The 4^{th} Congress of the Turkic World Mathematical Society (TWMS) Baku, Azerbaijan, 1-3 July, 2011

Numerical simulation of flows around two circular cylinders in the side-by-side and tandem by compressible gas

S.D. Mausumbekova
Al-Farabi Kazakh National University, Almaty, Kazakhstan,
e-mail:s_mausumbekova@mail.ru

In many areas of engineering, circular cylinders form the basic part of structures, for example, cooling systems for nuclear power stations, chimney stacks, transmission cables, heat exchange tubes, etc. Named engineering structures are exposed to either air or water flow, and therefore they experience flow- induced vibration, which could lead to destruction. To avoid the situations and to improve the structure design, it is necessary for engineers to understand the details of the flow- structure interaction and possess the ability to predict the force and response of the cylinder-like structures. Flow past one cylinder has been well studied and it is now considered as a classical case for a validating new numerical schemes. It is known that the flow field around one cylinder shows a wide class of models. From the point of view of a geometrical configuration flow around two cylinders can be considered as the expanded case of one isolated cylinder. Despite it, the corresponding problem is much more complicated. That is because the dynamic interaction between the shed vortices, shear layers and vortex sheets appears in the wake of the cylinders. Consequently, the wake behaves quite differently from one isolated cylinder. Interest to characteristics of flow past a pair cylinders has attracted many researches. The early experimental studies of flow around to circular cylinders were reviewed by Zdravkovich [1]. With fast development of computer technology, the flow around two circular cylinders is also numerically investigated [2, 3]. At numerical modeling of a flow of obstacles there are the difficulties connected with satisfies of boundary conditions on an obstacle. Approaches to the permission of these problems are known, the most effective among them is the immersed-boundary method [4], a method of the fictitious areas, different simple realization. In this paper is provided numerical solutions for the flows past two cylinders in the side-by-side and tandem arrangements by compressible turbulent gas in the field of the gravity, described by the non-stationary Navier-Stokes. For an exception of the difficulties arising at numerical integration of initial system of the equations for small Mach numbers, the model of hyposound flows is used .

The exact satisfaction of boundary conditions on an obstacle influences definition of the forces operating from a liquid on a body. Linear and bilinear interpolations are applied to increase of an order of approximation of dynamic characteristics on obstacles. It is already known that the wake interference behind two cylinders is influenced by the gap between the cylinders and there exists a so-called critical gap, which is used to classify the flow patterns behind the cylinder into several categories. To illustrate the detailed the structure of a stream near the wake region behind two circular cylinders, numerical visualization are presented in the form of vorticity contours and streamlines. To describe the flow pattern quantitatively, mean values of lift and drag coefficients are computed and compared with the results of other authors.

References

[1] Zdravkovich M.M.: Review of flow interference between two circular cylinders in various arrangements. // Journal of Fluid Engineering, 1977, Vol. 99. pp. 618–633.

[2] Mittal S., Kumar V.: Finite element study of vortex-induced cross-flow and in-line oscillations of a circular cylinder at low Reynolds numbers. // Int. J. Numer. Meth. Fluids. Vol. 31, 1999. pp. 1087–1120.

[3] Ding H., Shu C., Yeo K.S., Xu D.: Numerical simulation of flows around two circular cylinders by mesh- free least square based finite difference methods. // Int. J. Numer. Meth. Fluids. Vol. 5, 2007. pp. 305–332.

[4] Kim J., Kim D., Choi H.: An immersed-boundary finite-volume method for simulations of flow in complex geometries. // Journal of Computational Physics. 2001, Vol. 171. pp. 132–150.