ISSN: 2278-0181
NCRAIME-2015 Conference Proceedings

Investigating the Effect of Rear Spoiler and Rear Diffuser on Aerodynamic Forces using CFD

Mohankumar M | Hudson E Daniel Raj
[1](PG Scholar, Engineering Design of Mechanical PG department, Sri Krishna College of Technology, Coimbatore, INDIA,
[2](Assistant professor, Mechanical PG department, Sri Krishna College of Technology, Coimbatore, INDIA

***Abstract*−This research aims to develop an actively translating rear spoiler and rear diffuser device to reduce the aerodynamic drag experienced by TATA INDOGO CS passenger car. One of the features of the device is that it is ordinarily hidden under the rear bumper but slips out backward only under high-speed driving conditions. In this study, a movable arc-shaped semi-diffuser device, round in form, is designed to maintain the streamlined automobile's rear underbody configuration. The device is installed in the rear bumper section of a passenger car. Seven types of rear diffuser devices whose positions and protrusive lengths and widths are different (with the basic shape being identical) were installed and Computational Fluid Dynamics (CFD) analyses were performed under moving ground and rotating wheel conditions. The main purpose of this study is to explain the aerodynamic drag reduction mechanism of a passenger car cruising at high speed via an actively translating rear diffuser device. The base pressure of the passenger car is increased by deploying the rear diffuser device, which then prevents the low-pressure air coming through the underbody from directly soaring up to the rear surface of the trunk. At the same time, the device generates a diffusing process that lowers the velocity but raises the pressure of the underbody flow, bringing about aerodynamic drag reduction.**

Key Words: Automobile aerodynamics, Rear spoiler, Active rear diffuser device, Drag reduction.

1 INTRODUCTION

Aerodynamics is a branch of dynamics concerned with studying the motion of air, particularly when it interacts with a solid object. It is a sub-field of fluid dynamics and gas dynamics, and many aspects of aerodynamics theory are common to these fields. The term aerodynamics is often used with gas dynamics, with the difference being that "gas dynamics" applies to the study of the motion of all gases, not limited to air. The rules of aerodynamics explain how an airplane is able to fly. Anything that moves through air reacts to aerodynamics. A rocket blasting off the launch pad and a kite in the sky react to aerodynamics. Aerodynamics even acts on cars, since air flows around cars.

2 LITERATURE REVIEW

Chien-Hsiung Tsai, et al [1] studied the effect of installation of spoiler on aerodynamics and aero-acoustics of a vehicle, six cases were considered. Case 1 is the base model which corresponds to a car without a spoiler another five spoilers with different cross-sectional profiles was considered. Symmetrical boundary condition was applied on the plane where the actual computational domain was split into two.

On the plane upstream of the vehicle model, a constant velocity of 180 km/hr, which corresponded to Re = 3.6x106, was applied. This speed was used because 180 km/hr is almost the maximum speed reasonably achievable by a Honda S2000. This maximum speed was chosen over other speeds due to the fact that noise generation is more critical at a higher speed. Similarly, this boundary condition was also applied at the plane on top of the vehicle model as well as the plane opposite of the symmetrical plane. The plane downstream of the vehicle model was assumed a pressure outlet of 1 atm. Non-slip condition was applied on all of the solid surfaces. The combination of these boundary conditions implies the simulation of a vehicle model securely mounted in the test section of a wind tunnel.

Among these six cases, there are more positive pressure coefficients on the upper wing of the spoiler. Therefore, its lift force is the minimum among all models. For all the cases, the pressure coefficients in front of the bumper and the windscreen were positive, but that associated to the roof was always negative. Recall that case 1 corresponds to the vehicle model without the installation of a spoiler. If a spoiler (or a pair of spoilers) is installed on top of the trunk, the drag coefficient increases while the lift coefficient decreases.

R. B. Sharma, et al [2] in this investigation of paper Computational fluid dynamics (CFD) simulations of the steady flow field around passenger car models with and without Tail Plates presented and compared the simulated data to each other. The ANSYS-14.0 Fluent with the k-e steady model is used for the simulations of aerodynamics. In this analysis, the coefficient of drag has been reduced 3.87% and coefficient of lift is reduced 16.62%. Hence, the Tail Plates is the effective tool to reduce the drag force on vehicle.

The effects of different aerodynamic add-on devices on flow and its structure over a generic passenger car may be analyzed using CFD approach. The objective is to reduce aerodynamic drag acting on the vehicle and thus improve the fuel efficiency of passenger car. Hence, the drag force can be reduced by using add on devices on vehicle and fuel economy, stability of a passenger car can be improved.

Joseph Katz [3] studied the complexity of automobile and concluded that race car aerodynamics is comparable to airplane aerodynamics and is not limited to drag reduction only. The generation of downforce and its effect on lateral stability has a major effect on race car performance, particularly when high speed turns are involved. In the process of designing and refining current race car shapes, all aerospace-type design tools are used. Because of effects such as flow separations, vortex flows, or boundary-layer transition, the flow over most types of race cars is not always easily predictable. Due to the competitive nature of this sport and the short design cycles, engineering decisions must rely on combined information from track, wind tunnel, and CFD tests.

Ram Bansal, et al [4] analyzed different types of aerodynamic add-on devices used on the baseline car to get the results for the coefficient of drag and coefficient of lift. In the first case the spoiler is applied on the boot of the passenger car with the inclination angle 12∘.The coefficient of drag is 0.3441 and the coefficient of lift is 0.1985.The percentage reduction in drag coefficient in comparison with baseline car is 2.02%and in coefficient of lift is 6%.Hence, drag force and lift force on the passenger car are reduced as proportional to drag coefficient and lift coefficient, respectively. In the second case the vortex generators are applied on the rear side at roof of the baseline car with inclination angle 12∘. The coefficient of drag is 0.3471 and the coefficient of lift is 0.2085.

The percentage reduction in drag coefficient in comparison with baseline car is 1.17% and in coefficient of lift is 9.8%. Hence drag force and lift force on the passenger car are reduced as proportional to drag coefficient and lift coefficient, respectively. In the third case the tail plates are applied on the rear side: one is at the rear side of the roof garnish and the other is at the tail bumper of the passenger car.

The coefficient of drag is 0.3376 and the coefficient of lift is 0.1926. The percentage reduction in drag coefficient in comparison with base line car is 3.87% and in coefficient of lift is 16.62%. Hence drag force and lift force on the passenger car are reduced as proportional to drag coefficient and lift coefficient, respectively. In the fourth case the spoiler and VGs together are applied on the rear boot and rear side at the roof of the passenger car. The coefficient of drag is 0.3359 and the coefficient of lift is 0.1875.

The percentage reduction in drag coefficient in comparison with base line car is 4.35% and in coefficient of lift is 18.83%. Hence, drag force and lift force on the passenger car are reduced as proportional to drag coefficient and lift coefficient, respectively.

From the above analysis, it is found that spoiler with VGs is more effective add-on device to reduce the drag coefficient and lift coefficient which are applied on the passenger car when the car is running on the road. The drag coefficients and drag forces are proportional to each other so when the drag forces are reduced, lift forces are also reduced because it is proportional to the lift coefficient.

Tank Nilesh R, et al [5] presented that transient condition is inevitable for cross flow condition. Stagnation pressure is high at windward side compare to steady case, so more prone to yawing moment and side force which leads to instability of the vehicles. Tapered front section will slightly decrease the yawing moment and side force effects on overtaking vehicle. Notchback exhibits highest yawing moment and square back will lowest while reverse for side force.

Rounding of the edges of front end and hood and hood sides increase the yawing moment and leads to lower pressure on leeward side of front end. Increase in side projection area will leads to decrease in side force at rear end but will increase in yawing moment due to increment in length. Low placed cowl with less extent and greater wind shield angle reduce the front side force and yawing moment.

Jesper Marklund [6] in the paper presented that passenger cars are bluff bodies aerodynamically, with the pressure drag dominating the friction drag. Reducing the pressure drag has the greatest potential and the focus should be at this. The exterior body of a passenger vehicle, such as the upper and under-body, is responsible for more than 60 % of the drag. The rest is related to cooling flows and wheels. It is therefore relevant to study the vehicle body alone, and do so with simplified models of vehicle proportions. Simplified bluff bodies were used to study the overall effects of the body shape with regard to ground proximity, ground simulation, yaw and pitch angles.

The rear-end shape was seen to have a major effect on the flow distribution around the body. The effect of drag and lift dependence on ground proximity is very different depending on rear-end shape.

- ✓ A body of vehicle proportions will have reduced drag by a nose-down pitch as long as the flow does not generate a negative camber.
- ✓ There is an optimum drag relative pitch angle and yaw angle of a body.
- ✓ A square back model will have a larger drag increase by a yaw angle than other shapes.
- ✓ Drag reduction of an under-body diffuser, applied to a simplified bluff body is depending on the upper body. A model with large backlight angle will have better drag reduction by a diffuser than a square back model.
- ✓ Diffusers applied to passenger vehicles, with road vehicle boundary conditions, follow the same trend as diffusers applied to simplified bluff bodies. The main part of lift and drag of a passenger car is related to the main body.
- ✓ The sedan vehicle has greater potential to reduce drag by using a smooth floor and under-body diffuser than the wagon. The diffuser generates an upsweep and makes the wake more symmetrical in side view, for the sedan. For

Special Issue - 2015
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
NCRAIME-2015 Conference Proceedings

the wagon the wake is already symmetrical and the diffuser can even generate an asymmetric wake.

- ✓ Wake symmetry is very important for optimum pressure recovery of passenger vehicles.
- ✓ The effect of correct ground simulation is shown to be important since the flow around the body will be re-distributed to have more flow under the body.

Gopal P, et al [7] experimental investigated on the measurement of the variation of pressure coefficient and dynamic pressure on the roof of a utility vehicle with and without vortex generators (VG), the following conclusions were made:

- ✓ The value of pressure coefficient without VG is minimum whereas its value was observed to be maximum with VG having yaw angle of 15°.
- ✓ The pressure coefficient can be increased with the inclusion of VG by around 17% at a velocity of 2.42 m/s.
- ✓ The values of pressure coefficient don’t change significantly with increase in velocity for various values of yaw angle.
- ✓ Dynamic pressure over the surface of the vehicle roof increases with addition of VG which is favorable for avoiding flow separation and the consequent losses.
- ✓ The value of CD is reduced by 90% with the addition of VG at a velocity of 2.42 m/s and a minimum of 20% reduction in drag is obtained for VG with a yaw angle of 10°
- ✓ It is observed that VG with a yaw angle of 15° will be useful at lower velocity
- ✓ The value of CL decreases with increase in velocity with and without VG and the results revealed that at higher velocity the value of CL remains constant for VG with varying yaw angles.

Mahmoud Khaled, et al [8] concludes that unlike the lift force, the drag is smaller when the body lower face outlet is centered over the width of the model. Positioned on the left or the right, the body lower face air outlet induces the same effect on the aerodynamic torsor. The further towards the top the vertical air outlet is, the lower the induced drag and the higher the lift force. The aerodynamic drag is independent of the position of the air outlet in the wheel arch. However, placing this outlet at the rear of the wheel arch decreases the lift force.

The drag coefficient as well as the pitch momentum coefficient increase with the inlet/outlet section ratio up to 0.7, beyond which they become almost constant.

The lift coefficient decreases with the ratio between the inlet and outlet section also up to 0.7, from which it varies only slightly. It is also shown that increasing the distance in the *X* direction between the cooling module and the engine block from 6 to 20 cm reduces the drag coefficient by 1.4%, the cooling drag coefficient by 17.4%, and the lift coefficient by 1.8%. Also, shifting the cooling module in the right direction (through the *Y* direction) from the engine block, i.e., varying the distance *Y* between the two components, reduces the cooling drag by 1.1%, the cooling drag coefficient by 12.8%, and the pitch momentum coefficient by 1.5%.

With an air inlet opening centered in the *Z* direction, one reduces, with respect to classical air inlet position, the drag coefficient by 1.3%, the cooling drag coefficient by 56.4%, and the pitching momentum coefficient by 3.6%.

Xingjun Hu, et al (9) proposed that when the diffuser angle varied from 0□to 12□, the total aerodynamic drag coefficients of car first decrease and then increases, while the total aerodynamic lift coefficients decrease. There is a diffuser angle at which the sedan can obtain the minimum drag coefficient. From the calculation results, we can find when the diffuser angle varied from 0° to 12°, pressure distributing on the top of body and the front of body changed little, with the increase of diffuser angle, the distribution area of positive pressure on the rear of the body first increases and then decrease.

The distribution area of positive pressure reach peak at diffuser angle of 6°. Difference of positive pressure distribution on the rear of the body lead to differential pressure of the body surface varies from case to case, which results in the total aerodynamic drag coefficients of car first decreasing and then increasing while diffuser angle changes. Negative pressure is generated at the underbody interface and the region of the negative pressure become larger and larger. At the same time, the positive pressure generated at the edge of underbody decrease. When diffuser angle is changed to 9.8 degree, there is not any positive pressure distributing at the edge of underbody.

The increase of negative pressure distribution and the decrease of the positive pressure distribution on the underbody lead to increase in differential pressure of the body surface, which results in decreasing of total aerodynamic lift coefficients, found the negative pressure peak at the start of the diffuser. The streamline behind the vehicle can be seen for diffuser angles from 0 to 12 degrees. From the streamline and pressure contour of the sedan, it can be found when the diffuser angles varies from 0 degree to 12 degrees, the flow field after the sedan has an obvious change. It can be seen obviously the wake structures after the car of all cases are different.

3 OBJECTIVE

The main purpose of this project is to investigate the drag reduction phenomenon of the passenger car induced by a rear spoiler and rear diffuser device according to the diffuser length and the driving speed condition.

- ✓ Drag coefficient and lift coefficient plays a major role in affecting the forward motion of a car
- ✓ In this project work, rear spoiler and diffuser will be used and analyzed to reduce C_D,C_L

4. METHODOLOGY

4.1 Actively Translating Rear Diffuser Device

It was observed that drag reduction is achieved through the diffuser effect, resulting in an increase of the base pressure of the car body. Based on this concept of the diffuser effect, an actively translating rear diffuser device is suggested in this study, where a movable arc-shaped diffuser device is installed under the rear bumper of the passenger car. The device is rounded to maintain a smooth, streamlined rear underbody shape to obtain a greater diffusion effect when it is fully extracted. The basic concept of the actively translating rear diffuser device is shown in Figure 1. This device is hidden under the rear bumper while the car is parked and during low-speed driving, when the aerodynamic drag is negligible, because it is important not to alter the aesthetic design of the external configuration. Only at high speeds (above 70 km/h) does the rear diffuser device become active and control the rear flow of the automobile to reduce the aerodynamic drag. Once the rear diffuser device slides from its rear bumper position, the rear flow and wake patterns of the passenger car change significantly.

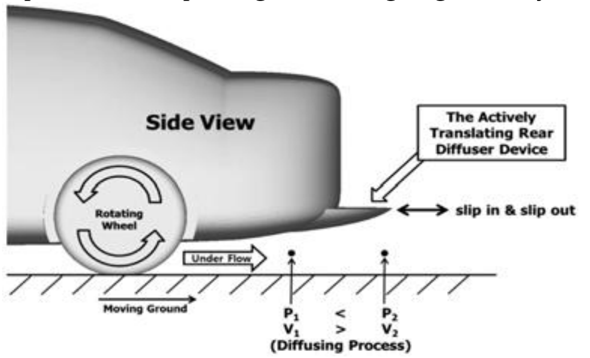


Figure1. Basic concept of the actively translating rear diffuser device.

4.2 CFD ANALYSIS.

In this study, the CFD solver ANSYS.

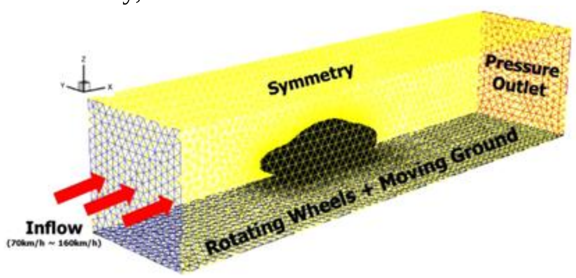


Figure2. Boundary conditions for the CFD analysis.

FLUENT was used. In the course of the analysis, a coupled equation system using a preconditioning method is used to improve the convergence performance. A second-order upwind difference scheme is used as the main scheme to solve the Navier-Stokes equations is used as the turbulence model. A realizable k-e model is chosen in the vicinity of walls because it is known to provide superior performance for flows involving rotation, boundary layers under adverse pressure gradients, separation and recirculation performed a validation of four turbulence models integrated in FLUENT and proved that a realizable k- e model gives the best match with the experimental results.

Finally, all CD values for a passenger car with seven types of rear diffusers are evaluated through steady analysis solutions that are fully converged.

5. RESULT AND DISCUSSION.

5.1 Modeling of Rear Diffuser Device

Seven types of diffuser devices with differing positions, protrusive length, width and height (but the same basic shape) were constructed for assessment in a CFD analysis.

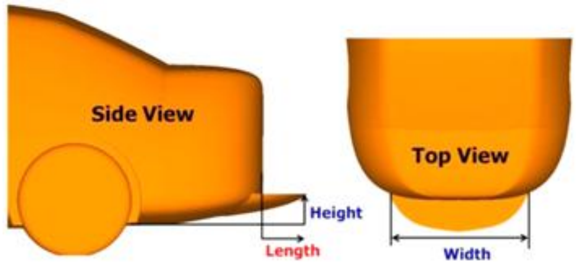


Figure 3. Definition of the rear diffuser size.

Table 1. Specifications of the seven rear diffusers.

	Length (mm)	Width (mm)	Height (mm)
Case 1	100	1518.2	144.5
Case 2	200	1318.2	154.5
Case 3	300	1218.2	174.5
Case 4	350	1218.2	194.5
Case 5	400	1218.2	204.5
Case 6	450	1158.2	224.5
Case 7	500	1078.2	244.5

The length, width and height are shown in Figure 3, and the specifications of the rear diffuser are demonstrated in Table 1. The maximum length of the rear diffuser device was set at 500 mm due to the storage constraint. The passenger car configuration developed in this study has a 500 mm trunk length; above this length, the diffuser device may have to be folded or bent. Except for the different configurations of the rear diffuser devices, all other factors in the automobile configuration and analysis conditions were identical. Figure 4 shows seven automobile configuration cases with the actively translating rear diffuser device compared with the baseline model for the CFD analysis.

5.2. *Aerodynamic Drag Analysis of Passenger Car with Rear Diffuser Device*

The aerodynamic performance of an automobile is estimated by measuring the drag force acting on the external car surface. However, each automobile has a different configuration, meaning that the drag force needs.

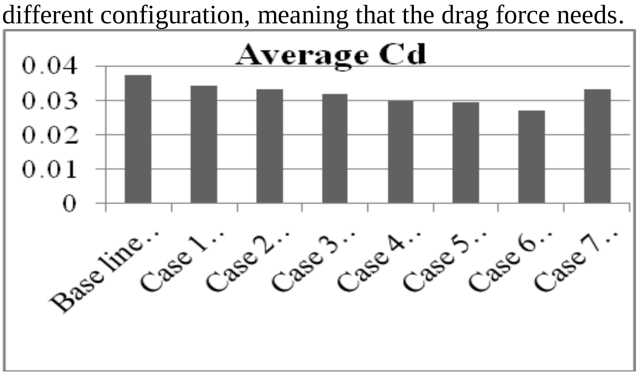


Figure 4. Average CD according to the case number.

To be normalized to compare one to another. The drag coefficient CD is regarded as a major indicator that represents the aerodynamic performance of an automobile. Table 2 demonstrates the CD values from the seven different cases and

Driving speed (km/h)	Base line (0mm)	Case 1 (100 mm)	Case 2 (200 mm)	Case 3 (300 mm)	Case 4 (350 mm)	Case 5 (400 mm)	Case 6 (450 mm)	Case 7 (500 mm)
	C_D	C_D	C_D	C_D	C_D	C_D	C_D	C_D
70	.0262	.0289	.0298	.0337	.0267	.0213	.0203	.0211
85	.0380	.0452	.0508	.0453	.0431	.0430	.0379	.0489
100	.0609	.0646	.0490	.0543	.0528	.0645	.0605	0.645
115	.0803	.0880	.0318	.0765	.0703	.0736	.0737	.0802
130	.0109	.0011	.0596	.0100	.0105	.0010	.0012	.0010
145	.0140	.00136	.0596	.0012	.0015	.0012	.0013	.0012
160	.0174	.0189	.0116	.0015	.0014	.0015	.0001	.0013
Average	0.0373	0.0341	0.0332	.0317	.0299	.0294	0.027	0.033

the baseline case at seven different driving speeds. The average CD values are also shown in the last row to compare the drag reduction effect according to the size of the rear diffuser device. As shown in Table 2, irrespective of the size of the rear diffuser device and the driving speed (70 km/h ~ 160 km/h), every diffuser device induces some CD reduction effects in the automobile.

Table2. Analysis results: CD of the passenger car.

Moreover, as the length of the diffuser increases, the CD reduction effect also increases. Here, Case 6 (450 mm) shows the most effective CD reduction rate, at a value of approximately 4.12%, compared with the baseline, as displayed in Figure 4. Focusing on Case 6, the drag reduction effect of the rear diffuser shows an increase as the driving speed increases, as confirmed in Figures 5. The gap in the required power to overcome the aerodynamic drag force between the baseline and Case 6 becomes wider as the driving speed increases, which indicates that the rear diffuser device is more useful as the automobile travels faster.

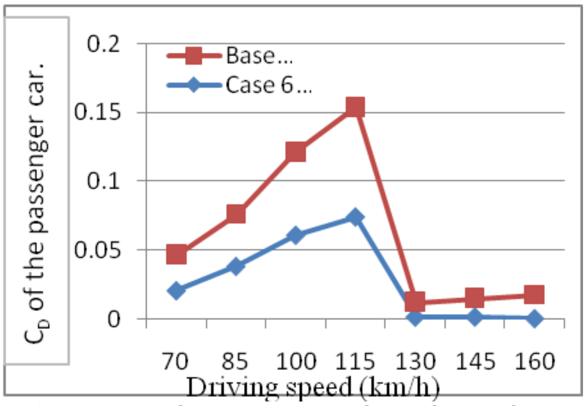
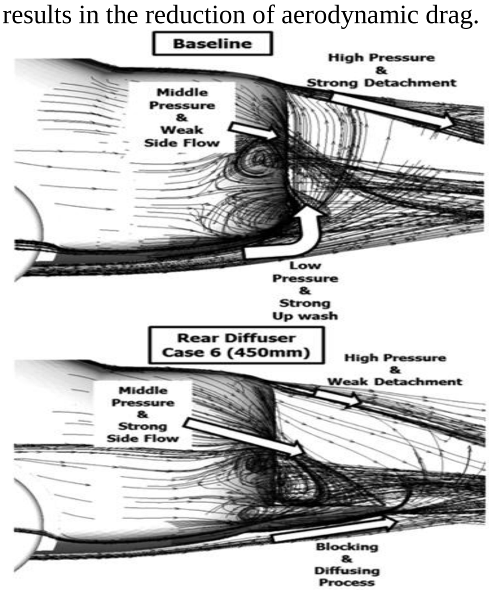


Figure5. Required power to overcome the aerodynamic drag (Base & Case 6).

5.3. Drag Reduction Mechanism Induced by the Rear Diffuser Device

To investigate why the drag reduction mechanism is induced by the rear diffuser device, the pressure distribution contour of the baseline condition was analyzed, as shown in Figure 14. Focusing on the rear body area, the pressure difference appears at the upper flow, the side flow and the under flow. The pressure of the upper flow is the highest, with that of the side flow second and that of the under flow third. As observed in Figure 15, the most evident characteristic of the streamline pattern in the baseline condition is the strong up wash flow that soars up the rear side of the trunk. However, the extracted rear diffuser device blocks out the up wash from the bottom. The space behind the rear trunk surface, which is supposed to be filled with the underbody flow, is instead filled with the side flow and the upper flow passing over the trunk. Therefore, the drag reduction phenomenon is explained in terms of a blocking out of the low-pressure air in its position behind the trunk surface, allowing for relatively high-pressure air from the side and the upper position of the trunk. Finally, the base pressure is increased, which results in the reduction of aerodynamic drag.



6. CONCLUSIONS

In this study, an actively translating rear diffuser device is proposed for automotive aerodynamic drag reduction. The performance of the device was tested through a CFD simulation. From the results, the following conclusions can be made:

(1) Once the rear diffuser device slides out from under the rear bumper, the aerodynamic drag of the passenger car is reduced within the speed range of 70 km/h~160 km/h. Among the various lengths of the diffuser, Case 6(450 mm) showed the best drag reduction performance, with an average reduction of more than 4%. Moreover, as the driving speed increases, the drag reduction effectual so increases.

(2) The automobile drag reduction mechanism is explained as follows. The diffuser device blocks out the low-pressure air from the underbody and allows relatively high-pressure air from the sides and the upper part of the trunk to fill the space behind the trunk, which results in an increase in the base pressure. According to its configuration, the device generates a diffusion effect that increases the underbody flow pressure, thus heightening the base pressure of the car body.

(3) An improvement of fuel efficiency is achieved through aerodynamic drag reduction using the actively translating rear diffuser device when the passenger car is travelling at a high speed. Obtaining the optimal configuration of the actively translating rear diffuser device through an optimization process is saved for future work.

REFERENCES

1. Chien-Hsiung Tsai, Lung-Ming Fu, Chang-Hsien Tai, Yen-Loung Huang, Jik-Chang Leong (2009): “Computational aero-acoustic analysis of a passenger car with a rear spoiler”, pp 3661–3673
2. R. B. Sharma, Ram Bansal (2013): “CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction”, pp 28-35
3. Joseph Katz (2008): “Aerodynamics of Race Cars”, pp 27-63
4. Ram Bansal, R. B. Sharma (2014): “Drag Reduction of Passenger Car Using Add-On Devices”
5. Tank Nilesh R, R. Thundil Karuppa Raj (2012): “Numerical Simulation of Aerodynamic forces acting on Passenger Vehicle While Overtaking”, pp 55-63
6. Jesper Marklund (2013): “Under-body and Diffuser Flows of Passenger Vehicles”
7. Gopal P, Senthilkumar T (2012): “Aerodynamic Drag Reduction In A Passenger Vehicle Using Vortex Generator With Varying Yaw Angles”
8. Mahmoud Khaled, Hicham El Hage, Fabien Harambat, Hassan Peerhossaini (2012): “Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body”
9. Xingjun Hu, Rui Zhang, Jian Ye , Xu Yan , Zhiming Zhao (2011): “Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body”