ABSTRACT: Numerical simulations have been carried out for a new vehicle model, created to analyze the behavior of unsteady wakes on bluff bodies. Reynolds-averaged Navier-Stokes equations are solved on unstructured grid using a near-wall low-Reynolds number turbulence closure, by using ISIS-CFD solver, developed by the Equipe Modélisation Numérique (i.e. CFD Department) of the Fluid Mechanics Laboratory. In the paper, only a large yaw angle ($\beta=30^\circ$) is investigated and the Reynolds number is close to $10^6$. Numerical results are compared with experimental data which show an excellent agreement.

KEYWORDS: Bluff body, car model, yaw angle, numerical simulations, turbulence model.

1 INTRODUCTION

The aerodynamic characteristics of a vehicle and their evolutions as a function of situations such as passing, crossing or the presence of an unsteady gust of wind are factors in the vehicle’s dynamic stability and driving security[1]. Several techniques allow the reproduction of a side gust of wind in a wind tunnel, assimilated to a pulse of velocity. The approach, retained in the experiments, consists of submitting the model to a periodic movement in a steady wind[2]. This approach does not directly simulate the side gust of wind, but it simulates the situation of passing and permits the analysis of the phenomena of phase shifting and hysteresis associated with an unsteady wind[3].

In this study, we only focus to the analysis of the steady behavior of the model. The numerical simulations are reproduced under the same experimental conditions as those described by Chometon et al.[4]. The upstream velocity, $V_0$, is 20 m/s, that is to say a Reynolds number of 900000. In the paper, only the yaw angle $\beta=30^\circ$ is presented.

2 NUMERICAL METHOD

The ISIS-CFD flow solver, developed by the EMN (Equipe Modélisation Numérique) of the Fluid Mechanics Laboratory of the Ecole Centrale de Nantes, uses the incompressible unsteady Reynolds-averaged Navier-Stokes equations (RANSE). 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.
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.

A second-order accurate three-level fully implicit time discretization is used and surface and volume integrals are evaluated using second-order accurate approximation\(^5\).

In this study, the turbulence model used is a quadratic explicit algebraic model (EASM)\(^6\). This model adopts the SSG pressure-strain rate model\(^7\) and solve the BSL $K-\omega$ model\(^8\) to determine the turbulence velocity and length scale. It takes into account the variation of the production-to-dissipation rate ratio. The numerical implementation is detailed by Duvigneau et al.\(^9\).

3 TEST MODEL AND MESH

The test model is designed using the following criteria\(^4\): 1) the geometry is realistic compared to a real vehicle, 2) the model's plane under-body surface is parallel to the ground, 3) the separations are limited to the region of the base for a moderated yaw angle ($\beta=10^\circ$), and 4) the digital definition of the model is analytical. The proportions, those of a classic van, are $L_1/H=3.386$, $L_2/H=1.2$ and $L_1=675$ mm (Figs 1-2). The model is mounted on cylindrical stilts of diameter 20 mm at a ground clearance of 29 mm. The model comprises under its floor a tube with a diameter 40 mm to allow the transfer of measurements of pressure towards outside. This tube is not present in experiments\(^4\) for the force and tomography measurements. Since, these measurements were remade by taking into account the central tube. At $\beta=0^\circ$, the trailing edge of the model is located at $x=345$ mm, $y=0$ is the symmetric plane and $z=0$ is the floor of the model.

![Figure 1. Side view of the model](image1)

![Figure 2. Front view](image2)

The computational domain starts $2.5L_1$ in front of the model and extends $4L_1$ behind the model. The width of the domain extends from $+1000$ mm to $-1000$ mm and the height is $1400$ mm. The mesh is generated by using HEXPRESS\(^\text{TM}\), an automatic unstructured mesh generator. This software generates meshes only containing Hexaedral. The mesh is composed to 19.5 millions of points with approximatively 500000 points on the model. This mesh is splitted into 110 blocks. The simulations are obtained for a Reynolds number of $Re=V_0L_1/\nu=900000$. Due to the near-wall low-Reynolds number turbulence model, the distance between the walls and the first fluid points is fixed to 0.006 mm. Then, the distance $y^+$ is near to 0.5 with a maximum of 0.7.

4 NUMERICAL RESULTS

4.1 Aerodynamic coefficient
Aerodynamic coefficients are presented in Table 1 for the yawing angle $\beta=30^\circ$. Forces and moment are non-dimensionalized by the maximum cross section of the model ($S_{\text{ref}}=41791.05 \text{ mm}^2$), its length ($L_1$) and the free-stream dynamic pressure ($q_0=\rho V_0^2/2$). The results are represented in the Lilienthal axes linked to the model. The drag coefficient ($C_x$), side force coefficient ($C_y$), and the yawing moment ($C_N$) are given in Table 1. The numerical results are compared to experimental data. The agreement is very well, particularly for $C_x$ and $C_N$.

| Table 1. Aerodynamic coefficients for $\beta=30^\circ$ |
|------------------------|--------|--------|
| **Numerics** | 0.314  | -1.49  | 0.193  |
| **Experiments** | 0.315  | -1.25  | 0.193  |

4.2 Wall results

The evolution of the pressure coefficient, $C_p=(p-p_0)/q_0$ where $p$ is the local pressure and $p_0$ is the static upstream pressure, on the curve A (Fig. 1), is presented in Figure 3. This figure shows that the windward side is well predicted and a weak positive pressure gradient at the leeward side exists as in experiments but pressure is underestimated according to experimental data.

Figure 4 presents the streamlines on the leeward side of the model. We notice that a vortex structure develops along the body and moves upwards.

4.3 Tomography of total pressure

The total pressure coefficient is $C_{pi}=(p_{i0}-p_i)/q_0$ where $p_i$ defines the local total pressure and $p_{i0}$ the total pressure associated with the upstream velocity $V_0$ and the free-stream dynamic pressure $q_0$. The tomographies of $C_{pi}$ show in Figure 4 and 5, respectively numerics and experiments, are obtained at $x=371.25 \text{ mm}$. The agreement between numerical and experiments is very well. Simulation reproduces the two vortices that are present in experiments. We can see the separation at the leeward side. The vortex located towards $y=-150 \text{ mm}$ results from the left front stilt and the central tube.
5 CONCLUSIONS

The flow simulation around a simple car model with a large yaw angle ($\beta=30^\circ$) is performed using Reynolds-averaged Navier-Stokes equations. Comparisons with experiments show excellent agreement with experimental data. Before to study an unsteady flow by submitting the model to a periodic movement in a steady wind, other yaw angles will be investigated.

6 ACKNOWLEDGEMENTS

The author gratefully acknowledges the Scientific Committee of IDRIS (project 0129) and of CINES (project dmn2049) for the attribution of CPU time. The author would also like to thank F. Chometon (Conservatoire National des Arts et Métiers) for have making available his experimental results.

7 REFERENCES