8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

COMPUTATIONS OF TURBULENT FLOWS AROUND OSCILLATING BODIES USING MULTI-BLOCK DEFORMING MESHES AND EDDY RESOLVING TECHNIQUES

***Dmitri K. Zaitsev[1], Evgueni M. Smirnov[1] and Nikolai A. Schur[1]**

[1] Dept. Aerodynamics, State Polytechnic University
29 Politechnicheskaya str, 195251 St.Petersburg, RUSSIA
aero@phmf.spbstu.ru

Key Words: *Deforming Meshes, Fluid Structure Interaction, Turbulent Flows, Eddy Resolving Techniques.*

ABSTRACT

In the present work, two fluid structure interaction problems are under consideration, namely, the vortex-induced vibration of an elastically mounted circular cylinder and the flow generated by oscillations of a flexible console blade. For both problems, the flow conditions adopted predefine development of turbulent flow regimes that are analyzed numerically on the base of eddy-resolving techniques, with the usage of parallel computing.

The *first part* of the contribution covers numerical aspects of the study. The computations have been performed on a cluster system using a well-validated in-house code [1, 2] that is a full Navier-Stokes finite-volume solver utilizing body-fitted block-structured grids and the second-order accuracy spatial and temporal discretization. Parallelization of the code is based on the MPI standard and the domain decomposition according to the grid blocks. A number of test computations (using grids up to 20 million cells and clusters up to 24 processors) showed the parallelization efficiency of above 0.8.

Flows around vibrating bodies are computed using the deforming mesh technique based on the arbitrary Lagrangian-Eulerian formulation. Special care is taken to ensure the maintenance of the conservation laws on deforming mesh. An efficient non-iterative algorithm is used to deform the block-structured grid in accordance with current position of the body and preserve the grid quality near the walls.

In both problems considered, the flow is dominated by large-scale eddies shed from the vibrating body. These eddies interact with each other, break apart, and provide intensive mixing that is not reproduced by widely used RANS turbulence models (like k-ε, k-ω, etc.). It gives a strong motivation to application of eddy-resolving turbulence techniques (DNS, LES, or RANS/LES hybridization). In the present study, a RANS/LES approach utilizing the ideas of the DES method [3] and based on the Menter SST turbulence model is adopted. According to this approach, the SST model is used for unsteady RANS computations close to the wall whereas farther from the wall the model shifts smoothly (depending on the cell size and the

local turbulence characteristics) to its sub-grid scale mode.

The *second part* of the contribution presents the results of parametric computations. It is noted that the vortex-induced vibration of circular cylinder was extensively studied during several decades; a comprehensive review of experimental results as well as the phenomenon discussion can be found in [4]. For the present work, the problem setup corresponds to the conditions of a particular experimental case reported in [4], namely the cylinder relative mass is m^*=3.3; the damping factor is ζ=0.004; the normalized velocity, U^*, changes from 2 to 11 so that the vortex-shedding frequency is around the cylinder natural frequency (accordingly, the Reynolds number varies from 1200 to 7000 that corresponds to regimes with laminar flow separation and turbulent wake). Despite the problem geometry is two-dimensional, a 3D formulation is used as the flow in the wake is known to be essentially three-dimensional (the domain extent along the axis is 2 to 6 times the cylinder diameter). The computation results exhibit an excellent agreement with experimental data (except a tight region of so called "upper" branch of the amplitude response). Unlike that, 2D RANS simulation underestimates considerably both the peak amplitude and the width of the "lock-in" region.

The flow induced by oscillations of a thin flexible blade (e.g. with piezoelectric excitation) can be used for cooling various electronic devices. Such piezoelectric fans are very low power and low noise devices that have recently emerged as an alternative to traditional fans. Presented in this paper, the results of 3D simulation of the flow produced by a piezoelectric fan seem to be the first ones. The computations have been performed for typical operating conditions of the device, with the blade width of a quarter of its length. It has been established that the vortices shedding from the side edges of the blade are responsible for the fan effect rather than the vortices shedding from the blade tip. Among other flow features discovered, a relatively rapid divergence of the jet (the half-angle is about 45°) and the presence of a backward stream near the blade tip should be mentioned.

REFERENCES

[1] E. Smirnov and D. Zaitsev, "Computations of internal flows using an artificial-compressibility solver enhanced with an elliptic pressure-correction procedure". In: *ECCOMAS-2004* (CD-ROM proceedings), 13p.

[2] E.M. Smirnov, A.G. Abramov, *et al*. "DNS and RANS/LES-computations of complex geometry flows using a parallel multiblock finite-volume code". In: *Parallel CFD. Advanced Numerical Methods Software and Application (Proc. ParCFD-03)*. Elsevier, 2004, pp.219-226

[3] M. Strelets, "Detached eddy simulation of massively separated flows", *AIAA Paper* 2001-0879, 18p., (2001)

[4] A. Khalak, C.H.K. Williamson, "Motions, forces and mode transitions in vortex-induced vibrations at low mass-damping", *J. Fluids and Structures*, Vol. 13 (1999), pp.813-851