Numerical analysis of drag and lift coefficient of a Sport Utility Vehicle (SUV)

G S Samy¹, S Thirumalai Kumaran¹*, M Uthayakumar¹, M Sivasubramanian¹ and Krishna Bhagavathi Sankar²

¹Faculty of Mechanical Engineering, Kalasalingam University, Tamil Nadu, India
²Department of Automobile Engineering, Kalasalingam University, Tamil Nadu, India

*Email: thirumalai.kumaran@yahoo.com

Abstract. The paper describes the modeling and analysis of a Sport Utility Vehicle (SUV). The present study mainly focuses on the reduction of aerodynamics drag and lift coefficient of a car, which helps to improve the vehicle performance and fuel efficiency. Usually the aerodynamic drag and lift is insignificant at low vehicle speed but the magnitude of air resistance becomes considerable with accelerating the vehicle. The Computational Fluid Dynamics (CFD) helps to analyze the flow stream around a SUV. The modeling software is used to design a 3D car model and the flow stream is analyzed by ANSYS Fluent. Initially, the first model was analyzed at various velocity conditions using Reynolds-Averaged Navier–Stokes (RANS) with standard k-ε model. The basic model of SUV has a drag coefficient and lift coefficient value of 0.62 and 0.099 respectively. Initially the first model is designed by introducing air splitter on roof and based on the results the second model is designed by introducing 15 degree boat tailing angle with first model. The drag reduction is reduced by 6.45% for first model and 19.35% for second model compared to the base model. Similarly, the lift coefficient is reduced by 2.02% for first model and 30.30% for the second model. Moreover the second model produces low drag and lift coefficient than first and base model. A significant reduction is observed in first model when compared to the base model. The vector plots are used to examine the streamline path of both the models.

1. Introduction
In a modern automotive vehicle, the speed is considerably as same as a sports car. It is also essential to reduce the aerodynamic drag and lift coefficient of a modern vehicle to enhance the performance [1]. The drag and lift forces can be expressed in a non-dimensional form - the drag ($C_D$) and lift ($C_L$) coefficients, defined as:

$$C_D = \frac{Drag\ force}{(\rho V^2 A)/2}$$  \hspace{1cm} (1)$$

$$C_L = \frac{Lift\ force}{(\rho V^2 A)/2}$$  \hspace{1cm} (2)$$

where $\rho$ describes the density of air, $V$ denotes the velocity of vehicle and $A$ represents the frontal area of car [2,3].

Recently many researches have been investigated through the numerical study as it is cost effective, less time consumed when compared to the experimental test. The Computational Fluid Dynamics
(CFD) is a tool used to analyze the aerodynamic performance of an automotive vehicle [4–7]. Reynolds- Averaged Navier – Stokes (RANS) is a model for simulating the aerodynamics profile of automotive vehicle. Since it can accurately capture the near-wake vertical structure of flow [8].

Sneh Hetawal et al. [9] numerically investigated the rear engine SAE race car with and without front spoiler and with firewall vents. The RANS with standard k-ε is used to analyze the model. The result reveals that the front wing modification provides a reduction of drag coefficient from 0.85 to 0.70 when compared to standard race car. The lift coefficient is increased to 0.25 for the model with firewall vents. Song et al. [10] used the artificial neural network to optimize the shape of rear end of the sedan car. The optimized shape provides 5.64% reduction when compared to the baseline model. Altinisik [11] has analyzed the engine cooling air flow with three configuration as follows: i) closed with smooth under body, ii) closed with detailed under body and iii) open with detailed under body. The detailed under body airflow increases the drag coefficient by 15%. The numerical result of the pressure distribution and drag coefficient exhibited excellent agreement with wind tunnel and coast down test. Song Wook Lee [12] studied the flow behaviour with respect to different wheel deflector non-dimensional heights and air jet deflector. They found that the air jet deflector effectively reduced the drag by 6.4% than conventional wheel deflector. Siva and Loganathan [13] analyzed the aerodynamic performance of Toyota Fortuner. Solid works modeling software is used to design the 2D car model and CFD is performed to analyze the designed model at a vehicle speed of 30 m/s. The C-post angle is selected to optimize the drag. The existing angle of 45 degree is analyzed and the results are compared with modified angle of 35 degree. The modified model offer better reduction in drag and increase in the vehicle performance.

2. Methodology and Numerical Procedure
The TATA SAFARI vehicle is selected to analyze the aerodynamic performance. The TATA SAFARI has a length of 4655 mm and a width of 1855 mm. The windshield angle of 32 degree is at 100 mm from the front wheel center. The boat tail angle of 0 degree is at 140 mm behind the rear wheel center. A one-third scaled down model is used to reduce the computational time. Figure 1 depicts the two dimensional design of the vehicle. Three models such as base, first and second are considered in this study. The actual design of TATA SAFARI is consider as a base model and the three dimensional view of the base model is shown in Figure 2 (a). In first model the air splitters is introduced from the B pillar to achieve the attached stream flow on the roof. The second model (Figure 2 (b)) is designed by inclining the boat tailing angle by 15 degree with air splitters to obtain effective vortex flow at rear end.

The ANSYS Fluent R 14.5 is used to analyze the aerodynamic drag and lift of SUV. The solver and model setting details are listed in Table 1 and 2. The inlet boundary conditions are set with a velocity inlet of 35 m/s and 34 degree atmospheric temperature. The gauge pressure of zero Pascal is the outlet boundary condition. The k-ε is selected to analyze the 3D model [14]. The car contour, top and bottom of the virtual wind tunnel are set as walls with no slip conditions. The air acts as a fluid medium with ambient condition of 1.225 kg/m³ density and 1.7894 x 10⁻⁵ kg/ms viscosity. The turbulent kinetic energy and turbulent dissipation rates are 1m²/s² and 1m²/s³ respectively. The meshing details of all the three models are shown in Table 3.

| Table 1. Solver details |
|-------------------------|
| CFD simulation: 3D       |
| Solver: Pressure based   |
| Space: 3D               |
| Formulation: Implicit    |
| Time: Steady            |
| Velocity Formulation: Absolute |
| Gradient Option: Cell-Based |
Figure 1. Two dimensional view of a SUV

Figure 2. (a) Basic model (b) Second model (air splitter with 15 degree boat tailing angle)

Table 2. Model settings

| Viscous Model | Turbulence Model | k-\(\varepsilon\) |
|---------------|-----------------|-----------------|
| k-epsilon Model | Standard Wall Functions |
| Near-Wall Treatment | Standard Wall Functions |
| Wall Zones | No Slip |
| Materials | Fluid - air |
| Operating Conditions | Ambient |
| Total Kinetic Energy Prandtl Number | 1 |
| Total Dissipation Rate Prandtl Number | 1.3 |
Table 3. Meshing details of various model

| Types of model | Number of elements | Number of nodes |
|----------------|--------------------|----------------|
| Base model     | 271900             | 50416          |
| First model    | 279968             | 51846          |
| Second model   | 299062             | 55382          |

3. Result and Discussions
The CFD simulation is initially performed to study and analysis the aerodynamic performance of a basic model. Further, it is used to identify the flow pattern and encounter the regions responsible for high drag and lift coefficient. The reduction in high pressure region and increases in attached flow over a vehicle are main key factor to reduce the drag and lift force. Initially the basic model was tested at an inlet velocity of 35 m/s with zero Pascal outlet pressure. The base model is analyzed and found that the high pressure generated at the front and the rear end. The analysis of the basic shape presents the value of drag coefficient as 0.62. Primarily the drag coefficient has been targeted to be reduced. In order to reduce the drag, the air splitter is introduced at the roof to achieve the attached streamline flow. The pressure contour plot Figure 3 (a) clearly shows the pressure distribution along the longitudinal length of the car. Also it is found that, the significant reduction of pressure at roof due to the acceleration off low velocity. In addition to that, at rear end and under body the pressure drop is absorbed which affect drag and lift coefficient. Figure 3 (b) shows the static pressure distribution of first model. The plot reveals that the maximum static pressure is caused in front end of the SUV. Furthermore, it is reduced over the bonnet and slightly increased in the intersection of bonnet and windscreen. The continuous reduction profile of static pressure is noted over the roof but at the end the pressure drastically increased due to the waking of under body flow interact with separation of roof flow.

Figure 4 (a) depicts the strong vortex of flow at end of the wind screen. Also it is noted that, the huge wake raised from the under body of the SUV [15]. This action create high pressure vertex on the rear end of the vehicle and it should be avoided to reduce the lift force of a vehicle. The velocity streamline plot explains that the flow around the vehicle as shown in Figure 4 (b). At the rear end, further decrease in the velocity which causes high pressure turbulence in the rear end of the vehicle. At the front of the vehicle the resistance created which continues at the rear of the vehicle. The turbulence created at the rear of the vehicle tends the vehicle to lift. Hence the flow separation must be happened at certain distance behind the vehicle in order to reduce the lift. Finally it is found that the significant reduction in drag coefficient of 0.58 and lift coefficient of 0.097. Hence the rear end modification has to be done on the profile to get reduce turbulence created at the rear end of the vehicle.
Figure 3. (a) Contour plot of first model, (b) Static pressure plot of first model

Figure 4. (a) Velocity contour of first model, (b) Streamlines trace of first model
Based on the result obtained from the first model it is decided to change the boat tailing angle. The 15 degree boat tailing angle is introduced to reduce turbulence in rear end by wake up the flow from boat tail. The similar model and boundary conditions are used to analyze the second model. The contour plot Figure 5 (a) shows the improvements in flow separation on the vehicle profile. The pressure drop was increased by allowing the underbody flow to the rear end of the vehicle. Also it is found that the increase in streamline flow region over the roof of a SUV. Figure 5 (b) depicts the static pressure plot of a second model SUV. At the front end the static pressure profile curve is similar to the first model. The plot reveals the reduction in turbulence at rear end by reduce the static pressure. At the end scattered static curve reveals the increase in velocity of upper body flow.

Figure 6 (a) shows the velocity contour plot of second model. From the image it is clearly noted that, the velocity increased at the end of under body due to the boat tailing angle. Also this flow is directed towards the downstream flow from air splitter. Due to this action the strong vortex is generated in rear end which causes high pressure in the rear body and reduces the lift coefficient. Since the flow from the air splitter is compressed and it is stretched towards the longitudinal length of a vehicle due to the generation of rear end strong vortex. Streamline profile of both sir splitter and 15 degree boat tailing angle is shown is Figure 6 (b). The flow from the air splitter is restricted in downwards direction and maintained its own streamline by the up wash moments of vortex [16]. This reduces the high pressure region in roof and rear end which reduce the drag and lift coefficient of 0.50 and 0.069 respectively.

Figure 5. (a) Contour plot of second model, (b) Static pressure plot of second model
Figure 6. (a) Velocity contour plot of second model, (b) Streamlines trace of second model

Figure 7. Drag and lift coefficient at V=35 m/s
Figure 7 shows the reduction of drag and lift coefficient of base, first (air splitter at roof) and second model (both air splitter at roof and 15 degree boat tailing angle). The drag is reduced by 6.45% for first model and 19.35% for second model compared to base model. Similarly, the lift coefficient is reduced by 2.02% for first model and 30.30 % for second model. Moreover the second model produces low drag and lift coefficient than first and base model.

4. Conclusion
The study numerically investigates the aerodynamic performance of SUV model. The modeling software is used to design a 3D model. The original shape of the model is considered as a base model and it’s analyzed to find the actual drag and lift coefficient. Later, the air splitter is introduced to analyze the flow behaviour and aerodynamic losses. Based on the result obtained from the first model the second model is designed by introducing a 15 degree boat tailing angle. The same boundary conditions are used for the entire model at a vehicle velocity of 35m/s. The basic model has a drag coefficient and lift coefficient value of 0.62 and 0.099 respectively. The drag is reduced by 6.45% for first and 19.35% for second model compared to base model. Similarly, the lift coefficient is reduced by 2.02% for first model and 30.30 % for second model. Moreover the second model produces low drag and lift coefficient than first and base model. The slight significant reduction is observed in fist model then base model. The vector plot are shows the streamline path of both models.

References
[1] Mosaddeghi F and Oveisi M 2015 Journal of Central South University 22 4645
[2] Hamut H S, El-emam R S and Dincer I 2014 International Journal of Numerical Methods for Heat & Fluid Flow 24 627
[3] Kim I, Chen H and Shulze R C 2006 SAE Technical Paper 13
[4] Murukesan P, Mu'tasim M A N and Sahat I M 2013 IOP Conf. Ser.: Mater. Sci. Eng. 50 012039
[5] Essaghouri Abdellah and Bo Wang 2017 IOP Conf. Ser.: Mater. Sci. Eng. 231 012173
[6] Huminic A and Huminic G 2017 International Journal of Automotive Technology 18 397
[7] Emmanuel Guilmineau 2008 Journal of Wind Engineering and Industrial Aerodynamics 96 1207
[8] Christopher Roy, Jeffrey Payne and Mary McWherter-Payne 2006 Journal of Fluids Engineering 128 1083
[9] Hetawal S, Gophane M, Ajay B K and Mukkamala Y 2014 Procedia Eng. 97 1198
[10] Song K S, Kang S O, Jun S O, Park H I, Kee J D, Kim K H and Lee D H 2012 International Journal of Automotive Technology 13 905–14
[11] Altinisik A 2017 International Journal of Automotive Technology 18 245
[12] Lee S W 2018 Microsystem Technologies https://doi.org/10.1007/s00542-018-3992-1
[13] Siva G and Loganathan V 2016 Middle-East Journal of Scientific Research 24 133
[14] Kieffer W, Moujaes S and Armbya N 2006 Mathematical and Computer Modelling 43 1275
[15] Sterken L, Sebben S and Löfdahl L 2017 Journal of Fluids Engineering 138 14
[16] Hassan S M R, Islam T, Ali M and Islam Q 2014 Procedia Eng. 90 308