iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL
FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 Issue: IV Month of publication: April 2018

DOI: http://doi.org/10.22214/ijraset.2018.4655

www.ijraset.com
Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue IV, April 2018- Available at www.ijraset.com

Aerodynamic Analysis of Side Body Panels of Lancer EVO VIII

S.R.Benin[1], M. Sumathi [2], A. Jacob Moses [3], Renjin J Bright[4], A. Satheesh Kumar [5]
[1, 3 ,4 5] Assistant Professor, [2] Professor, Department of Mechanical Engineering, Jeppiaar SRR Engineering College, Chennai.

Abstract: The purpose of this paper is to modify the side camber of the car and analysing the drag coefficient by taking various models without creating a prototype. In this the side camber is increased with various models. . The aim of this project is the presentation of a series of aerodynamic improvements for the cars, which reduce the aerodynamic resistance of these vehicles and consequently result in a positive impact on fuel consumption which is substantially reduced. Initially the car model is created by using pro-E software. To make the analysis the computational fluid dynamics (CFD) methodology using Ansys has been used since it allows simulating some geometries and modifications of the geometry without making physical prototypes that considerably increase the time and the economic resources needed.
Keywords: Drag Force, Lift Force, camber, Computational Fluid Dynamics

I. INTRODUCTION

The technology to develop a vehicle body with high strength, high rigidity and lightweight has become increasingly necessary for the automotive industry to improve crashworthiness, reduce CO_2 emissions and to reduce the fuel consumption of the vehicle. At present the fuel consumption causes major influences in the automotive industry. The aim of this project is the presentation of a series of aerodynamic improvements for the cars, which reduce the aerodynamic resistance of these vehicles and consequently result in a positive impact on fuel consumption which is substantially reduced. The growing demand for further reduced fuel consumption as a means of combating global warming, oil-resource depletion, and other environmental problems has created a critical need for automotive manufacturers to produce lighter vehicles. Measures to minimize aerodynamic drag are vital for fuel economy. Indeed, aerodynamic drag at high speeds causes most of a vehicle's running resistance. Although the body reflects comprehensive measures to minimize aerodynamic drag, it also has attractive styling and permits efficient packaging. The combination of superior aerodynamic efficiency, styling, and packaging is one of the most significant attributes. Aerodynamics is the branch of dynamics that treats the motion of air (and other gaseous fluids) and the resulting forces acting on solids moving relative to such fluid. Fluid flow over a moving vehicle falls in three categories.

A. Flow of air around the vehicle.
B. Flow of air through the vehicle.
C. Flow within the vehicle's machinery.

The first two fields constitute the aerodynamics are closely related. Thus vehicle aerodynamics is the study of air, its effect over the moving vehicle, the shape of the vehicle and its reaction to the flow field around it during the vehicle motion. The major constituents of aerodynamic effect are drag, lift, side wind, pitching moment, yawing moment, and rolling moment.

II. LITERATURE SURVEY

N. A. Ahmed[7] predicted that the aerodynamic body can be generated by reducing the total drag. The commonly called momentum method for total drag determination. In this method, theoretical analysis and experimentation has been combined to produce a powerful tool for efficient and cost-effective aerodynamic investigation of total drag. In this he used the momentum method to find the total drag.

Elias E. Panagiotopoulos, Spyridon D. Kyparissis[1] predicted the separation movements of the external store weapons carried out on military aircraft wings under transonic Mach number and various angles of attack is an important task in the aerodynamic design area in order to define the safe operational-release envelopes. They analyzed the store separation from the aircraft. They analsed the store separation with six DOF with the help of the computational fluid dynamics. Because normally this store separation test is done by the flying test and it was too expensive. This study has shown that CFD with unstructured dynamic meshing can be an effective and successful tool for modelling transonic store separation at various angles of attack.

©IJRASET (UGC Approved Journal): All Rights are Reserved

M. J. Yelland and B. I. Moat, R.W. Pascal, D. I. Berry[4] conducted an CFD analysis on the two ships in order to place the anemometer. The presence of the ship causes the airflow to a particular instrument site to be either accelerated or decelerated, displaced vertically, and sometimes deflected slightly in the horizontal. Although recognized for some time, it is only recently that the problem has been addressed using three-dimensional computational fluid dynamics (CFD) models to simulate the flow over particular ships, quantify the effects of flow distortion, and hence correct the ship-based measurements. Wind tunnel studies are costly and time-consuming. So they preferred CFD.

R. Verzicco, M. Fatica, G. Iaccarino, P. Moin§, and B. Khalighi[11] were conducted an experiment on the drag reduction. The large eddy simulation is used to simulate the unsteady separated flow around a bluff body .They experimented in numerical method also with the help Reynolds effects. They attached two drag reduction devices also. There are a cavity at the rear side and a boat tail at the base are attached to the vehicles. So they experimentally proved that by altering the shape and attaching a device will also reduce the drag.

Fu-Hung Hsu, Roger L. Davis[3] were conducted an experiment on a bluff body such as tractor-trailors for drag reduction. They changed the size of the tractor by add on humps and also they add a curved boat tail flaps. The computational fluid dynamics in the form of unsteady Reynolds averaged navier stokes equation and detached eddy simulation were used to determine the viable design strategies. Thus the results from the optimized design were shown to have a 50.9% reduction in drag coefficient.

Yuichi Kuya, Kenji Takeda, Xin Zhang, Scott Beeton, and Ted Pandaleon[6] were investigated the flow separation control using vortex generators on an inverted wing in ground effect. Counter-rotating and co-rotating rectangular-vane type vortex generators are tested on the suction surface of the wing. They done this experiment in a wind tunnel. From this work they shown that the vortex generators is used for controlling flow separation with a improvement in downforce for relatively low drag penalty.

Joseph Katz[9] conducted an aerodynamic analysis in race cars. He explained clearly about the factors that involves in aerodynamics. Typical design tools such as wind tunnel testing, computational fluid dynamics, and track testing, and their relevance to race car development, are discussed as well. The results show that the computational fluid dynamics are well suited for aerodynamic analysis.

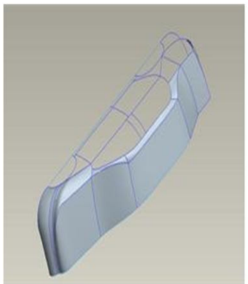


Fig 1. Wire frame model

Kyoungwoo Park, Byeong Sam Kim, Juhee Lee, and Kwang Soo Kim[5] Predicted the aerodynamic characteristics and shape optimization of airfoil under the ground effect have been carried out by integration of computational fluid dynamics and the multi objective Pareto-based genetic algorithm. NACA0015 airfoil is considered as a baseline model. The main flow characteristics around an airfoil of WIG craft are lift force, lift-to-drag ratio and static height stability (H.S). These are all calculated by the CFD solver.

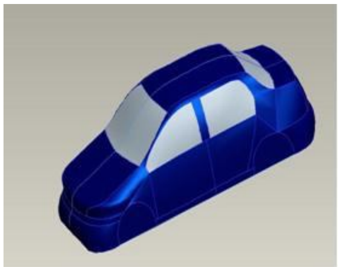


Fig. 2 Solid Car Model

©IJRASET (UGC Approved Journal): All Rights are Reserved

Ramon Miralbes Buil, Luis Castejon Herrer[10] conducted an aerodynamic analysis of a vehicle tanker by CFD. They changed the vehicle tanker geometry. They altered the front line and the rear side in order to reduce the drag coefficient. The result shows that the CFD programs are very useful tools for performing aerodynamic modifications in vehicles at a very low cost, and that they allow the achievement of satisfactory results in a short period of time.
Xin Zhang, Willem Toet, Jonathan Zerihan[2] were analysed the Ground Effect Aerodynamics of Race Cars in particular open wheel race cars. There are three aerodynamic simulation test available. They are full scale track tests, CFD simulation, and wind tunnel model tests. CFD is playing an increasingly important role in ground effect aerodynamics.
Sinisa Krajnovi.c and Lars Davidson[8] were conducted an experiment on Large-Eddy Simulation of the Flow Around a Ground Vehicle Body. The large eddy simulation is used to simulate the unsteady separated flow around a bluff body. The experiment in done and it is compared with the numerical values.
Wolf- Heinrieh Hucho[12] explained the impact of sides in drag coefficient on cars. By providing camber in the top-view tapered and rear end boat tailed contour the sides of the cars changed to a streamlined shape. Then a stream of airflow over the car gives the drag force and drag coefficient. By varying the geometry of the car change in drag coefficient is analyzed. Thus the sides of car are taken as a source of drag force.
Sun Yongling, Wu Guangqiang and Xieshuo[13] gives the procedural steps to solve the aerodynamic problem in a CFD software package. From the output pressure and velocity field the drag coefficient and drag forces are calculated for a particular velocity.

III. 3D NUMERICAL ANALYSIS

A. Basic Steps in ANSYS

The following are the basic steps used in the ANSYS software for the analysis of the car model,

1) Preprocessing phase
2) Solution phase
3) Post processing phase

B. Preprocessing Phase

Preprocessor consist the input of a problem to an ANSYS program by means of a graphics user interface and the subsequent transformation of this input into a form for use by the solver. The user activities at the pre-processing stage involve

C. Building a Model

Building a finite element model requires more or your time than any other part of the analysis. First, specify the job name and analysis title. ANSYS includes the title on all graphics displays and on the solution output. Utility menu> file> change job name > Main menu >Preferences > Flotran CFD

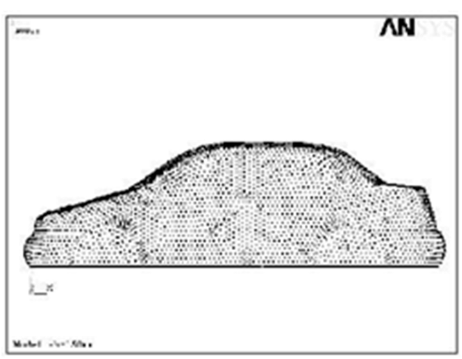


Fig.3 Nodes attached to the surface of the car

D. Defining Element Types

While defining the actual elements, select the appropriate type from the preprocessor> element type and select the FLUID142 element which is suitable for three-dimensional computational fluid dynamic analysis. Main menu> Preprocessor> Elementtype> Add/Edit/Delete> FLUID142.

©IJRASET (UGC Approved Journal): All Rights are Reserved

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue IV, April 2018- Available at www.ijraset.com

E. Defining FLOTRAN Setup

Flotran setup is the special solver in the ANSYS program to solve the computational fluid dynamics problem either two or three-dimensional. In this setup initially the type of flow laminar or turbulent, compressible or incompressible is defined. Then the number of iterations to solve the problem is defined. Then the fluid properties are defined. In this problem the fluid used is air at 20ºC.BMain menu> Preprocessor> Flotran setup> Solution option> steady state, laminar, incompressible. Main menu> Preprocessor> Flotran setup> Fluid properties> AIR-SI.

F. Creating the Model Geometry

Car model designed in the Pro/E software is imported in the ANSYS package through IGES file format. Then create the block, which is three times larger than the main dimensions of the car (i.e. length, height, breadth). Then subtract the car volume from the block using Boolean operations. Utility menu> file> import> IGES> model Main menu> Preprocessor>Modeling> Create>Volume> Block> By Dimension.

G. Meshing

Before meshing we define the default attributes. For example the minimum edge lengths of the area are to be set.
Main menu> Preprocessor> Meshing> Mesh Tool.

H. Creating the Model Geometry

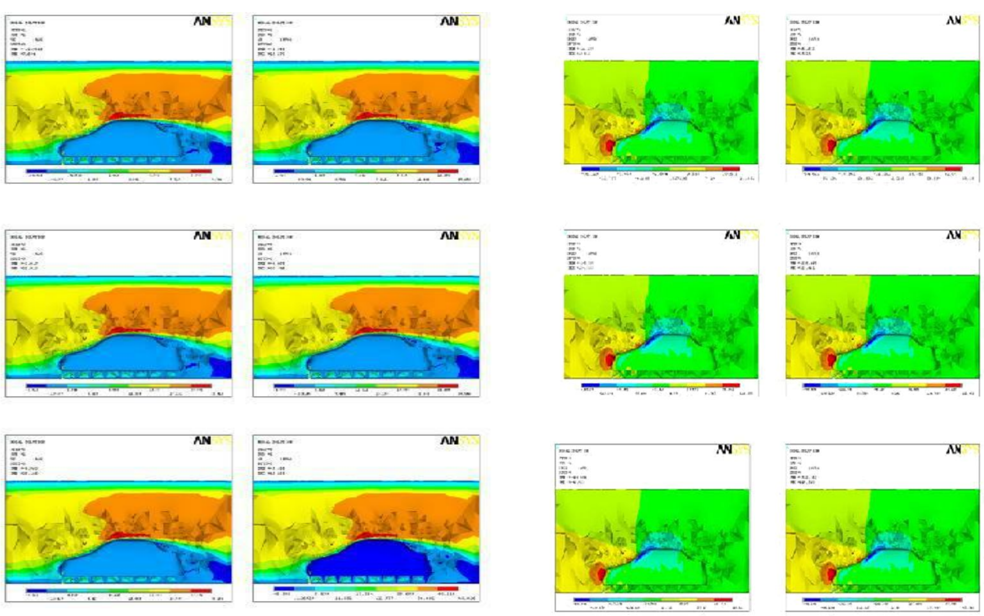
Car model designed in the Pro/E software is imported in the ANSYS package through IGES file format. Then create the block, which is three times larger than the main dimensions of the car (i.e. length, height, breadth). Then subtract the car volume from the block using Boolean operations. Utility menu> file> import> IGES> model Main menu> Preprocessor>Modeling> Create>Volume> Block> By Dimension.

I. Meshing

Before meshing we define the default attributes. For example the minimum edge lengths of the area are to be set.
Main menu> Preprocessor> Meshing> Mesh Tool.

IV. RESULTS AND DISCUSSIONS

Based on the above process the drag coefficient (CD), Drag force (D) and fuel consumption rate for different models of the car with varying camber are predicted. The results are listed below and the comparisons are plotted in a graph.



(a) (b)

Fig.4 (a) Velocity distributions (Vx) for model at different velocities in m/s
(b) Pressure distributions for model at different velocities in m/s

V. CONCLUSION

From figure (g) it is clearly defined that providing camber to the vehicle gradually reduces the percentage of fuel consumption for the car model by providing camber of 200 mm the drag force is reduced by 5%.

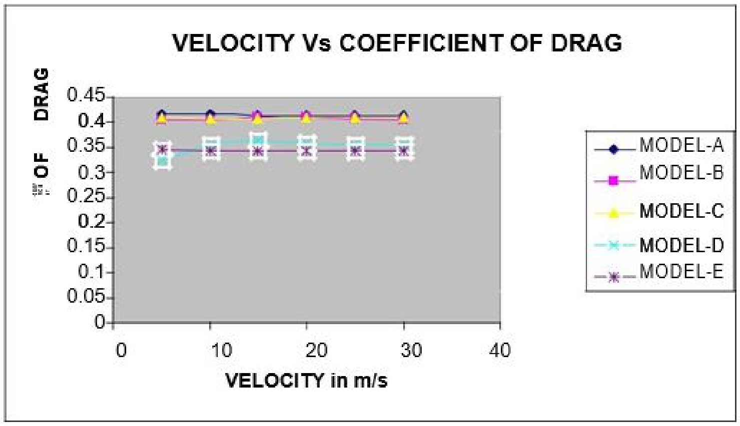


Fig.5 Comparison of drag coefficient for different car models

VI. VALIDATION

From the analysis carried for the different models it is inferred that providing camber gradually reduces the coefficient of drag for the models.

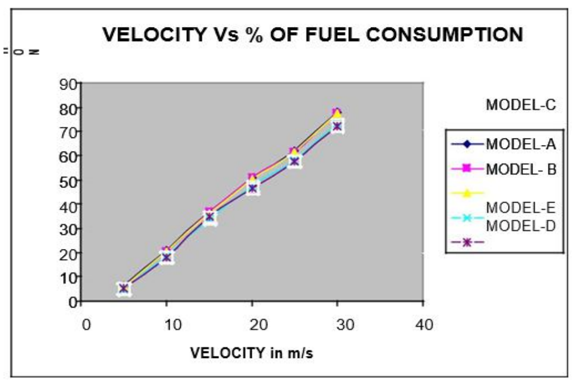


Fig.6 Comparison of Aero Dynamic drag force for different car models

As the camber increased from 0 mm to 200mm the coefficient of drag is reduced from 0.41 to 0.34. The change in drag coefficient is in the order of 0.01. From the drag coefficient the drag force for different models are calculated and the percentage reduction in drag force is around 16%. Finally the percentage of fuel consumption by the engine to overcome the drag force is calculated. The fuel consumption is reduced to 5% by providing the camber.

REFERENCES

[1] Elias E. Panagiotopoulos,Spyridon D. Kyparissis(2002),"CFD Transonic Store Separation Trajectory Predictions with Comparison to Wind Tunnel Investigations" International Journal of Engineering (IJE), Volume (3): Issue

[2] Xin Zhang, Willem Toet, Jonathan Zerihan (2006), "Ground Effect Aerodynamics of Race Cars" Vol. 59 / 33 by applied mechanical reviews.

[3] Fu-Hung Hsu, Roger L. Davis (2010), "Drag Reduction of Tractor-Trailers Using Optimized Add-On Devices" Journal of Fluids Engineering Vol. 132 / 084504-1.

[4] M. J. Yelland and B. I. Moat, R.W. Pascal, D. I. Berry (2002), "CFD Model Estimates of the Airflow Distortion over Research Ships and the Impact on Momentum Flux Measurements" journal of atmospheric and oceanic technology vol. 19.

[5] Kyoungwoo Park, Byeong Sam Kim, Juhee Lee, and Kwang Soo Kim (2009), "Aerodynamics and Optimization of Airfoil Under Ground Effect" International Journal of Science, Engineering and Technology vol 52.

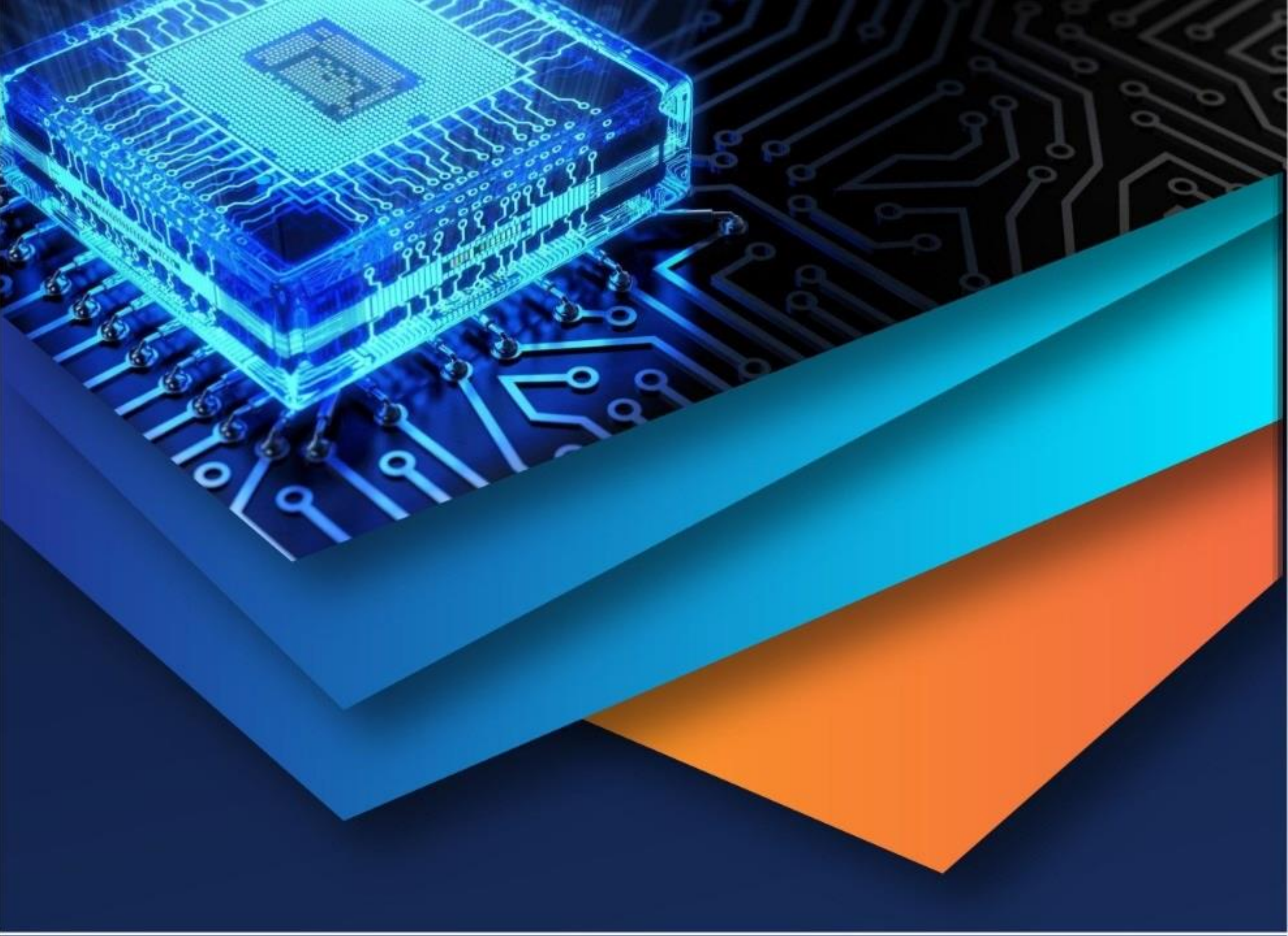
[6] Yuichi Kuya, Kenji Takeda, Xin Zhang, Scott Beeton, and Ted Pandaleon (2009), "Flow Separation Control on a Race Car Wing With Vortex Generators in Ground Effect" Journals of fluid engineering Vol. 131 / 121102-1.

[7] N. A. Ahmed (2001), "Implementation of a momentum method for total drag determination for undergraduate students" International Journal of Mechanical Engineering Education Vol 30 No 4.

©IJRASET (UGC Approved Journal): All Rights are Reserved

[8] Sinisa Krajnovi.c and Lars Davidson (2001), "Large-Eddy Simulation of the Flow Around a Ground Vehicle Body" SAE 2001-01-0702.
[9] Joseph Katz (2008), "Aerodynamics of Race Cars" Journal of aerospace engineering. Ramon Miralbes Buil, Luis Castejon Herrer (2009), "Aerodynamic Analysis of a Vehicle Tanker"Journal of fluids engineering Vol. 131 / 041204-
[10] R. Verzicco, M. Fatica, G. Iaccarino, P. Moin§, and B. Khalighi(2002), "Large Eddy Simulation of a Road Vehicle with Drag-Reduction Devices" AIAA Journal Vol. 40, No. 12
[11] Wolf- Heinrieh Hucho (2003), "Aerodynamics Of Road Vehicles" SAE Vol 269/No. 42 Sunyongling,WuGuangqiangand Xieshuo(2003),"Numerical Simulation of the External Flow Field Around a Bluff Car".

©IJRASET (UGC Approved Journal): All Rights are Reserved



10.22214/IJRASET

45.98

IMPACT FACTOR:
7.129

IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)