

Numerical Simulation with a DES Approach

2010-01-0758

Published
04/12/2010Emmanuel Guilmineau
Ecole Centrale De Nantes

Copyright © 2010 SAE International

ABSTRACT

This paper presents a finite-volume-based detached-eddy simulation for the prediction of flow around a passenger vehicle. The flow solver used is ISIS-CFD developed by the CFD Department of the Fluid Mechanics Laboratory of Ecole Centrale de Nantes. The validation is carried out by a crosswind simulation around the squareback Willy model. The model was designed in order that separations are limited to the region of the base for a moderate yaw angle. This model without sharp corners on the fore body and a square base is more convenient for the analysis of unsteady separations limited on its leeward side and base. The angle between the upstream velocity and the direction of the model varies between 0° and 30°. The results are compared to a previous numerical study obtained with a RANS simulation and experimental data. All comparisons (aerodynamic forces, wall pressures, and topology of total pressure) show that DES simulations give a better agreement with experimental data, particularly for the large yaw angles. The second model is the classical Ahmed body with the slant angle of 25°. With a RANS simulation, we have a massive separation in the wake while with DES simulation, the agreement is better with the experimental data. This paper shows that DES simulations give an improvement of the wake prediction for the automotive flow.

INTRODUCTION

The external aerodynamics of a car determines many relevant aspects of an automobile such as stability, comfort and fuel consumption at high cruising speeds [1]. The flow around vehicles is characterized by highly turbulent and three-dimensional separations, and there is a growing need for more insight into the physical features of these dynamical flows on the other hand, and powerful numerical tools to analyze them on the other hand. Computations based on

Reynolds-Averaged Navier Stokes Equations (RANS) are common in industry today. Although they are very successful in predicting many parts of the flow around a vehicle, they are unable to predict unsteadiness in the wake region. The failure in predicting the base pressure is the major reason for the large discrepancy in drag prediction between experiments and numerical simulations.

In an attempt to improve the predictive capabilities of turbulence models in highly separated regions, Spalart et al [2] proposed a hybrid approach, which combines features of classical RANS formulations with elements of Large Eddy Simulations (LES) method. The concept has been termed Detached Eddy Simulation (DES) and is based on the idea of covering the boundary layer by a RANS model and of switching the model to a LES mode in detached regions. Compared to classical LES methods, DES save orders of magnitude of computing power for high Reynolds number flows, due to the moderate costs of the RANS model in the boundary layer region, but still offers some of the advantage of an LES method in separated regions.

The capabilities of DES turbulence model for automotive applications will be evaluated and discussed in this paper. Two car models will be investigated. The first is the Willy model which is designed to understand the physics of separated flow with application to side gust wind effects. This generic model has no sharp corners on the fore body and a square base which are more convenient for analyzing unsteady separations limited to its leeward side and base. Numerical simulations with RANS are available but for the large yaw angles, the agreement between numerical results and experiments is less accurate than for the small yaw angles [3]. Consequently, DES may constitute a valuable way to compute the Willy body flow with a large yaw angles.

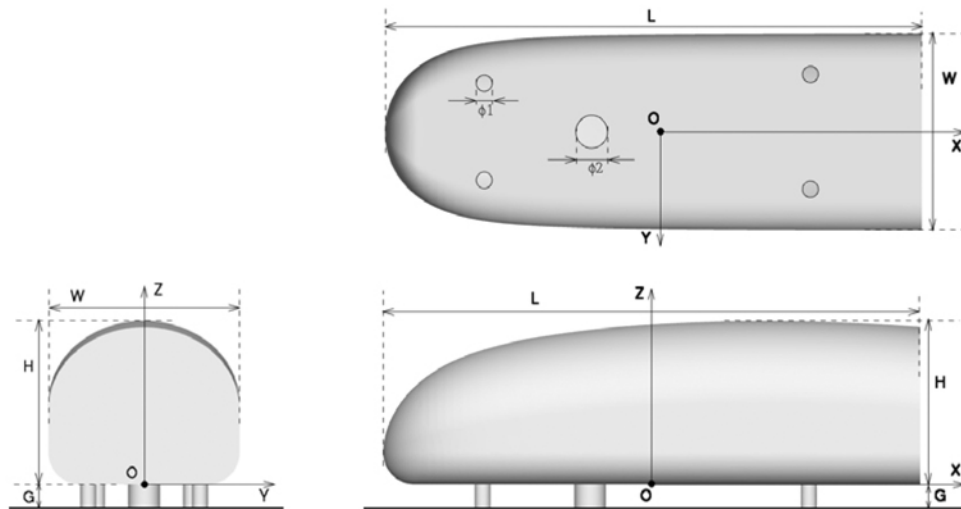


Figure 1. Willy - View of the model

The second car model is the Ahmed model. This model has been selected due to the availability of the experimental results [4, 5]. The flow is three-dimensional and shows separations that can be followed by reattachments depending on the slant angle. Numerous attempts to compute this body flow by RANS approaches or Unsteady RANS (URANS) approaches have been proposed [6, 7]. Comparisons with the experiments show that these approaches only succeeded to simulate the flow at the slant angle 35° . The flow computed for the slant angle 25° does not match the experimental results. Generally, it is the partial detachment of the flow on the rear slant that provides the main difficulty. Consequently, LES attempts have been carried out. The first LES results [8, 9] show that the accuracy of the solution is undeniably improved compared to RANS studies. But the flow does not reattach over the slant and turbulent stresses profiles show large differences with experimental data. A new LES using a spectral numerical method [10] provides predictions in good global agreement with the reference experimental results. However, the drag is highly over-estimated. Recently, a Lattice-Boltzmann approach [11] has provided more satisfactory results, but no turbulent stress profiles have been presented. DES simulations have also been carried out [12]. Small and large scale structures of instability on the slant are reported. However, the statistics do not compare favorably with the experimental results. The mesh used for these DES simulations is not sufficiently fine. It is a possible explanation of the disagreement.

In this paper, the side wind effects on the Willy model and the flow over the Ahmed body with the slant angle 25° are investigated. The turbulence model is a DES simulation. All simulations have been computed with the flow solver ISIS-CFD.

SIMPLIFIED CAR MODELS

WILLY MODEL

The Willy model is a square-back model, which is a realistic representation of a van-type vehicle. This model was used first time by Chometon et al [16]. This model is presented in Figure 1. The length of the model (L) is 675 mm, the width 240 mm, the maximum height 192 mm and the reference surface (S_{ref}) is 41,791 mm². The ground clearance is $G = 29$ mm and the diameter (ϕ_1) of the four feet (f) which are used to secure the model to the floor of the wind tunnel is 20 mm. A cylinder (c) of diameter (ϕ_2) is 40 mm is used to protect the pressure tubes passing from the pressure taps to the multi-channel manometer. The other dimensions are defined in [3]. The reference axis (X_o, Y_o, Z_o) is the Eiffel axis where the axis X_o is parallel to the upstream. The reference axis (X, Y, Z) is linked to the model. The origin of these axis lies at the point O located on the floor of the model, see Figure 1. This point O is the rotation center of the model.

The upstream velocity (V_o) is 20 m/s. The Reynolds number, based on the length of the model, is 900,000. The yaw angle β is positive when the right side of the car model is windward.

<figure 1 here>

The computational domain starts $3.9 L$ in front of the model and extends to $5.4 L$ behind the model. The width of the domain is $3 L$ and its height is $1.6 L$. The cross-section of the computational domain is such that the blockage ratio is 1.9% at $\beta = 0^\circ$ and 4.3% at $\beta = 30^\circ$. The mesh is generated using HEXPRESSTM, an automatic unstructured mesh generator. This software generates meshes containing only hexahedrals. For the surface of the car model, we use a no-slip boundary condition, and the wall normal resolution is set 0.0006 mm,

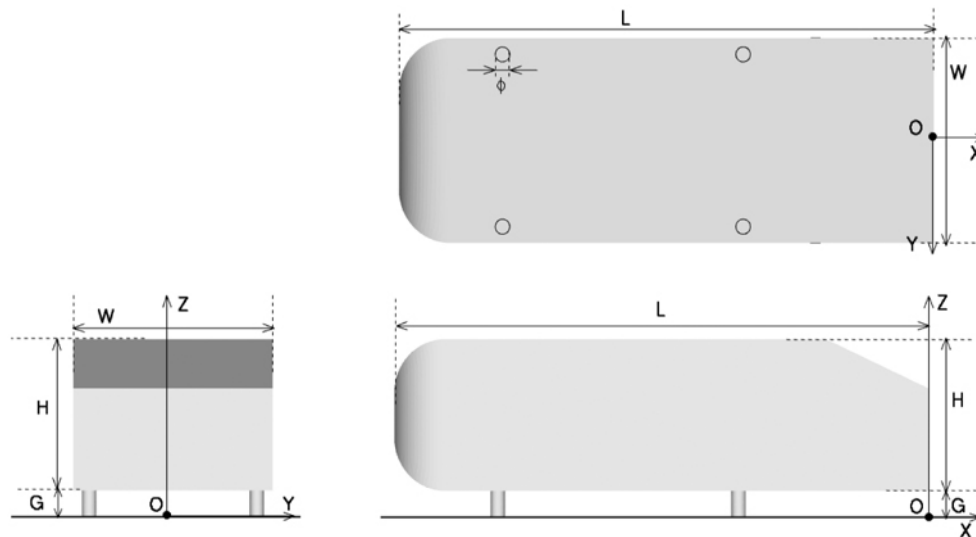


Figure 2. Willy - View of the model

i.e. $y^+ < 0.5$. For the ground of the wind tunnel, we use a wall function, and the distance between the ground and the first fluid points is set to 0.6 mm. This approach, used to take the walls into account, is the same as the approach proposed by Krajnovic and Davidson [17], who used a LES approach around a generic ground model. At the outlet of the computational domain, the pressure is prescribed, while for the other boundaries, we use a Dirichlet condition for the velocity and turbulence. The total number of nodes is 6.6 million, and the model is described by 103,000 faces.

The numerical results obtained by DES presented in this paper will be compared with experimental data and previous numerical results that have been presented recently [3]. These previous numerical results have been obtained with a non-linear turbulence model, the Explicit Algebraic Stress Model (EASM), and a very fine mesh, approximatively 20 million nodes.

AHMED BODY

The geometry of the Ahmed body is shown in [Figure 2](#). The slant angle is adjustable and is the main variable model-parameter in the experimental investigations. Only, the 25° slant angle was investigated. The Ahmed body, of length (L) 1044 mm, height (H) 288 mm, and width (W) 389 mm, was placed at 50 mm from the ground (G), as presented in [Figure 2](#). The model is mounted on four cylindrical struts with a diameter (ϕ) of 30 mm. The reference axis (X,Y,Z) is linked to the model. The origin of these axis lies at the point O located on the floor of the ground of the wind tunnel, on the base of the model and in the symmetry plane of the model, see [Figure 2](#).

The upstream velocity is $V_0 = 40$ m/s. The Reynolds number, based on the length of the model, is 2.78×10^6 .

The computational domain starts $2 \times L$ in front of the model and extends to $5 \times L$ behind the model. The width of the domain is 1.87 m and its height is 1.4 m. These dimensions are recommended for the ERCOFTAC workshop on Refined Turbulence Modelling [6]. The cross-section of the tunnel gives a blockage ratio of 4.3%. The mesh is also generated using HEXPRESSTM. The boundary conditions are similar to those used for the Willy model, except for the ground of the wind tunnel. For this boundary condition, we use a no-slip boundary condition. The total number of nodes is 23.6 million, and the model is described by 384,000 faces.

<[figure 2](#) here>

NUMERICAL APPROACH

The ISIS-CFD flow solver, developed by the EMN (Equipe Modélisation Numérique) of the Fluid Mechanics Laboratory of the Ecole Centrale of Nantes, uses the incompressible unsteady Reynolds-averaged Navier-Stokes equations. The solver is based on the finite volume method to build a spatial discretization of the transport equations. The face-based method is generalized to two-dimensional or three-dimensional unstructured meshes for which non-overlapping control volumes are bounded by an arbitrary number of constitutive faces. The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure-equation. A second-order accurate three-level fully implicit time discretization is used and surface and volume integrals are evaluated using second-order accurate approximation, Jasak et al. [18]. In the case of turbulent flows, additional transport equations for modeled variables are solved in a form similar to that of the momentum equations and they can be discretized and solved using the same principles.

In this study, the DES-SST turbulence model presented by Menter and Kuntz [12] is the model used. The idea behind the DES model of Strelets [19] is to switch from the standard k- ω SST of Menter [13] to an LES model in those regions where the turbulent length, L_t , predicted by the RANS model is larger than a local grid spacing. In this case, the length scale used in the computation of the dissipation rate in the equation for the turbulent kinetic energy is replaced by the local grid spacing Δ . The destruction term, that is equal to $\varepsilon = \beta^* \omega^3 k$ in RANS model, becomes $\varepsilon = \beta^* \omega^3 k F_{DES}$ in DES where the function F_{DES} is defined by

$$F_{DES} = \max\left(\frac{L_t}{C_{DES} \Delta}, 1\right)$$

The turbulent length is defined by $L_t = \sqrt{k}/(\beta^* \omega)$, $\Delta = \sqrt{\Delta x^2 + \Delta y^2 + \Delta z^2}$ and $C_{DES} = 0.78$.

In this study, older numerical simulations [3] used the quadratic explicit algebraic model (EASM) [14]. This model adopts the SSG pressure-strain rate model [15], and solves the BSL k- ω model [13] to determine the turbulence velocity and length scales. It takes into account the variation of the production-to-dissipation rate ratio. This model can be integrated down to the wall and gives correct log law behaviors without any recirculation. The equations of this model are presented in [3].

RESULTS

WILLY MODEL

The time step used for the simulations is $\Delta t = 3 \times 10^{-4}$ s, which corresponds to 112 time steps for one passage of the car at freestream velocity. The averaging time, $t \times V_0/L$, in the simulations is 28.

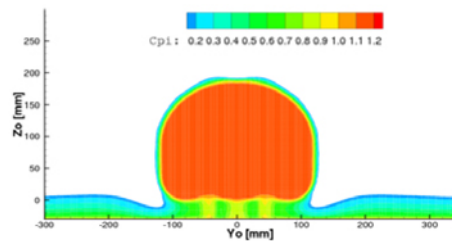
Figures 3, 4 and 5 represent the total pressure coefficient in a slice perpendicular to the upstream velocity at $X_0/L = 0.60$ for the yaw angles 0° , and 20° and at $X_0/L = 0.55$ for the yaw angle 30° . For the yaw angle $\beta = 0^\circ$, the flow is attached to the body. Only, a recirculation bubble appears in the wake. Its length depends on the turbulence model chosen. With the EASM model, its length is $0.50 \times L$ while with the DES simulation, the length is $0.44 \times L$. In another experiment [20], where the cylinder (c) is not present, the length of this bubble, is $0.4 \times L$. In the experiments, Fig. 3(c), the small loss of total pressure noted (V) is due to an interaction of an upstream obstacle with the boundary layer of the wind tunnel. When the yaw angle increases, the boundary layer thickens until a separation zone, designated V2, is created as shows Figure 6 for the yaw angle $\beta = 10^\circ$. This vortex (V2) is more stretched in the results obtained with a DES simulation, Fig.

6(b), than the results obtained with the EASM turbulence model. We notice also a second vortex, located under the vortex V2, designed V3, close to the bottom of the model and attached to the model. As the yaw angle increases, the size of the vortex V2 increases also.

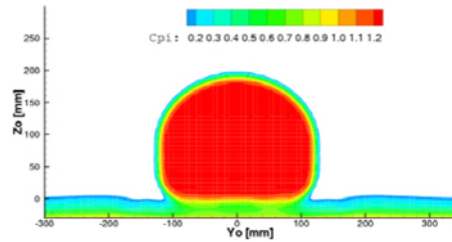
For the yaw angle $\beta = 20^\circ$, the size of the vortex V2 located on the leeward has increased. We notice also a zone, towards $Y_0 = -100$ mm, where the total pressure has increased. We see that when the yaw angle increases, the position of the vortex V3 depends of the turbulence model. With the non-linear model, see Fig. 7(a), this vortex is completely separated from the model. It is not the case for a DES simulation.

For the yaw angle $\beta = 30^\circ$, the position of the vortex V2 and its size is different for both turbulence models, that it is clearly visible on Figures 5 and 8. We note that the vortex V3 is very far from the body with the EASM modelization while in the DES simulation, this is diffused. In Figure 7, the zone located between $Y_0 = -300$ mm and $Y_0 = -200$ mm corresponds to the vortex V3, that arises from the interaction of the wakes of the front feet with the central cylinder (c), as shown by Figures 9. We note that the wake of the feet is more intense in the results obtained with the EASM model than the results obtained with DES simulations.

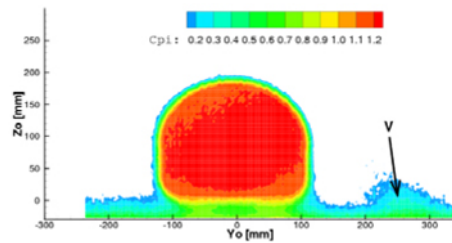
The evolution of the pressure coefficient (C_p) along curve (Pt), see [3] for more details, is drawn in Figure 10 for different yaw angles (β). The results show that the pressure at the stagnation point is leeward and as the yaw angle increases, the stagnation point moves windward. The wall pressure is correctly simulated by both simulations on the windward side for all yaw angles. On the leeward side, the flow separates when the yaw angle is greater that 10° . The negative pressure gradient increases and simulations and experiments align until the change in the sign of the pressure gradient. At approximatively $X = -200$ mm, the rise of adverse pressure is the indication of a flow separation for the results obtained with the EASM turbulence model. This is confirmed in Figure 11, which shows a view of the pressure on the leeward side for the yaw angle $\beta = 30^\circ$. We see that the zone of low pressure extends until the top of the model. We notice also that the wake of the central foot (c) disturbs the flow more in the simulations with the EASM model than the DES simulations.



(a). EASM

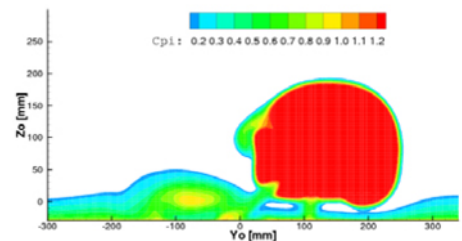


(b). DES-SST

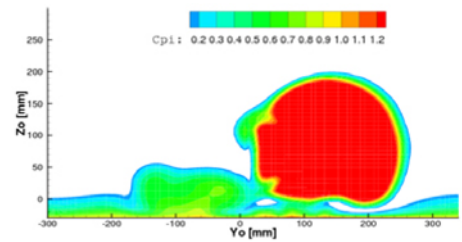


(c). Experiments

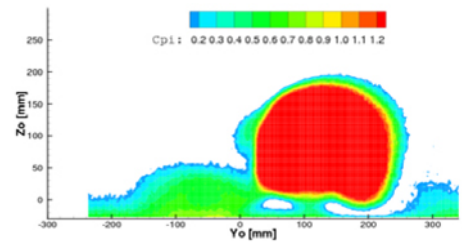
Figure 3. Willy - Total pressure coefficient at $Xo/L = 0.60$ for $\beta = 0^\circ$



(a). EASM



(b). DES-SST



(c). Experiments

Figure 4. Willy - Total pressure coefficient at $Xo/L = 0.60$ for $\beta = 20^\circ$