CFD analysis of the flow variation due to addition of aerodynamic devices on underbody of Renault Duster using Fluent - Ansys 14.5

Akshay Parab
Ammar Sakarwala
Bhushan Paste
Vaibhav Patil
Department of Mechanical Engineering
Rajiv Gandhi Institute of Technology
Mumbai, India.

Amol Mangrulkar
Department of Mechanical Engineering
Rajiv Gandhi Institute of Technology
Mumbai, India.

Abstract - Vehicle aerodynamics play an important role in the performance characteristics of a vehicle. As the speed of a vehicle increases, parameters such as lift and drag force play a much more significant role. Other than wind tunnel testing, the other method available to obtain these parameters is through computational fluid dynamics (CFD) analysis. In recent years, with an increase in the computational power available, many manufacturers are looking towards CFD to reduce testing time and keep the costs of R&D low. In this paper, the flow of air around a production vehicle Renault Duster has been analyzed using the analysis software Fluent-Ansys 14.5. Once the lift and drag parameters were determined, a diffuser and then vortex generators were added to the vehicle underbody to improve its aerodynamics. The results of these modifications have been published in this paper.

Keywords—cfd; drag; downforce; diffuser; vortex generator; renault duster; ansys; fluent.

I. INTRODUCTION

The current economic growth trend in India predicts the number of vehicles on road to increase rapidly in the next 15 years. Such a growth, however, would also increase the CO_2 emissions considerably which would have drastic effects on the environment. Automobile manufacturers are therefore, looking for new ways and developing new technologies to reduce fuel consumption and improve vehicle efficiency.

In terms of vehicle efficiency, drag is an important factor which is why vehicle aerodynamics is such an active area of research for automobile manufacturers. While wind tunnel testing was the most profound way of testing vehicle aerodynamics in the 20^{th} century, recent growth in the available computational power has led to more and more adaptation of numerical simulations. Computational Fluid Dynamics (CFD) helps study the flow behaviour without having to create a physical model and thus helps reduce R&D costs while simultaneously saving time.

II. OBJECTIVE

This paper focuses on the CFD analysis of a production vehicle. The vehicle selection criteria is: a) Vehicle availability in India. b) Officially provided value of vehicle drag coefficient (as a reference to check the accuracy of the simulation results). c) Availability of vehicle blueprints (necessary to create CAD model).

The vehicle Renault Duster satisfies the above requirements and is hence selected considering its high popularity in India. The vehicle model, created in Solidworks 2013, is analyzed using analysis software Fluent- Ansys 14.5. 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.

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 analyze 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. In this flow regime, a Reynolds Averaged Navier-Stokes formulation can be used, which is based on the observation that the flow field over time contains small, local oscillations that can be treated in a time-averaged sense.

A. RANS

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. k-epsilon model

The k-epsilon model is one of the most commonly used turbulence models. It is a two equation model that employs two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy. This model, however, does not perform well in cases of large adverse pressure gradients.

C. Realizable k epsilon model

The realizable k-epsilon model addresses the well-known deficiencies of the traditional k-epsilon model by incorporating:

- A new eddy-viscosity formula involving a variable C_μ originally proposed by Reynolds.
- A new model equation for dissipation based on the dynamic equation of the mean square velocity fluctuation.

This model makes it possible to achieve good results in terms of integral values (eg. Cd) which are within 2-5% of the actual value. It is also very stable and converges quickly.

D. 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 sublayer, 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.

E. Aerodynamic Devices

1) Diffuser

The wake area behind a vehicle is a low pressure region which has a retarding effect on the vehicle. A diffuser is a modification in the vehicle geometry which helps counter this effect by gradually increasing the air pressure from the underbody towards the rear. At the beginning of the diffuser, the air accelerates due to the sudden change in the shape of the underbody and creates a low pressure point. As we move towards the rear bumper, the diffuser expands which causes the pressure to increase gradually upto ambient pressure at the rearmost of the vehicle. Thus, a diffuser has a combined effect of increasing the downforce at the vehicle rear while simultaneously reducing the low pressure wake region created behind the vehicle.

2) Vortex Generators (VGs)

VGs are small projections attached to a surface to prevent boundary layer separation. Towards the rear of a vehicle, the boundary layer loses energy and slows down due to the retardation caused by skin friction. This process continues till a point (stagnation point) at which the boundary layer separates from the surface. This separation is accompanied with a corresponding drag penalty. VGs help generate swirls or vortices which take energy from the high velocity flow outside the boundary layer and impart it to the boundary layer thus building the pressure gradient. This prevents the separation from occurring.

IV. PRE PROCESSING

A. Vehicle Geometry Modeling

For modeling the geometry, 3D modeling software Solidworks 2013 was used. The modeling process involved importing the vehicle blueprints into Solidworks 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 fig. 1) before importing it to Ansys.

Fig. 1. Top : Actual model view ; Bottom : Solidworks model.

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. Fig. 2 shows the final mesh.

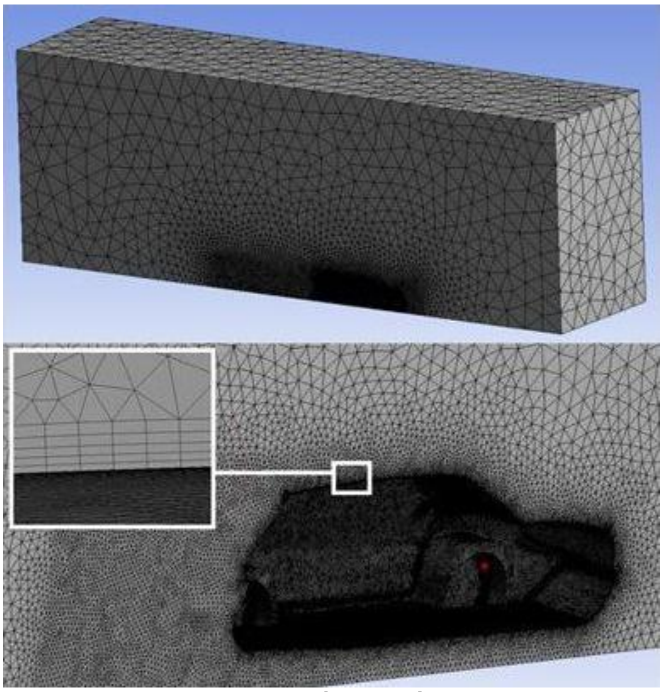

Fig. 2 Mesh generated

D. *Boundary Conditions*

The enclosure inlet plane was named "velocity-inlet". Air coming through the inlet was given a velocity of 150 kmph 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

For this analysis, a pressure based steady state solver was used. The solution methods, equations used along with the input data are listed below:

- Pressure based steady state solver.
- Realizable k- epsilon model with non-equilibrium wall functions.
- Air velocity at inlet: 150 kmph or 41.67 m/s.
- Reference area to determine drag and lift coefficients – Frontal Area: 1.17425 m^2.

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 ACCURACY

A. *Test Model*

In order to test the accuracy of the results obtained by employing the above mentioned solver settings, a test case – Ahmed body was first analyzed. The drag coefficient obtained in this simulation (Cd = 0.291) closely related to the wind tunnel data for Ahmed body (Cd = 0.285). The results were within the range of accuracy desired.

B. *Applicability of Turbulence Model*

The results obtained using the k-epsilon model (which employs wall functions) depend heavily on the wall Y-plus. Y-plus is non-dimensional parameter which defines the height of the first cell in contact with the surface. For the k-epsilon model, the value of Y-plus obtained on the vehicle surface should lie between 25-300. This was obtained by a trial and error approach wherein the mesh surface sizing was varied for different areas on the car until the desired Y-plus value was achieved in each area.

VII. 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 Renault Duster is 0.42. The results of the analysis were thus deemed accurate. The solver settings along with the results after each stage is shown in Table I.

TABLE I. RESULTS OF MODEL ANALYSIS

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 Processor – Ansys CFX

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

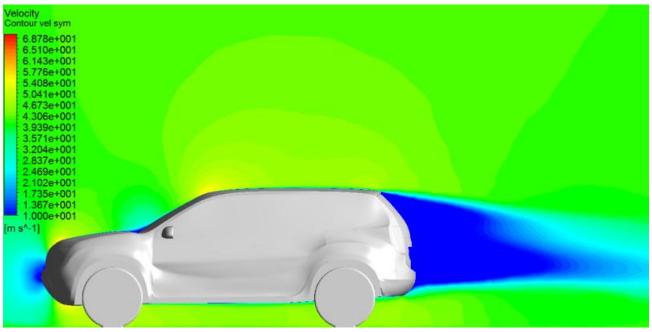

Fig. 3. Velocity contour.

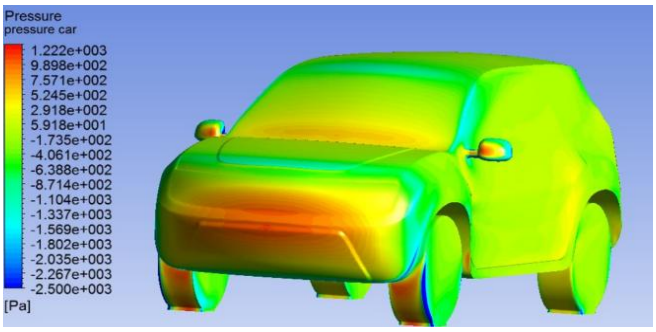

Fig. 4. Pressure contour on vehicle.

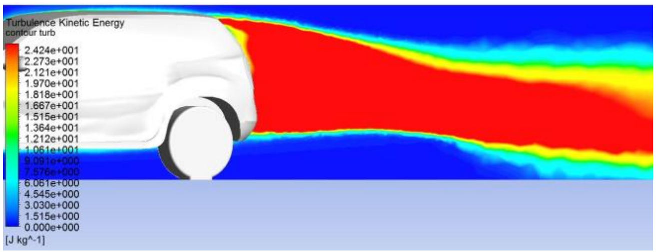

Fig. 5. Turbulence kinetic energy contour.

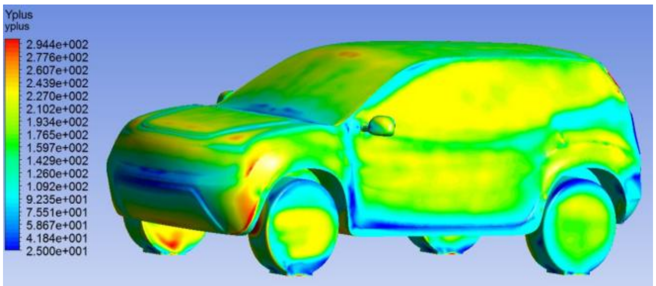

Fig. 6. Contour showing cell Y plus.

VIII. GEOMETRY MODIFICATION

A. Addition of Diffuser

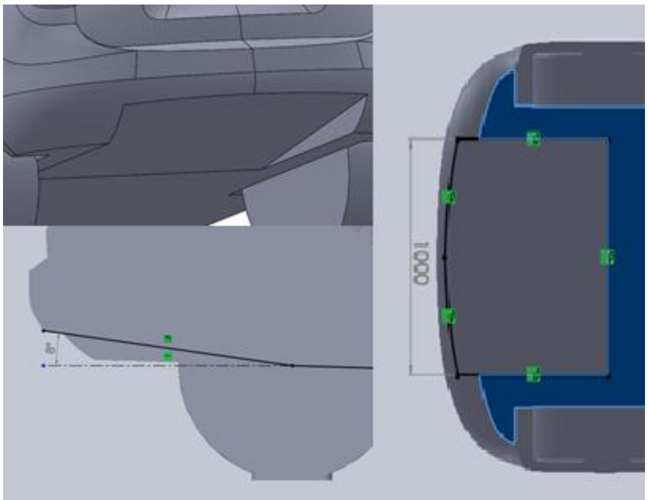

Fig. 7. Dimensions and shape of diffuser.

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 analyzed were 8 deg, 10 deg and 15 deg. The vehicle geometry after addition of a diffuser is shown in the fig. 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 fig. 8. The streamlines (refer fig. 9) show an increase in the upward deflection of the air in the vehicle wake after the addition of a diffuser.

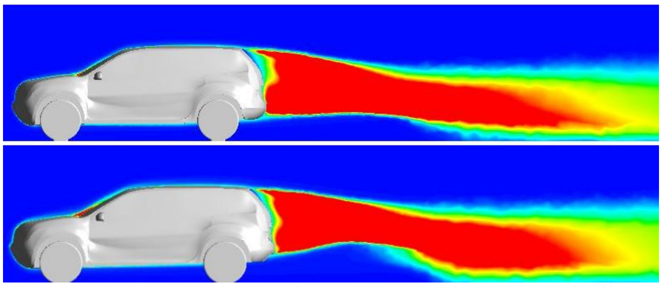

Fig. 8. Turbulence K. Energy before (above) and after addition of diffuser.

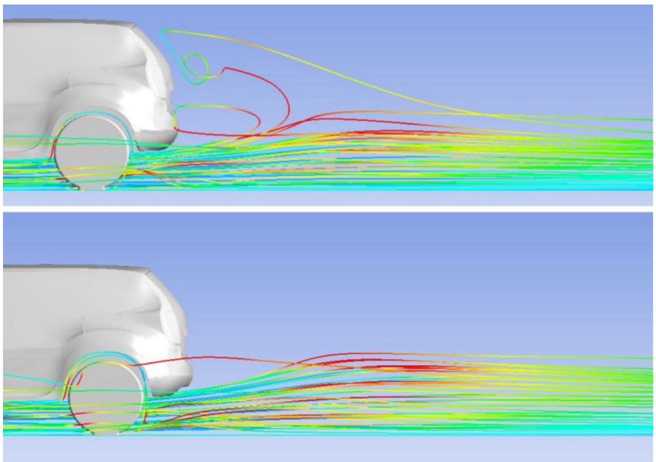

Fig. 9. Streamlines before (above) and after addition of diffuser.

C. Addition of Vortex Generators (VGs)

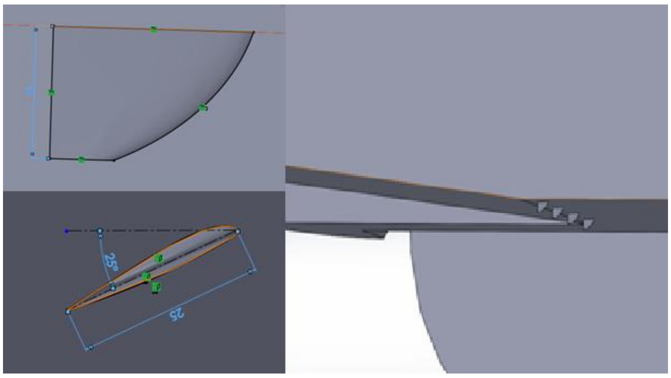

Fig. 10. Dimension, shape and position of Vortex generators.

In order to eliminate possible boundary layer separation occurring near the middle of the diffuser, VGs were added on the underbody of the vehicle. Delta shaped VGs were fitted on the underbody, each having a height of 15 mm and an angle of 25 deg. to the vertical plane (refer fig.10). Since the least drag was observed in the model with the 8 degree diffuser, that model was selected for the addition of VGs.

D. Solver Results – with Diffuser and Vortex Generators

The drag and lift coefficients before and after addition of VGs are tabulated below:

TABLE III. COMPARISON OF CD AND CL VALUES OBTAINED AFTER ADDITION OF VGS

Model	Cd	Cl
Original model	0.3569	0.05815
8 deg. diffuser	0.3588	0.03848
8 deg. diffuser and VGs	0.3596	0.03608

The addition of VGs, thus, had a detrimental effect on the overall drag while negligibly reducing lift. The pressure and turbulent kinetic energy contours are shown in fig. 11, 12, 13.

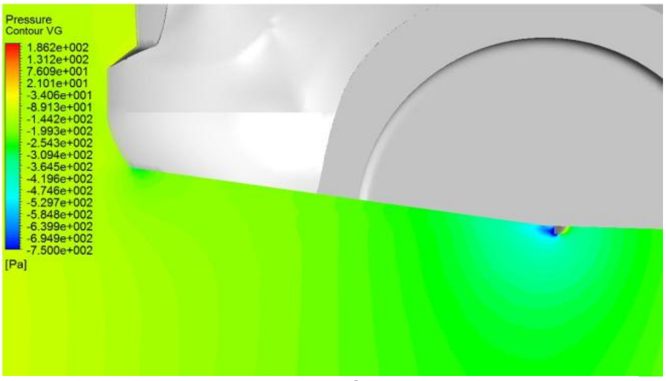

Fig. 11. Pressure contour with Vortex generators.

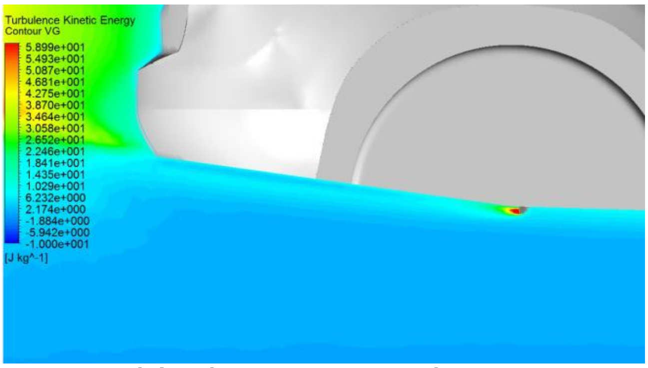

Fig. 12. Turbulence kinetic energy contour with Vortex generators.

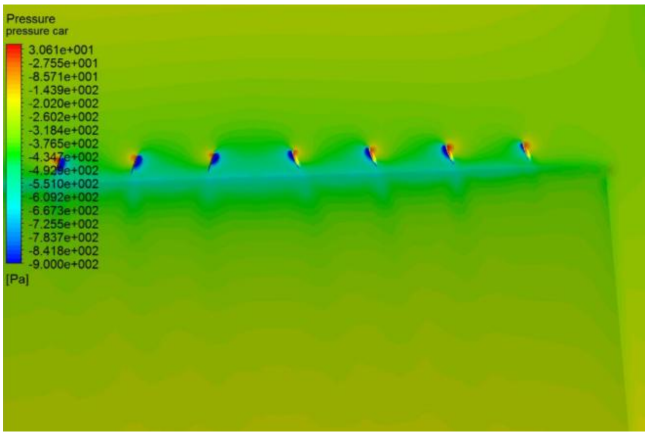

Fig. 13. Pressure contour of car underbody (near Vortex generators).

IX. CONCLUSION

CFD analysis was successfully carried out on the production vehicle Renault Duster. 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.

A potential cause for the increased drag coefficients observed was flow separation occurring in the diffuser region. To keep the flow attached, vortex generators were attached near the diffusers. Their addition on the 8 deg. diffuser model did not show any visible improvement in the overall drag or lift characteristics. However, more research on vortex generators needs to be done before any conclusions are drawn in that regard.

The addition of a diffuser 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 Renault Duster 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] Yu Hao , Yang Zhigang, "Numerical analysis on effect of back/front windshied and hood angle on automotive aerodynamic drag", IEEE 978-1-84919-641-3.

[3] 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

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

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

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

[7] Kang, S. O., Jun, S. O., Park, H. I., Song, K. S., Kee, J. D., Kim, K. H., and Lee, D. H. (2012) "Actively translating a rear diffuser device for the aerodynamic drag reduction of a passenger car", International Journal of Automotive Technology, Vol.13(4), pp583-592.

[8] 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.

[9] Debojyoti Mitra, "Design Optimization of Ground Clearance of Domestic Cars", International Journal of Engineering Science and Technology(2010).

[10] Dubey, A., Chheniya, S., and Jadhav, A. "Effect of Vortex generators on Aerodynamics of a Car: CFD Analysis." International Journal of Innovations in Engineering and Technology (IJIET), Vol. 2(1), pp137-144.

[11] K. Sai Sujith, G.Ravindra Reddy, "CFD analysis of sedan car with vortex generators", International Journal of Mechanical Engineering applications Research – IJMEAR, pp. 179-184, Vol 03, Issue 03, July 2012.

[12] 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.

[13] Cooper, K. R. Bertenyi, T. Dutil, G. Syms, J. Sovran, G. "The Aerodynamic Performance of Automotive Underbody Diffusers", SAE 980030, 1998.

[14] J.R. Callister, A.R. George, Wind Noise, "Aerodynamics of Road Vehicles", W.H. Hucho (Ed.), SAE International, Warrendale, PA, 1998.

[15] R. B. Sharma1, 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