CFD Study of Aerodynamic Flow around Generic Sports Utility Vehicle

Wael Mokhtar and Samira Ishrat Jahan,
School of Engineering, Grand valley State University,
Grand Rapids, Michigan, USA.

Abstract

Sports Utility Vehicle, commonly known as SUV is one of the most popular vehicles which dominates the global automobile market. Due to having a number of useful features like large passenger and goods carrying capacity, robust structure, better towing capacity, aesthetics etc., it is gaining attention from both automakers and customers. Starting from Willys Jeep Station Wagon (1946), SUVs have gone through versatile change in model to become more aerodynamic, aesthetic and to have modern features. This study focuses on the flow around a generic Sports Utility Vehicle which has been modeled based on the SUV models available in market. Although the model has been kept simple, it covers all the basic features of a modern Sports Utility Vehicle. For the CFD simulation, a mesh model consisting of polyhedral mesh, prism layer and surface re-mesher, has been developed. The CFD study performed in this study, has used K- ε turbulence model with Renolds Average Navier Stokes (RANS) equations. This study gives a clear idea about the flow pattern around a generic SUV along with aerodynamic drag and lift forces.

Key words: SUV, CFD, aerodynamic, flow structure

Introduction

SUV became popular in the United States, Canada, India, and Australia in the 1990s and early-2000s. U.S. automakers could enjoy profit margins of $10,000 per SUV, while losing a few hundred dollars on a compact car [1]. In the United States, four-wheel-drive SUVs are considered light trucks.

Though SUV sale declined temporarily because of its low fuel efficiency in 2010, it has become one of the world's largest automotive sections since 2015. It dominates almost 23 % of global light vehicle sales and almost 37 % of world's passenger car market [2].

The size of SUV varies in a wide range from small size weighing 1 ton to large size weighing up to 3 tons. Today automobile companies are trying to build a good image of SUVs by making them more aerodynamically efficient to reduce aerodynamic drag without compromising the aesthetics.

Because of size and stronger chassis, it can carry more weight and offer more comfortable seating with spacious interior, and ample leg and head room. Due to its higher ground clearance, it makes driving smoother on rugged terrain. SUV is compatible for all kinds of weather conditions, as they weigh more enabling better handling. The bigger and wider wheels allow better grip and more stability on the road making driving easier in difficult conditions.

Literature Reviews

I. What is CFD?

CFD (Computational Fluid Dynamics) is a branch of Fluid Dynamics, that defines a mathematical model of a physical problem and the flow is analyzed through numerical method with the help of software [12]. The physical properties of the fluid like pressure, temperature, velocity, viscosity, density play significant role in generating solution for a physical phenomenon related to fluid flow.

II. What is Aerodynamic Drag?

The major objective of vehicle aerodynamic analysists is decreasing drag and keeping lift at a desired level without compromising its speed or any other significant features. Sometimes producing downforce for better traction and stability becomes a matter of concern. This study focuses mainly on the aerodynamic drag of SUV.

Aerodynamic drag can be defined as friction force or resistance that acts in the opposite direction of the relative motion of an object [3]. Drag force depends on the properties of both the fluid and the object. The drag force can be computed from the following equation:

$$F_D = \frac{1}{2}\rho v^2 C_D \tag{1}$$

where,
F_D is the aerodynamic drag force
ρ is the fluid density
v is the speed of the object moving relative to the fluid
C_D is the coefficient of drag which depends on Reynolds number (R_e)

$$R_e = \frac{\mu D}{\rho v} \tag{2}$$

where,
Re is the Reynolds number
D is the characteristic diameter
μ is the fluid viscosity

From equation (1) it can be easily understood that drag force will be higher when the vehicle will move at higher speed.

SUVs having a greater ground clearance have more aerodynamic drag. It's due to the turbulence created underneath the chassis. Also when the wheels/tires are massive, the rolling of the wheels/tires create more spiral motion of air and thus more turbulence is developed causing more drag force [4].

The greatest challenge for the vehicle manufacturers is to design and develop smoother and aerodynamic shapes to reduce the SUV's coefficient of drag. If any sudden change of the cross-sectional area can be avoided the flow pattern becomes aerodynamic.

Model Design

One of the most important parts of this study was to model a SUV that goes similar with the present models available. Nowadays the models have been made to be more aerodynamic compared to the box type SUVs used before as discussed earlier in this study. In this study a generalized model has been designed based on the contemporary SUV models in market like Toyota Rav4, Honda CRV, Subaru Cross trek etc. The model is designed in such a way that the air flows more smoothly around the vehicle. The model used in this study has been kept as simple as possible. It has the side mirrors and a simple front radiator.

The over-all length of the model is 3982.41 mm, the width is 1896.68 mm and height is 1449.02 mm. The wheel to wheel distance in lengthwise direction is 2620 mm for the model and in widthwise direction the wheel to wheel distance is 1396.68 mm.
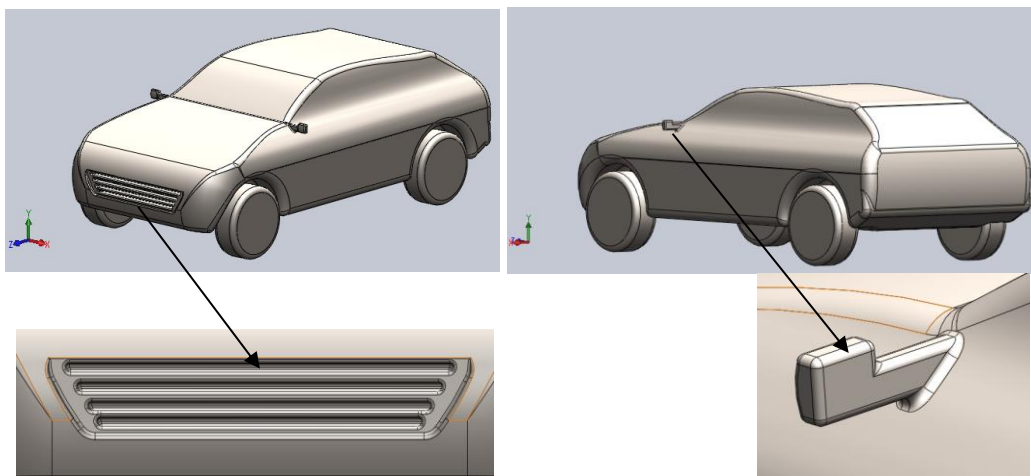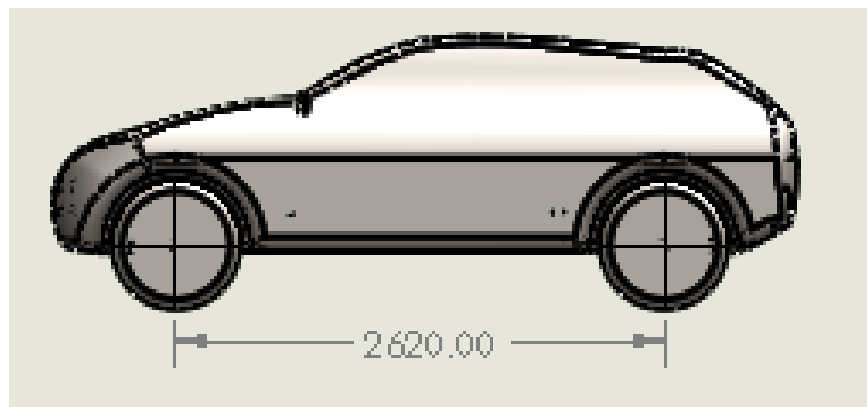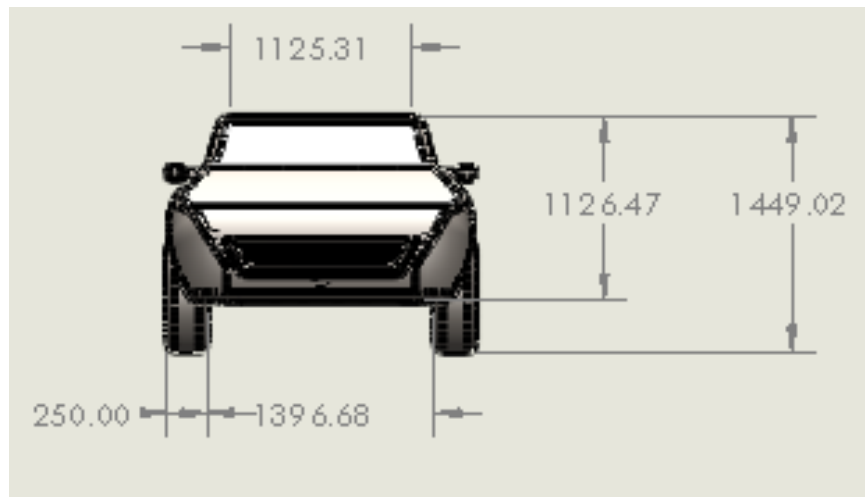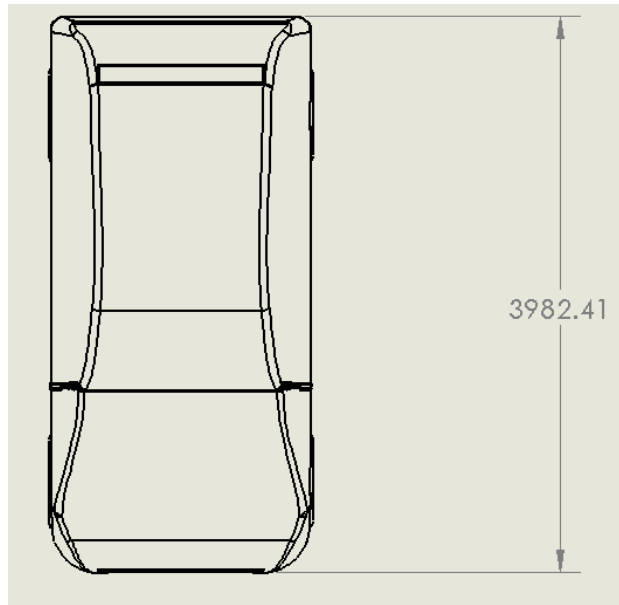

Figure 1: Isometric View

Figure 2: Dimention Details

CFD simulation

The domain is 6 times longer than the SUV, 3 times in the front and 3 times in the back. The height of the domain is 4.3 times to the height of the SUV. The width of the domain is 3 times the width of the SUV [Figure 3]. The reference area for calculation of drag coefficient is 2.081617 m² and 1.017707 m² for full and half-symmetric model respectively. The coordinate system used for the study is also shown in Figure 4. The free stream flow is in negative x-axis. The y-axis is in vertical direction and z-axis is in lateral direction.
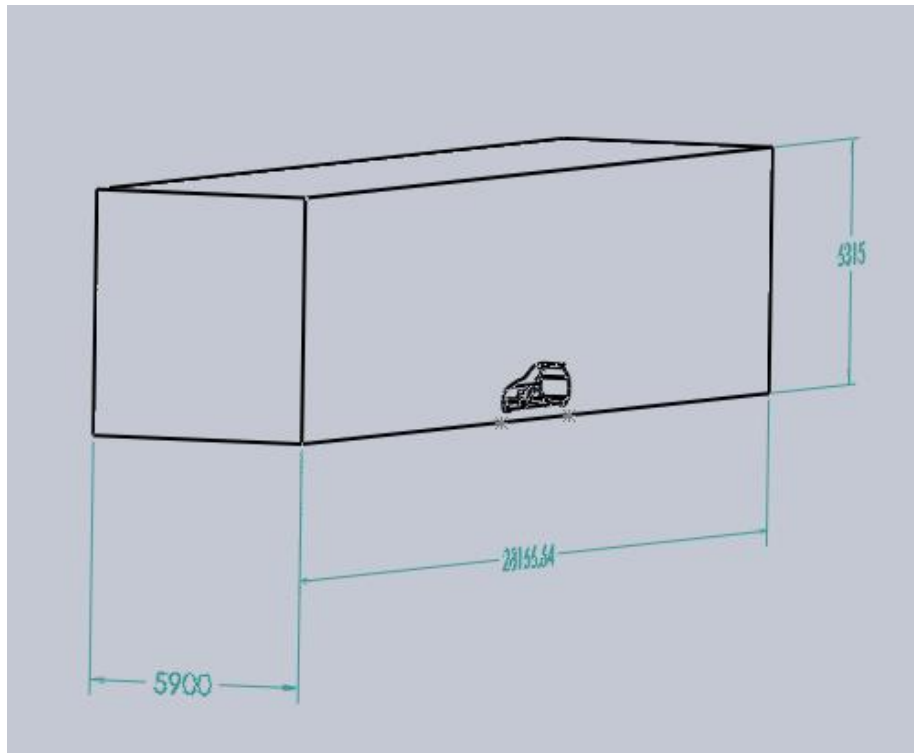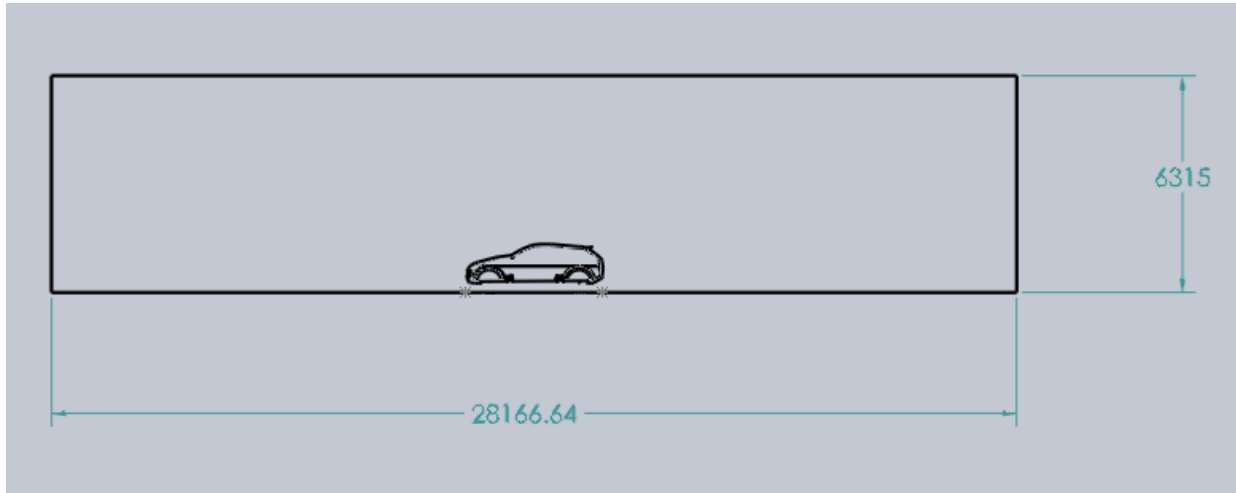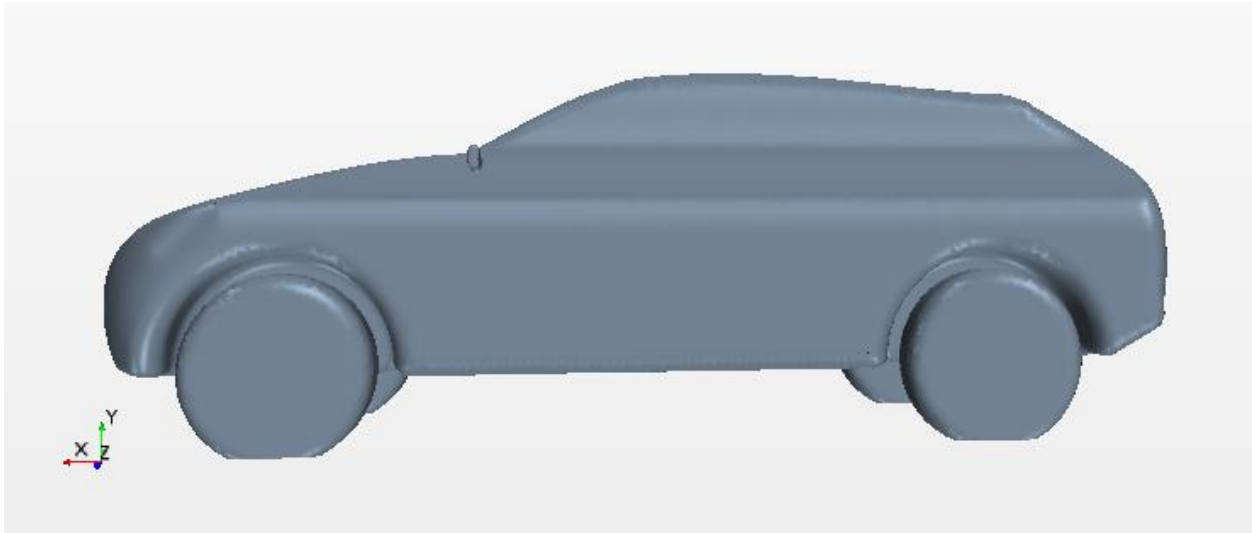

Figure 3: Domain dimension

Figure 4: Axis Orientation

III. Mesh Model

The mesh model was generated to keep the number of cells as small as possible. But at the same time capturing the detail properly was also a challenge. Surface remesher, Polyhedral mesher and Prism Layer mesher have been used in this study to develop the mesh.

   a. To improve the overall quality of the surface and also to optimize it in the volume mesh, the surface remesher has been used. It re-triangulates the existing surface [5].
   b. In polyhedral mesher, cells are less sensitive to stretching. Cells are able to be joined, split or modified by using additional points, edges and faces by the advantage of limitless possibilities offered by smart grid generation and optimization techniques [6].
   c. Prism layer mesher has been used to capture the effects of shear and boundary layer physics near surface.

The mesh parameters in this study has been kept the same as the study titled *A Parametric CFD Study of a Generic Pickup Truck and Rear Box Modifications* [11] where the mesh model was chosen to capture all the fine details of the model using an optimum number of cells (4.8 millions). In that study [11] a good convergence of the numerical simulation was achieved while using less computational power.

For capturing flow patterns around SUV and wake zone in detail with the least possible number of cells, volume control has been used. So the overall number of cells was only 1.97 million for the base model with a reasonably refined mesh around the SUV.
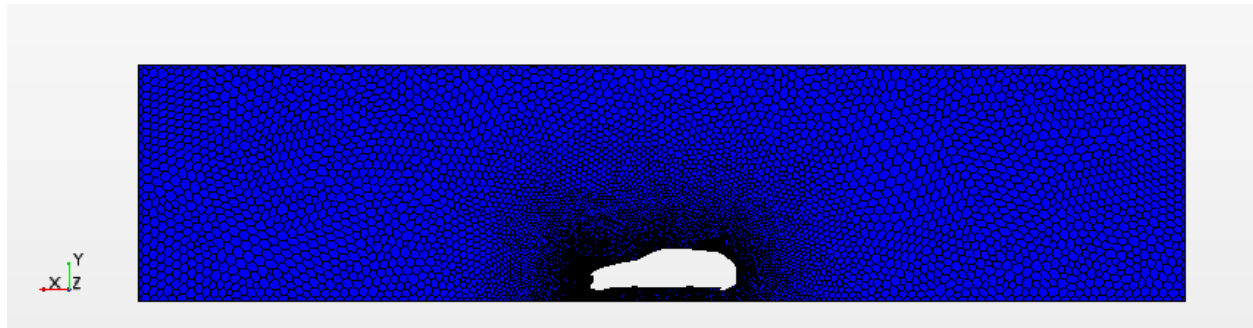
Figure 5: Symmetry Plane

Volume Controls have been used around the front radiator and side mirror. Although they have increased the number of cells by a small amount, the fine details were able to be captured.
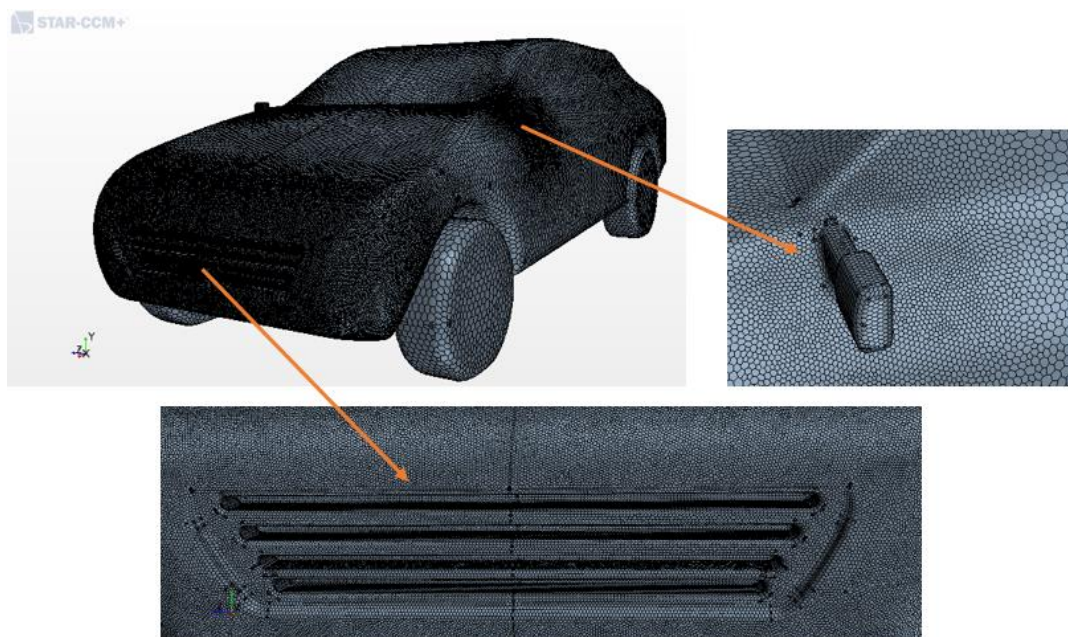

Figure 6: Details captured by volume controls

IV.   Boundary Conditions
Boundary conditions define the finite end of a problem. They are required to understand the flow at a certain point.

The study has been performed with half symmetric model. The entire domain has been divided into four regions: a. SUV, b. Ground, c. Air stream, d. Symmetry plane.

a. **Ground:** Since the vehicle will be moving relative to the ground, the ground has been considered as moving and the wheels as stationary.
b. **Air Stream:** The air stream has been considered as flowing at a speed of 70 mph in a direction opposite to the vehicle's motion.
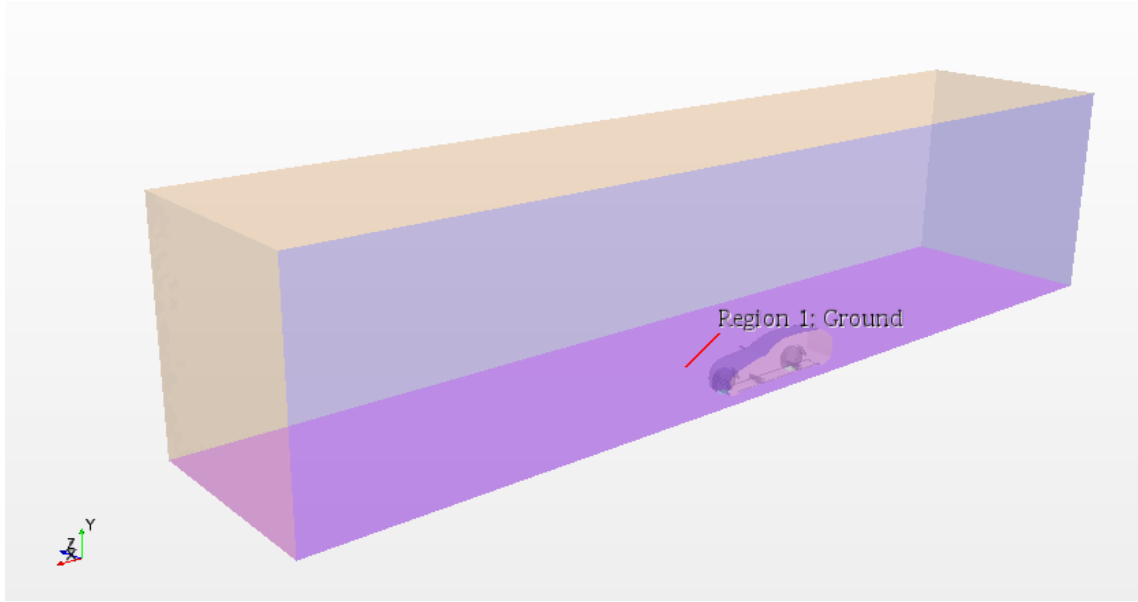c. **SUV:** The SUV has been considered as solid wall.

d. **Symmetry Plane:** To save the computational power, half symmetric model has been considered. A plane has been considered that is situated exactly at the middle of the model. By using the symmetry plane, the equations are needed to be solved for half number of cells of the entire model.
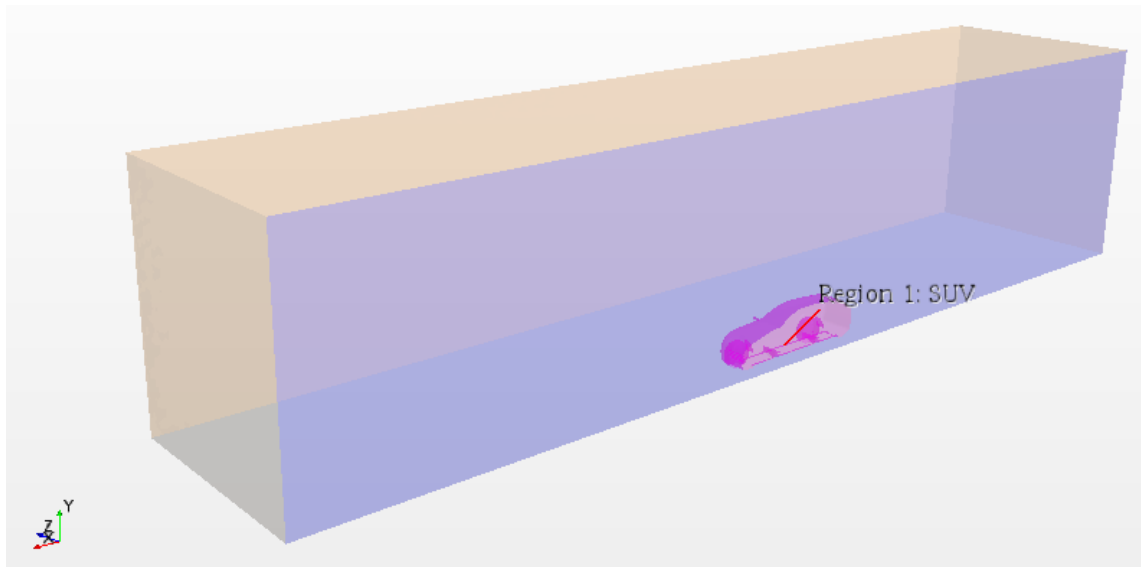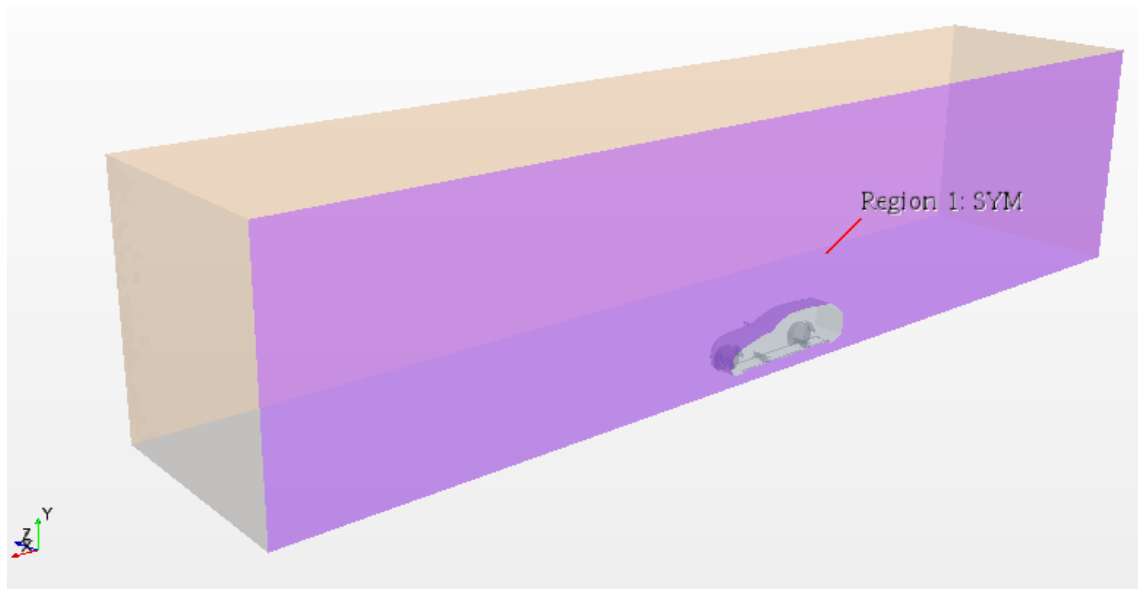

a. SUV
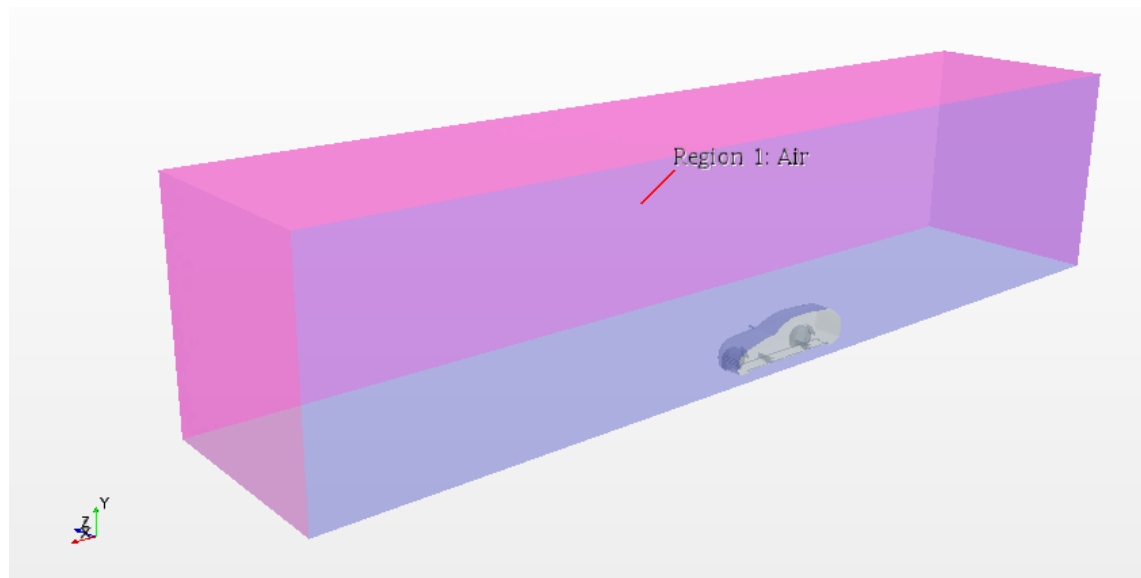
b. Air

c. Ground

d. Symmetry plane

Figure 7: Region divided into boundaries

The boundary condition applied by Mokhtar et al in "*Further analysis of pickup trucks aerodynamics*" [2] has been reproduced in this study for the full domain, since the results of that study [2] was validated in an actual wind tunnel. In the full domain the coefficient of drag was found nearly same as the half symmetric model.

V.   Solver Settings

For the present study, the following things have been considered:

- Segregated and
- K- ε model has been used for better prediction far from the boundaries (wall).
- RANS (Reynolds Averaged Navier Stokes) has been used to reduce computational requirements.

VI.   Results and Analysis

In this study, baseline simulation has been done at 70 mph air stream velocity. The aerodynamic drag has been calculated at 0°, 3° and 6° yaw angle.

Figure 8 shows the pressure distribution for 0° yaw angle at 70 mph. The high-pressure zone exists at the front of the car where the air stream first come in contact with the car, stagnation region. At the roof, two negative pressure zones can be seen. Also, low pressure exists at the underbody. The figure shows the balance between kinetic energy and pressure energy in the air flow interaction with the vehicle. The areas where the air slows down generate high pressure where the opposite happens at high speed areas such as over the roof.
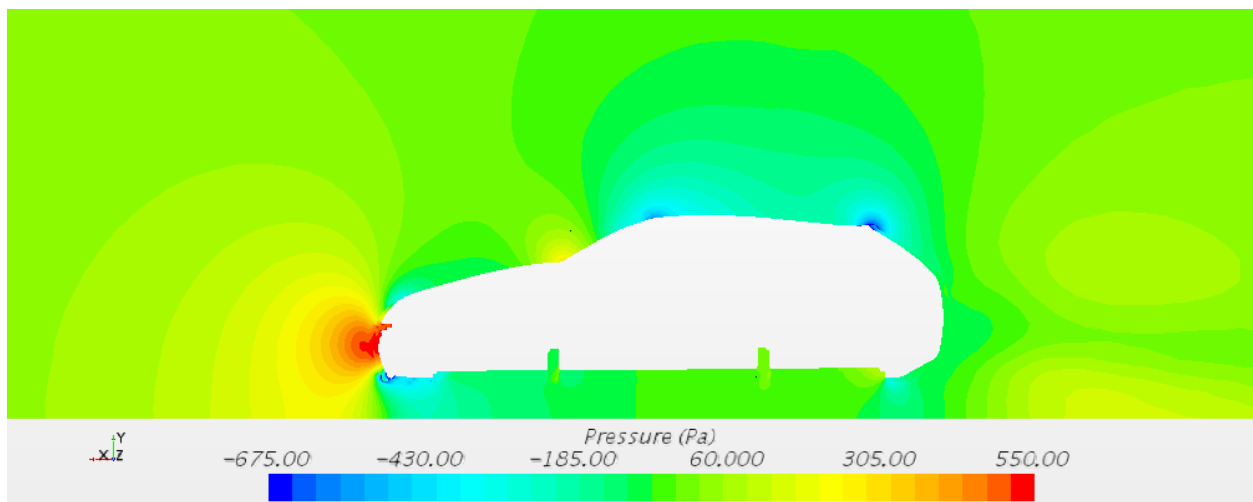

Figure 8: Pressure Distribution

This effect is demonstrated further in the velocity vector field shown in Figure 9. In the wake zone at the rear end the flow separates from the vehicle surface and encloses low speed wake full of vortices. This region generated relative a rotating fluid that decrease the pressure downstream to car and increases the pressure drag. On the other hand, the stagnation region at the front end of the car has a high pressure generated by the air flow slowdown over the surface. This in turn increases the pressure drag. Some of the aerodynamic modifications and add-on tools have been developed to address these two zones.

Figure 9 shows the stream lines around the car. The flow is separated from the two corners of the car and this separation continues through the wake zone. Two main important points need to be observed from this figure. First is the regions of high pressure and low pressure in from and in the wake of the vehicle respectively. These are key factors in the formation of pressure drag. The second point is the areas of high velocities near to the surfaces such as the roof and the car sides. These are very important contributors to friction drag. Typically pressure drag and friction drag are the main two components of aerodynamic drag.
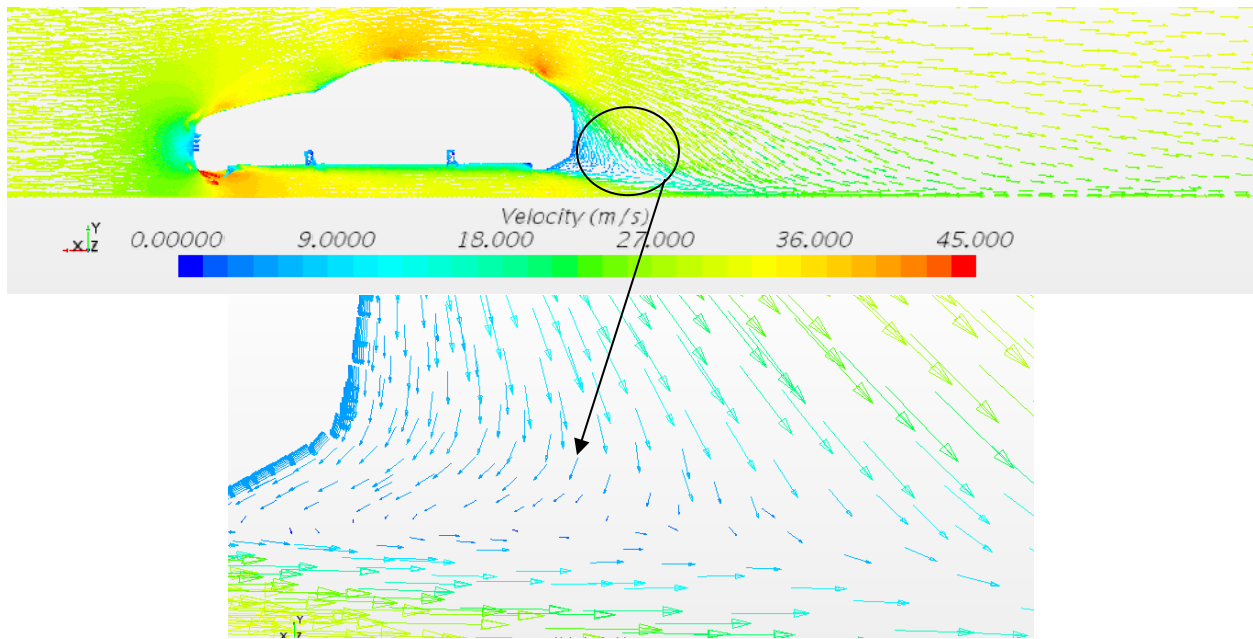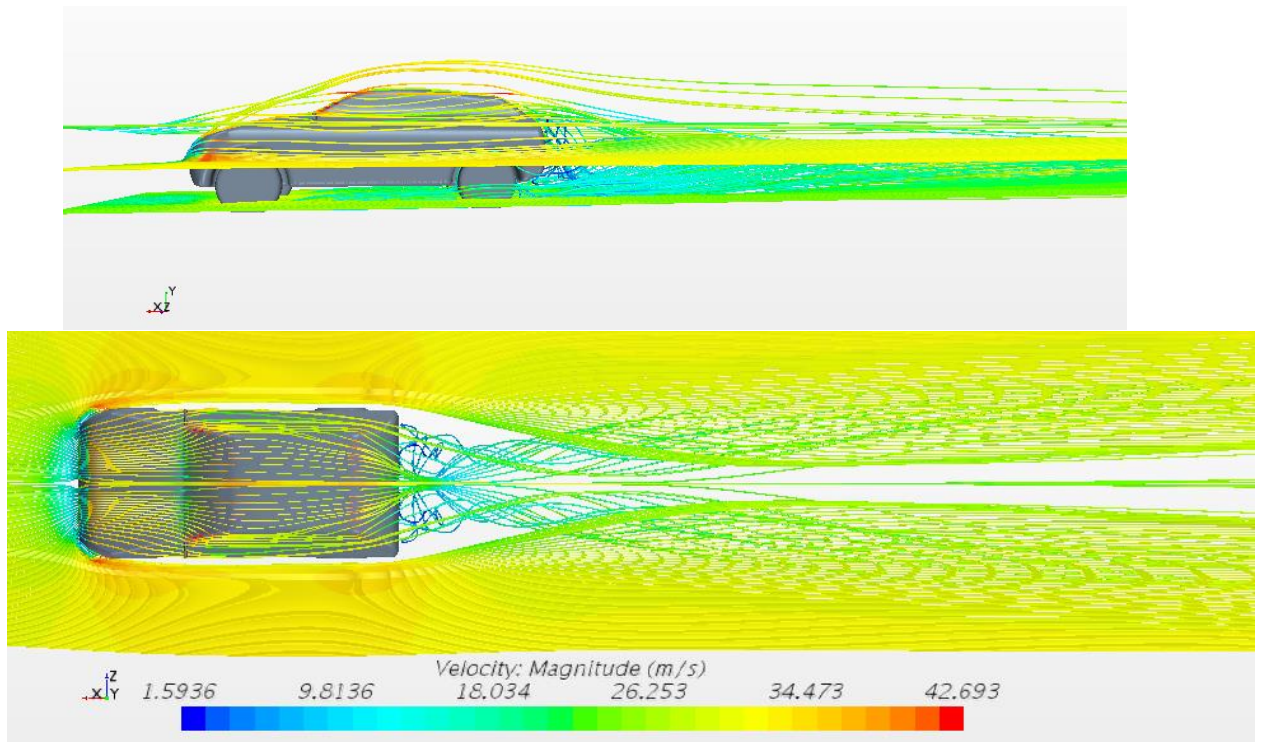

Figure 8: Velocity Vector

Figure 9: Flow Separation

Another way to observe the stagnation effect is the use of pressure coefficient contours as shown in Figure 10. The red region with pressure coefficient close to 1.0 shows that nearly all the flow energy in this region is transformed into pressure. A similar effect can be seen at the side mirrors and at parts of the front wheels. Regions of low pressure can be seen at some portion of the front wheel since the air stream velocity is very high when it passes the corners. In these areas the surface curvature accelerate the air flow, as a result the pressure coefficient becomes low. Regions of negative Cp indicate that the local air velocity is higher than the free stream velocity (car speed).
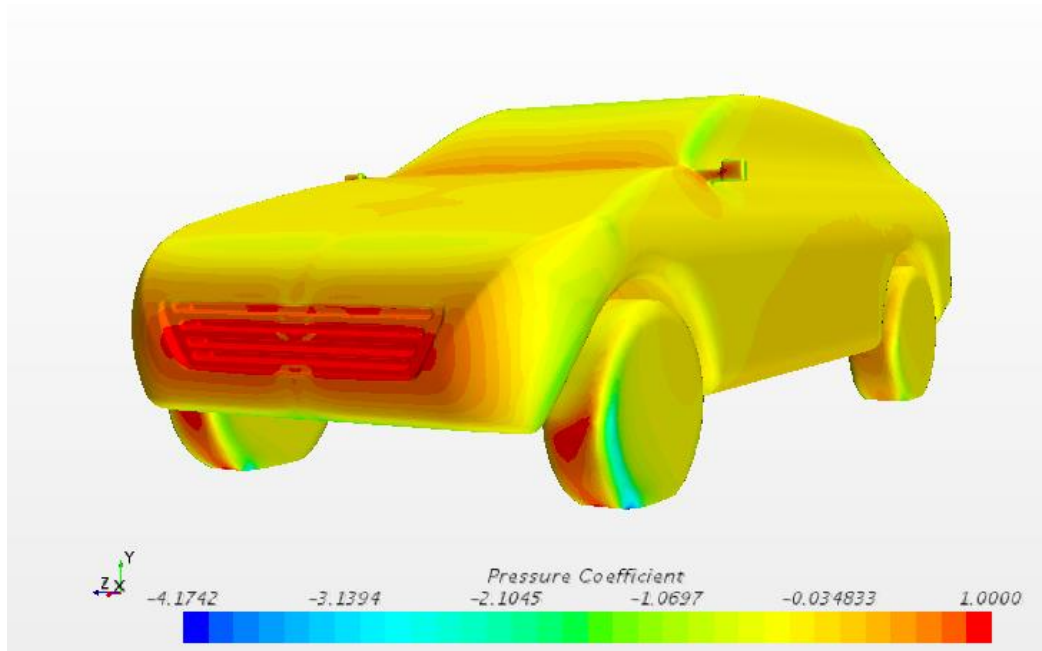
Figure 10: Coefficient of Pressure

Figure 11 show the wall Y+ value is around 4.5 almost all over the car except around the front radiator and side. That increase in Wall Y+ value can be due the shear layers around the radiator and mirror. This is one of the indications for shear stress in the areas of high velocity.
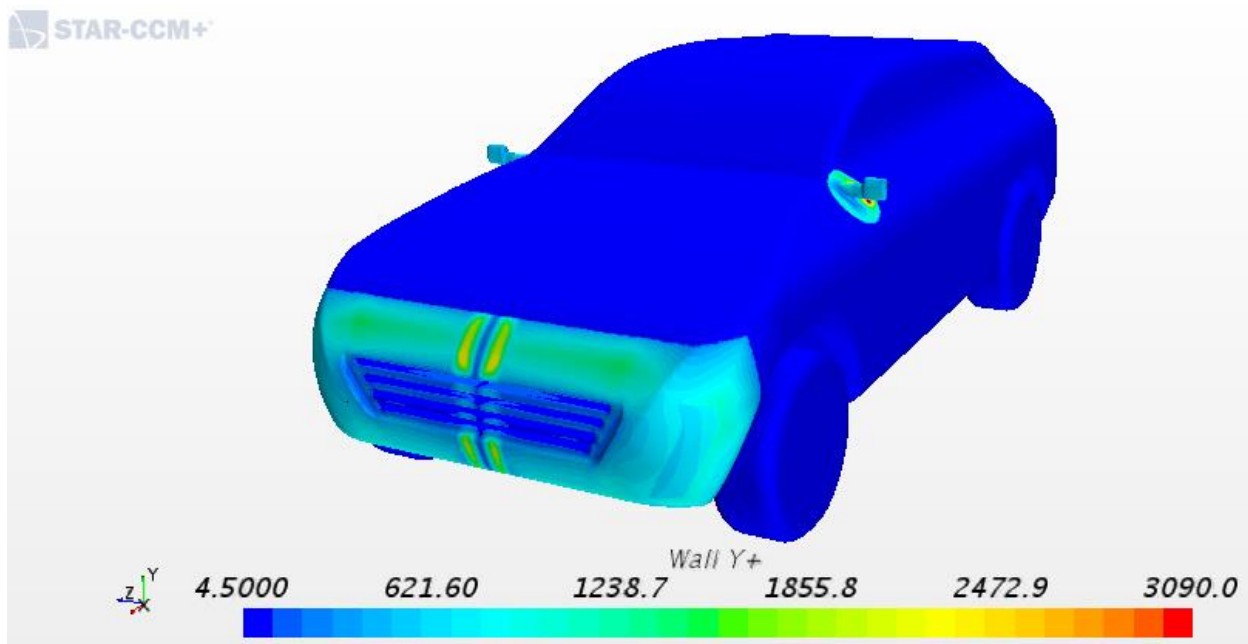

Figure 11: Wall Y+

One of the important parameter to study in vehicle aerodynamics is the effect of cross-wind. In CFD simulations, this is represented as change in the car yaw angle as show in Figure 12. The coefficient of drag for the vehicle at not cross-wind (0° yaw angle) is 0.3797. This value compares well with the published data for this type of cars. [10], [13], [14]

As the cross-wind increases (yaw angle increases) higher projection areas of the car are exposed to the air which increases the pressure drag. The car side that experiences higher velocity will generate higher skin friction. The wake size increases as result of the changed air angle and further separation regions appear in the flow. Figure 12 shows two yaw angles 3 and 6°. These are corresponding to cross-wind of 70 mph. Although the magnitude of the cross-wind is not that high, it has a significant impact on the drag. An increase of almost 21% in the drag is observed in the yaw angle of 6°. It confirms the importance of the cross-wind analysis and the need to design cars to meet these conditions.

Conclusion

This study shows the flow pattern and drag coefficient C_d of the baseline model of a generic Sports Utility Vehicle at highway speed and at different yaw angles. SUVs have become very popular over the years. Despite having several advantages like good towing capacity, spacious seating arrangements, good hauling capacity etc, its fuel economy can be further improved by the reduction of its aerodynamic drag.

Important regions to be considered for a better aerodynamic design include stagnation regions, acceleration flow over the roof, separated wake and effect cross-wind. The study shows the flow structures in each of these regions and the significant effect of cross-wind on the drag value. The next step of this study can be the observation of change in aerodynamic drag with some add-on device(s).
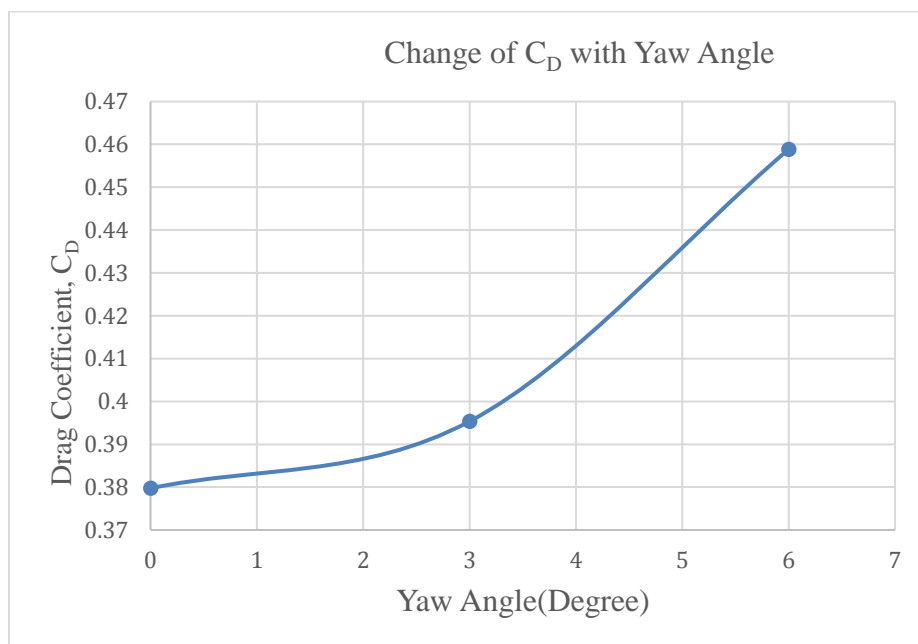

Figure 12: Change of Cd with yaw angle

References

1. Mokhtar, Wael, and Robert Camp. "Pickup Trucks-Box Configuration and Drag Reduction." *28th AIAA Applied Aerodynamics Conference. 2010*. DOI: <u>10.2514/6.2010-4954</u>

2. Mokhtar, Wael A., Colin P. Britcher, and Robert E. Camp. "Further analysis of pickup trucks aerodynamics*". No. 2009-01-1161. SAE Technical Paper, 2009*. DOI: <u>10.4271/2009-01-1161</u>

3. Drag. (n.d.). Retrieved from <u>https://www.merriam-webster.com/dictionary/drag</u>

4. Al-Garni, A. M. (2003). *Fundamental investigation of road vehicle aerodynamics*.

5. What is the Surface Remesher? (2008, October 26). Retrieved March 12, 2019, from <u>http://www.southampton.ac.uk/~zxie/SESS6021/CCMP/online1/064-workingWithSurfaceMesh-02.html</u>

6. Advantage of Polyhedral Meshes. (n.d.). Retrieved from <u>https://pdfs.semanticscholar.org/51ae/90047ab44f53849196878bfec4232b291d1c.pdf</u>

7. Moussa, A. A., Yadav, R., & Fischer, J. (2014). Aerodynamic Drag   Reduction for A Generic Sport Utility Vehicle Using Rear Suction. *International Journal of Engineering Research and Applications*, *4*(8), 101-107.

8. Wahba, E., Al-Marzooqi, H., Shaath, M., Shahin, M., & El-Dhmashawy, T. (2012). Aerodynamic drag reduction for ground vehicles using lateral guide vanes. *CFD Letters*, *4*(2), 68-79.

9. Sevilla, Y. A. (2012). *Drag reduction of sport utility vehicle using vortex generators* (Doctoral dissertation).

10. Krishnani, P. N., & Zhou, D. (2009, January). CFD Analysis of Drag Reduction for a Generic SUV. In *ASME 2009 International Mechanical Engineering Congress and Exposition* (pp. 589-598). American Society of Mechanical Engineers.

11. Mokhtar, Wael A., Hossain, Md Maruf and Jahan, Samira Ishrat. A Parametric CFD Study of a Generic Pickup Truck and Rear Box Modifications. *American Society of Engineering Education - North Central Section Spring Conference 2018*

12. What is CFD | Computational Fluid Dynamics? ¶. (n.d.). Retrieved from <u>https://www.simscale.com/docs/content/simwiki/cfd/whatiscfd.html</u>

13. *Sirenko, Volodymyr & Rohatgi, Upendar. (2012). Methods of Reducing Vehicle Aerodynamic Drag. 10.1115/FEDSM2012-72491.*

14. Ramya, P., Hemanth Kumar, A., Moturi, J., & Ramanaiah, N. (2015, November 25). Analysis of Flow over Passenger Cars using Computational Fluid Dynamics. Retrieved from https://www.researchgate.net/publication/299529645_Analysis_of_Flow_over_Passenger_Cars_using_Computational_Fluid_Dynamics