SPECTRAL ACCURATE LES OF TURBULENT FLOW OVER BLUFF BODIES : APPLICATION TO THE ’AHMED’ CAR MODEL

Minguez Matthieu*, Pasquetti Richard† and Serre Eric*

*Lab. M2P2, UMR CNRS 6181, Univ. Aix -Marseille
IMT La Jeté - Technopôle de Château-Gombert
38, rue Frédéric Joliot Curie, 13451 Marseille Cedex 20, France
e-mail: minguez@L3M.univ-mrs.fr, Eric.Serre@L3M.univ-mrs.fr

†Lab. J.-A. Dieudonné, UMR CNRS 6621, Université de Nice-Sophia Antipolis
Parc Valrose, 06108 Nice Cedex 02, France
e-mails: Richard.PASQUETTI@unice.fr

Keywords: Ahmed Body Aerodynamics & Wake, Turbulence, High-order LES

1 INTRODUCTION

The constant increase of the computational power will allow in a close future to frequently perform Large-Eddy Simulation (LES) of complex flows with an industrial rendering (time and cost). Consequently, numbers of LES methodologies are explored and applied for high Reynolds number flows in complex geometries. Among these configurations, flows over bluff bodies are related to many aerodynamic applications and represent ones of the most current test cases for the numerical validation. As it is typical for bluff body wakes, drag prediction and its control are among the main objectives of aerodynamics studies.

In the present work, we propose a high-order LES methodology based on spectral approximations to perform turbulent flow over complex geometries. The LES-methodology is based on the Spectral Vanishing Viscosity (SVV) stabilization method, as developed in [7]. Basically, the energy cascade from the low frequencies to the higher ones is taken into account as an increased diffusion in the higher frequency range. However, an appropriate treatment of the near wall region with the SVV (SVV-NW) appears here quite necessary. Two ERCFOTAC test cases have been investigated: the classical square cylinder and the more complex Ahmed body. The flow over the square cylinder at \( Re=21400 \), [4], has been first addressed and has shown in particular the efficiency of the SVV-NW correction to describe the turbulent boundary layers around bluff bodies. Then, the more tricky Ahmed body test case [1] which is now considered as a reference car model for the study of flows around road vehicles has been studied for a slant angle of 25° (value for which the flow partially separates on the slant) and a Reynolds number of 768000.

The flow that develops in this case is very complex, i.e., fully three-dimensional, unsteady, with separations and very thin turbulent boundary layers. Consequently, only few the numerical works agree with recent experimental results [3]. Moreover, as it clearly appeared during the
ERCOFTAC workshop [5], significant variations can be observed between numerical simulations based on statistical methods (RANS) or LES approaches. Especially, it seems difficult to predict accurately the partial detachment on the slant, the dynamic of the flow and the mean velocity or turbulent kinetic energy profiles above the slant and in the wake.

In this work, we show that a LES based on a high-order numerical method (spectral method) can provide good results in these complex configurations in agreement with experimental results of reference [4, 3].

2 NUMERICAL MODELING

The flow is governed by the incompressible Navier-Stokes equations written in the velocity-pressure formulation. The scheme is globally second order accurate in time and is based on a fractional step method (3 steps). The approximation in space makes use of a Fourier-Galerkin method in the homogeneous spanwise direction (z-direction) and a Chebychev-collocation method in the non-homogeneous streamwise and crossflow directions (x and y-directions). Moreover, a domain decomposition is implemented in the elongated streamwise direction. An influence matrix technique is used to enforce the transmission conditions between the different sub-domains.

Each of 8 subdomains is discretized with \( N_x = 40, N_y = 190 \) and \( N_z = 340 \) grid-points, so that the total number of mesh-points is about 22 millions (see figure 1(a)). The Ahmed body is modeled using a “pseudo-penalization” technique [8]. Finally, the code is parallelized with MPI, each sub-domain being associated to a vectorial processor of a NEC-SX8 computer. Our LES methodology is performed through a Spectral Vanishing Viscosity (SVV) stabilization method, allowing a damping of the highest frequencies of the solution [7]. The Navier-Stokes equations are completed with a supplementary diffusion term and may then be expressed with a SVV-modified diffusion operator \( \Delta_{SVV} \), obtained from \( \nu \Delta_{SVV} = \nu \Delta + \hat{\Delta} \),

\[
\hat{\Delta} = \nabla (\varepsilon_N Q_N \nabla)
\]

and where \( N \) is the degree of the polynomial approximation (Chebychev or trigonometric polynomials). The parameter \( \varepsilon_N \) controls the amplitude of the dissipation (usually \( \varepsilon_N = O(1/N) \)) and \( Q_N \) represents the SVV-operator kernel. In the Chebyshev or Fourier space, this operator kernel is such that \( Q_k = 0 \) if \( k < m_N \) and \( 1 \geq Q_k \geq 0 \) if \( k \geq m_N \) (see [7] for details). Thus \( m_N \) is a control parameter which allows to select the range of frequencies on which the SVV acts (usually \( m_N = O(\sqrt{N}) \)).

The flow around the Ahmed body is strongly governed by the boundary layers that develop at the walls, where are located the production and the transfer of turbulence at the smallest scales of the flow. These boundary layers being too thin to be resolved, a near wall treatment has been developed, in the frame of the SVV which initially is a global approach. Here, the near wall treatment (SVV-NW) permits to relax locally the SVV dissipation within the boundary layer through the activation parameter \( m_N \). To this end, we complete the SVV stabilized Navier-Stokes equations with an additional force term defined as \( f_{BL} = \chi_{BL}(\hat{\Delta}_{BL} - \hat{\Delta})u \) where \( u \) is the velocity and with \( \chi_{BL} \), a new characteristic function equal to 1 in the NW region and to 0 elsewhere.

3 RESULTS

The SVV-LES performed for the flow over the square cylinder have pointed out a greater ability of the SVV-NW correction for under-resolved boundary layer configurations. The too high dissipation conjointly due to the coarse grid and to the SVV appeared to decrease with the SVV-NW approach. The increase of the local turbulent Reynolds, \( Re_{\tau} \), in the vicinity of the wall is consequently correlated to a better prediction of the turbulent phenomena as shown by
the turbulent statistic quantities. The SVV-LES with the previous NW correction have been then applied to the ”Ahmed” car

Figure 1: (a) Visualization of the mesh in the median vertical plane. (b,c,d) Visualization of the vortical structures in the wake of the Ahmed body, \(Re = 768000\) : (b) Iso-surfaces \(< p >= 0.25\) and \(< p >= -0.07\) of the mean pressure colored by the dimensionless turbulent kinetic energy \(k\) and showing the velocity field on the slant and in the wake; (c) Mean three-dimensional turbulent kinetic energy colored by the mean streamwise velocity \(< u >\); (d) Iso-surface of the pressure fluctuations colored by the mean streamwise velocity \(< u >\) on the slant and in the wake. (e,f) Mean velocities and turbulent quantities in the symmetry plane \(z = 0\) over the slant and in the near wake of the ahmed body at \(Re = 768000\) : (e) profile of the mean streamwise velocity, (f) profile of the turbulent kinetic energy. Comparisons of the numerical results, with and without near wall treatment, to the experimental results of [3].

model furnishing hopefully results. The topology of the flow, both on the slant and in the wake, is in close agreement with the one described in the earlier experiments [1]. Indeed, the partial detachment of the boundary layer on the slant back, controlled by two strong contra-rotative trailing vortices, is very well captured (see figure1 (b,c)). The main geometrical characteristics, such as the position of the centers of the vortical structures, are also very close to the ones described experimentally in [3]. A finer analysis of the slant part of the flow (see figure1(c)) allows to point out a more complex topology of the partial recirculation on the hatch back such
as the two foci constituting it and previously observed in [8].
Moreover, the near wall treatment developed in this present work permits to capture a more realistic turbulent level within the boundary layer around the bluff body. This confinement of the turbulent kinetic energy favors the development of the slant mixing layer which is a source of instabilities and unsteadiness [2, 6]. Vortices, parallel to the separation line, form on the slant and are convected downstream as shown figure1(d). These vortical structures are slightly lifted within the shear layer at about half-way down the slant back, taking a more tilted shape reminiscent to the hairpin vortices form usually observed in boundary layers or wake problems. These structures can interact with the trailing vortices coming from the slanted side edges of the slant to form helicoidal patterns spreading farther in the wake. Moreover, statistical quantities, as the mean velocity (figure1(e)) and turbulent kinetic energy (figure1(f)) profiles on the slant as in the wake are in good agreement with the experimental results of [3]. This LES study provides an original and successful application of a high-order method in the numerical modeling of turbulent flows for typical industrial complex geometries.

REFERENCES