptions are renovated and the prototypical is enclosed with a synthetic province that characterises the unique atmospheric or wind tunnel circumstances.

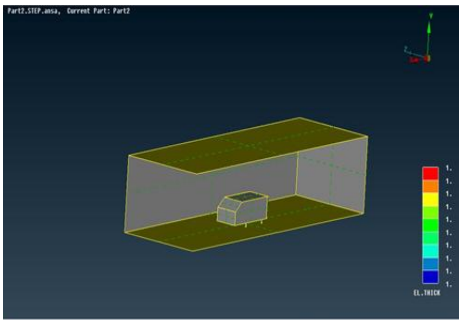


Fig. 2: Ahmed Model with Computational Province

B. Grid Generation

The Ahmed Body is discretized using Beta CAE-ANSA 13.0.1, an extremely well-organized general pre-processing tool used comprehensively for mesh generation in Manufacturing. The entire Ahmed body and wind tunnel is meshed with tetrahedron elements. Advancing front grid generation code is used to generate the grid accurately so as to capture velocity and pressure flow fields inside the province. The entire wind tunnel with Ahmed body is considered as a single fluid province. The quality of the element is fixed at 0.6 and mesh quality check and clean-up is performed for all models.

Ansys T-Grid is used to produce volume mesh privileged the computational province. Octree and Proceeding forward-facing process is recycled to create grid. The process delivers improved results in evaluation with the further obtainable codes. There are two significant characteristics of the meshing. The major is to determination the flow in the wake. To realize this, supplementary mesh controller entities are obtainable in the geometry.

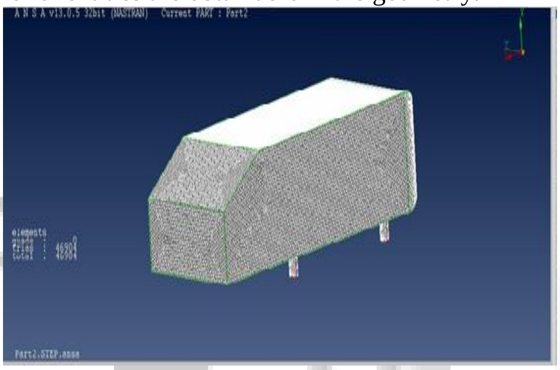


Fig. 3: Surface Mesh on Ahmed body

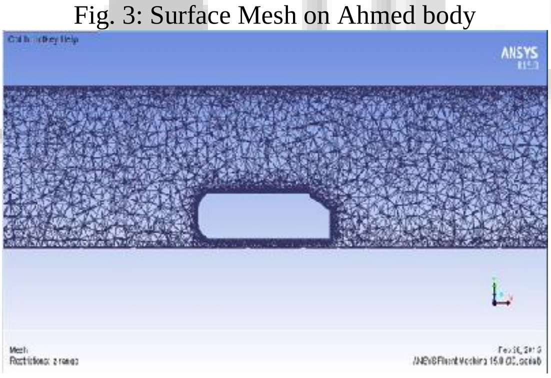


Fig. 4: Cut Plane along Y-Axis Showing Volume Meshing.

In the surface mesh provides a function template to compute a triangular mesh approximating a surface. In the volumetric meshes are polygonal representation of the interior volume of an object. In improving mesh quality of multi-material tetrahedral and hexahedral meshes while minimizing changes to the mesh charateristics and to the discrete boundary surfaces.

C. Governing Equations

Conservation of Mass and Momentum

Rate of increase of mass in fluid element equals the net rate of mass flow into the element.

$$\frac{\partial p}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$$

For incompressible fluid flow

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

Momentum is conserved in x, y, z direction & from the Newton's second law (F=ma) the momentum equations in all three direction is derived as,

$$\frac{\partial(\rho u)}{\partial t}+\nabla(\rho u V)=-\frac{\partial p}{\partial x}+\frac{\partial \tau xx}{\partial x}+\frac{\partial \tau yx}{\partial y}+\frac{\partial \tau zx}{\partial z}+\rho f x$$

Conserved in X direction

$$\frac{\partial(\rho v)}{\partial t}+\nabla(\rho v V)=-\frac{\partial p}{\partial y}+\frac{\partial \tau xy}{\partial x}+\frac{\partial \tau yy}{\partial y}+\frac{\partial \tau zy}{\partial z}+\rho f y$$

Conserved in Y direction

$$\frac{\partial(\rho w)}{\partial t}+\nabla(\rho w V)=-\frac{\partial p}{\partial z}+\frac{\partial \tau xz}{\partial x}+\frac{\partial \tau yz}{\partial y}+\frac{\partial \tau zz}{\partial z}+\rho f z$$

Conserved in Z direction

D. Pre-Processing

Fluid province abstraction and the grid generation is the most important portion of a pre-processing effort with respectable mesh quality. The Ahmed Body model with province is demonstrated with Solid works CAD modeling tool and analysis the file in ANSA. The analysis the file should perfect as case file or data file or case and data file. In this we have to analyse case and data file and scaling the grid. Checking the skewness of the grid and describing the models. Model must describe whether it is steady or unsteady and whether it is sticky. The model is well-defined here is steady and viscous. The module that explains the CFD problem is called the Solver. It creates the necessary outcomes in a non-interactive/collection process. A CFD problem is solved as follows

- Describing the material properties
- Boundary condition of that material and control over the body.
- Initialize and refinement is merge free nodes, deleted unused nodes and improve surface mesh.
- In monitoring the surface by pressure drop and temperature drop in report type area weighted average and field variable is pressure and total pressure.

E. Post Processing

The post-processor is the module used to investigate, imagine and current the effects interactively. Post-processing comprises something from attaining fact standards to multifaceted animated arrangements. Some important features of post processing are Conception of the geometry and control volumes, Vector designs presenting the direction and magnitude of the flow, Animation and Measureable mathematical controls.

F. Boundary Condition

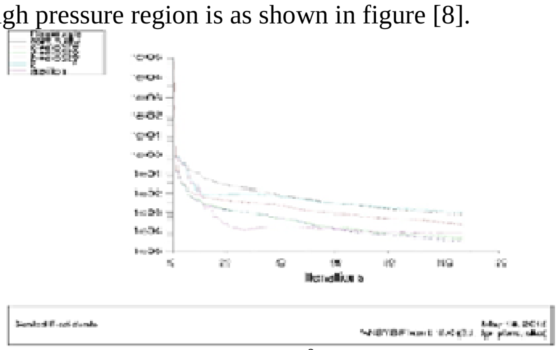
Pre-processor for Ansys FLUENT the fluid domain is defined. In this study there is no solid domain involved. The flow in this study is turbulent, hence K-epsilon (K-ε) model is chosen. The boundary conditions are specified in Ansys FLUENT pre-processor and then the file is distributed to the solver. The Reynolds number, based on the length of the model, is 2.78×10^6. Besides, following Conditions were applied for solving the case: Material: Air, Inlet is specified with below details

- Inlet velocity – 40 m/s
- Boundary type – Velocity inlet
- Outlet is specified with below details
- Outlet pressure – 0 Pa

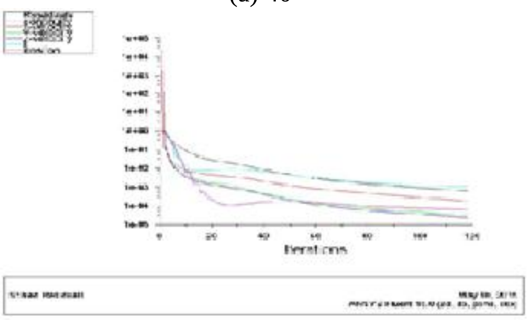
- Boundary type – Pressure outlet
- Car surface is specified as walls
- Road surface is specified as wall with roughness constant of 0.5.
- Wind tunnel sides specified as Symmetry.

III. Results and Discussions

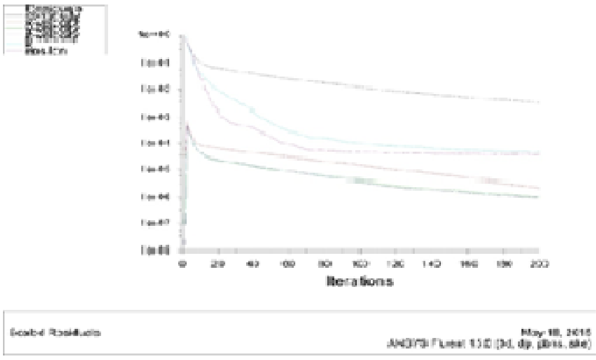
In the automobile manufacturing gives the significant reputation to study the three dimensional pulverised vehicle. It's improving the fuel economy and enhancing the body nature to decrease the aerodynamic drag of vehicle. In wind tunnel experiment measure the aerodynamic drag coefficient with four different slant angles. The figure 1 shows the residual for the velocity X, Y and Z directions for rear slant angle 10^0, 20^0, 30^0 & 40^0. Residuals are the difference valves between the consequent iterations. The residual tends to decrease near to 0 and the solution starts converging is as shown in figure [5]. The wake region and drag around the Ahmed body can be observed by observing the velocities around the body. The increasing the different rear slant angles in the junction position of the upper and lower flow tend to move upward and near to the truck-deck and Which may decrease the recirculation region and the drag. The time averaged velocity vector in the longitudinal central section at four different rear slant angles are shown in figure [6]. for rear slant angles by using standard K-ε turbulence model gives the result of the flow is fully separated over the slant and strong vortices do not form and also indicates the velocity is almost zero at the nose of ahmed body and the velocity is minimum at the slant side due to formation of vortices is shown in figure [7]. The pressure at front is none as there is obstacle and there will be stagnation point and flow meters of the ahmed body the pressure will be steady and equal at all points but near the rear slant angles the pressure will be negative which will pull the vehicle and the fluid known as drag and also the fluid will tend to more upwards that inserts will create the vehicle to lift at high speeds. As the fluid tries to more from low pressure region to high pressure region is as shown in figure [8].



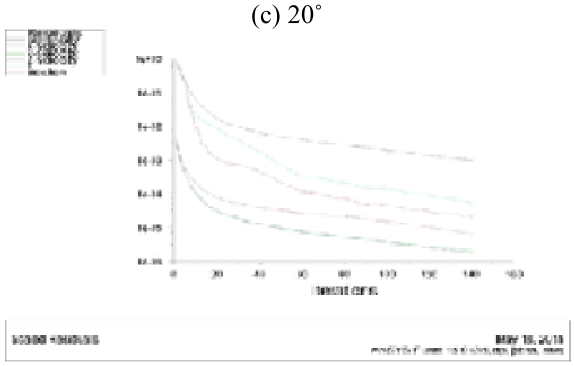
(a) 40^0



(b) 30˚

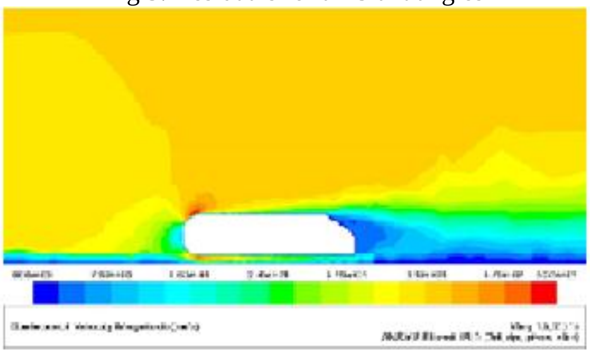


(c) 20˚

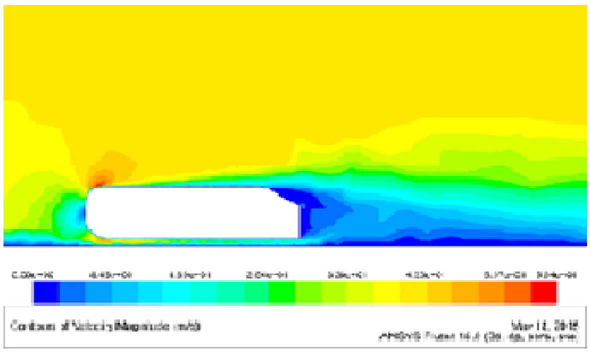


(d) 10˚

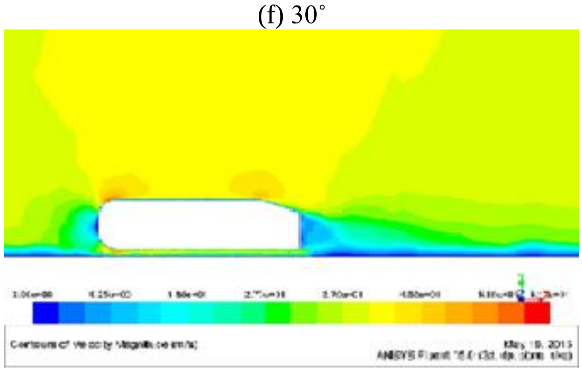
Fig 5: Residuals for all Slant angles



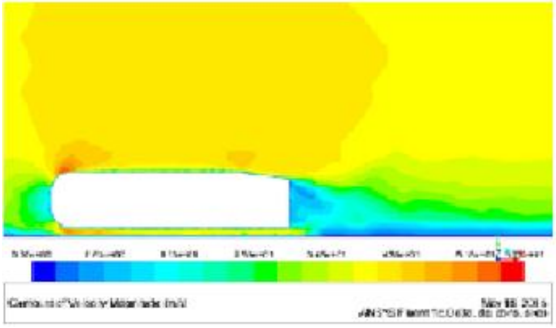
(e) 40^0



(f) 30˚

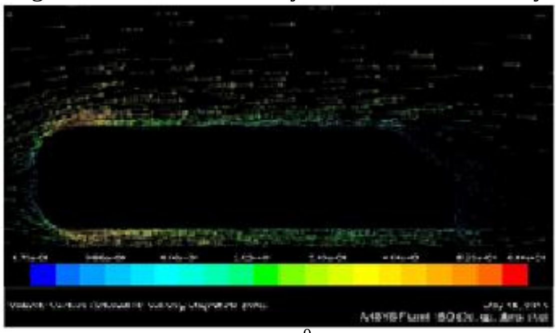


(g) 20˚

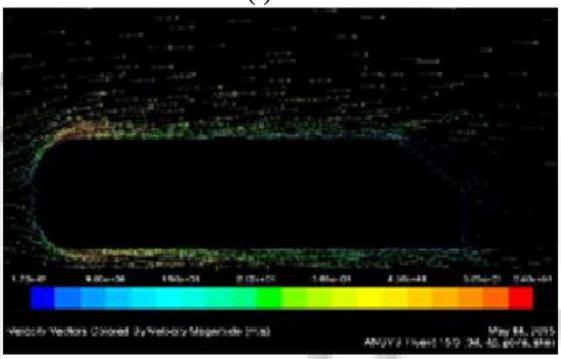


(h) 10˚

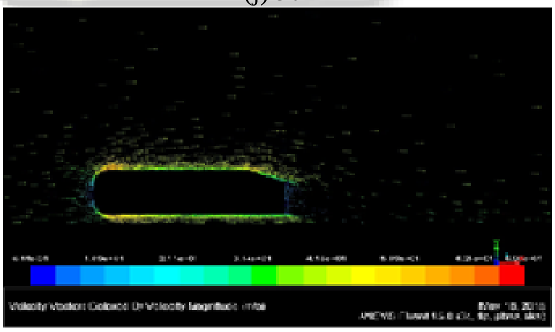
Fig 6: Contours of Velocity over the Ahmed Body



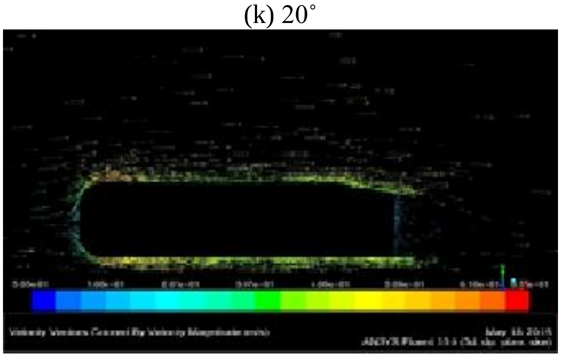
(i) 40^0



(j) 30˚

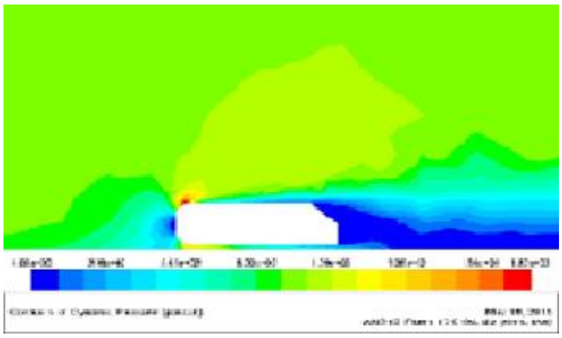


(k) 20˚

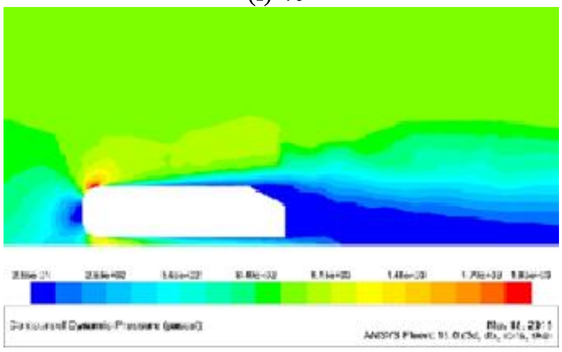


(l) 10˚

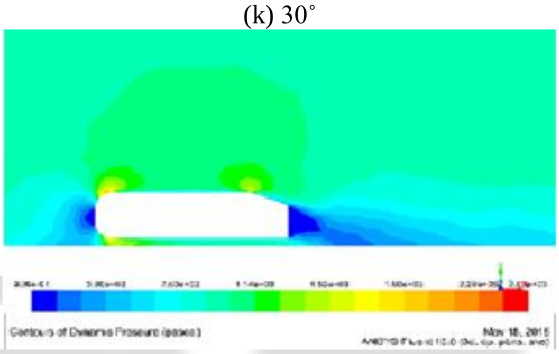
Fig. 7: Velocity vectors colored by velocity magnitude



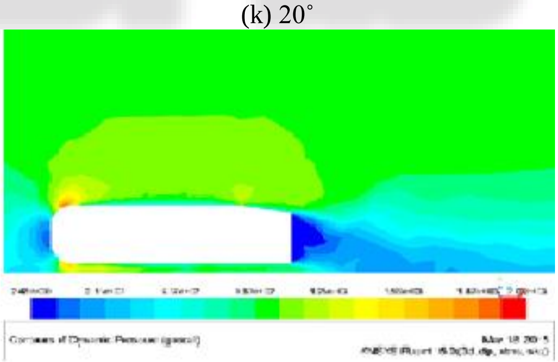
(l) 40˚



(k) 30˚



(k) 20˚



(l) 10˚

Fig. 8: Contours of Dynamic Pressure

IV. Conclusion

In this paper the analysis of four different rear slant angles are concluded as follows,

- The aerodynamic drag of a vehicle can be influenced by rear slant angle with increasing rear slant angle, the drag first decreases and then increases.
- From the below results we can conclude that the optimised angle in all the angles given above, the drag and lift minimal for the better working of an Ahmed body with good aerodynamic characteristics is 30^0 angles.

– It creates minimal drag which means lesser power is required for a vehicle to overcome the drag by this the forward thrust of a vehicle will be more. Also the lift being minimum of all the above in 30^0 allows the vehicle to operate freely with good aerodynamic characteristics.

– Slant Angles	Drag	Lift
40˚	0.451	0.312
30˚	0.311	0.265
20˚	0.511	0.42
10˚	0.6	0.45

Table 1: Coefficient of Drag & Lift of All Rear Slant Angles

ACKNOWLEDGMENT

I would like to thank my guide Mahesh T S and their encouragement, guidance and support. I would like to thank my classmates for their support. Also I would like to thank Department of Mechanical Engineering, Government College of Engineering for support.

REFERENCE

[1] Ahmed, S.R. Ramm G., and Faltin, G., “Some Salient Features of the Time-Averaged Ground Vehicle Wake” SAE Technical Paper-1984
[2] Rehan Salahuddin Khan, Sudhakar Umal., “CFD Aerodynamic Analysis of Ahmed Body” Volume 18 Number 7- Dec 2014
[3] W. Meile, G. Brenn, A. Reppenhagen, B. Lechner, A. Fuchs C. “Experiments and numerical simulations on the aerodynamics of the Ahmed body” Vol. 3 (1) March 2011
[4] Chauhan Rajsinh B. and Thundil Karuppa Raj R. “Numerical Investigation of External Flow around the Ahmed Reference Body Using Computational Fluid Dynamics”. 30th March 2012
[5] Chenguang Lai, Yasuaki Kohana., “experimental and numerical investigation on the influence of vehicle rear diffuser angle on aerodynamic drag and wake structure”. June 28 2010
[6] Emmanuel Guilmineau, “Computational study of flow around a simplified car body”. 6 August 2007
[7] Václav Uruba, Ondrej Hladik., “On the ahmed body wake”. Institute of Thermomechanics AS CR, Prague, October 21 - 23, 2009
[8] D.E. Aljurea, O. Lehmkhul., “Flow and Turbulent Structures around Simplied Car Models” March 5, 2014

