International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 09, September-2020

Drive Through Water Simulation A Comparative Study on the Effect of Drag Force on the Ahmed Body with A Commercial Car

Shivam Prajapati
National Institute of Technology,
Agartala, India.

Yogesh Upadhyay
Zakir Husain College of Engineering and Technology,
Aligarh Muslim University, Aligarh, India

***Abstract*— Computational Fluid Dynamics can provide valuable insight into the flow field around a car (water and air). Particularly when the chassis experiences the drag due to water. When any vehicle moves in the flooded roads, it experiences the drag produced by the water because of this drag, the velocity keeps on decreasing. After heavy rain events, streets may be flooded, and the maximum wading depth of the car can be reached quickly. Furthermore, even at low velocities, a bow wave can develop, which increases the risk of water getting into the combustion system.The work focuses on a study on the variation of velocity and the effect of drag force with the displacement of the Ahmed body in water and the splashing of water over the chassis. To compare the results the geometry of a four-wheeler (i.e. car) is also considered. This complete simulation is based on multiphase in which two phases were selected as air and water.it is performed in Ansys fluent taking K-omega SST model which is generally used for two-equation eddy viscosity models which helped in the simulation for calculating drag force as well as the drag coefficient.**

Keywords—Computational Fluid Dynamics, Solidworks, Ansys, Ahmed body.

I. INTRODUCTION

The drive through water means the movement of the vehicle in water. When any vehicle moves in water say in the rainy season, it experiences a drag force. The effect of drag caused by air affects the performance of the vehicle. .Khapane et.al presented a study on the Deepwater Wading simulation of automotive vehicles. He proposed a new concept named the co-simulation of CFD and MBD. In this approach, the MpCCI server makes use of different codes to take data like force and torque from the CFD and linear and angular velocity from the MBD and then produce results.[1] M. Nakisa et.al presented a Numerical estimation of shallow water effect on multipurpose amphibious vehicle resistance. He used the water pumps to pump out the uncontrolled water ingress during the river crossing mission. He modelled and stimulated in ansys cfx using FVM based on the RANS equation. the hydrodynamic pressure distribution on the ship hull has an important role in ship resistance. Due to an increase in flow under the keel, there is a reduction in pressure in that region, as a result, buoyancy decreases and results in sinkage and trim. As the sinkage of the ship occurs, the wetted surface area increases, as a result, viscous drag increases.[2] K.M Tan et.al presented Drag Coefficient Estimation Model to Simulate Dynamic Control of Autonomous Underwater Vehicle (AUV) Motion. This work defines the model that estimate the drag coefficient at any provided condition for any shapes and sizes. The proposed method covers body shapes, angle of attack, and typical low-speed scenario in a standard underwater AUV maneuveranoeuvre environment. The basic model of a cylinder was obtained as an example in the work to exhibit the accuracy and robustness of the proposed model. Further investigation is required to identify other fitting methods such as locally weighted smoothing regression (i.e. LOWESS and LOESS methods) to predict the drag coefficient accurately.[3] Morteza Anbarsooz et.al presented a study on underwater vehicles for drag reduction by means of hydrophobic surfaces. To reach an efficient underwater vehicle, a novel approach is introduced in this article to reduce the drag of underwater hulls. Total, friction, and pressure volumetric drag coefficients for the L-20 profile, as a function of the sliding coefficient. Variations of the slip velocity on the wall for various values of the sliding coefficient. Anbarsooz 307 phenomena of fluid slippage on the hydrophobic surfaces, which can drastically reduce the skin friction drag. In this regard, the drag reduction of an underwater hull with an unseparated flow profile is investigated numerically. For validation purposes, the numerical results for the slip flow over a micron-sized spherical particle are compared with the available analytical results, where an acceptable agreement is observed.[4] Phillips et.al presented A package of CFD simulations is proposed to aid the designer in these areas. The studies suggested reflecting the fact that AUVs are designed on a limited budget, normally by small multi-disciplinary teams with access to limited computational resources. The approaches built are mainly steady-state and appropriate for moving on a workstation personal computer.[5] Phillips et. al presented the Use of Computational Fluid Dynamics to Assess the Hull Resistance of Concept Autonomous Underwater Vehicles. Three AUV's have been modelled in CFX to determine straight-line resistance. This has then been validated against existing experimental data. The ease with which shape/appendage modifications were made as well as the inclusion of the free surface demonstrates the success of the parametrization strategy and its suitability for considering concept hull forms. CFD can be applied to the design of AUVs specifically.[5] Tian et.al presented Layout Optimization of Two Autonomous Underwater Vehicles for Drag Reduction with a Combined CFD and Neural Network Method. The CFD results indicated that the drags of the AUVs in the fleet are defined by the relative position of the two AUVs in various regions based on the drag behaviour of the two AUVs: the Parallel Region, the Tandem Region, the Pull Region, and the Push Region. [6]Kumar et.al presented

Design and a research On Unmanned Aquatic Vehicle This work presents the modelling, design and numerical simulation of a fixed-wing configured aquatic UAV. The fundamental idea about UAV's designs emerged from the flying fish and the remaining construction was constructed with the help of existing submarine design parameters to design high forward speed UAV for detecting and count the fishes. The maximum speed of the fish is around 30 m/s and the average speed is 25 m/s so this work forced towards the hydrodynamic analysis on the modelled UAV with the help of Ansys Fluent. [7]

In the above literature, there have been many research approaches done in the field of drive through water. It was found that in all researches, the research lacks a proper comparison of the vehicles and their effects of drag on the vehicles.

This work presented the following objectives-

1) A comparative study is done on the Ahmed body and common four-wheelers (i.e. car)
2) The drag is calculated, and its effect is studied on the vehicles.

II. METHODOLOGY

A. Outline

Computational Fluid Dynamics can be used broadly in the study of aerodynamics as well as the study of Hydrodynamics. Many investigations and researches have been done already for drag reduction of the vehicle moving in the water. This research focuses on a mainly comparative study of Ahmed body and a general car body moving in the water. The Ahmed body is a generic car body. The airflow around the Ahmed body captures the essential flow features around an automobile and was first defined and characterized in the experimental work of Ahmed[8]. Although it has a very simple shape, the Ahmed body allows us to capture characteristic features that are relevant to bodies in the automobile industry. The models of the vehicle were designed in Solidworks and then they were imported in Ansys fluent 2019R3. An enclosure containing the vehicle model was made to define the finite volume in which the simulation was carried out. The Boolean operation was performed in which the vehicle body was subtracted from the fluid domain. Meshing was done for the complete system which comprised of triangular as well as tetragonal mesh. After meshing, named selections were given to different walls and to the vehicle which was enclosed inside the enclosure.

The entire simulation was based on multiphase so two faces (primary and secondary) were chosen (air as primary and water as secondary). The model used in the simulation was the k-omega SST model having open channel flow. Boundary conditions of inlet and outlet of the water flow were given and in the report definition, a drag was considered which would occur due to resistance between the vehicle model and water. The entire simulation was carried out in steady-state and hybrid initialization was done before running the calculations.

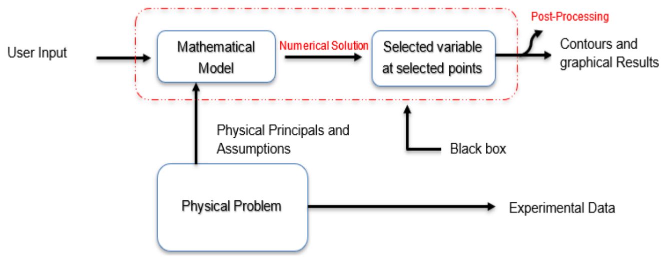


Figure 1: Methodology of the Approach

B. Geometry and Meshing

The Geometry that is used to compare the drag of the Ahmed body with the Car is shown in figure 2 and figure 3. Ahmed body is a basic and simple car model designed by Ahmed in the 1980s to inspect the behaviour of newly developed turbulence models for complex geometry cases. The Ahmed body consist of front part, a movable slant plane placed in the rear of the body to study the separation phenomena at different angles, and a rectangular box, which connects the front and rear slant plane. Ahmed body is a simplified model of a car body, but it shows all the flow features involved in an actual case of a moving car. Due to the considerable deviation of its geometry from normal vehicles, the body represents the basic aerodynamic properties of a vehicle, especially in the rear part. The rear slant angle ϕ has a robust effect on the aerodynamic drag and lift at the back. Both the drag and lift coefficients change abruptly at the angle of ϕ=30°. Therefore, this angle is called the critical slant angle

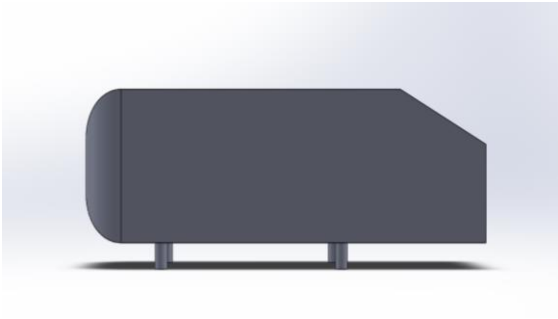


Figure 2: Ahmed Body

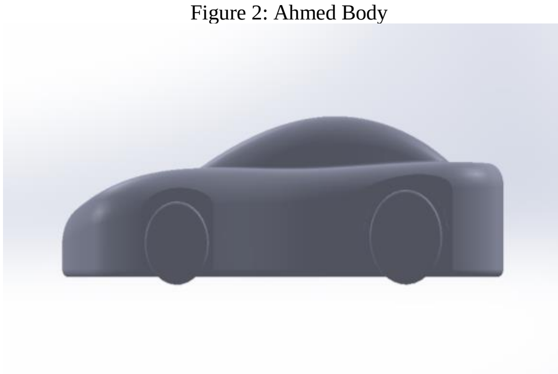


Figure 3: Car Body

Meshing is an integral part of the engineering simulation process where complex geometries are divided into simple elements that can be used as discrete local approximations of the larger domain. Mostly the elements presents are Tetrahedral both in Ahmed body and in the model of the car.NO. of elements and nodes present in the car model are

1287761 and 228422 respectively. The number of elements and nodes present in the Ahmed body are 315954 and 57314 respectively. We also check the orthogonal mesh matrix quality statics and it was found to be good and can be seen from figure 6 and figure 7. It can be seen that most of the elements have an index that lies between 0.7 to 1 which is expected from the orthogonal index.

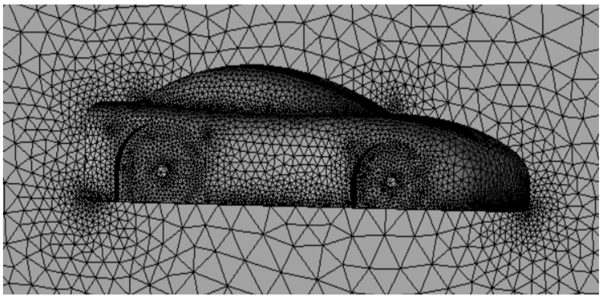


Figure 4: Meshing of the car body

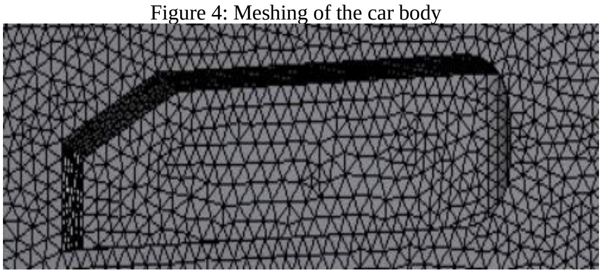


Figure 5: Meshing of the Ahmed body

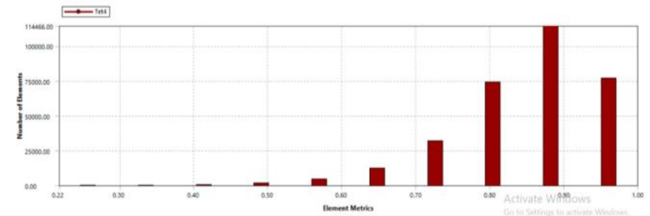


Figure 6: Mesh quality of the car

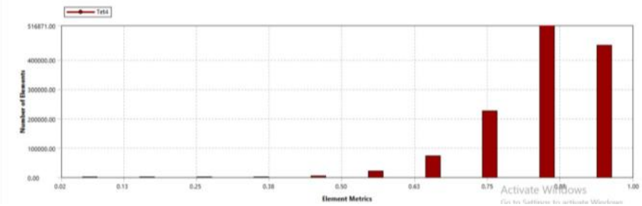
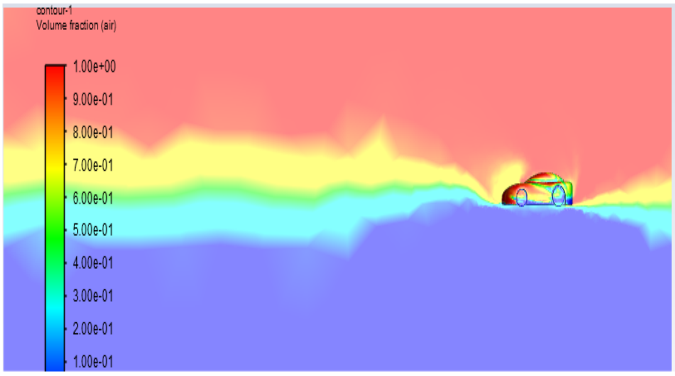


Figure 7: Mesh quality of the Ahmed body

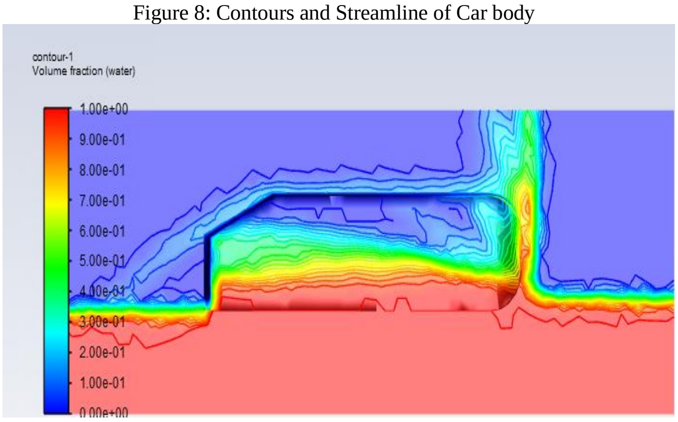
III. RESULTS AND DISCUSSION

The results obtained after simulating both geometries are analyzed. The different contours of the flow are shown in figure 8 and figure 9. It was seen from the contour and the streamline of both car and Ahmed body that the velocity is low in the front in the car due to more drag and in case of the Ahmed body the velocity was not reduce



2

Figure 8: Contours and Streamline of Car body



.

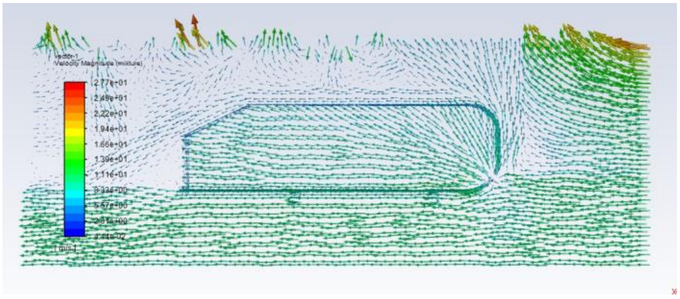


Figure 9: Contours and Streamline of the Ahmed body

.The contours in figure give us an idea of the different regions which experiencing more drag and have less velocity. It can be seen that the Ahmed body experiences a nearly constant drag with the more velocity at the bottom potion, Whereas in the car the less amount of velocity was observed in front and bottom. Also, it was found that the roof experiencing less drag with more velocity vectors which is obvious as it experiencing air drag instead of water drag. It can also be compared from figure 10 the drag is found to be more in Ahmed body.

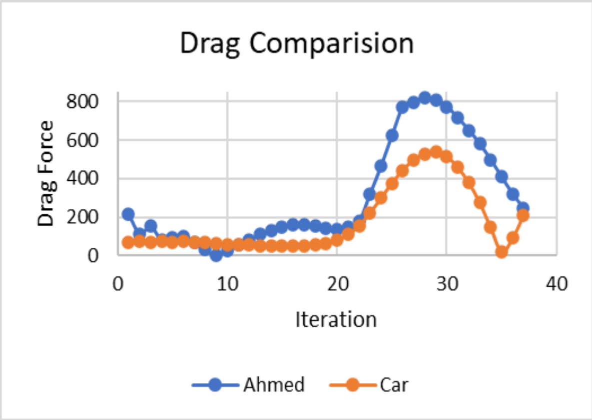


Figure 10: Drag force comparison

WADING RESULTS

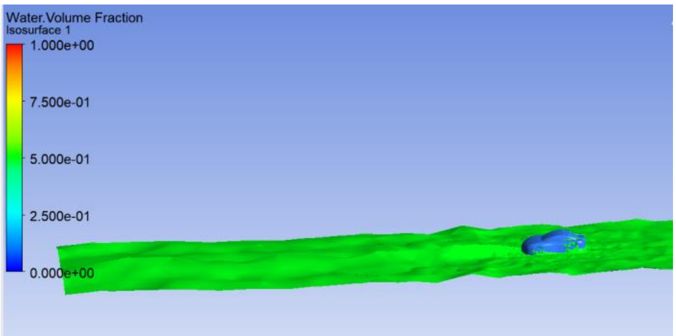


Figure 11-car travelling through wade Trough

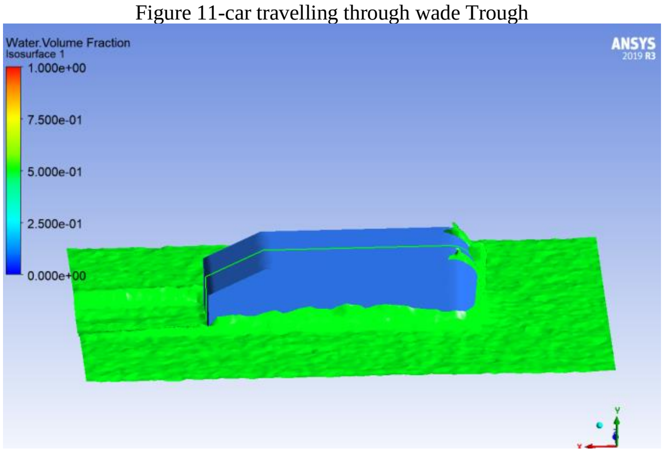


Figure 12-Ahmed body travelling through wade trough

Figure 11 and figure 12 shows us the water wading when the vehicle passes tough it. vehicle wading can be defined as the state where vehicle crosses through the water at distinct speeds. Vehicle water wading is a standard test procedure for JLR vehicles. The experiment method involves a set of combinations with various depths of water level and entry speeds into the wading trough. The different failure checks are the key indicators for the physical test scenario.

IV CONCLUSION

This research investigated the drag force and drag coefficient comparison in water between an Ahmed body and a four-wheeler ie car. Following were the observations-

1. The velocity variation in the profile of both the models. It was witnessed that the Ahmed body suffered more drag force as compared to the car.
2. We could also see the comparison between the wading trough and splash by water in both the models.
3. It was observed that due to the sharp geometry of the Ahmed body the drag was more compared to the smooth curves of the car which gives an advantage of less drag.

This work discusses the drag comparison of two geometry. It can be further expanded to more geometries. Also the regions that experience more drag in the Ahmed body and car can further be studied to optimize the design.

REFERENCES

[1] D. I. Kalmykov, I. P. Bayrasy, and K. Wolf, "DEEP WATER WADING SIMULATION OF AUTOMOTIVE."

[2] M. Nakisa, A. Maimun, Y. M. Ahmed, F. Behrouzi, and A. Tarmizi, "NUMERICAL ESTIMATION OF SHALLOW WATER EFFECT ON MULTIPURPOSE AMPHIBIOUS VEHICLE RESISTANCE Abstract :," vol. 8535, pp. 1–8, 2017.

[3] K. M. Tan, T. F. Lu, and A. Anvar, "Drag coefficient estimation model to simulate dynamic control of autonomous underwater vehicle (AUV) motion," *Proc. - 20th Int. Congr. Model. Simulation, MODSIM 2013*, no. December, pp. 963–969, 2013, doi: 10.36334/modsim.2013.c10.tan.

[4] M. Anbarsooz, "A numerical study on drag reduction of underwater vehicles using hydrophobic surfaces," *Proc. Inst. Mech. Eng. Part M J. Eng. Marit. Environ.*, vol. 233, no. 1, pp. 301–309, 2019, doi: 10.1177/1475090217740470.

[5] A. B. Phillips, S. R. Turnock, and M. Furlong, "The use of computational fluid dynamics to aid cost-effective hydrodynamic design of autonomous underwater vehicles," *Proc. Inst. Mech. Eng. Part M J. Eng. Marit. Environ.*, vol. 224, no. 4, pp. 239–254, 2010, doi: 10.1243/14750902JEME199.

[6] W. Tian, Z. Mao, F. Zhao, and Z. Zhao, "Layout Optimization of Two Autonomous Underwater Vehicles for Drag Reduction with a Combined CFD and Neural Network Method," *Complexity*, vol. 2017, 2017, doi: 10.1155/2017/5769794.

[7] V. Praveen Kumar, S. Kishor Kumar, K. R. Sankaresh Pandian, E. Ashraf, K. Thanga Tamil Selvan, and R. Vijayanandh, "Conceptual design and hydrodynamic research on unmanned aquatic vehicle," *Int. J. Innov. Technol. Explor. Eng.*, vol. 8, no. 11 Special Issue, pp. 121–127, 2019, doi: 10.35940/ijitee.K1027.09811S19.

[8] F. J. Bello-Millán, T. Mäkelä, L. Parras, C. del Pino, and C. Ferrera, "Experimental study on Ahmed's body drag coefficient for different yaw angles," *J. Wind Eng. Ind. Aerodyn.*, vol. 157, pp. 140–144, 2016, doi: 10.1016/j.jweia.2016.08.005.