Numerical Investigations of the DrivAer Car Model using Opensource CFD Solver OpenFOAM

Gopal Shinde[1], shinde.gopal@tcs.com, Aniruddha Joshi, joshi.aniruddha@tcs.com and Kishor Nikam, kishor.nikam@tcs.com

Abstract:

A lot of the investigations in automotive aerodynamics are still based on strongly simplified generic bodies such as the Ahmed Body or the SAE body. To close the gap between these strongly simplified models and highly complex production cars the new generic DrivAer body model is introduced by the Institute of Aerodynamics and Fluid Mechanics, Technische Universitat Munchen (TUM). This current study is focused on three different DrivAer body models namely Fastback, Estateback, and Notchback and two different underbody types for each model, smooth and detailed. Hence, total 6 different models are simulated using open source CFD solver OpenFOAM at two different ground conditions, with ground and without ground effect (WGS and WoGS). All the models used in simulations are 2.5 scaled down models as compared with the actual car dimensions. The vehicle velocity considered for this numerical study is 40 m/s, Reynolds number is 4.87M and turbulence model used is k-w-SST. The mesh is generated using SnappyHexMesh (SHM) tool of OpenFOAM and it is around 11 million volume cells for the smooth underbody and 14 million volume cells for the detailed underbody. The coefficients of drag (C_d) values are within 0.5% to 12% error band as compared against the experimental values published by the TUM. The coefficients of pressure (C_p) plots are comparable with experimental results and also the contribution of individual body part in overall C_d values is obtained in this study. All the simulations are carried out using OpenFOAM 2.1.1 on Tata Consultancy Services (TCS) High Performance Computing facility.

Keywords : DrivAer body, External Aerodynamics, OpenFOAM, SnappyHexMesh, CFD, HPC, TCS.

---

[1] Author for correspondence, Gopal Shinde, shinde.gopal@tcs.com, Tata Consultancy Services, Pune, India

Introduction

Nowadays with the help of Computational Fluid Dynamics (CFD) and High Performance Computing (HPC) Technology, vehicle aerodynamics engineers are reducing the wind tunnel experiments and accelerating the vehicle design cycle. A lot of the investigations in automotive aerodynamics are still based on strongly simplified generic bodies such as the Ahmed Body. In an experimental work, Ahmed et al. [1] set a bluff body, called the Ahmed Model, in the Gottingen open section wind tunnel and analysed the time averaged flow behaviour for different slant angle variations. Gilli´eron et al. [2] performed computational simulation and compared against the experimental ones for the reference Ahmed model. Lienhart et al. [3] performed experiments for two slant angles, 25^0 and 35^0, with slightly lower velocity 40 m/s but of the same order, with Re = 2.8E06. Later, Kapadia et al. [4], Hinterberger et al. [5], Sinisa Krajnovic and Lars Davidson [6] and Ehab Fares [7] also contributed to understand the Ahmed body flow physics better. Recently, Ronak Pandya et al. [8] and Angelina Heft et al. [9] also contributed numerical observations on Ahmed body.

These simple car models like Ahmed Body or SAE body make it easy to relate the observed phenomena to specific areas and thus help to understand basic flow structures. At the same time, more complex flow phenomena, e.g. at the underbody, wheels/wheelhouses and around the rear view mirrors etc., cannot be reproduced due to the over simplification of these geometries. On the other hand, it is usually not feasible to investigate these phenomena on a specific production vehicle, as, due to its short life span and restricted access, typically little validation data is available. Recognizing the need for a model combining the strengths of both approaches, various more or less generic models, such as the VW reference car and the MIRA reference car, have been proposed [10]. However, while these reference cars mark a step in the right direction, these models are still too generic to completely understand the complex phenomena occurring at realistic vehicles.

To close this gap, the Institute of Aerodynamics and Fluid Mechanics of the Technische Universitat Munchen (TUM), in cooperation with two major car companies, the Audi AG and the BMW Group, has proposed a new realistic generic car model called “DrivAer Model” [11]. The body is based on two typical medium class vehicles (Audi A4 and BMW3 series) and includes three interchangeable tops and two different underbody geometries to allow for a high universality. To encourage the use of the DrivAer model in independent research projects, TUM research group is open to share the geometry and a comprehensive experimental database is published in different papers [12, 13].

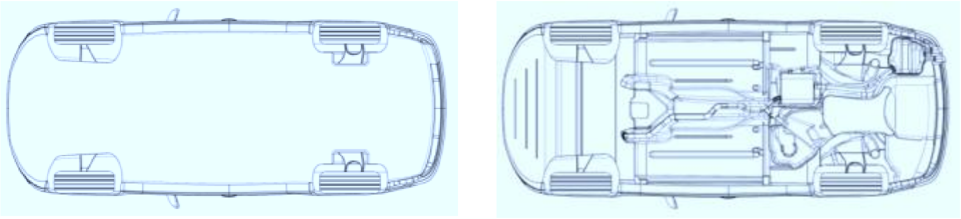
The aim of the current work is to use different DrivAer body models and their experimental results to validate the opensource CFD solver OpenFOAM.

Problem Description

This current study is focused on three different DrivAer body models namely Fastback, Estateback, and Notchback and two different underbody types for each model, smooth and detailed as shown in Figure 1. Hence, total 6 different models are simulated using open source CFD solver OpenFOAM at two different ground conditions, with ground and without ground effect (WGS and WoGS). In case of WGS, car wheels are rotating anticlockwise and road is moving in the direction of air and in case of WoGS both car wheels and road are stationary. All the models used in simulations are 2.5 scaled down models as compared with the actual car dimensions. Figure 2 show a sketch and main dimensions of the fastback configuration of the 1:2.5 DrivAer model. Different parts of the Fastback model considered for the simulation study are as shown in Figure 3. The individual drag contribution of these components is also calculated along with the total car drag.

The vehicle velocity considered for this numerical study is 40 m/s, Reynolds number is 4.87E10^{6} and turbulence model used is k-w-SST.

Fastback Estateback Notchback



Smooth underbody Detailed underbody

Figure 1: DrivAer body models and different underbody types

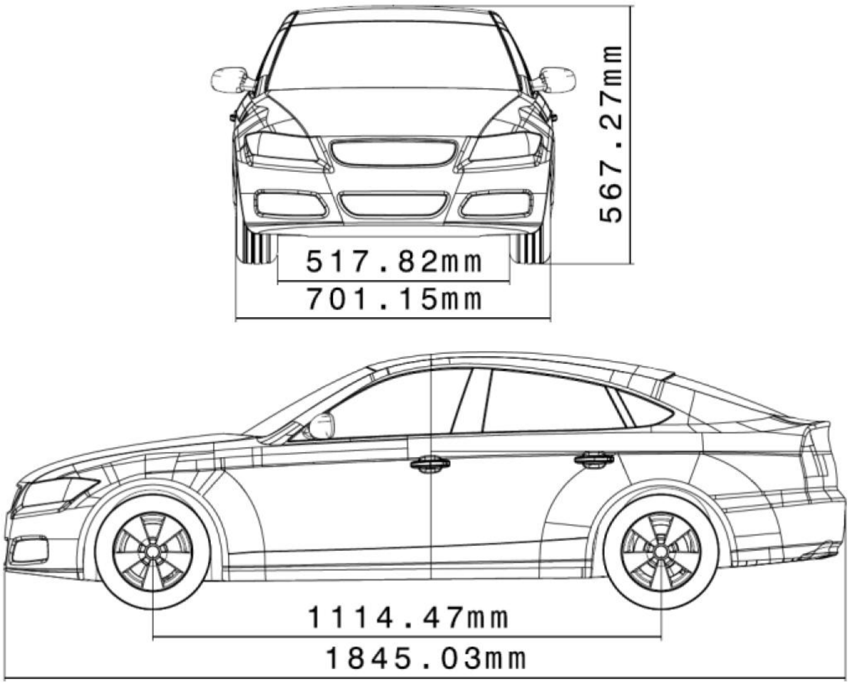


Figure 2: Typical dimensions of the Fastback model

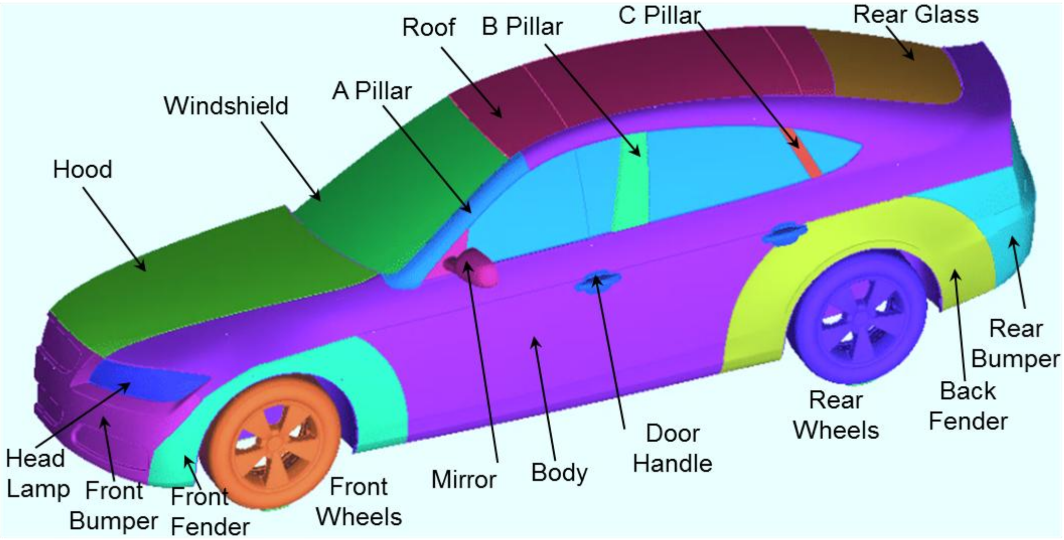


Figure 3: Different parts of the Fastback model

Computational Domain and Mesh Generation

To capture the flow around the car model the computational domain is created around the car which often called as numerical wind tunnel as shown in Figure 4. This computational domain is like physical wind tunnel test section. The length of the computational domain is 28 m, width is 10 m and height is 7 m. While the length of physical wind tunnel test section is 4.8 m, width is 2.4 m and height is 1.8 m. The blockage ratio used in computational domain is 0.57% while in physical wind tunnel it was 6.17%.

To capture flow physics around the car more accurately the mesh size kept near the car is fine and becomes coarse when go away from the car as shown in Figure 5. To achieve this variable mesh size distribution various refinement boxes are created with different dimensions as shown in Figure 4. To capture the boundary layer around the car, multiple layers of very fine and fine element sizes are kept around full car and along the road which is shown in Figure 6. The y+ obtained is 30 which is supposed to be good for incompressible simulations using OpenFoam.

SnappyHexMesh (SHM) tool was used in parallel mode for the grid generation. The meshing was carried out on 4 cores of 32 GB workstation. The mesh generated on smooth underbody models is about 11 million volume cells and 14 million volume cells for the detailed underbody.

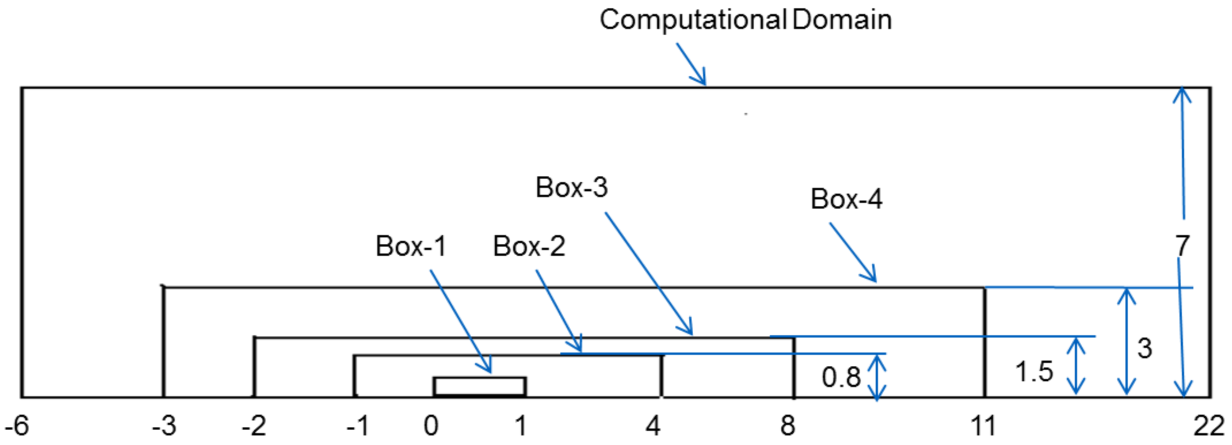


Figure 4: Different boxes of refinement around the Fastback model

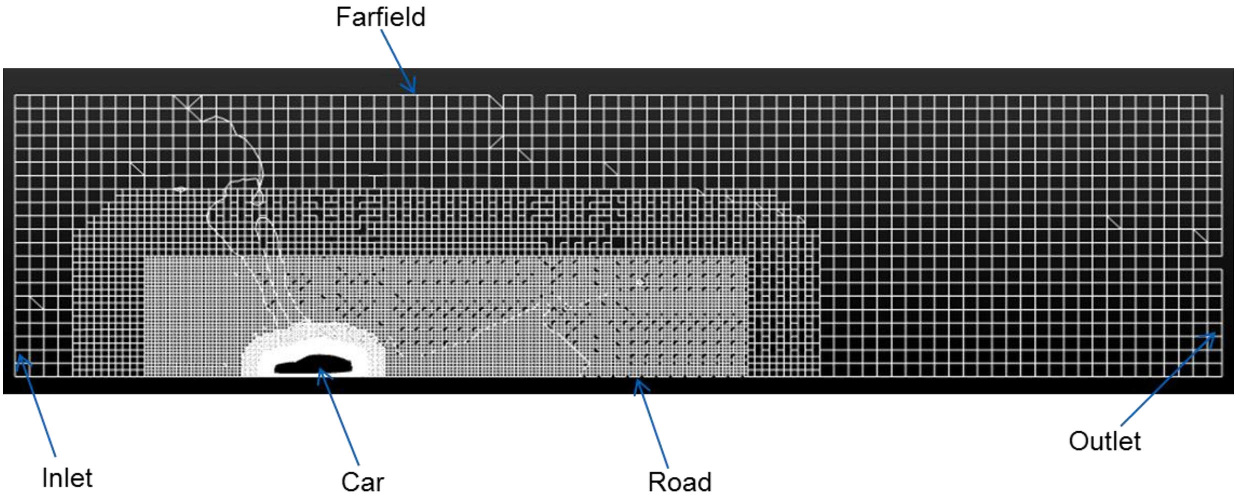


Figure 5: Z-Cut plane of the grid around the Fastback model

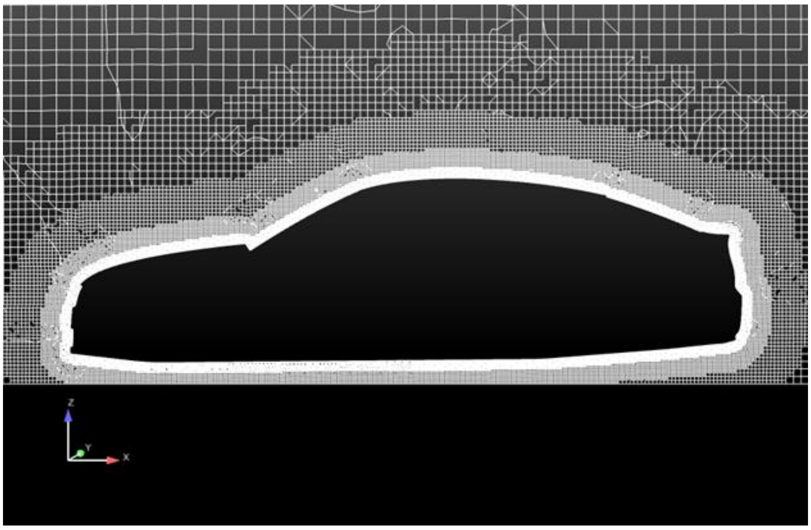


Figure 6: Zoomed view of the grid cut plane around the Fastback model

Boundary Conditions and Solver Settings

Boundary conditions are implemented on the computational domain, car body, car wheels and road. Velocity was specified at the inlet and pressure was specified at outlet. Road was given translation speed 40 m/s in the direction of flow in simulations with ground effect and considered stationary for without ground simulations. Car wheels are rotating at 320 rad/s anticlockwise in simulations with ground effect and considered stationery for without ground simulations. All other car parts are defined as stationary walls.

“simpleFoam” solver from OpenFoam has been used for simulations. “potentialFoam” solver is used to set pressure field in the domain. Post-processing has been carried out using the in-built utilities sampleDict, streamlines, and cuttingPlane. The models considered here are with wheels and mirrors and were simulated for 10,000 iterations each. The convergence achieved was 5 decades fall for flow variables P and U and 6 decades fall for turbulence quantities omega and k. Convergence of drag force is also monitored and achieved upto 15 counts in last 100 iterations.

Some of the solver’s important settings are given below.

- Time stepping = Steady state
- Gradient scheme = Second order
- Divergence scheme = Upwind (First order)
- Laplacian scheme = Linear (Second order)
- Interpolation scheme = Linear (Second order)
- Turbulence model = K-omega-SST[15]
- Pressure solver = GAMG

- Velocity solver = GaussSiedel
- Non-orthogonal correctors = 2

Result Analysis

The total drag values obtained for the three models with smooth underbody considering with and without ground effect are given in the Table 1. Table 2 shows the drag values obtained for the three models with detailed underbody considering with and without ground effect. Percentage error found out in the simulated drag values is in the range of 0.5% to 12% as compared to the experimental drag values. Simulated drag value is taken as the average of drag values of last 100 iterations.

The component wise drag plot for all configurations is as shown in Figures 7 and 8. This plot shows the drag produced by each component, which gives us the important insight where we should look for design changes in order to reduce the drag. Reduction of drag directly contributes towards increase of fuel efficiency. These component wise drag values are not readily available from experiments; however, it is easily possible in simulations. C_p plots for the top surface of the models at y=0 are plotted against the experimental C_p distribution as shown in Figure 9 and 10. We have observed deviation in the simulated C_p distribution at the center of the model. In experimental setup, at center of the model a vertical strut is mounted.

Table 1: Total drag values for Smooth Underbody models

Model Type	Ground Effect	Expt. C_d	CFD C_d	% Error
Fastback	WoGS	0.254	0.2597	-2.24
	WGS	0.243	0.2599	-6.95
Estateback	WoGS	0.296	0.2684	9.32
	WGS	0.292	0.2840	2.73
Notchback	WoGS	0.258	0.2567	0.5
	WGS	0.246	0.2565	-4.26

Table 2: Total drag values for Detailed Underbody models

Model Type	Ground Effect	Expt. C_d	CFD C_d	% Error
Fastback	WoGS	0.284	0.2925	-2.99
	WGS	0.275	0.3029	-10.14
Estateback	WoGS	0.318	0.3199	-0.6
	WGS	0.319	0.3546	-11.16
Notchback	WoGS	0.286	0.3042	-6.36
	WGS	0.277	0.3097	-11.8

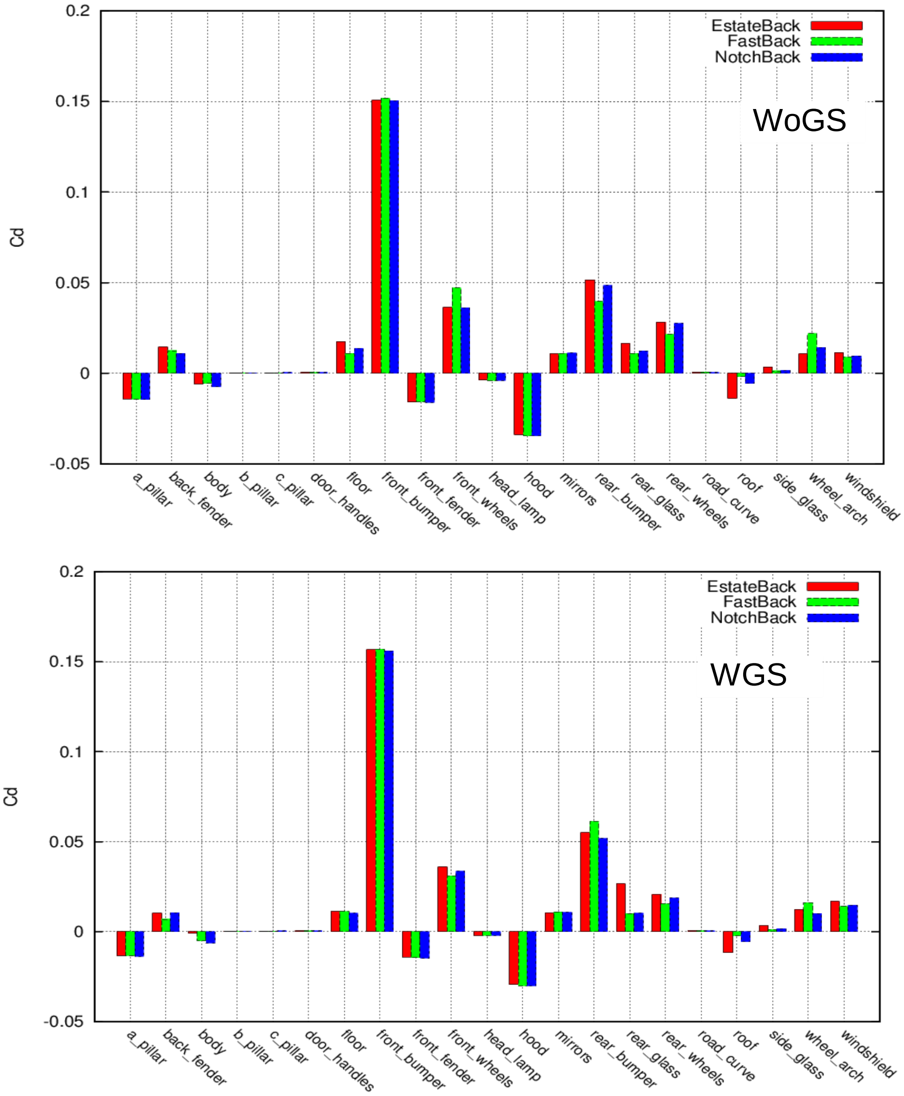
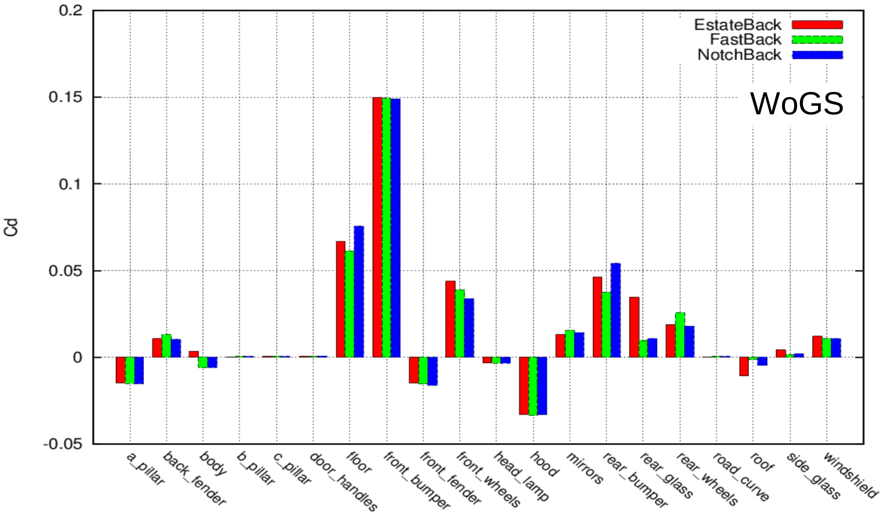


Figure 7: Component wise drag contribution plot for Smooth Underbody models



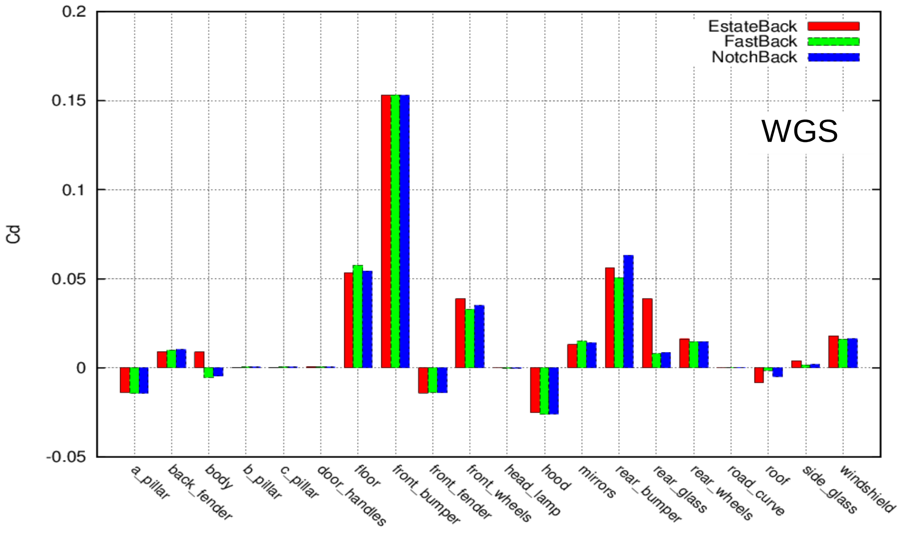


Figure 8: Component wise drag contribution plot for Detailed Underbody models

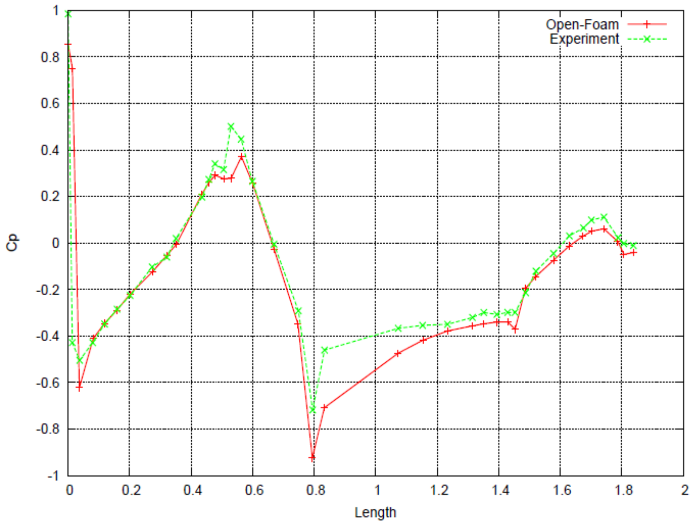


Figure 9: Coefficient of pressure (C_p) plot for Fastback model, WoGS

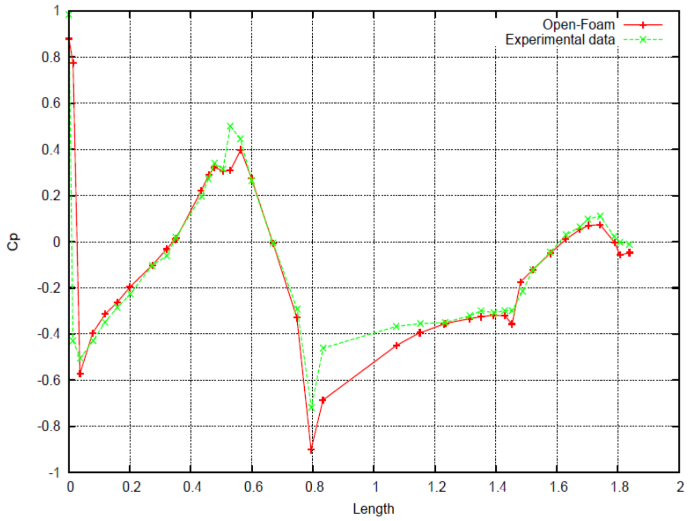


Figure 10: Coefficient of pressure (C_p) plot for Fastback model, WGS

Conclusion

This current study is focused on 3 different models of DrivAer body namely FastBack, EstateBack, and NotchBack and 2 different underbodies, smooth and detailed underbody. Hence, total 6 different configurations are simulated using open source software OpenFOAM at two different conditions with ground and without ground effect. The coefficients of drag (C_d) values are within 0.5% to 12% error band as compared against the experimental values published by the TUM. The coefficients of pressure (C_p) plots are comparable with experimental results and also the contribution of individual body part in overall C_d values is obtained in this study. There are huge flow separation / recirculation zones on the back-side of the models. In order to capture the flow features more accurately, we need to refine the grid in these zones. Since the flow features in these zones keep on changing with respect to time, current steady state simulations may not mimic the exact flow situation. Unsteady state flow simulations with region-wise refined grid and with more stringent convergence criterions may give more insights.

Future Work

In this current work, steady state simulations using k-w-SST turbulence model are carried out. We would like to investigate this further for few more turbulence models and also with transient calculations. Grid refinement may be necessary at the back side of the models to capture the flow separation / recirculation zones more accurately, which needs to be investigated.

References

1. S. R. Ahmed, G. Ramm, and G. Faltin, 1984. Some salient features of the time averaged ground vehicle wake. SAE paper no. 840300.
2. P. Gilli´eron and F. Chometon, 1999. Modelling of stationary three-dimensional separated air flows around an Ahmed reference model. In ESAIMproc., volume 7, pages 173-182.
3. H. Lienhart, C. Stoots, and S. Becker, 2000. Flow and turbulence structures in

the wake of a simplified car model (Ahmed Model). In DGLR Fach Symp. der AGSTAB.

4. S. Kapadia, S. Roy, and K. Wurtzler, 2003. Detached eddy simulation over a reference Ahmed car model. AIAA paper no. 2003-0857
5. M. Hinterberger, M. Garcia-Villalba, and W. Rodi, 2004. Large Eddy Simulation of flow around the Ahmed body. Lecture notes in Applied and Computational Mechanics, The Aerodynamics of heavy vehicles: Trucks, Buses, and Trains, R. McCallen, F. Browand, J. Ross (Springer, New York, 2004).
6. S. Krajnovic and L. Davidson, 2005. Flow Around a Simplified Car, Part 1: Large Eddy Simulation. J. Fluids Eng. 2005, Volume 127, Issue 5.
7. Ehab Fares, 2006. Unsteady flow simulation of the Ahmed reference body using a lattice Boltzmann approach. Computers & Fluids, Volume 35, Issues 8-9, 2006.
8. Pandya, R., Pavitran, S., Nikam, K., Singh, A., A Turbulence based Computational Study on the Drag Breakdown of Ahmed Body, IUTAM Symposium on Bluff Body Flows, December 12-16, 2011, IIT Kanpur, India. Pg. 337-340.
9. A. I. Heft, T. Indingery, and N. A. Adams, Investigation of Unsteady Flow Structures in the Wake of a Realistic Generic Car Model, 29th AIAA Applied Aerodynamics Conference, 27 - 30 June 2011, Honolulu, Hawaii
10. Le Good, G. M., and Garry, K. P., On the Use of Reference Models in Automotive Aerodynamics, SAE Technical Paper 2004-01-1308, 2004, doi: 10.4271/2004-01-1308.
11. Angelina I. Heft, Thomas Indinger and Nikolaus A. Adams, Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations, SAE International, 2012-01-0168
12. Mack, S., Indinger, T., Adams, N. A., and Unterlechner, P., “The Ground-Simulation Upgrade of the TUM Wind Tunnel,” SAE Technical Paper 2012-01-0299, 2012, doi:10.4271/2012-01-0299.
13. J. Wojciak, P. Theissen, K. Heuler, T. Indinger, N. Adams, R. Demuth: Experimental Investigation of Unsteady Vehicle Aerodynamics under Time Dependent Flow Conditions - Part 2, SAE 2011 World Congress, April 12-14, 2011, Detroit, Michigan, USA, Paper 2011-01-0164
14. OpenFoam User Guide, http://www.openfoam.org/docs/user/
15. Sebastian MÖLLER, Daniele SUZZI, Walter MEILE, Investigation of the flow around the Ahmed body using RANS and URANS with various turbulence models, 3rd OpenFoam Workshop, Politecnico Di Milano, Milan, Italy, 10-11 July 2008.

Acknowledgements

The authors are thankful to Angelina I. Heft et al. (Institute of Aerodynamics and Fluid Mechanics, Technische Universitat Munchen, Germany) for sharing the DrivAer body CAD files and for valuable suggestions during this course of work. Authors would like to thank the TCS-HPC team for their continuous support to carry out this exercise. Authors also acknowledge the encouragement and support extended by Mr. Rajaravisankar Shanmugam and EIS leadership team in pursuing this initiative.