

Numerical Simulations of Flow around a Realistic Generic Car Model

Emmanuel Guilmineau
Ecole Centrale de Nantes

ABSTRACT

This paper focuses on the numerical simulations of flow around a realistic generic car model called the DrivAer body. This new open-source model is based on the geometries of two medium sized cars, the Audi A4 and the BMW 3 series, and possesses more representative car features as the well-known generic Ahmed body. In this paper, only the fastback geometry is investigated. The flow solver used is ISIS-CFD developed by CNRS and Ecole Centrale de Nantes. This solver is based on a finite-volume method, and two turbulence modelizations are used: the Explicit Algebraic Reynolds Stress Model (EARSIM) and a Detached Eddy Simulation (DES). Two meshes are used. For one, the walls are described with a wall function and the mesh contains 19 million cells. This mesh is called "Mesh 1". For the second mesh, a low-Reynolds number turbulence model for the walls is used. In this case, the mesh contains 39 million cells, and is called "Mesh 2". For the simulation with the EARSIM model, the drag coefficient is well predicted while the lift coefficient is over-predicted with however a value lower with the "Mesh 2". With both meshes, an automatic grid refinement has been used. In this case, from the "Mesh 1", the new mesh contains 25 million cells while from the "Mesh 2", the new mesh contains 45 million cells. With these new meshes, the drag coefficient is similar and the lift coefficient is lower than the values obtained with the initial meshes but the predicted values over-predict the experimental value. DES simulations have been performed with all meshes. The aerodynamic coefficients are similar and in good agreement with the experimental values, even for the lift coefficient.

CITATION: Guilmineau, E., "Numerical Simulations of Flow around a Realistic Generic Car Model," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(2):2014, doi:10.4271/2014-01-0607.

INTRODUCTION

The external aerodynamics of a car determines many of its relevant aspects of an automobile such as stability, comfort and fuel consumption at high cruising speed, see Hucho [1]. The study of vehicle aerodynamics is commonly based on generic models such as the Ahmed body, see Ahmed et al. [2], the SAE model, see Cogotti [3]. Generic models help to analyze basic flow structures as vortices at the C-pillar. However, they also fail to reproduce important parts of the flow, as the flow in the wheelhouses in combination with rotating wheels. For realistic applications, more geometric details should be considered. But for a lot of researchers, it is difficult to have access to database for realistic geometries because these geometries are often confidential.

Recently, the Institute of Aerodynamics and Fluid Mechanics of the Technical University of Munich, in cooperation with Audi AG and the BMW Group, propose a new realistic generic car model, called the DrivAer model, and the corresponding experimental data for comparison, see Heft et al. [4], [5]. The body is based on two typical medium-class vehicles and includes interchangeable tops and two different underbody geometries to allow for a high universality. Some first numerical simulations have been performed with the open source software

OpenFOAM™, see Heft et al. [6]. While the overall agreement between the experiments and the simulations is quite good, several discrepancies can be seen. This model has been also studied in crosswind situations, see Strangfeld et al. [7].

This paper presents a short description of the DrivAer model. Afterwards, the flow solver will be presented, followed by a short description of the numerical setup. Then, a comparison between the numerical and experimental data will be presented.

DrivAer MODEL

The geometry of the DrivAer model is based on the geometries of two medium sized cars: the Audi A4 and the BMW 3 Series, see Heft et al. [4]. To be able to investigate the specific flow structures of the different vehicle types, the DrivAer model is designed as a modular concept with three interchangeable rear end forms (fastback, estate back and notchback), see Figure 1, and two different underbody geometries: a smooth underbody and a generic underbody based on the simplified underbody geometry of the Audi A4, see Figure 2. The model's curvature is based on a realistic design, and this model generates more realistic and much more complex flow structures when compared to the Ahmed body.

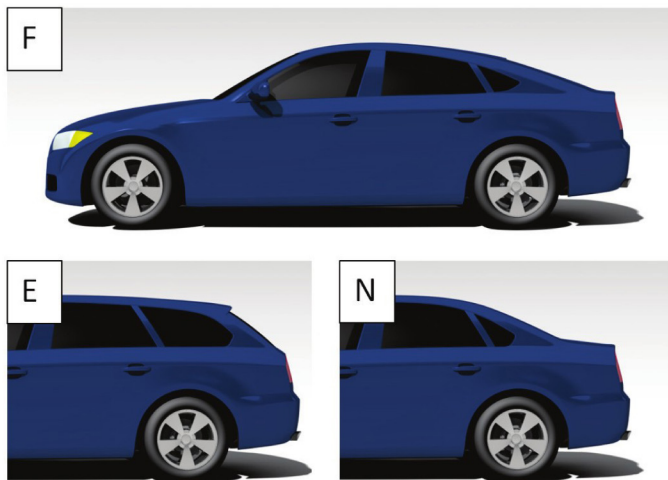


Figure 1. Different configurations of the DrivAer model: Fastback, Estate, and Notchback (courtesy of A.I. Heft).

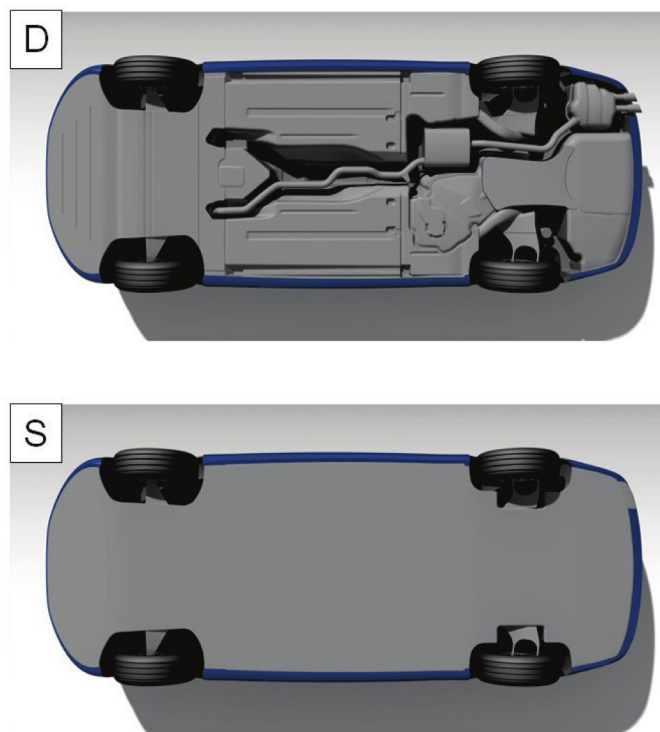


Figure 2. Different underbody configurations: Detailed and Smooth (courtesy of A.I. Heft).

The results presented in this paper are obtained using the configuration of the fastback model and the smooth underbody geometry. Figure 3 shows a sketch of the fastback configuration of the DrivAer model used in this paper. The length of the model is $L = 1845$ mm, the maximum height $H = 567$ mm, and the maximum width $W = 728$ mm. The frontal area is $A_{ref} = 0.3457$ m².

THE ISIS-CFD FLOW SOLVER

ISIS-CFD, developed by the “Ecole Centrale de Nantes” and CNRS and available as a part of the FINE™/Marine computing suite, is an incompressible unsteady Reynolds-averaged Navier-Stokes (URANS) method. The solver is based on the finite volume method to build the spatial discretization of the transport equations. The unstructured discretization is face-based, which means that cells with an arbitrary number of shaped faces are accepted. A second order backward difference scheme is used to discretize time which is treated in a fully implicit manner. Thus, there is no stability limit on the CFL number. The solver can simulate both steady and unsteady flows. A detailed description of the solver is given by Duvigneau et al. [8] and Queutey and Visonneau [9].

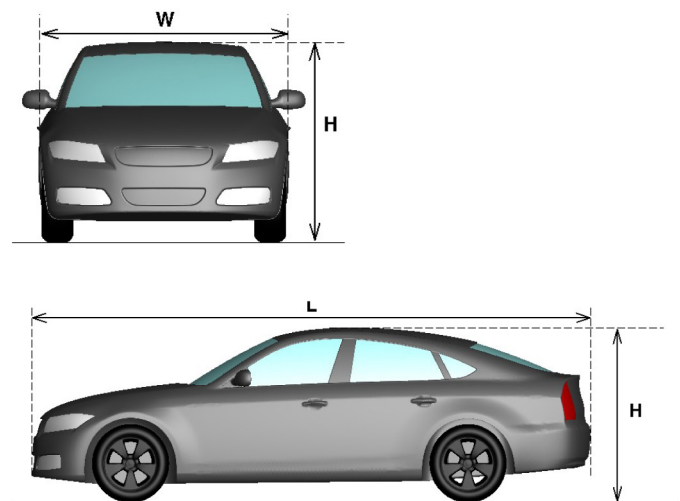


Figure 3. DrivAer model used for the numerical simulation.

The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass equation constraint, or continuity equation, transformed into a pressure equation. In the case of turbulent flows, transport equation for the variables in the turbulence model are added to the discretization. As an implicit formulation is used, the CFL number is not used.

The method features sophisticated turbulence models: apart from the classical two-equation $k-\epsilon$ and $k-\omega$ models, the anisotropic two-equation Explicit Algebraic Reynolds Stress Model (EARSIM), as well as Reynolds Stress Transport Models, are available, see Duvigneau et al. [8] and Deng and Visonneau [10]. All these models can be called RANS simulations. Recently, a Detached Eddy Simulation (DES) approach has been introduced, see Guilmineau et al. [11].

An automatic adaptive grid refinement (AGR) technique is included in the solver, see Wackers et al. [12]. The method supports the isotropic and anisotropic refinement of unstructured hexahedral meshes. The refinement can be done in order to adapt the grid to unsteady problems. The refinement criterion, which indicates where the grid must be refined, can

be modified very easily. Different refinement criteria have already been tested. The grid refinement is performed in parallel and includes an automatic dynamic local balancing in order to redistribute the refined grid over the processors when some partitions are refined more than the others. Recently, this approach has been used for the flow simulations around simplified car models, see Guilmineau and Wackers [13].

NUMERICAL SETUP

Computational Domain and Boundary Conditions

The DrivAer model is positioned 2L from the domain inlet. The total length of the computational domain is 10L. The computational domain has a height of 8H, and a width of 11W. Hence, the blockage ratio calculates to 2%, whereas it is 8% in the experiment.

At the inlet, a uniform velocity $V_0 = 40$ m/s, which correspond to a Reynolds number of $Re = 4.87 \cdot 10^6$ with respect to the vehicle length, is imposed as well as the sides and the top of the computational domain. At the outlet, the pressure is prescribed. For the walls, i.e., the geometry of the car and the floor, two conditions are used depending on the mesh used.

Grids

The generation of meshes is performed using Hexpress™, which the mesh generator of FINE™/Marine. This software generates meshes containing only hexahedron. Two meshes are generated where all walls are a different boundary condition. The "Mesh 1" is a mesh where all the walls are a wall function as boundary conditions. In this case, the non-dimensional wall distances, y^+ , of the first cell center in wall-normal direction are around 6.2 in average. This mesh contains 19.3 millions points and the DrivAer model is described by 2.7 million faces. For the second mesh, called "Mesh 2", all the walls have a no-slip condition, and so the number of points is 38.7 millions. The number of faces on the model is 2.9 millions, and the average value of y^+ is 0.3. For each meshes, a numerical simulation with an automatic adaptive grid refinement technique is carried. In this case, the refinement criterion used is based on the second spatial derivatives of the pressure and the minimum cell size is 2 mm. In this case, two new meshes are created, and they are called "Mesh 1 + AGR" and "Mesh 2 + AGR".

Figure 4 presents a comparison of the mesh obtained with the automatic grid refinement procedure and the initial mesh "Mesh 1". The wheelhouse is refined, see Figure 4.a. In Figure 4.b, the wake of the side mirror contains more cells than the initial mesh. It is the same thing for the wake of the model. The same remarks can be made if the initial mesh is the mesh "Mesh 2".

Settings

In this paper, two turbulence model are used. The first is the EARSM model. In this case, it is a RANS simulation. The second model is a DES approach.

For the RANS simulation, the time step used is $\Delta t = 0.00046$ s and 8 non-linear iterations are performed for each time step to reduce the non-linear residuals by two orders of magnitude. This approach is used to predict the unsteadiness of the flow, particularly at the beginning of the simulation.

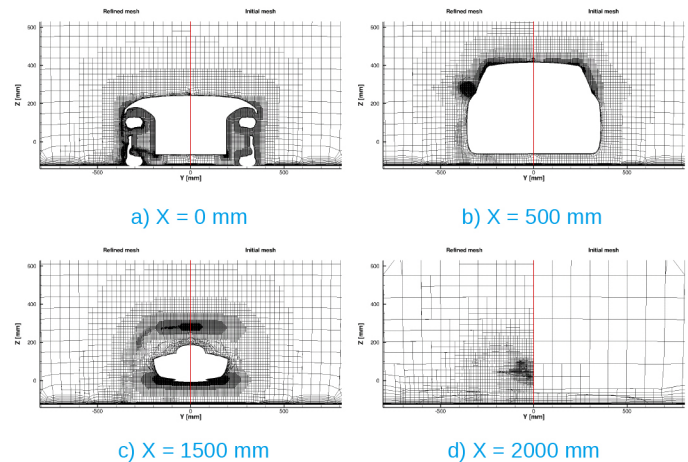


Figure 4. Comparison between the initial mesh 1 and the mesh obtained with the automatic grid refinement procedure.

For the DES simulation, the time step used is set to $\Delta t = 0.0003$ s and 20 non-linear iterations are performed. Initial transient is carried out, and then time-averaged results are calculated for approximatively an additional non-dimensional time $tV_0/L = 23$, based on examining the development of the turbulence kinetic energy in the symmetry plane.

RESULTS

All simulations are performed without using moving ground and rotating wheels.

Flow Structures

Figures 5, 6, 7 show the vortex structure around the model using an iso-surface of the second invariant of velocity gradient (Q) for the numerical simulation using the EARSM turbulence model. The flow detaches at the mirrors and the A-pillars. A vortex is generated at the windshield beginning. For all figures, the flow characteristics are similar. However, the wake shedding behind the mirror is different, as the structures around the front wheels.

Figure 8 presents the vortex structures for the numerical simulation using DES. This figure shows the averaged flow. The same structure of the flow as that obtained with a RANS simulation is obtained. However, the length of the A-pillar