Simone Camarri, Annabella N. Grozescu, Maria Vittoria Salvetti

Dipartimento di Ingegneria Aerospaziale, Università di Pisa

Numerical simulation of bluff-body flows

The flow around bluff-bodies and the characterization of the aerodynamic loads acting on their surface are of interest for many engineering applications. Evident examples are the flow past a submarine or a motor vehicle, wind blowing over a bridge or over a building, long pipes or cylindrical structures in water, etc. Moreover, bluff-body flows represent a challenging problem for numerical simulation. The flow around bluff-bodies is characterized by a complex three-dimensional and intrinsically unsteady dynamics. At low Reynolds numbers (of the order of a few thousands, based on the body cross dimension and on the freestream velocity), direct numerical simulation of the full Navier-Stokes equations is possible. As the Reynolds number increases and the wake becomes turbulent, a critical issue is the choice of turbulence modeling. It is known that RANS methods encounter difficulties in accurately predicting bluff body flows, while large-eddy simulation (LES) is certainly more promising from the viewpoint of accuracy. However, large-eddy simulations imply much higher computational costs than RANS. Bluff-body flows are thus typical target flows for LES methods and are often used as benchmarks to assess their capabilities. However, the assessment of the quality and reliability of the results of LES is still an open issue and is the object of a wide research activity.

The first objective of the project is a systematic assessment of the accuracy of the numerical results obtained through large-eddy (LES) and variational multiscale (VMS) LES simulations for different bluff-body flow configurations, with the aim of giving a contribution to the more general issue of the assessment of the quality and reliability of LES and VMS-LES. Moreover, the analysis of the results of numerical simulations carried out in this project is useful to enhance the physical understanding of the flows under investigation, because numerical simulation can give complementary information to that obtained in experiments.

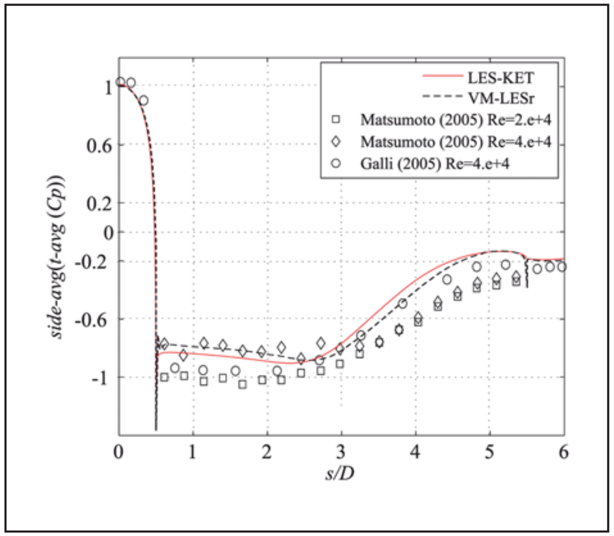

Fig. 1. – Comparison between the mean pressure coefficient at the cylinder: symbols are experimental data, the dashed black line corresponds to one of our VMS-LES simulations, the red solid line corresponds to a LES simulation carried out by another group by means of OPENFOAM.

To this aim, we considered first the flow around a rectangular cylinder characterized by a breadth to depth ratio equal to 5. This configuration is the object of an international benchmark launched with the support of ANIV (Italian National Association for Wind Engineering), IAWE (International Association for Wind Engineering) and ERCOFTAC (European Research Community On Flow, Turbulence And Combustion) (http://www.aniv-iawe.org/barc). Different simulations were carried out by using the VMS-LES approach and a proprietary research code (AERO). This is a numerical solver of compressible flows; the space discretization is based on a mixed finite-element/finite-volume formulation, applicable to tetrahedrical unstructured grids. This scheme is a variational one relying on a finite-volume formulation for the convective terms, associated with the finite-volume cell centered on each grid vertex. A finite-element formulation is used for the diffusive term, with basis and test functions continuous, linear by element, equal to 1 at one vertex and vanishing at other vertexes. The Roe scheme represents the basic upwind component for the numerical evaluation of the convective fluxes. A preconditioning term is introduced to avoid accuracy problems at low Mach numbers. To obtain second-order accuracy in space, the Monotone Upwind Scheme for Conservation Laws reconstruction method (MUSCL) is used, in which the Roe flux is expressed as a function of reconstructed values of the flow variables at each side of the interface between two cells. The introduced numerical dissipation is made of sixth-order space derivatives and, thus, concentrates on a narrow-band of the highest resolved frequencies. Finally, an implicit linearized time-marching algorithm is used, based on a backward-difference scheme for the discretization of the time derivative. The numerical method is second-order accurate in space and time. More details can be found, for instance, in references 1 and 2. The VMS-LES approach was used for turbulence modeling. The main idea of VMS-LES is to decompose, through Galerkin projection, the scales resolved in LES into the largest and smallest ones and to add the closure model (SGS model) only to the smallest ones. This is aimed at reducing the excessive dissipation introduced by eddy-viscosity SGS models also on the large scales. In several applications in the literature, it has been observed that predictions were as accurate with the VMS-LES approach as with dynamic SGS models, which are much more computationally demanding especially on unstructured grids.

The impact on the flow and on the aerodynamic loads of different parameters, viz. the Reynolds number, the closure model and the grid refinement, were investigated. Moreover, since, as previously mentioned this is an international benchmark, the numerical results obtained within the present project could be compared with those of other groups all over the world and with experimental data. The main results can be summarized as follows:

- The different computational models used by different groups and based on LES or VMS-LES provide results in excellent agreement, proving the reliability and quality of LES results for a flow configuration of high complexity (see figure 1).
- The scatter between the different computational simulations is comparable to, and even lower than, the measurements obtained in different experimental facilities in (almost) homogeneous nominal set-up conditions (see figure 1).
- Our VMS-LES formulation proves its efficiency versus classical LES approach, because analogous accuracy is obtained with a coarse grid having a number of cells seven times lower than the ones adopted in classical LES approach by other groups.
- Finally, numerical simulations suggest the possibility of a flow dynamics leading to an asymmetric mean flow and that this dynamics is characterized by extremely small scales in the spanwise direction. This point will be the object of future investigations.

The results of the simulations carried out within the present project for the flow around a rectangular cylinder are documented in publications 3 and 4. Part of the results will also be included in a paper in preparation to be submitted for publication in an international journal.

Another activity carried out in the present project is an investigation of the capabilities of the VMS-LES approach to capture Reynolds num-

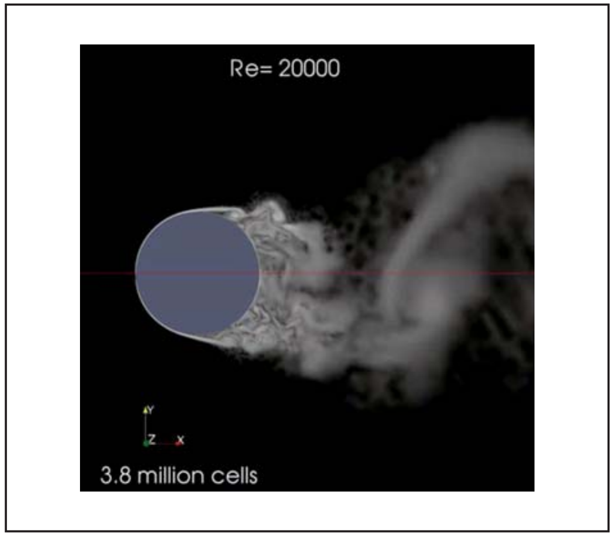

Fig. 2. – A snapshot of the vorticity modulus at Re = 20,000.

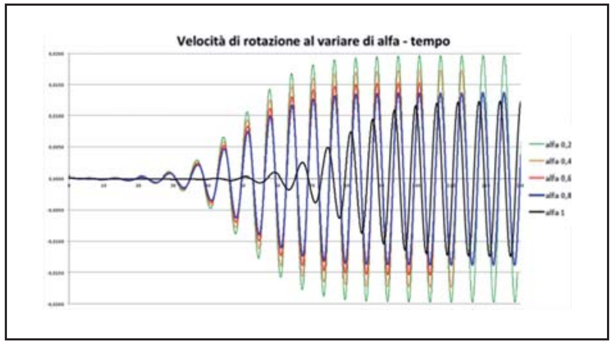

Fig. 3. – The angular velocity of the cylinder vs. time for different values of the density ratio, at a Reynolds number at which the wake is unstable.

ber effects for the flow around a circular cylinder of infinite length in the subcritical regime. The same code and the same approach as in the previous test case was used. Simulations at different Reynolds numbers, viz. Re = 3900, 10,000 and 20,000, were carried out without any other change in the simulation parameters. The trends observed in the literature for Reynolds numbers in the considered range are correctly reproduced in our simulations. As an example, figure 2 shows a snapshot of the vorticity modulus at Re = 20,000. Besides the main vortex shedding, the small-scale Kelvin-Helmotz vortices, which characteristic feature in the considered Reynolds number range, are clearly detectable. The quantitative agreement of our results with available experimental and numerical data is also generally good. The results of the simulations carried out in the present project are included in reference 5.

Finally, we have concentrated our attention on the case of a cylinder which is free to rotate in an incoming flow. This configuration is particularly attractive for the cases in which a small cylinder is used as a device for a passive flow control. In the considered kind of flow the free parameters are the flow Reynolds number and the ratio alpha between the densities of the solid body (its density is assumed constant) and of the surrounding fluid, respectively. Before carrying out a linearized stability analysis of the flow, in order to characterize the vortex shedding instability vs alpha, a preliminary set of DNS simulations have been run in order to quantify the influence of the free rotation on the onset of the primary wake instability. For this flow configuration, the incompressible flow equations are discretized by a second-order centered finite-difference scheme and advanced in time by a third-order Runge-Kutta scheme. The velocity and pressure are solved together, avoiding approximate decoupling techniques. Solid boundaries are reproduced by an immersed-boundary method. At this stage we have found out that the free rotation has a negligible influence on the wake instability, at least for the explored density ratios (alpha > 0.2). As an example, we show in figure 3 the angular velocity of the cylinder vs. time for different values of alpha, at a Reynolds number at which the wake is unstable.

Acknowledgement

We acknowledge the CINECA award under the ISCRA initiative, for the availability of high performance computing resources and support.

References

(1) S. Camarri, M.V. Salvetti, B. Koobus and A. Dervieux, Large-eddy simulation of a bluff-body flow on unstructured grids. *Int. J. Num. Methods Fluids* **40** (11): 1431-1460, (2002).
(2) S. Camarri, M.V. Salvetti and B. Koobus and A. Dervieux, A low-diffusion MUSCL scheme for LES on unstructured grids. *Computers & Fluids* **33** (9): 1101-1129, (2004).
(3) Grozescu A.N., Salvetti M.V., Camarri S. and Buresti G., Variational multiscale large-eddy simulations of the BARC flow configuration, *13th International Conference on Wind Engineering, Amsterdam* (The Netherlands), 2011
(4) Grozescu A.N., Bruno L., Fransos D. and Salvetti M.V., Large-eddy simulations of a benchmark on the aerodynamics of a rectangular 5:1 cylinder, *Atti del XX Congresso dell'Associazione Italiana di Meccanica Teorica e Applicata*, Bologna (Italy), 2011.
(5) Wornom S., Ouvrard H., Salvetti M. V., Koobus B. and Dervieux A., Variational multiscale large-eddy simulations of the flow past a circular cylinder: Reynolds number effects, *Computers & Fluids*, **47**, pp 44-50, 2011.