CFD Analysis of Aerodynamic Aspects of the Generic Car Model

A. Sarkar[1], A. Chittawar[1*], M.S. Khan[2]

[1]B.E. Graduate, St. Vincent Pallotti College of Engineering and Technology, Maharashtra, India
[1]B.E. Graduate, St. Vincent Pallotti College of Engineering and Technology, Maharashtra, India*
[2]Professor, Dept. of Mechanical Engineering, St. Vincent Pallotti College of Engineering and Technology, Maharashtra, India

---

Abstract - *In this research project, modeling, numerical and computational analysis was performed on the rudimentary car geometry for the estimation of all related aerodynamic aspects, and the significance of additional components in reducing the drag values was proven. The focus of this study was to investigate the streamlined airflow patterns over proposed car geometry. Based upon the initially assumed dimensional frame, various possible car geometries were created under specified constraints using the designing software. The analysis was performed considering various boundary conditions and all the considered cases were analyzed at a certain constant velocity condition using Reynold's Average Navier-Stokes (RANS) with the standard k-ϵ model. The calculated values of the drag coefficient and the drag force of the baseline model (Case 1) were found to be 0.964 and 382.2 N respectively. In Case 2, the modifications include the mounting of a continuous contact wing-type spoiler from the rear trunk of the car, and its drag coefficient and drag forces were calculated as 0.7747 and 311.53 N respectively. However, to improve the model specifications, even more, we experimented with other components and the final drag values were found to be 0.6385 (Cd) and 256 N (Fd). The vector plots were used for the examination of the streamlined path for all the models and the drag values were also calculated numerically. It was concluded that additional components were indeed successful in improving the flow pattern of the car and reducing the drag force of the baseline model.*

Key Words: Reynolds – Averaged Navier Stokes, Computer Aided Design, Computational Fluid Dynamics, Drag Coefficient, Drag Force.

1. INTRODUCTION

Vehicles began utilizing streamlined body shapes in the early piece of their history. As engines turned out to be all the more remarkable and vehicles turned out to be faster, automobile engineers understood that air resistance essentially affected their speed. Engineers use computational simulations and air stream explores different avenues regarding scale models and genuine vehicles to tweak the streamlined features of autos so they produce the ideal measure of descending power to the front and back wheels with the least conceivable measure of drag. This can be a far less expensive process than awaiting track results later within the development.

Aerodynamics Drag is a force wherein the approaching air applies to a moving body. It is the opposition offered by the air to the development of the body. The total drag offered by the vehicle is the result of the frontal region and drag coefficient. It is the Cross-Sectional area perpendicular to the direction of motion of the drag on the car. Cd is the drag coefficient and depends on things like the windscreen angle, airflow through the radiator, turbulence on the wheels, etc. So, reducing either the drag coefficient or the frontal area can reduce the total drag.

Reduction of the fuel consumption, particularly on the high velocities throughout vehicle motion on the roads, will be obtained by further vehicles coming up with to cut back air coefficient. This paper shows all the probabilities for the determination of the air coefficient of drag and issues concerning air flows around the vehicle model. This project also deals with one vehicle model with certain magnitude relation and with the procedure to determine the air coefficient of drag. The values of the air coefficient of drag of the car model are derived through the wind tunnel and values of air drag forces for various airspeeds. Analysis of the construction/wind tunnel data started initially with investigating the drag coefficient in isolation. The results of the air coefficient of drag are compared to the constant real value of the original car. The models were analyzed at various velocity conditions with significant implementation of Reynold's average Navier Stokes (RANS) with the standard k-ϵ model [1].

2. LITERATURE SURVEY

The illustration of modeling and analysis of a Sports Utility Vehicle (SUV) for the reduction of aerodynamics drag and lift coefficient which improves its vehicle performance and fuel efficiency. Discrete velocity conditions were considered using Reynold's average Navier Stokes Equations (RANS) with the standard k-ϵ model [1]. Mohan et al. performed a numerical analysis for the estimation of fuel economy for a basic and a modified truck trailer, where all the specifications and basic dimensions of the truck–trailer were recorded and reconstructed accordingly. Numerical Simulations were carried out on the streamlines and the velocity vectors at different α angles which resulted in large variations in the velocity of air near the curves on the surface and the vortexes at the backside of the rear end [2]. A generic model of a Sedan Car was designed with a wing-type spoiler and performed the analysis on the designed vehicle to calculate the changes in drag and lift coefficient acting on it, with and without the spoiler to improve the stability of the vehicle. A virtual wind tunnel was created around the car to investigate the behavior of the streamlined flow of air velocity at the rear end of the vehicle [3]. Parab et al. performed an analysis to calculate the lift and drag force acting on a vehicle at three different angles of the infused diffuser. Increasing the diffuser angle leads to a greater reduction in the lift coefficient but on the other hand, the drag coefficient value was a little higher than expected [4]. Singh et al. focused on redesigning the outer body of Maruti-800 to attain a low drag coefficient value. The secondary aim was to investigate the aerodynamically linked characteristics of the vehicle without using any aerodynamic devices. Observations concluded that the shape of the object and increasing velocity affect the Drag-Coefficient and Drag-force respectively [5]. An effective numerical model was proposed to obtain the flow structure around a passenger car with different add-on devices. Testing and simulation were performed for the calculation of the drag coefficient for passenger vehicles resulting in reducing the drag and lift coefficient value [6]. Mohebbi et al. proposed the research that deals with the numerical analysis of the unsteady aerodynamic performance of a regional train for crosswind stability. The conclusion drawn from the study showed that massively separated flow for the higher yaw angles on the leeward side of the train justifies the use of three-dimensional Reynolds-averaged Navier-Stokes equations (RANS) and turbulence model, where the numerical results show good agreement with test results [7]. A study was carried out to demonstrate the effect of the spoiler on the pressure difference between front and back of passenger cars which affects its motion in the long run. Moreover, the investigation was further extended to the spoiler effect on aerodynamic streamlines, vectors, and drag force [8]. Ruia et al. proposed a numerical simulation of airflow around a car, focusing on the considered lift and drag predictions for the vehicle outer body shape in standard specifications and with an air dam with front splitter and rear wing attached first individually and then in combination. The air dam with the front splitter working in combination with the rear wing was found to be the most aerodynamically efficient configuration [9]. Raina et al. focused on finding the impact of aerodynamic force like bluff bodies on cars with simulation with the help of different turbulence. Different experimental observations with the combination of simulations helped to estimate the flow behavior [10].The impact of the rear spoiler on the Sedan and the front spoiler placed above the front windshield of a hatchback model. The outcomes were compared between both with and without spoiler, and the analysis was carried out using three turbulence models of K-є, RNG k-є, and k-Ø. It was observed that other devices can also be used such as splitters, canards, and rear diffusers, but spoilers can be very effective at high speeds by increasing the downforce [11]. Sudin et al. concentrated on the effect of active and passive flow control on the car's drag reduction. Passive and active methods were used to couple the flow control methods that improved their aerodynamic design. The percentage values of drag coefficient reduction were calculated by the various flow control methods. This was illustrated by the implementation of both active and passive methods to improve vehicle performance [12].

3. DESIGN ARCHITECTURE

The drafting process is the fundamental stage which has various segments which must be followed systematically to induce the planning which is practical to use. Design and drafting provide a step-by-step approach by supporting the wants for building projects.

In this dissertation, we considered three cases to prove the significance of some additional components in improving the aerodynamic efficiency of the car. These additional components (spoilers) are mounted on the existing body of the car.

The outline chosen for the car is a simple geometry that will help to indicate its probable outcomes in complex body type situations too. Modeling of various cases was carried out using the software name CATIA V5 R20. All the models were reduced to the scale of 1:30 to simplify the further calculations.

3.1 Case 1:

In the first case, the car geometry is quite simple with all sharply inclined faces throughout its surface. This surface model was later given a thickness of 0.0025 m on each face and converted to a solid part model. Based on the observations obtained on the above model, the modified versions of the existing geometry would be drafted and imported for further analysis.

The chosen geometry is so simple that it is versatile enough to be worked upon and hundreds of modifications are possible in its geometry; added to this, various additional components can be mounted on the cars. So, taking advantage of the same, it was decided that two of the modification ideas should be adopted in the existing geometry as explained in a later study.

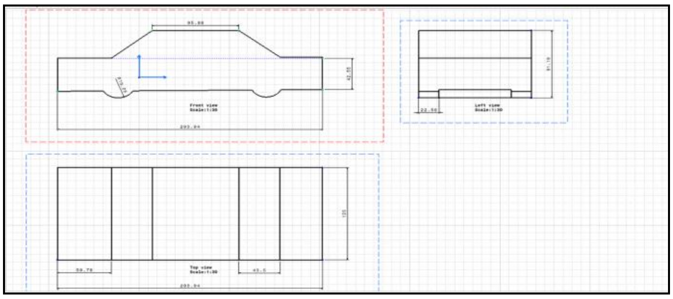

Fig. 1 2D draft of the Generic car geometry

The dimensions of the car are given in the following:

- **Length**: 0.353 m
- **Width:** 0.185 m
- **Height**: 0.151 m
- **Frontal Area of the generic car**: 0.0279 m^2
- **Density of the Medium**: 1.225 kg/m^3

3.2 Case 2:

In the second case, the original car is modified by mounting a continuous contact wing-type spoiler at the rear end of the car.

Wing-type Spoilers create vital downforce and therefore give dependability decreasing lift. Particularly in the rear, there is stream partition because of which vortices are created and they cause a pressure drop in the aft region. The spoiler used for this case is a bit unconventional in nature as it is not open to airflow beneath the upper surface as we usually see. This step was taken to justify the rigidity of the spoiler at higher speeds.

A 2D outline of this simple geometry was drafted using standard dimensions and extruded to a significant thickness using the designing software.

This form of the model was then fit to be imported in the analysis programming for required task computations.

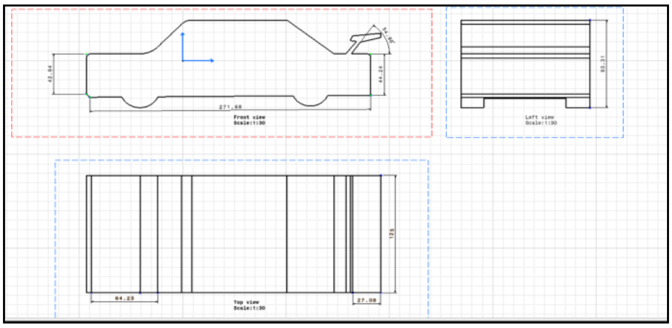

Fig. 2 modeled car geometry having wing type spoiler

The dimensions of the car are given in the following:

- **Length**: 0.345 m
- **Width:** 0.185 m
- **Height**: 0.153 m
- **Frontal Area of the car** : 0.0283 m^2
- **Density of the Medium**: 1.225 kg/m^3

3.3 Case 3:

In the second case, the original car was modified by mounting a continuous contact wing-type spoiler at the rear end of the car. We observed some changes in the flow pattern over the surface of the car. It was also observed that the drag coefficient and the force of drag were also improved after the modification. In all, the car's aerodynamic stability improved. But there were still some limitations linked to the modified model used in the second case.

So, considering the noted limitations, it was decided that the baseline model should be improved by the mounting roof and ducktail spoilers this time.

This may reduce drag in specific cases and generally increase high-speed stability due to the reduced rear lift. This model was imported for analysis and underwent aerodynamic analysis and the required output dealing with the aerodynamic aspects of the car was drawn using suitable boundary conditions.

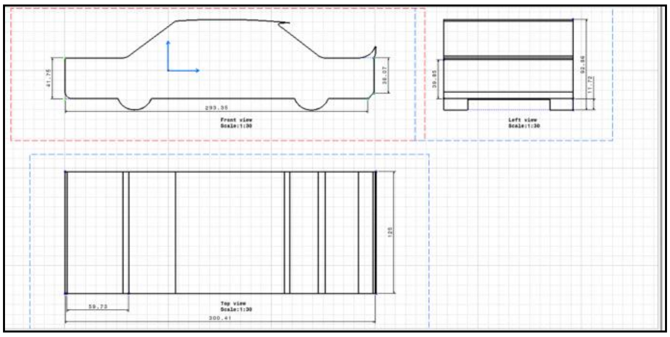

Fig.3 modified car-geometry with Rooftop & Ducktail spoiler

The dimensions of the car are given in the following:

- **Length**: 0.36133 m
- **Width:** 0.185 m
- **Height**: 0.152 m
- **Frontal Area of the car**: 0.0282 m^2
- **Density of the Medium**: 1.225 kg/m^3

4. COMPUTATIONAL ANALYSIS

A generic model of a car was considered with and without a spoiler and it was used for determining its optimum aerodynamic structure. The analysis was performed for discrete models at same velocities and the drag coefficient and drag force was determined. The main variables in the analysis were the frontal area of the car, the drag coefficient and our prime objective was to reduce the drag force by drafting a model having a spoiler. To obtain the streamline air flow structure around the car body with and without the rear spoiler, this study was carried out on a numerical model using the fundamentals of Computational Fluid Dynamics (CFD).

Boundary conditions were considered as an essential component of a mathematical model. They directed the motion of the streamlined air-flow which led to a unique solution. The formula used for the calculation of drag force value is shown below:

$$F_d = \frac{1}{2} \times \rho \times V^2 \times C_d \times A \quad \text{... (1)}$$

In the above equation, density of the medium and velocity of airflow would be considered as constant entities whereas the frontal area and the coefficient of drag value would be a variable entity while determining the drag force for all the cases.

There are some certain assumptions that are to be made and certain values that are need to be calculated from the simulation software in order to compute the drag force.

- ρ is the density of the medium created in which the object is subjected to be in motion. In our study, to know the density of air, we need the temperature of the medium, which is assumed to be at 288.16 K. Referring the Engineering Toolbox table, 1.225 kg/m^3 is a good approximation for the density of air at this temperature.

- V, the velocity of the airflow in the medium against the car is assumed to be 100 km/hr i.e., 27.77 m/s.
- Cd, is the coefficient of drag of the car and it is a variable for all the cases.
- A, is considered as the frontal area of the car.

5. RESULTS

The computational and numerical analysis results for the three cases mentioned above are as follows:

5.1 Case 1:

The (Cd) value is calculated using the analyzing software in which the drag coefficient value is determined by performing 50 iterations. The graph between the Cd value and the no. of iterations performed is shown below. In the y-axis we are considering the Cd value and in the x-axis we are considering the no. of iterations performed. Therefore, In this case, the drag coefficient is found to be 0.964. For this value of Cd, the force of drag is calculated.

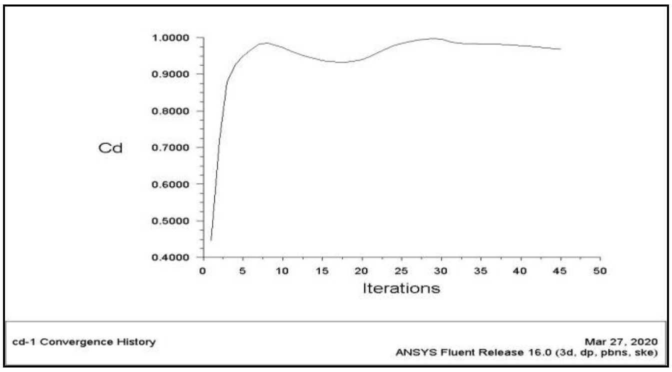

Fig. 4 Cd vs. no. of iterations performed for Case 1

The drag force value varies with the drag coefficient and the frontal area of the car. As these values change with the boundary conditions, the force of drag also changes. Whereas in this case, the drag force calculated in the analysis of the generic model of the car is given below:

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
vehicle_body	12.415553	0.32745291	12.743006	0.93923503	0.024771771	0.9640068
Net	12.415553	0.32745291	12.743006	0.93923503	0.024771771	0.9640068

Fig. 5 Net drag force acting on the body for Case 1

In this case, the force of drag calculated by the software analysis is equal to 12.74 N for a drag coefficient of 0.964 at 100 km/hr (27.77 m/s) velocity of the airflow. The airflow motions are shown in the following subsections.

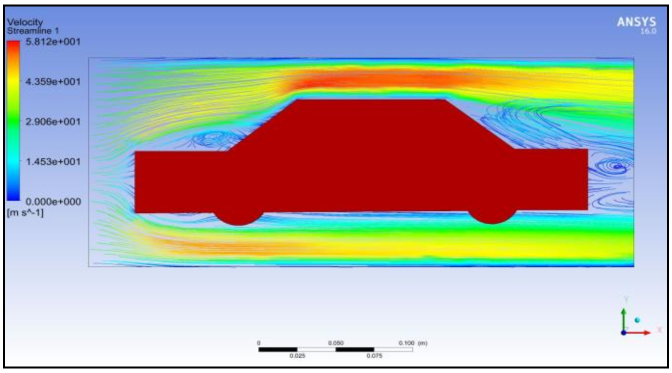

Fig. 6 Streamline airflow of the isosurface plane for Case 1

It was nearly impossible to calculate the force of drag numerically at every instant of the varying velocity of the car. So, for our study, we calculated the drag force of the car at a certain fixed velocity with the following observations.

Now, when we substitute the values in our main equation (1), force of drag value was found out to be,

$$F_{drag} = \frac{1}{2} \times (1.225) \times (27.7)^2 \times (0.964) \times (0.0279) = 12.74 \text{ N}$$

As mentioned earlier, in order to simplify the calculations, the actual dimensions were reduced to the scale of 1:30.

Actual Force of Drag = 12.74 ×30 = 382.2 N

5.2 Case 2:

The drag coefficient was found to be 0.774. For this value of Cd, the force of drag was calculated.

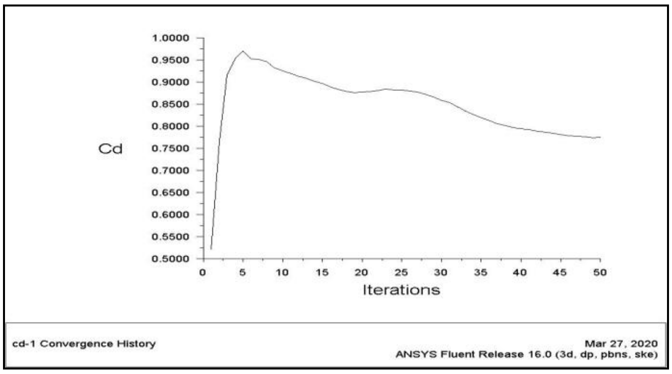

Fig. 7 Cd vs. no. of iterations performed for Case 2

The drag force calculated in the analysis of this car model having wing type spoiler at its rear end is given below:

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
vehicle_body	9.9924912	0.39255553	10.385047	0.74547217	0.029285913	0.77475808
Net	9.9924912	0.39255553	10.385047	0.74547217	0.029285913	0.77475808

Fig. 8 Net drag force acting on the body for Case 2

The drag force obtained by the analysis is equal to 10.38 N for drag coefficient of 0.774 at 100 km/hr air velocity.

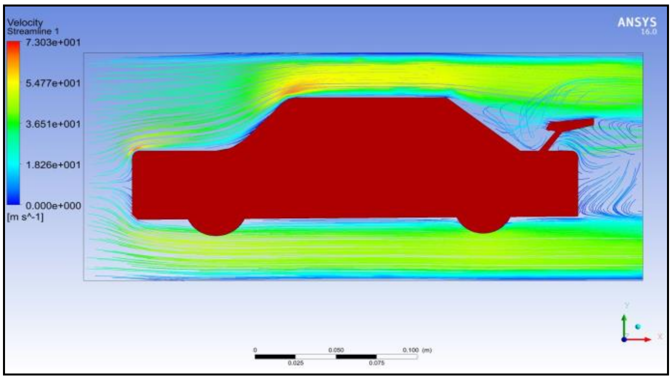

Fig. 9 Streamline airflow of the isosurface plane for Case 2

As we have calculated the value of drag force for the baseline model, we are attempting to reduce it by adding a rear spoiler to the car and changing the flow patterns of the air nodes. While analyzing the flow pattern of the modified car, it was observed that the flow patterns changed significantly and the drag coefficient value decreased provided other values of the equation remained remarkably unchanged.

Substituting all the values in the equation (1), the new drag force was found to be,

$$F_{drag} = \frac{1}{2} \times (1.225) \times (27.7)^2 \times (0.774) \times (0.0283) = 10.38 \text{ N}$$

To convert the values in accordance with the real dimensions,

Actual Force of Drag = 10.38 ×30 = 311.53 N

It was observed that the spoiler we employed improved the aerodynamic aspects of the vehicle by reducing drag force by 2.5 N. However, the overall structure of this spoiler still had some limitations and there was a visible scope of improving the results even more if a rather better model was taken into consideration.

5.3 Case 3:

The drag coefficient is found to be 0.63853696, for which the drag force was calculated, for the car having wing type spoiler at its rear end is given below:

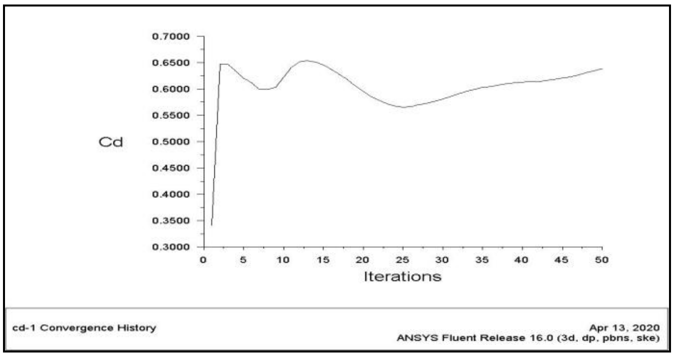

Fig. 10 Cd vs. no. of iterations performed for Case 3

Forces - Direction Vector (1 0 0)

	Forces (n)			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
vehicle_body	8.130597	0.40333179	8.5339288	0.60835833	0.030178627	0.63853696
Net	8.130597	0.40333179	8.5339288	0.60835833	0.030178627	0.63853696

Fig. 11 Net drag force acting on the body for Case 2

The force of drag obtained by the computational analysis was equal to 8.53 N for drag coefficient of 0.638 at 100 km/hr (27.77 m/s) air velocity. The airflow motions are shown in the following subsections.

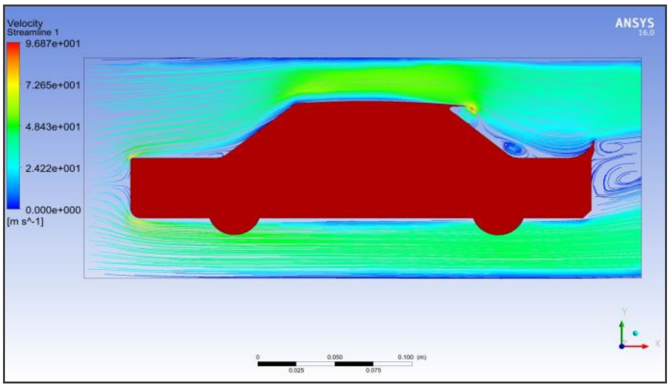

Fig. 12 Streamline airflow of the isosurface plane for Case 3

We calculated the drag force for the baseline model with a wing type spoiler mounted on the rear trunk of the car, the results were satisfactory, but our main aim still lies with reducing the drag force of the same car even more by getting rid of the limitations possessed by the wing that we employed. So, the wing type spoiler was now replaced by the roof spoiler added with a ducktail spoiler mounted on the rear trunk of the car to acquire the below observations. While analyzing the flow pattern of the modified car, it was observed that the flow patterns changed remarkably and the drag coefficient value decreased provided with use of the rear wing. But when roof and ducktail spoilers were used as a replacement for the rear wing, following results were obtained.

Substituting all the values in the equation (1), the desired drag force was found to be,

$$F_{drag} = \frac{1}{2} \times (1.225) \times (27.7)^2 \times (0.638) \times (0.0282) = 8.53 \text{ N}$$

To convert the values in accordance with the real dimensions,

Actual Force of Drag = 8.53 ×30 = 256.00 N

6. CONCLUSION

On performing the analysis with the generic model of the automobile body using three different cases at a given constant speed of 27.77 m/s, the drag coefficients acting on it were determined by performing 50 iterations for each case. For the first case, we considered the generic model of the car geometry in which, the drag coefficient and drag force was found to be 0.964 and 382.2 N respectively. For the second case, the car geometry was further improved with the addition of an extra components i.e., a wing spoiler mounted on the rear trunk of the car. For this case, drag force and drag coefficient calculated was equal to 311.53 N and 0.774 respectively.

It was evident that the results surpassed after the modifications implemented on the baseline geometry model. But, to improve the aerodynamic stability even more, the wing type modification was replaced by roof and ducktail spoilers and the results obtained were 256.00 N of drag force and the drag coefficient was equal to 0.638.

From the study, it was concluded that the additional components mounted on the base model of the car improved the aerodynamic stability of the car at a uniform speed. They were successful in improving the flow pattern of the car and reducing the drag force of the baseline model.

The following table shows all the analysis results obtained in this study:

Table -1: Analysis of Drag Force values Drag coefficient values for all the 3 Cases

Case	Car type	v (m/s)	A (m^2)	C_d	F_d (N)
1	Generic car	100	0.0279	0.964	382.2
2	Car with wing type spoiler	100	0.0283	0.774	311.53
3	Car with rooftop spoiler and ducktail	100	0.0282	0.638	256.00

REFERENCES

[1] G. S. Samy, S. T. Kumaran, M. Uthayakumar, M. Sivasubramanian and K. B. Sankar, "Numerical analysis of drag and lift coefficient of a Sport Utility Vehicle (SUV)," International Conference on Recent Advances in Fluid and Thermal Sciences(ICRAFTS), vol. 10, 2019, no. 1088,pp.1- 8, doi:10.1088/1742-6596/1276/1/012013.

[2] J. K. M. Mohan and B. Venkateshwarlu, "Numerical investigation on the aerodynamics and fuel consumption of a truck-trailer," International Journal of Applied Engineering Research(IJAER), vol. 9, 2014, no. 24, pp. 28957 - 28969.

[3] D. Patel, J. Jangid, Y. Devani, S. Goswami and M. Vachhani, "CFD Analysis on Aerodynamics of an Automobile Body," International Journal of Scientific Research in Science, Engineering and Technology (IJSRSET), vol. 4, 2018, issue 4, pp. 778-783.

[4] A. Parab, A. Sakarwala, B. Paste, V. Patil and A. Mangrulkar, "Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5," International Journal of Recent Technologies in Mechanical and Electrical Engineering (IJRMEE), vol. 1, 2014,issue 4, pp. 007-013.

[5] S. P. Singh and A. Singh, "Design and Simulate an Aerodynamic Car Body for The Maruti Suzuki 800 With Less Coefficient of Drag," International Research Journal of Engineering and Technology (IRJET), vol. 03, 2016, issue 06, pp. 299-303.

[6] R. Bansal and R. B. Sharma, "Drag Reduction of Passenger Car Using Add-On Devices," Journal of Aerodynamics, vol. 2014, no. 678518, pp. 1-13, doi: 10.1155/2014/678518.

[7] M. Mohebbi and M. A. R. Baboli, "Numerical Analysis of Aerodynamic Performance of Regional Passenger Train under Crosswind Conditions," International Journal of Vehicles Structures & Systems (IJVSS), vol. 5, 2013, no. 2, pp. 68-74, doi: 10.4273/ijvss.5.2.05.

[8] A. B. Mohamed, M. S. Khatab and I. M. Ibrahim, "Computational Investigation of Spoiler Effect on the Aerodynamic Performance of Passenger Cars," International Undergraduate Research Conference (IUGRC), 2018.

[9] S. Ruia and A. Dixit, "CFD Study of Aerodynamic performance of a popular Vehicle's outer body Shape and Analysis of the effect of Aerodynamic Aids," International Journal of Mechanical Engineering and Technology (IJMET), vol. 6, 2015, issue 10, pp. 171-193.

[10] A. Raina and A. Khajuria, "Assessment of Flow Control using Passive Devices around Bluff Bodies," International Journal of Research in Applied Sciences and Biotechnology (IJRASB), vol. 6, 2018, issue 1, pp. 1-5.

[11] V. Yakkundi and S. Mantha, "Effect of Spoilers of Aerodynamics properties of a car," International Journal of Science Research and Review (IJSRR), vol. 7, 2018, issue 3, pp. 271-280.

[12] M. N. Sudin, M. A. Abdullah, S. A. Shamsuddin, F. R. Ramli, M. M. Tahir,, "Review of Research on Vehicles Aerodynamic Drag Reduction Methods", International Journal of Mechanical & Mechatronics Engineering (IJMME), Vol. 14, 2014, No. 2, pp. 35-47.