FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
April 22, 2022

Flow And Turbulent Structures Around Simplified Car Model Using OpenFOAM®

Aditya Anand Kulkarni
Department of Mechanical Engineering, IIT Bombay

Synopsis

The external car aerodynamics is vital in determining the car efficiency , comfort, and car ride-ability. The flow over and under the car body shows three dimensional and unsteady turbulent characteristics. Moreover, vortex shedding , flow reattachment and re-circulation bubbles are formed around the bluff body. These influence the lift and drag parameters which directly affect the efficiency and car ride-ability. The main aim of this study is to validate the results through OpenFoam and match with the results as obtained by Aljure et al., using Ansys ICEM CFD software. The author has used several modules of LES models and compared the results against the obtained experimental analysis. The snappyHexMesh utility is used here to generate 3-Dimensional mesh to carry out this research study through OpenFoam .

Keywords : Vortex Shedding , Flow Reattachment , LES model , snappyHexMesh, OpenFoam

1 Introduction

CFD (Computational Fluid Dynamics) has greatly influenced the productivity in the industrial and academia research in carrying out the Research and Development in these few couple of decades. CFD technology along with HPC has provided great results in the domain of Aerodynamics , process engineering and Automobile Engineering especially in the Engine development and engineering the BIW contours. These advances make high efficiency and drive stability vehicle. The present study is carried out to validate the results by Aljure et al., in OpenFoam , using FVM. The referred research study was focused on the assessment of LES model , as well as to show the capabilities of capturing the large-scale turbulent flow structures in car-like bodies using relative coarse grids, using ANSYS ICEM CFD. Asmo Car was made by Mercedes-Benz in the 90s to investigate low drag bodies in automotive aerodynamics and testing CFD codes.

The present study is carried out in OpenFoam , using FVM. The referred research study was focused on the assessment of LES model , as well as to show the capabilities of capturing the large-scale turbulent flow structures in car-like bodies using relative coarse grids. The rectangular computational domain of 8.1meter x 2meters x 2meters is considered. Asmo Car was made by

Mercedes-Benz in the 90s to investigate low drag bodies in automotive aerodynamics and testing CFD codes.

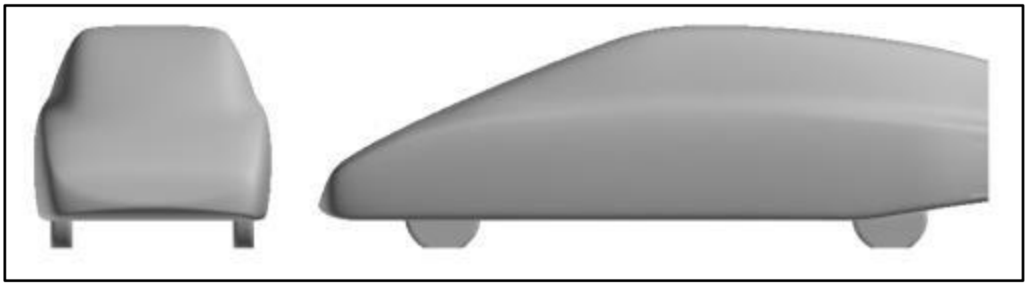


Fig 1. ASMO Car Model

The finite volume method adopts the idea of control volumes used to create models of physical systems. A control volume represents a region of space, which is generally fixed, enclosed by a surface through which fluid flows in and out. It applies conservation equations, e.g., of mass, momentum, and energy, by balancing fluxes, due to inflow and outflow at the bounding surface, with additional sources within the volume.

2 Governing Equations and Models

Incompressible Navier Stoke's Equations are solved:

$$\frac{\partial \bar{u}}{\partial t} + (\bar{u} \cdot \bar{v})\bar{u} - \nu\nabla^2\bar{u} + \rho^{-1}\nabla\bar{p} = -\nabla \cdot \tau \qquad (1)$$

$$\nabla.(\bar{\mathbf{u}}) = 0 \qquad (2)$$

$\bar{\mathbf{u}}$: Filtered Three-Dimensional Velocity Vector
$\bar{p}$: Filtered pressure scalar filed

$$\tau = {-2\nu_{SGS}\bar{S} + (\tau{:}\,\mathbf{I})\mathbf{I}}/{3} \qquad (3)$$

ν_{SGS} : Turbulent or sub-grid viscosity

$\bar{S}$: filtered rate-of-strain tensor

$$\bar{S} = \frac{1}{2}[\nabla(\bar{u}) + \nabla^T(\bar{u})]$$

The governing equations are discretized on an unstructured mesh by means of finite volume techniques.

3 Simulation Procedure

3.1 Geometry and Mesh

The Computational domain refers to a simplified form of the physical domain both in terms of geometrical representation and boundary condition imposition. This simplified form should retain all physically important features of the problem but can ignore minor details. For example, when simulating fluid flow about a car, one can ignore gaps between doors while the actual boundary of the fluid flow domain may be replaced by a simple box-like boundary.

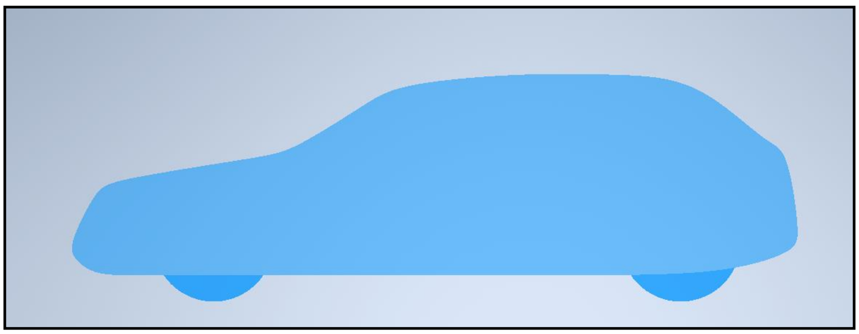


Fig2. CAD ASMO Car

The snappyHexMesh utility generates 3-dimensional meshes containing hexahedra (hex) and split-hexahedra (split-hex) automatically from triangulated surface geometries, or tri-surfaces, in Stereolithography (STL) or Wavefront Object (OBJ) format.

An optional phase will shrink back the resulting mesh and insert cell layers. The specification of mesh refinement level is very flexible, and the surface handling is robust with a pre-specified final mesh quality. It runs in parallel with a load balancing step every iteration.
The number of Cells generated for given geometry is : **1.92×10^6**

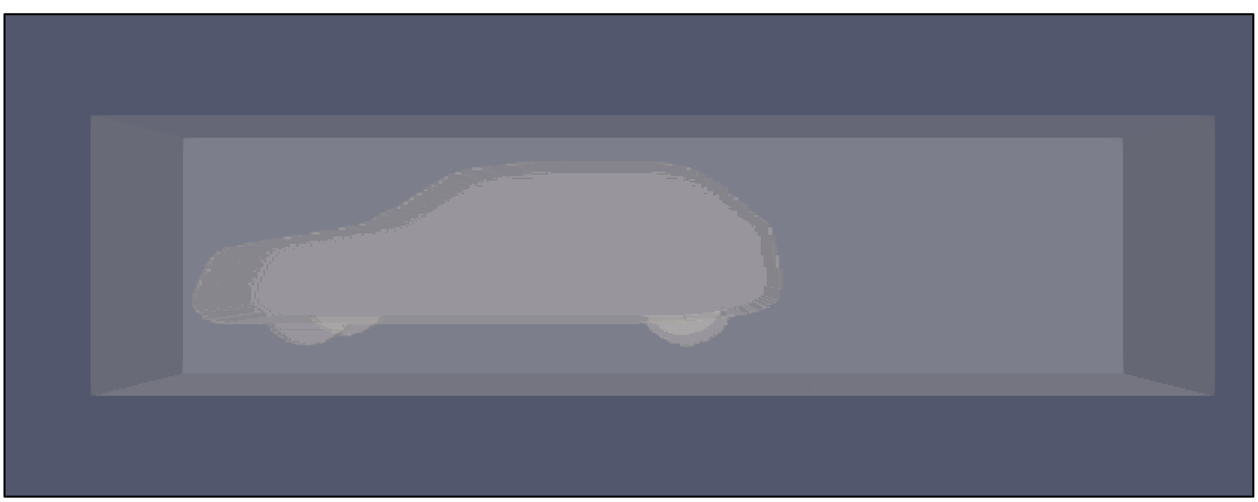
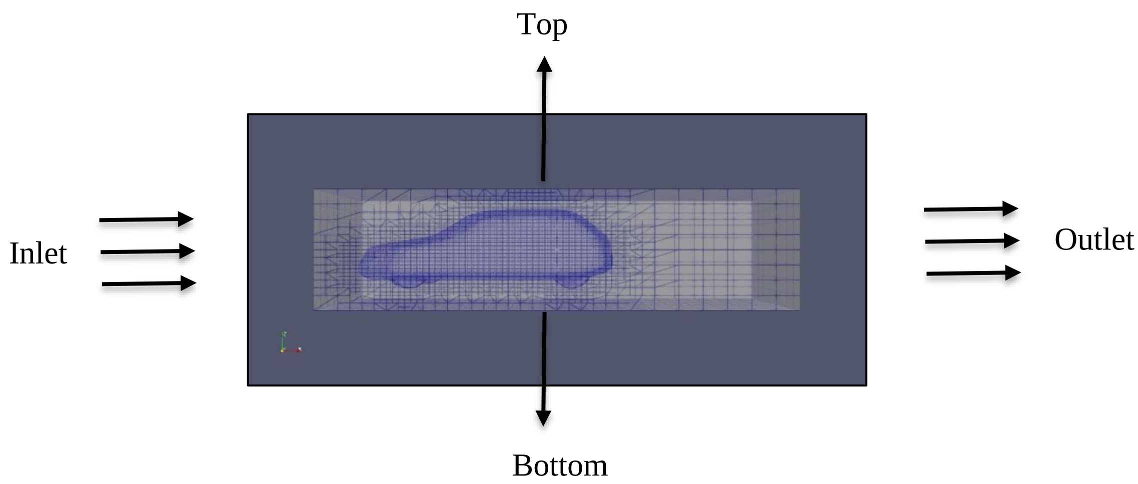


Fig3. snappyHexMesh generation

3.2 Initial and Boundary Conditions

- Constant Velocity inlet.
- The outlet of the domain is modelled using a convective boundary condition.
- No-slip conditions are set for the bottom and car surfaces.
- $Re = u_{ref} * h_{ref} / \nu = 7.68 \times 10^5$



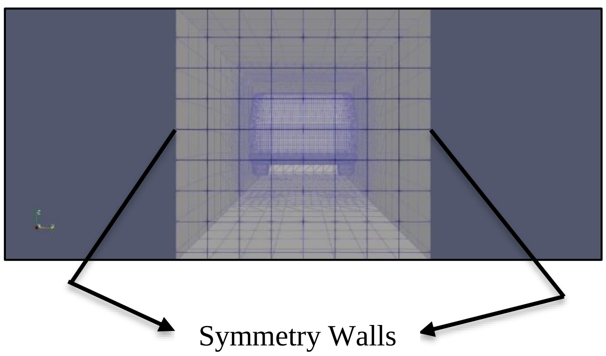


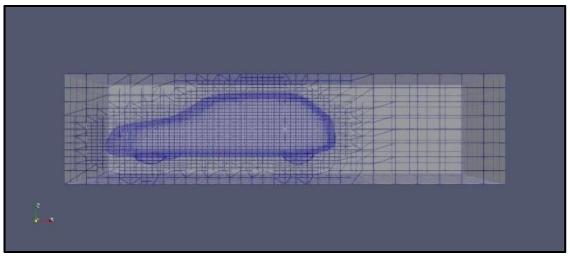
Fig4. Boundary Conditions

3.3 Solver

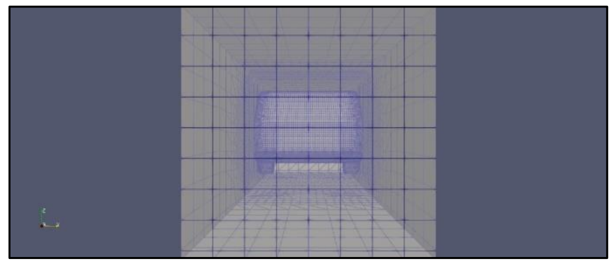
- The geometry is slightly modified to create on CAD as solid modelling.
- The CAD is at 25 deg slanted at backside.
- The rectangular computational domain of 8.1meter x 2meters x 2meters is considered.

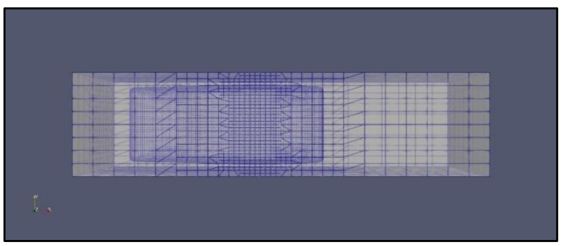
simpleFoam -Steady-state solver for incompressible, turbulent flows.

Front View

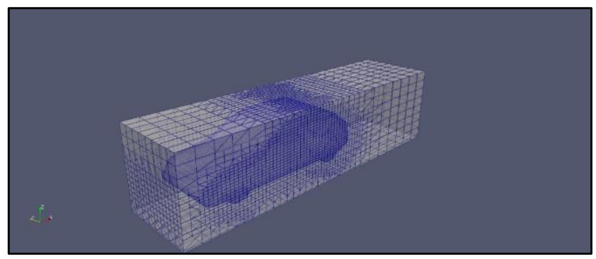


Side View





Top View



Isometric View

Fig 5. Computational Domain Views

4 Results and Discussions

Solver : simpleFoam (Steady-state solver for incompressible, turbulent flows)
Start Time : 3
End Time : 1000

Fig 6 shows the mean streamlines in the location where the main longitudinal vortices are generated. In this case the following can be observed:
(i) a high-speed stream moving along the side wall of the car;
(ii) a slow speed stream in the back of the car and, and
(iii) the main flow traveling along the top of the body.

The vortex is generated by the interaction between these three streams, as the high-speed stream passes the end of the side wall it flows towards the mid plane (pushed by the low-pressure zone found in the back), crashing into the slow speed stream, decelerating before colliding against the main flow closer to the mid plane (where the pressure has a higher value).
This second collision changes the mixed stream direction giving it a twirling motion, creating the longitudinal vortex.

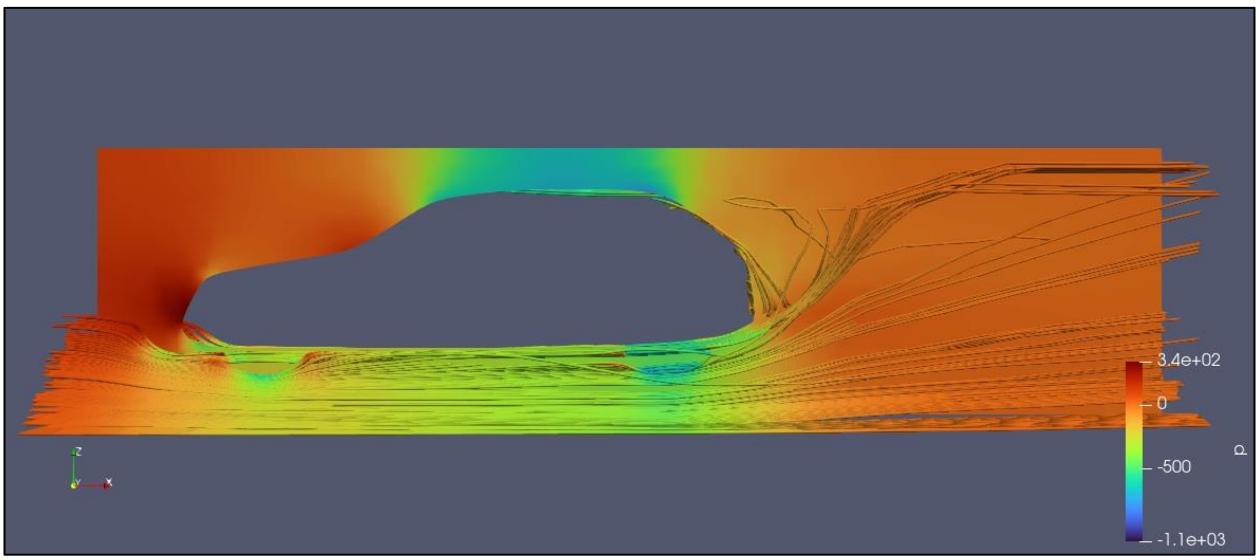


Fig 6.Time Averaged Streamlines at the back if the CAR Model

Fig 7 shows the recirculation bubbles behind the body. The vortex takes the shape of a horseshoe toroidal vortex bounded by the flow coming from the sides and the top of the body. The flow from the underbody, after passing through the diffuser, reduces its speed which decreases its influence in the recirculation bubble.

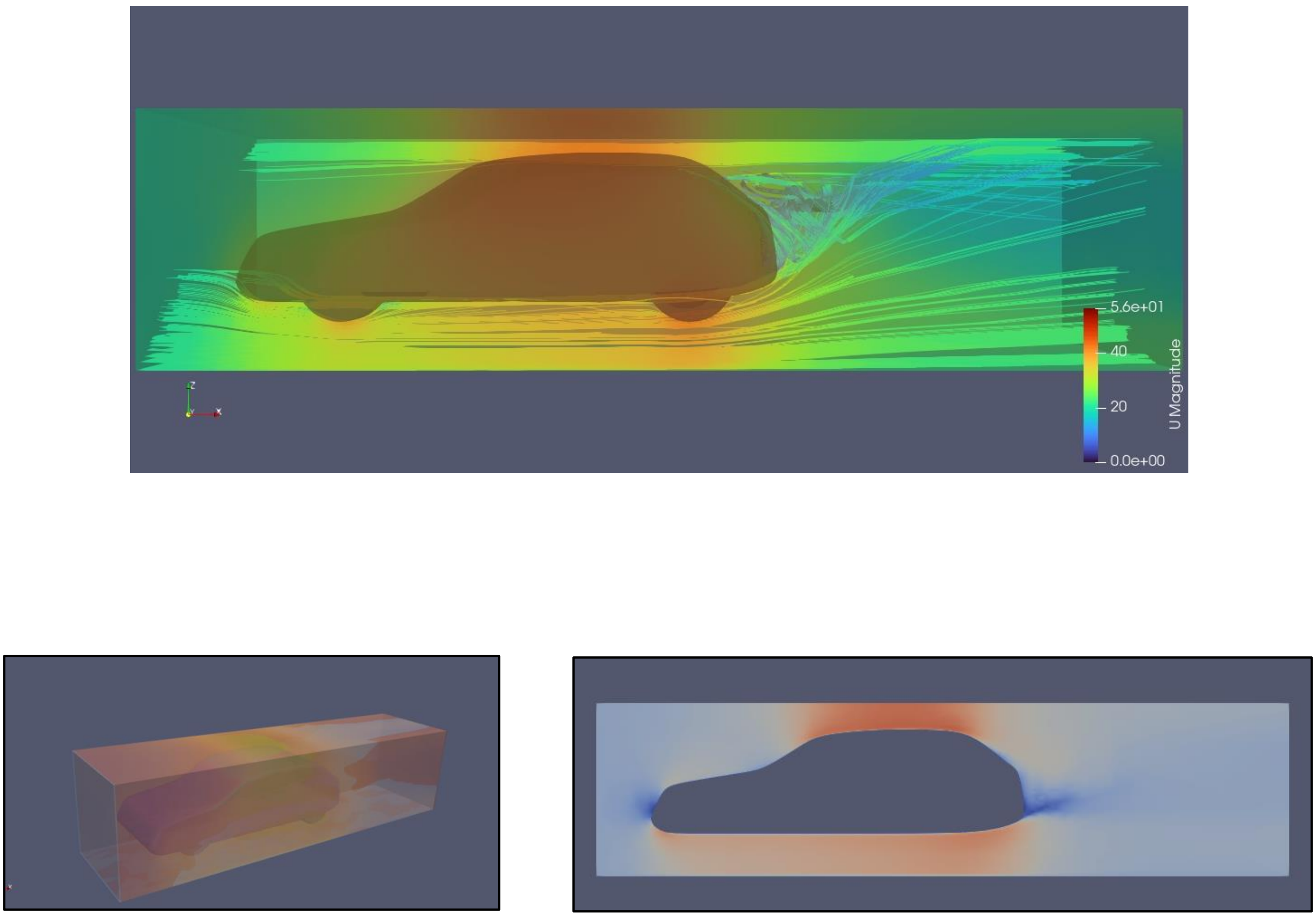


Fig 7. Recirculation Bubble behind the CAR geometry

The high- and low-pressure bubbles around the bodies can be inferred. The low-profile nose of the car allows the formation of a smoother flow pattern, diminishing the stagnation in the flow and reducing the size of the high-pressure bubble.

It is also important to notice that at the slant back, the edges are in a low-pressure zone with high flow separation. This accounts for the formation of the big lifted longitudinal vortices shown before.

There is also a mid-body low pressure zone present in the car geometry. This zone is spread wider in the given geometry, accounting for the smoother pressure gradients of this geometry.

The effects of pressure gradients can be classified in two major groups, positive and negative relative to the direction of the flow. These zones can readily be seen in Fig. 8.

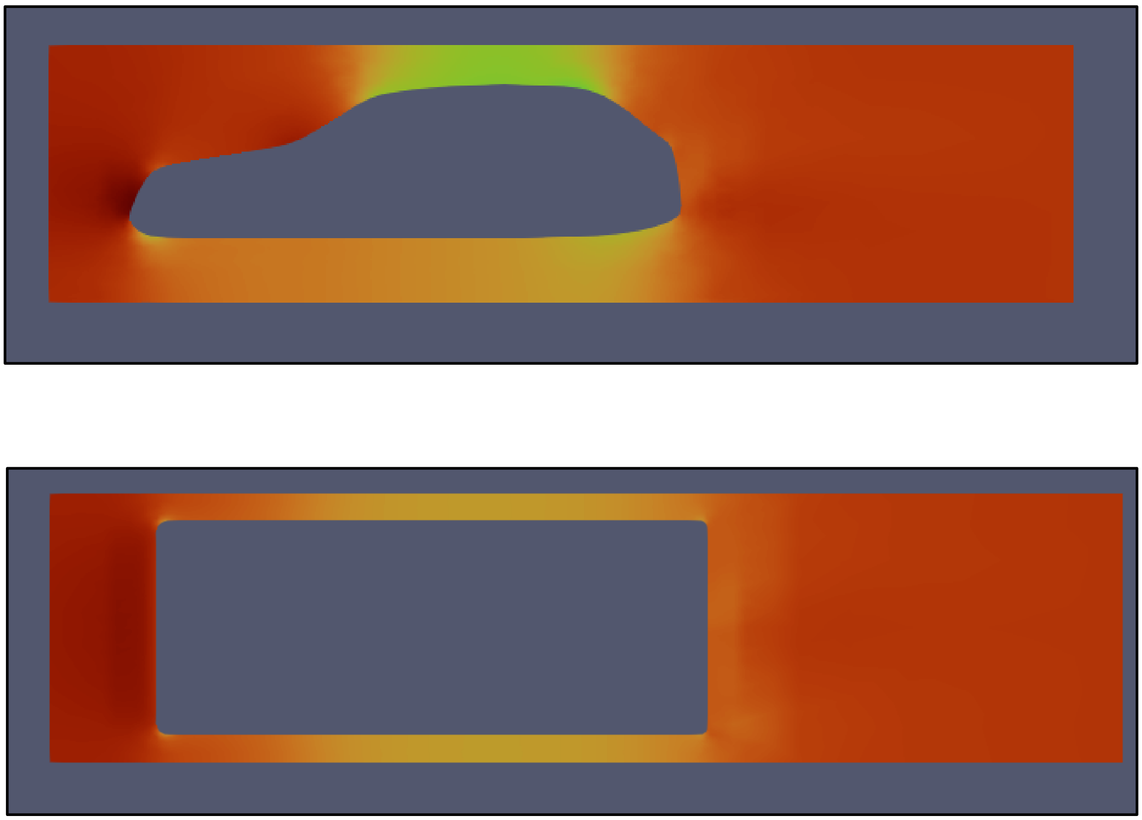


Fig 8. Time-averaged non-dimensional pressure contours

Fig. 9 shows the average pressure coefficient ($C_p = (p-p\infty)/(0.5\rho u_{ref}^2)$) profile alongside the symmetry plane of the underbody and in the back of the car.
In the upwind section the results are close to the Daimler- Benz data, whereas in the downwind section the results closely resemble those of the other LES Models.

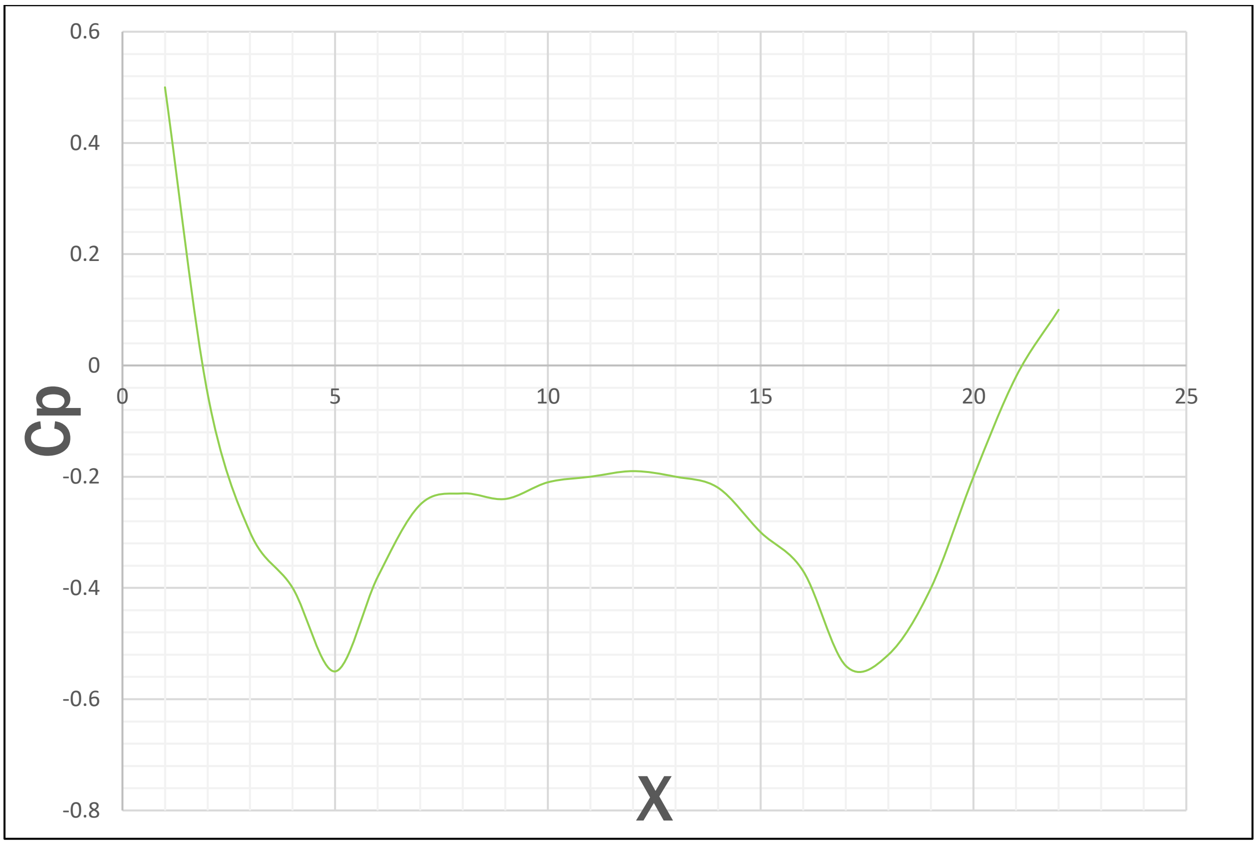


Fig 9: Time-averaged non-dimensional pressure contours

Fig. 10 shows the results in the symmetry plane along the roof of the model. there is a low-pressure bubble wrapping around the car and centered in the middle of the roof. This produces a negative pressure spike in the geometry , something which is also shown in the Fig. 7.

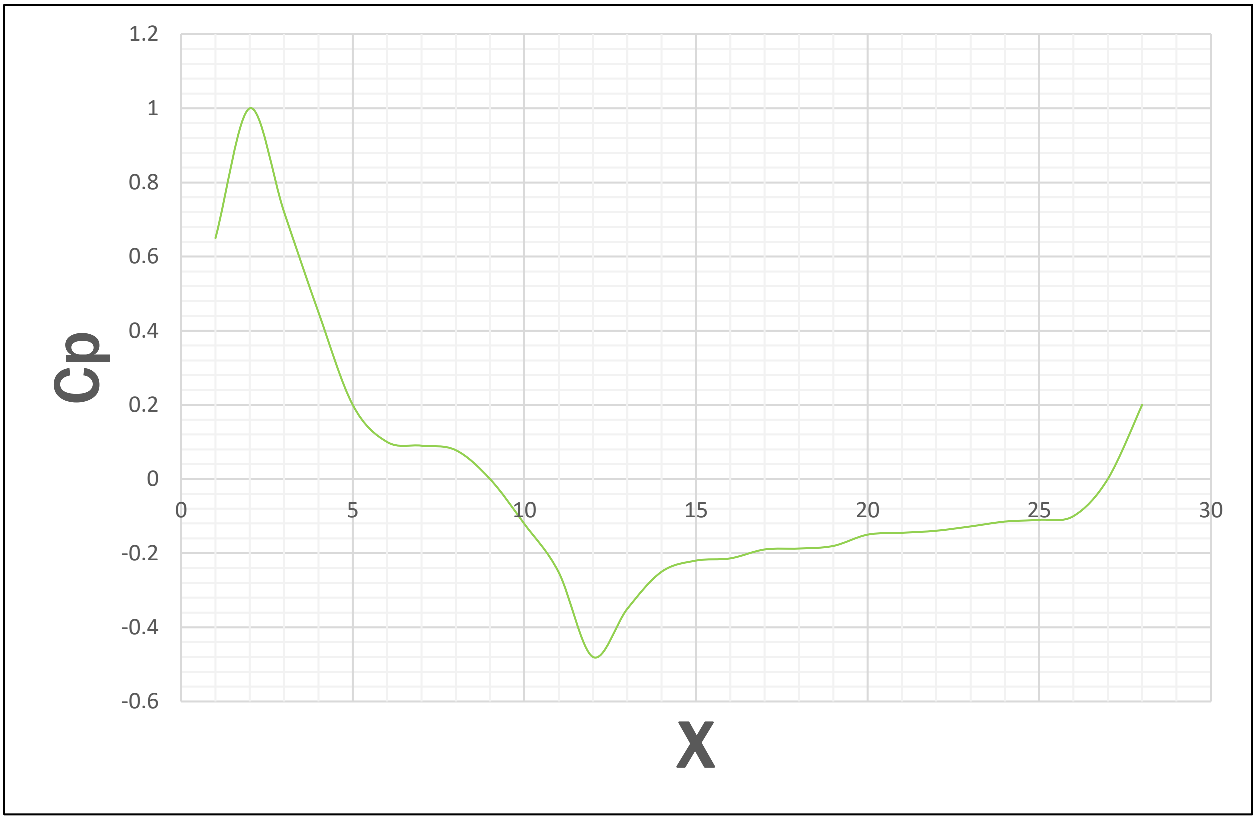


Fig 10 Pressure coefficients for car on the roof

A. The wake region will tend to form more downstream if the downward angle gets increased to , say 30 degrees.
B. Smooth machining and curvature at the ROOF CONSOLE and ROOF RAIL can result in the drag reduction.
C. Since the given study is done at low velocity, therefore temperature study is not executed.
D. This study also does not include the drag force resulting due to fixtures like side mirrors , door handles etc. These fixtures are eliminated in the CAD cleanup.

The study has been performed on k-omega SST model against the LES model to reduce the computational time and limited machine running facility.

References

1. Ahmed SR, Ramm G, Faltin G. Some salient features of the time averaged ground vehicle wake, SAE paper no 840300.
2. Krajnovic´ S, Davidson L. Flow around a simplified car: part1: large eddy simulation. ASME: J Fluids Eng 2005;127():907–19.
3. Minguez M, Pasquetti R, Serre E. High-order large-eddy simulation of flow over the Ahmed body car model. Phys Fluids 2008;20(9):095101.
4. Serre M, Minguez E, Pasquetti R, Guilmineau E, Deng G, Kornhaas M, et al. On simulating the turbulent flow around the Ahmed body: a French–German collaborative evaluation of LES and DES. Comput Fluids.
5. Krajnovic´ S, Davidson L. Flow around a simplified car: part2: understanding the flow. ASME: J Fluids Eng 2005;127:919–28.
6. Lienhart H, Stoots C, Becker S. Flow, and turbulence structures in the wake of a simplified car model (Ahmed model). In: DGLR Fach. Symp. der AG STAB. Stuttgart University; 2000.
7. https://www.openfoam.com/documentation/guides/latest/doc/index.html
8. https://cfd.direct/openfoam/cfd-book/
9. https://cfd.direct/openfoam/user-guide/v6-snappyhexmesh/

DISCLAIMER: This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.