|| Volume 2 || Issue 2|| SEPTEMBER 2017 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH AND ENGINEERING TRENDS

CFD Analysis of an Automobile to Improve the Aerodynamics

Manoj Kumar D. Birajdar[1] Suresh Choudhary[2], Prof. Vivek Mane[3]
Assistant Professor, (M.E Machine Design), Department of Mechanical, VDF School of Engineering & Technology Latur, Swami Ramanand Teerth Marathwada University Nanded, India[1]
Mechanical, SSJCET/ Mumbai University, India.[2]
Asst. Prof, K J S I E I T, Sion, Mumbai, India.[3]

Abstract- An Engineer is always focused towards challenges of bringing ideas and concepts to life. Therefore, Software and Computer Techniques must be constantly developed and implemented for Virtual and Economic Analysis. At the same time, we must take care that there is no compromise with the Accuracy of the Techniques used. In the Age of Speed, Automobiles have become an integral part of human routine. By designing Aerodynamically Sound Automobiles, overall efficiency of an Automobile can be increased, by reduction in the Drag Force (Losses) and too by the reduction in the Fuel Consumption, eventually developing an Environment-Friendly Vehicle. The Engineer is constantly conformed to the challenges of bringing ideas and design into reality. The Computational Fluid Dynamics is a step towards bringing numerous Hypothetical Concepts to reality, as the Analysis done gives better results compared to the Physical Analysis (Wind Tunnel Testing), without any costly setup and space requirement. In this Project, Modelling of Automobile (Car) was done using the Software-Solid works. The Lift and the Drag of the car were determined by the Analysis of Fluid (Air) Flow around it using Software-Ansys Fluent. After that, with modifications like Air Vents, Rear Spoiler, the Analysis was repeated. Based on the values of Lift Force and Coefficient of Drag (C_d), optimal solution was considered as the success of the Project.

Keywords: *CFD, Drag, Downforce, Lift, Solidworks, Ansys*

I INTRODUCTION

Speed is the need of today's world. The ironical saying that, 'Distance must be measured in minutes, rather than in miles' is turning out to be a reality in the current world of pace. The ambitious construction of the Autobahns in Germany which began in 1920s and which was carried forward by the Nazi Supremo - Adolph Hitler, encouraged the people in that region to shift from using the conventional carts to mechanically powered vehicles. Gradually the trend spread around the world and reached our country, which resulted in the construction of Flyovers, Expressways, Sea Links, etc. which connected the vastly spread India. But along with the availability of good infrastructure, the Automobile Manufactures started feeling the need to develop faster vehicles, which they have managed to do well in the past two decades. But, challenges never end. Lower efficiencies of the vehicles have troubled the Automobile world for years, as to produce good results, huge power inputs are required. Hence, an effective way known, to better the outputs without regulating the inputs is – '*Aerodynamics*'

II OBJECTIVE

This paper focuses on the CFD analysis of a car. The goal is to simulate the air flow around the vehicle and obtain an accurate value of its drag and lift coefficient. The next step would be to make modifications to the vehicle geometry which could improve its lift and drag characteristics making the vehicle handle better at cruise speeds and also improve its fuel efficiency. The major objectives are – Getting optimal solution with maximum number of iterations, reducing the lift, working out with the coefficient of Presentation drag in a way that it facilitates stability and better handling of the car, developing pressure and velocity contours and gaining proficiency in the modelling and analysis software.

III THEORY

CFD or Computational fluid dynamics is a branch of fluid mechanics that, with the help of computers, uses numerical methods to solve and analyse problems involving fluid flows. Computers are used to carry out calculations using an iterative procedure wherein the solution accuracy improves with every iteration. The underlying equations that are solved in CFD problems are the Navier-Stokes equations. In the laminar regime, the flow of the fluid can be completely predicted by solving the steady-state Navier-Stokes equations, which predict the velocity and the pressure fields. As the flow begins its transition to turbulence, it is no longer possible to assume that the flow is invariant with time. In this case, it is necessary to solve the problem in the time domain. As the Reynolds number increases, the flow field exhibits small eddies, and the timescales of the oscillations become so short that it is computationally unfeasible to solve the Navier-Stokes equations, so in such flow regime, Reynold's Average Navier Stokes Formulation is used to cope up with the issue.

A. RANS Equation

The Reynolds Averaged Navier-Stokes equations (also known as RANS equations) are equations used to predict the fluid flow using a time averaged formulation. The primary concept applied is Reynolds decomposition which involves decomposing an instantaneous quantity into its time averaged and fluctuating quantities. The time averaged nature of its equations makes it an attractive choice while simulating turbulent flows. Considering certain approximations based on the knowledge of properties of turbulent flows, these equations can be used to give time averaged solutions to the Navier– Stokes equations.

B. Non-equilibrium wall function (NWF)

For high Reynolds number flows, such as in external flow around vehicles, resolving the near wall region down to the wall is not practical. To overcome this, wall functions are used. NWF takes into account the effects of local variation in the thickness of the viscous sub layer, when computing the turbulent kinetic energy budget in wall adjacent cells. Besides this, NWF is also sensitized to adverse pressure gradients which are common in flow around vehicles. Compared to traditional wall functions, NWF provide more realistic predictions of the behaviour of the turbulent boundary layers, including flow separation, and they do so without a significant increase in either CPU time or dynamic memory.

C. Aerodynamic Devices – Rear Spoiler

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 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 some Aerodynamic benefit. The term "Spoiler" is often mistakenly used interchangeably with "Rear Wing". An automotive wing is a device whose intended design is to generate downforce as air passes around it, not simply disrupt existing airflow patterns. As such, rather than decreasing drag, automotive wings actually increase drag.

IV PRE PROCESSING

A. Vehicle Geometry Model

For modelling the geometry, 3D modelling software Solid works was used. The modelling process involved importing the vehicle blueprints into Solid works with the help of which, 3D curves were projected. These curves then acted as boundaries to generate surfaces. The final surface model was converted into a solid part (refer Figure. 1) before importing it to Ansys.

Figure 1 Top: Actual Model View

B. Creating Fluid Enclosure

In order to simulate the air flow around the vehicle, a fluid volume needs to be created which will encompass the vehicle. This was done by creating an enclosure around the vehicle and subtracting the vehicle body. This enclosure acts as the air domain. To reduce the overall computational cost and time, the vehicle was considered symmetric laterally. The size of the enclosure was taken to be 3 car lengths each; ahead of the car, above the car and beside the car whereas 5 car lengths spacing was left between the car rear and the end of the enclosure.

C. Mesh Generation

While generating the mesh, sizing functions were used wherever necessary in order to obtain accurate lift/drag parameters. Two bodies of refinements were added to properly capture the flow in the region closest to the vehicle and also capture the flow in the wake. Since boundary layer separation has a significant effect on drag, five layers of inflation were added to the vehicle surface to properly resolve the boundary layer. The total number of elements obtained was 4.835 million. Figure 2 shows the Mesh Generation.

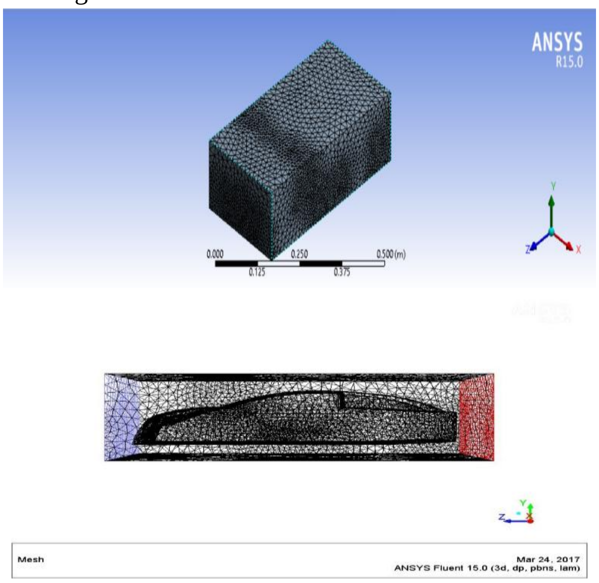

Figure 2 Mesh Generated

D. Boundary Condition

The enclosure inlet plane was named "velocity-inlet". Air coming through the inlet was given a velocity of 150 km/h which equates to 41.67 m/s. The road and the vehicle body were both made walls. The surrounding enclosure surfaces, being imaginary surfaces, were all named symmetry planes having a „no slip" condition. The outlet was named a "pressure-outlet" with its pressure set constant and equal to atmospheric pressure.

V SOLVER

Following conditions were considered for the Analysis: Air velocity at inlet: 150 km/h or 41.67 m/s. Reference area to determine drag and lift coefficients – Frontal Area: 1.17425 m2. The final solution was obtained by performing the iterations in three stages. With each progressive stage, the solver accuracy was raised by employing higher order equations. In the first stage, first order equations were used to prevent the solution from diverging. The Pseudo Transient Scheme was selected to speed up convergence. Once sufficient convergence was achieved, the equation order was raised. The iterations were carried up to the point where the change in the value of drag coefficient was found negligible.

VI SOLVER RESULTS

A final drag coefficient of 0.3569 was obtained. The analysis of the aerodynamic effects of the radiator and a rough underbody was out of the scope of this study. These two factors, though, tend to have a significant drag penalty and are known to contribute between 15-20% of the total drag. Adding their drag penalty to the obtained value gives a Cd value ranging between 0.4198 – 0.4461. The official drag coefficient for the car is 0.42. The results of the analysis were thus deemed accurate.

Stage	**I**	**II**	**III**
Convergence Criteria	Residuals 10e – 4	Residuals 10e - 4	Residuals 10e - 4 & Stable value of Cd
Iteration number	95	296	600
Order of Momentum, Turbulence K.E. and Turbulence Dissipation Eq.	First Order	Second Order	Second Order
Pressure equation	Second Order	Second Order	PRESTO
Scheme	Pseudo Transient	-	-
Relaxation Factors	-	0.25	0.4
Drag Coefficient	0.3915	0.3688	0.3569
Lift Coefficient	0. 1192	0.0819	0.0581

A. Post Professor – Ansys CFX

The velocity, pressure, turbulence kinetic energy and Y-plus contours are shown below (refer Figure. 3, 4, 5, 6).

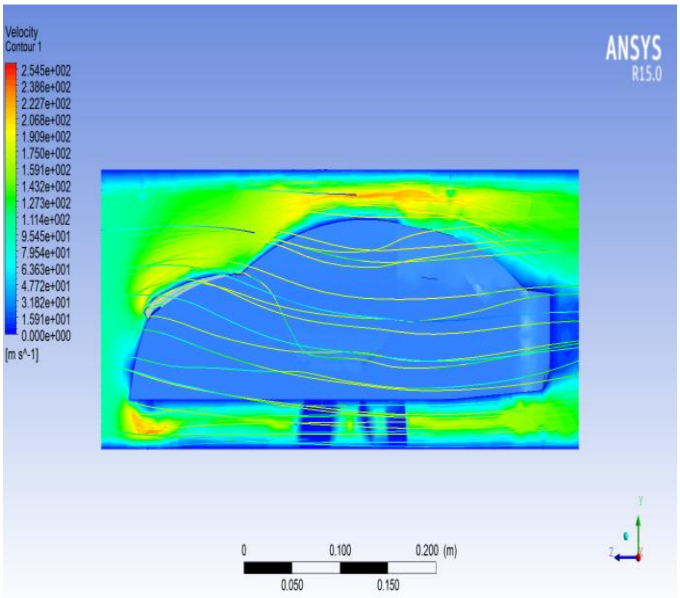

Figure 3 Velocity contour.

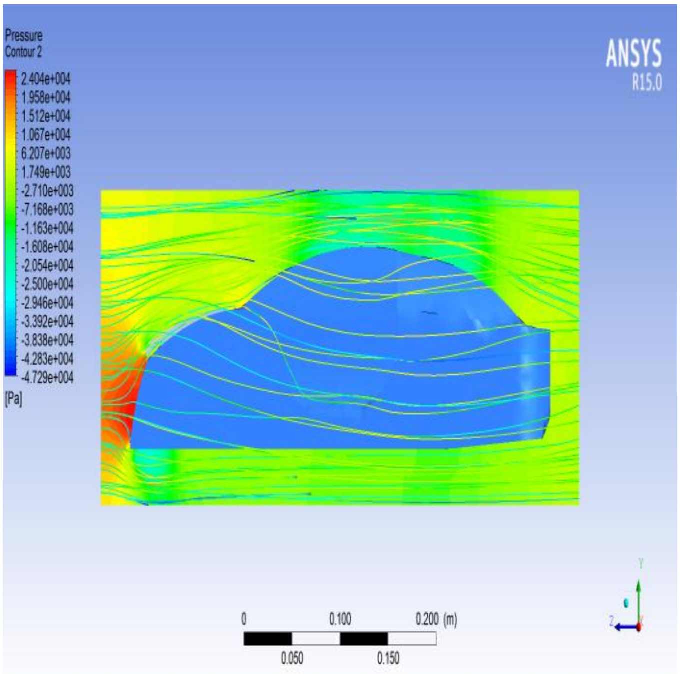

Figure 4 Pressure contour on vehicle.

|| Volume 2 || Issue 2|| SEPTEMBER 2017 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH AND ENGINEERING TRENDS

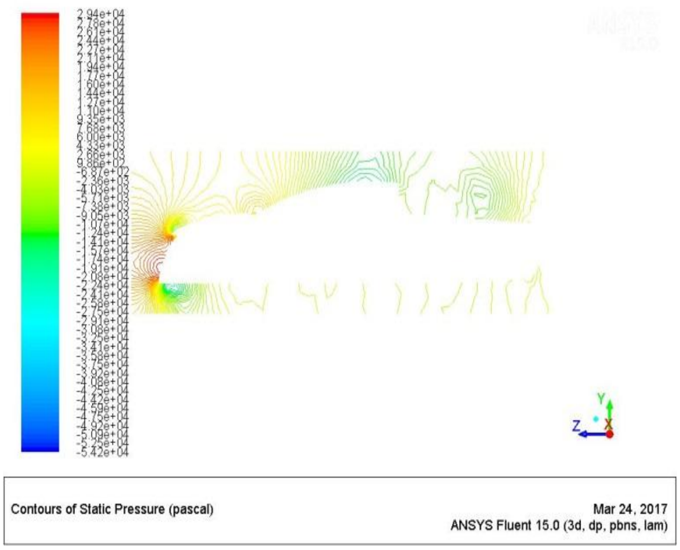

Figure 5 Turbulence Pressure contour.

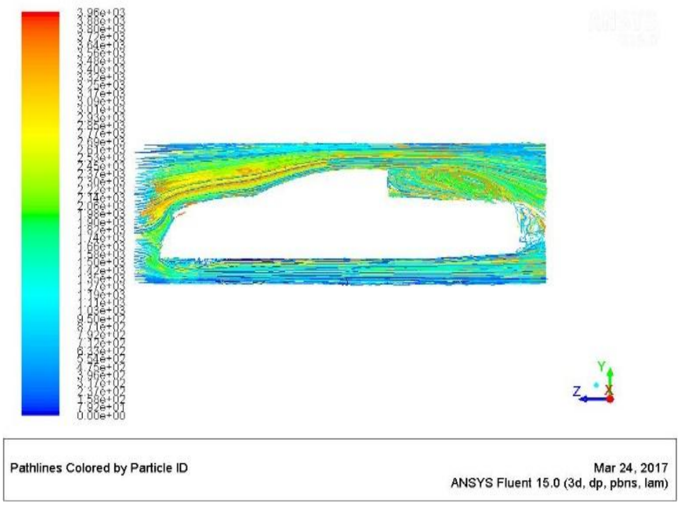

Figure 6 Particle Path line Contour.

VII GEOMETRY MODIFICATION

A. Addition of Diffuser

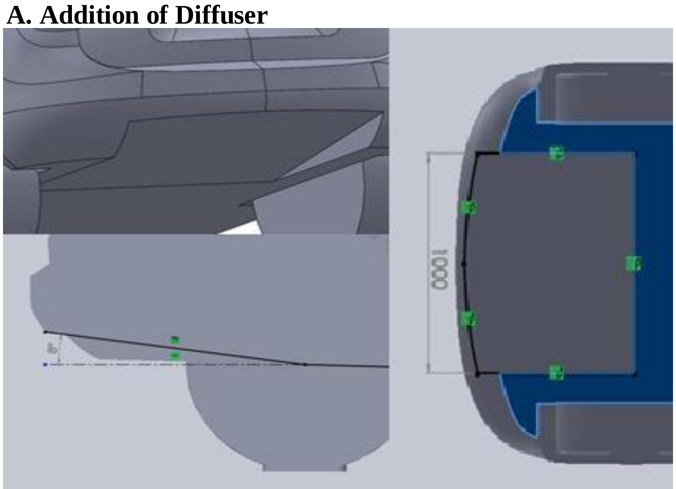

Figure 7 Dimensions and shape of diffuser

In order to improve the drag/downforce characteristics of the vehicle, the geometry was modified and a diffuser was added at the rear. The goal was to reduce the wake area created behind the car and to redirect the air upwards to reduce the lift coefficient. When considering diffuser characteristics, the variable having the most prominent effect on the air flow is the angle made by the diffuser to the horizontal. Three simulations were carried out, varying the diffuser angle in each case. The diffuser angles analysed were 8 deg, 10 deg and 15 deg. The vehicle geometry after addition of a diffuser is shown in the Figure. 7.

B. Solver Results – with Diffuser

The iterations in all the three diffuser cases were carried out following the same three stage procedure as carried out in the first simulation. The drag and lift coefficients obtained from the diffuser variations are tabulated below.

Note that the difference in coefficients is with respect to the original model.

Table II Comparison between Values of Cd and Cl before and after addition of Diffuser

Diffuser angle	Cd	Cl	Δ Cd	Δ Cl
8	0.3588	0.03848	+ 0.0019	- 0.01968
10	0.3609	0.02242	+ 0.0040	- 0.03574
15	0.3659	0.01788	+ 0.0090	- 0.04028

The results obtained can be summarized as follows:

Addition of a diffuser showed an increase in drag and a corresponding decrease in lift in all three cases. The reduction in lift was maximum in the 15 deg. diffuser model which also had the maximum drag penalty. The least drag coefficient after the original model was observed in the 8 deg. diffuser model.

The comparison between the turbulence kinetic energy before and after the addition of diffuser is shown in Figure 8. The streamlines (refer Figure. 9) show an increase in the upward deflection of the air in the vehicle wake after the addition of a diffuser.

|| Volume 2 || Issue 2|| SEPTEMBER 2017 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH AND ENGINEERING TRENDS

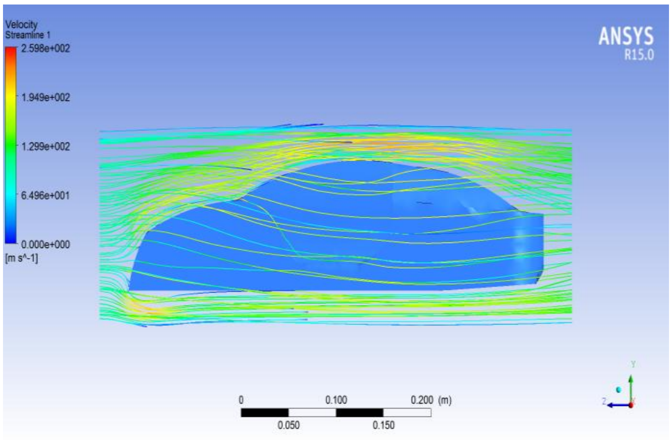

Figure 8 Velocity Streamline Flow before spoiler

VIII TRANSIENT FLOW ANALYSIS

Transient flow is the flow, wherein, the flow velocity and pressure are changing with time. When changes occur to fluid systems such as during starting or stopping, in such a situation transient flow conditions exists. Otherwise the system is in steady state. Often, transient flow conditions persist as oscillating pressure and velocity waves for some time after the initial event that caused it.

In the laminar regime, we can assume that the velocity field does not vary with time, and get an accurate prediction of the flow behaviour. As the flow begins to transition to turbulence, it is no longer possible to assume that the flow is invariant with time. Such problems can be solved by transient flow analysis which solves it by considering a time based domain. It will solve the problem in different time steps with variation of flow.

From the steady state result obtained, we see that the 8 degree diffuser attached underbody shows the least drag coefficient. A transient flow analysis was therefore performed on that model to check the accuracy of the results.

A. Mesh Generation

The model with 8 degree diffuser is imported to Ansys fluent and a fluid enclosure was created around it similar to that of the steady state analysis done above. Sizing functions were used wherever necessary in order to obtain accurate lift/drag parameters. One body of refinement was added to properly capture the flow in the region closest to the vehicle and also capture the flow in the wake. The total number of elements were set to 1.977 million to decrease the computational time.

B. Solver

Boundary condition and named selection were same as that of steady state analysis. For this analysis, a pressure based transient state solver was used. The solution methods, equations used along with the input data are listed below:

Realizable k- epsilon model with non-equilibrium wall functions.

Air velocity at inlet: 100 m/s.

Reference area to determine drag and lift coefficients – Frontal Area: 1.17425 m^2.

Second order implicit equations for Pressure, Momentum, Turbulence K.E. and Turbulence Dissipation Energy were used. Generally the PISO scheme may aid in accelerating convergence for many unsteady flows thereby increasing the speed of the solver. However, in order to be able to compare the results with those obtained from the steady state solution, the solver setting were kept same as that of the steady state analysis. A Coupled scheme was used for the solution. For fast convergence, the value of Courant number was set to 60 and K.E and pressure coefficients to 0.5 in residual settings. Time step size (Δt) must be small enough to resolve time-dependent features observed in transient flow and to make sure convergence is reached within the number of Max Iterations per Time Step. The setting selected were:

Time step value: 0.04 second

Max iterations per time steps: 100

Number of time steps: 100

With these settings, the transient flow solution for a flow duration of 4 seconds would be obtained after a total of 10000 iterations. The advancing of the time step occurs either when Max Iterations / Time Step is reached or convergence criteria is satisfied. Time steps are converged sequentially until the total number of time steps is reached.

C. Solver Results

From the transient analysis, a final drag coefficient of **0.3649** was obtained. The value of drag coefficient obtained in the steady state analysis was **0.3588**. Thus, the steady state and transient state analysis solutions were in close agreement which confirmed the accuracy of the solution.

VIII CONCLUSION

CFD analysis was successfully carried out on the production vehicle. The results of the simulation (drag coefficient) were found to be in close agreement with the officially provided values. Once the validity of the simulation was achieved, the next step was to make modifications in the geometry of the original model which could positively affect performance characteristics (lift and drag). A diffuser was added to the rear end of the vehicle and further simulations were performed.

The addition of an 8 deg. diffuser helped reduce the lift considerably (34% reduction in Cl) while only slightly increasing the drag coefficient (0.5% increase in Cd). Increasing the diffuser angle to 10 deg. and then 15 deg. led to a greater reduction in lift. However, the corresponding drag penalties were also higher. The results obtained by performing the steady state analysis were then confirmed by performing a transient simulation, both providing a similar value of Cd.

The addition of a spoiler to the original model at an angle of 8 degrees showed a definite improvement in the vehicle lift characteristics with a negligible drag penalty. Such modifications could be carried out in the selected car to improve its handling capabilities at higher speeds thereby improving the overall safety of the vehicle.

REFERENCES

[1] “Racing Car Wheel Aerodynamics – Comparisons between Experimental and CFD Derived Flow-Field Data”, SAE 2004-01-3555.

[2] Johan Levin and Rikard Rigdal, “Aerodyanamic analysis of drag reduction devices for SAAB 9-3 by CFD”, Master's thesis in Automotive Engineering 2011:30.

[3] Castelli-Dezza, Mastinu G and. Mauri M., “An urban vehicle with very low fuel consumption: realization, analysis and optimization”, IEEE 978-1-4799-3786-8.

[4] Satheesh A., “Computational drag analysis in the under-body for a sedan type car model”, IEEE 978-1-4673-6149-1.

[5] Ping Hu , Shanbin Lu, “Numerical analysis of the effect of ground clearance on a simplified car model”, IEEE 978-1-4244-2692-8.

[6] Mechanical Engineering Faculty in Slavonski Brod, Josip Juraj Strossmayer University of Osijek, Croatia, “CFD Analysis of concept car in order to improve Aerodynamics”, 2011.

[7] Ahmed H. and Chacko S., “Computational Optimization of Vehicle Aerodynamics” 23rd International DAAAM Symposium, Volume 23, No.1, ISSN 2304-1382 ISBN 978-3-901509-91-9.

[8] Akshay Parab, Ammar Sakarwala, Bhushan Paste, Vaibhav Patil, Department of Mechanical Engineering, Rajiv Gandhi Institute of Technology, Mumbai, India, “Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5”, November 2014.

[9] J. Sita Priyadarshini, A.V.S. Abhinav, B. Sharath Chandra and R.S. Swathi, “Use of Aerodynamic Lift in Increasing the Fuel Efficiency of Heavy Vehicles”, 2015.

[10] Kevin M. Peddie, Luis F. Gonzalez, “CFD Study on the Diffuser of a Formula 3 Racecar” School of Aerospace, Mechanical and Mechatronic Engineering University of Sydney.