Computers & Fluids 96 (2014) 122–135

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Flow and turbulent structures around simplified car models

D.E. Aljure [a], O. Lehmkuhl [a,b], I. Rodríguez [a], A. Oliva [a,*]

[a] Heat and Mass Transfer Technological Center (CTTC), Universitat Politècnica de Catalunya-BarcelonaTech (UPC), ETSEIAT, Colom 11, 08222 Terrassa, Barcelona, Spain
[b] Termo Fluids, S.L. Av. Jaquard, 97 1-E, 08222 Terrassa, Barcelona, Spain

ARTICLE INFO

Article history:
Received 21 February 2013
Received in revised form 6 March 2014
Accepted 13 March 2014
Available online 21 March 2014

Keywords:
Automotive aerodynamics
CFD
Challenging LES
Turbulence models

ABSTRACT

External car aerodynamics study has great importance in overall car efficiency and ride stability, being a key element in successful automotive design. The flow over car geometries shows three dimensional and unsteady turbulent characteristics. Additionally, vortex shedding, flow reattachment and recirculation bubbles are also found around the bluff body. These phenomena greatly influence the lift and drag coefficients, which are fundamental for ride stability and energy efficiency, respectively. The aim of the present study is focused on the assessment of different LES models (e.g. VMS or SIGMA models), as well as to show their capabilities of capturing the large scale turbulent flow structures in car-like bodies using relative coarse grids. In order to achieve these objectives, the flow around two model car geometries, the Ahmed and the Asmo cars, is simulated. These generic bluff bodies reproduce the basic fluid dynamics features of real cars. First, the flow over both geometries is studied and compared against experimental results to validate the numerical results. Then, different LES models are used to study the flow in detail and compare the structures found in both geometries.

1. Introduction

Computational fluid dynamics (CFD) has evolved greatly in the past two decades becoming a vital tool in industrial research, development and investigation. Due to the nature of fluid flows, most practical applications deal with turbulent motion. The modeling of this phenomenon is of importance within the CFD industry where vast resources are invested into its research. Within the different possible applications of the CFD technology come aerodynamics and automotive design. The automotive industry makes great advances every year; engine performance has increased greatly in the past decade, along with weight reduction and safety measures. These advances make aerodynamics more important to ensure high efficiency and vehicle drive stability.

The bodies to be studied in the present paper are the widely studied Ahmed car model with a 25° angle of the rear slanted surface (Ahmed et al. [1], Krajnović and Davidson [2], Minguez et al. [3], Serre et al. [4], Lehmkuhl et al. [5], among others), and the Asmo car model (Perzon and Davidson [6], Nakashima et al. [7], among others). The Ahmed body car is a semi-rectangular vehicle with a rounded front and a slanted back. The simplified topology of this model allows easy modeling, meshing and comparisons between experimental and numerical results. As for the Asmo car, it has a square back rear, smooth surface, boat tailing and an under body diffuser. This model is characterized by no pressure induced boundary layer separation and low drag coefficient.

The Ahmed car was originally used in the experiments of Ahmed et al. [1]. They concluded that the flow structure in the wake was controlled, for a given Reynolds number, by the inclination of the slanted back. Lienhart et al. [8] performed further experiments with the 25° and 35° slant back geometries using a laser Doppler anemometer to make detailed measurements of the velocity profiles around the bodies. Numerous numerical studies have been carried out using the aforementioned geometry. The 25° and 35° geometries were used in the 9th ERCOFTAC workshop [9]. Results presented by different groups varied greatly, mainly by insufficient grid resolution and convergence. Hinterberger et al. [10] performed large eddy simulations (LES) for the 25° Ahmed geometry. The authors used two grids of 8.8×10^6 and 18.5×10^6 control volumes (CV). Flow comparison with the experiments performed by Lienhart et al. [8] revealed that the flow structures were well captured. Kapadia et al. [11] used a Spalart–Allmaras based Detached-Eddy Simulation (DES) to model the flow around the 25° and 35° Ahmed car using 2.3×10^6 CV, 3.1×10^6 CV and 4.6×10^6 CV meshes. The authors compared the Reynolds-Averaged Navier–Stokes (RANS) model, the DES and experimental results concluding that their simulations were not very satisfactory

* Corresponding author. Tel.: +34 93 739 81 92; fax: +34 93 739 89 20.
E-mail addresses: davidal@cttc.upc.edu (D.E. Aljure), termofluids@termofluids.com (O. Lehmkuhl), ivette@cttc.upc.edu (I. Rodríguez), oliva@cttc.upc.edu, cttc@cttc.upc.edu (A. Oliva).

http://dx.doi.org/10.1016/j.compfluid.2014.03.013
0045-7930/

as the flow separations were not correctly predicted. Krajnović and Davidson [2,12], in two papers, performed large eddy simulations at a lower Reynolds number (2×10^5) to decrease the computational requirements. Three grids containing 3.5×10^6, 9.6×10^6, and 16.5×10^6 were used, whereas a SIMPLEC algorithm solved the velocity–pressure coupling. The authors concluded that the influence of the Reynolds number in the wake, after the separations at sharp edges, is small. Furthermore, flow visualization results, including time averaged and instantaneous data, showed structures not observed in experimental set ups. Minguez et al. [3] performed high-order LES for the 25° slant back case using a mesh of 21×10^6 CV. They used a spectral vanishing viscosity technique to perform the LES around the geometry. The results, conducted at the Reynolds number $Re = 7.68 \times 10^5$ previously discarded by Krajnović and Davidson [2,12], showed improvement in the overall flow resolution and allowed the visualization of all relevant structures. Recently, Serre et al. [4] compared different approaches (the Smagorinsky subgrid scale model with a wall-function and a mesh with 18.5×10^6 CV, the dynamic Smagorinsky with near-wall resolution and a mesh with 40×10^6 CV, a LES based on spectral approximations with a 21×10^6 CV mesh, and a Detached Eddy Simulation (DES) with a $k - \omega$ SST model and a mesh with 23.1×10^6 CV mesh) for computing the flow over the Ahmed geometry at Reynolds number $Re = 7.68 \times 10^5$. Advantages and disadvantages of the different methods were addressed in the paper. Lehmkuhl et al. [5] used a coarser mesh to simulate the 25° slant back case with a 8.32×10^5 CV mesh, and different LES models. Results from this paper showed the good stability and results a conservative formulation of the governing equations can achieve.

As for the Asmo car, it is a model created by Daimler-Benz in the 90s to investigate low drag bodies in automotive aerodynamics and testing of CFD codes with a geometry not related to the development of Mercedes cars. Wind tunnel experiments were made by both, Daimler-Benz and Volvo. Aronson et al. [13] studied the flow in the under body region of the car and the differences in drag different geometries (wheels and diffuser) achieved. Their analysis showed that the rear wheels contributed greatly to the drag coefficient. They also saw a significant reduction in drag when the model was equipped with a rear diffuser. Perzon and Davidson [6] performed stationary simulations using RANS models on a 3.8×10^5 CV mesh and transient simulations in a 7.6×10^5 CV mesh using the standard $k - \epsilon$ model and the SZL-model among others. Results of these simulations showed good agreement in pressure and high over prediction in the drag coefficient. Nakashima et al. [7] simulated the flow over the Asmo car using three tetrahedral meshes of 1.3×10^6, 5.5×10^6 and 24×10^6 CV and both RANS and LES approaches. The LES simulations in this paper showed overall better results over the RANS models. Tsubokura et al. [14] performed LES and RANS simulations on the Asmo car using meshes with 5.5×10^6 and 24.3×10^6. Their study on this geometry was aimed to validate the turbulence models for simulations on more complex geometries concluding at the end that LES is a powerful tool within vehicle aerodynamics. Simulations performed within the Xflow project in vehicle aerodynamics [15] were carried out using a Reynolds number, based on body length, of $Re = 2.7 \times 10^6$. They used a LES with a wall- adapting local eddy diffusivity (WALE) model within a Lattice-Boltzmann algorithm and concluded that results for the drag coefficient are highly improved when using LES models.

Several difficulties arise when modeling turbulent flows. RANS models often fail to correctly reproduce flow dynamics, especially in detached flows. On the other hand, LES demand high computational resources when dealing with industrial-type high Reynolds flows. In the present paper we aim at modeling the flow past aerodynamic simplified cars by means of large-eddy simulations using relative coarse grids. A second-order symmetry-preserving and conservative formulation is used for the discretization of the governing equations. The properties of the present formulation ensure stability and conservation of the kinetic energy balance even with coarse grids at high Reynolds numbers. Thus, in the present paper challenging large-eddy simulations (CLES) are performed. In these CLES, coarse meshes are used in conjunction with conservative and symmetry preserving numerical schemes to reduce the computational requirements necessary to simulate these cases. The capabilities of CLES for reproducing the complex fluid flow phenomena present in automotive aerodynamics using simplified car models are investigated. Both geometries, i.e. the Ahmed and the Asmo cars, have been widely studied in the literature and flow structures around them are well known. By means of different SGS models, the flow structures obtained using different grids are compared to those identified in experimental and numerical works from the literature. In addition to the mean and instantaneous flows, aerodynamics coefficients are also compared. The performance of the different models for predicting these complex aerodynamics forces in car-like geometries is discussed in detail.

2. Definition of the case

2.1. Geometries and computational domain

The geometries to be considered are the 25° slant back Ahmed car shown in Fig. 1a and the Asmo car depicted in Fig. 1b.

Both cases are solved in a rectangular computational domain of $9.1944 \times 1.87 \times 1.4m$ shown in Fig. 2. The front of the car is located at 2.1014 m downwind from the inlet boundary. The outlet boundary is at a distance of 6.048 m for the Ahmed car and of 6.282 m for the Asmo car, measured from the rear end of the body. To simulate the same case conditions as those measured by Lienhart et al. [8] a 3/4 open wind tunnel is considered in both geometries.

3. Mathematical and numerical model

In order to study the flow, the filtered incompressible Navier–Stokes equations are solved:

$$\frac{\partial \overline{\mathbf{u}}}{\partial t} + (\overline{\mathbf{u}} \cdot \nabla)\overline{\mathbf{u}} - \nu \nabla^2 \overline{\mathbf{u}} + \rho^{-1} \nabla \overline{p} = -\nabla \cdot \tau \quad (1)$$

$$\nabla \cdot \overline{\mathbf{u}} = 0 \quad (2)$$

where $\overline{\mathbf{u}}$ is the filtered three-dimensional velocity vector, $\overline{p}$ is the filtered pressure scalar field, ν stands for kinematic viscosity and ρ for the density of the fluid. τ corresponds to the subgrid-scale (SGS) stress tensor:

$$\tau = -2\nu_{SGS}\overline{S} + (\tau : \mathbf{I})\mathbf{I}/3 \quad (3)$$

where ν_{SGS} is the turbulent or subgrid viscosity and $\overline{S}$ is the filtered rate-of-strain tensor, $\overline{S} = \frac{1}{2}[\nabla(\overline{\mathbf{u}}) + \nabla^T(\overline{\mathbf{u}})]$.

To close the formulation, the subgrid-scale viscosity should be modeled. Turbulence modeling is carried out in the present paper by using different SGS models: the wall-adapting local-eddy viscosity, WALE, [16], the WALE model within a variational multiscale framework, VMS, [17], the singular values subgrid model, SIGMA, [18], and a model which uses the invariants (q and r) of the filtered strain tensor to model eddy viscosity, QR, [19]. Hereafter, a small description of each model is given.

3.1. QR model

This model was proposed by Verstappen [19] responding to the question of damping subfilter scales properly. It is based on the invariants of the rate-of-strain tensor. This eddy viscosity model