Fluid Flow Analysis for cross-flow around four cylinders arranged in a square configuration

Anu Dutta*, Priyanshu Goyal, R.K. Singh, K.K. Vaze
Reactor Safety Division
Bhabha Atomic Research Centre
Mumbai-400085.
adutta@barc.gov.in

Abstract: Cross-flow around a group of cylinders is a very common phenomenon in engineering, such as flow around heat exchanger tube arrays. The cross-flow-induced vibration might cause a reduction of equipment life and might even lead to the occurrence of severe accidents. Hence, it is necessary to understand the mechanism of flow-induced vibration and the associated fluid–structure interaction in order to improve the design of such equipment. The flow of fluid behind a blunt body is difficult to compute due to the unsteady flows. The wake behind such a body consists of unordered eddies of all sizes that create large drag on the body. Over the past 30 years, a great deal of attention has been focused on research on flow around cylindrical structures, especially on flow around one or two cylinders. Nevertheless, investigations of the flow past more than two cylinders are still relatively scarce because of the numerous parameters such as geometric parameters related to cylinder arrangement, Reynolds number (Re), and boundary conditions that could affect the flow patterns. In the present paper, the flow around four cylinders in an in-line square configuration is simulated using a finite-element method based multiphysics code COMSOL.

Keyword: Cross-flow, Reynolds number, Flow Induced Vibration, Strouhal number, vortex shedding

1. Introduction

The wake interference around the cylinder arrays has been widely investigated in the past because of its inherent importance and practical significance in engineering applications. The more complex wake flow around the four-cylinder arrays configuration has not been studied as extensively as the single and two cylinders configurations because of the numerous parameters such as geometric parameters related to cylinder arrangement, Reynolds number (Re), and boundary conditions that could affect the flow patterns. In the present paper, the case of four cylinders in an in-line square configuration is investigated because a configuration with four equally spaced cylinders is a fundamental element in any tube array. Flow-induced vibration of cylinders is closely related to fluctuating flow pattern given rise by vortex shedding from the cylinders. Therefore, a numerical investigation of the relation between changes in flow pattern of the cylinders in an in-line square configuration could enhance the understanding on the relation between vortex shedding behaviour around the cylinders. From this analysis, Strouhal number has been estimated which is an important parameter for evaluation of flow induced vibration.

2. Governing equations and numerical simulation

The governing equations are the incompressible unsteady Navier–Stokes equations and the continuity equation, which can be written in dimensionless vector form as

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{\mathrm{Re}} \nabla^2 \mathbf{u},$$

$$\nabla \cdot \mathbf{u} = 0$$

where $Re = U_\infty D/\nu$ is the Reynolds number, U_∞ is the oncoming mainstream velocity, ν is the fluid kinematic viscosity, u is the nondimensional velocity vector in the Cartesian coordinate system (x, y) with its two velocity components u and v and p is the non dimensional static pressure. The calculation domain and the cylinder configuration are shown in Fig. 1, where (x, y) denote the coordinates along the stream direction, the transverse direction respectively. As per the guidelines suggested by various authors (Ferrant et. al. (2000), Tezduyar and Shih (1991) and Behr et al. (1995)), a computational domain of 24Dx24D is chosen for the present simulation (Fig.1) with an upstream distance of 8D, a downstream distance of 16D, and a distance of 12D on both sides of the cylinders. The numerical simulation is carried out using finite-element method based multiphysics code COMSOL.

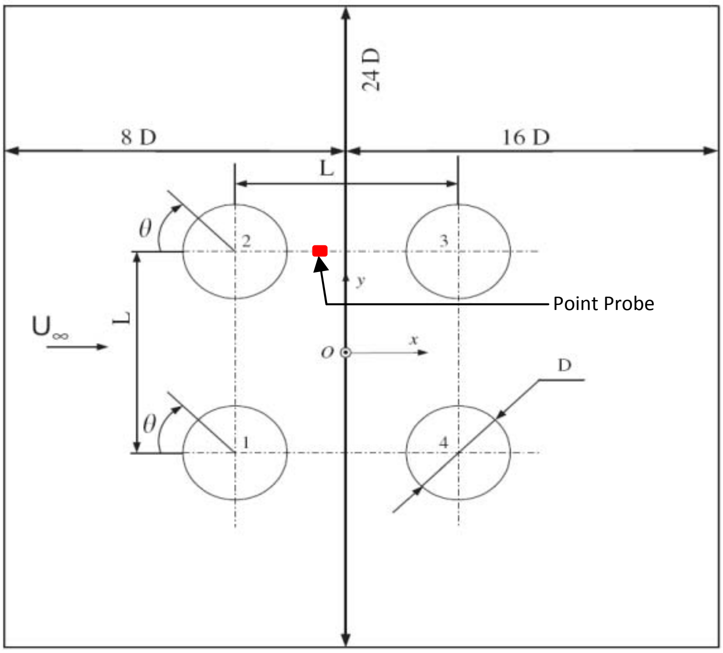

Fig. 1. Arrangement and computational domain for a four-cylinder array

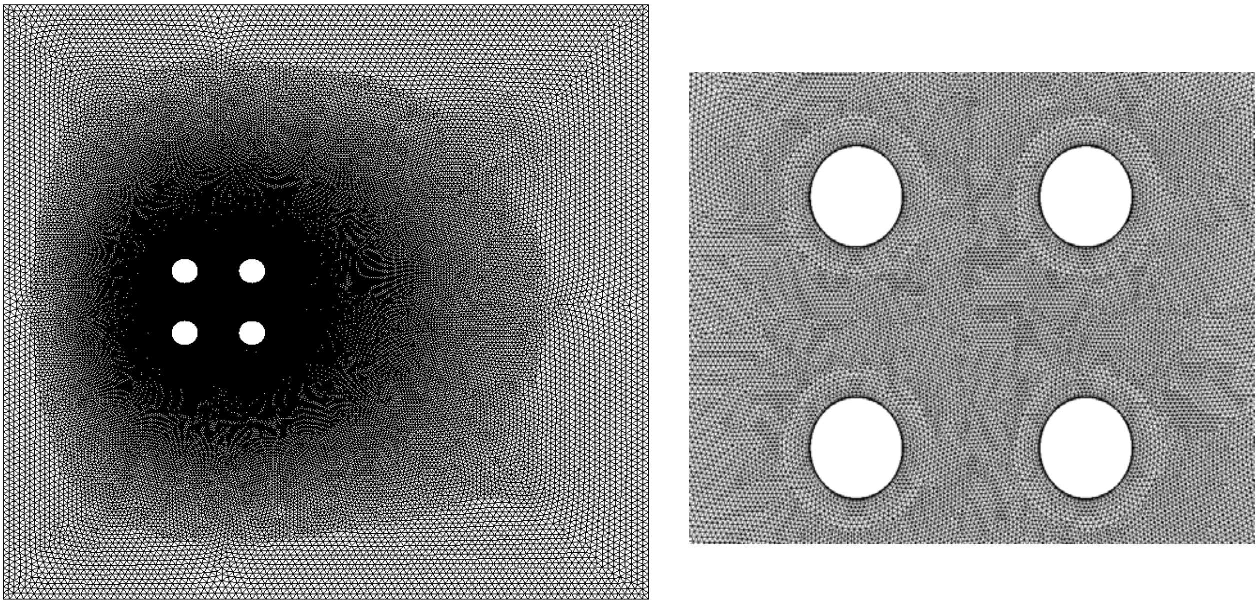

Fig.2 (a) Mesh used for computation (b) Zoom View of computational mesh

To capture the vortex shedding phenomenon, mesh plays a crucial role. For this purpose, an unstructured mesh is generated which is very fine near the cylinders and gets coarse as we move away from the cylinders (Fig.2). A uniform and constant mainstream velocity is specified at the inlet of the computational domain. Constant pressure at the other two sides and at the outlet is specified. No-slip boundary condition is invoked on all cylinder surfaces. The initial condition is assumed to be a uniform incoming flow.

3. Results and Discussions

The instantaneous streamlines in Figures 3(i)-(viii) show the detailed views of the laminar vortex shedding near the four cylinders arranged in square pitch at Re=200 for L/D = 2.5, for eight successive moments of time which span over the whole period. It can be observed that Fig. 3(i) is repeated after Fig. 3(viii) for the next cycle of vortex shedding. It can be observed that two inner side free shear layers from the upstream cylinders reattach onto the two downstream cylinder surfaces. However, the outside free shear layers do not reattach on the downstream cylinder surfaces. Instead, they engulf the downstream cylinders completely. The flow pattern is almost steady during the whole process. Fig. 4 shows the time history of vertical velocity component at the selected point probe shown in Fig. 1. From Fourier Series analysis of the time history, the Strouhal number (which signifies vortex shedding frequency) of 0.17 is obtained which matches very well with the literature quoted value (K. Lam et. al., 2008)

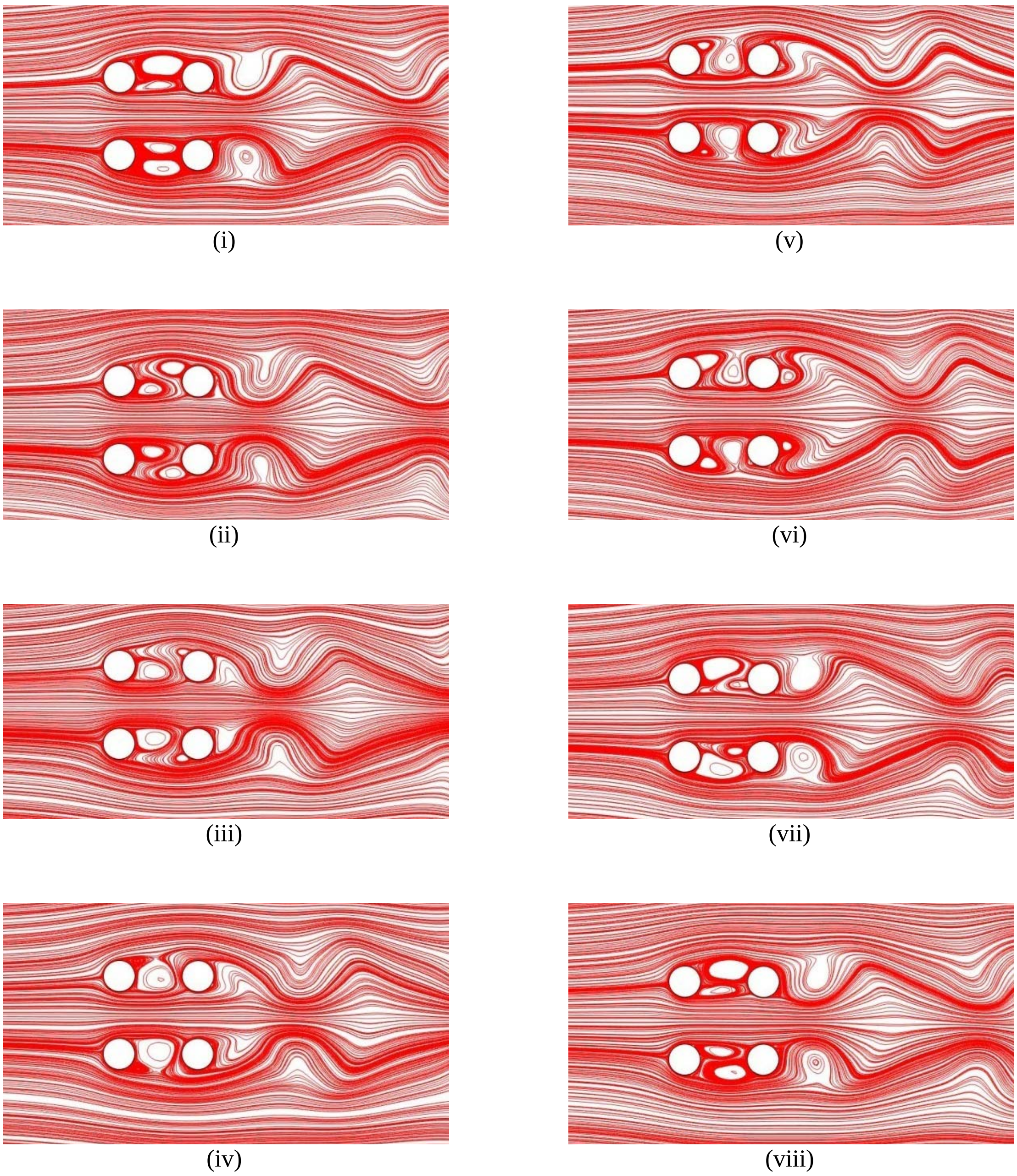

Fig. 3. Flow patterns (i)–(viii), shown by distribution plots around four cylinders for 2-D simulation at Re = 200 for L/D = 2.5

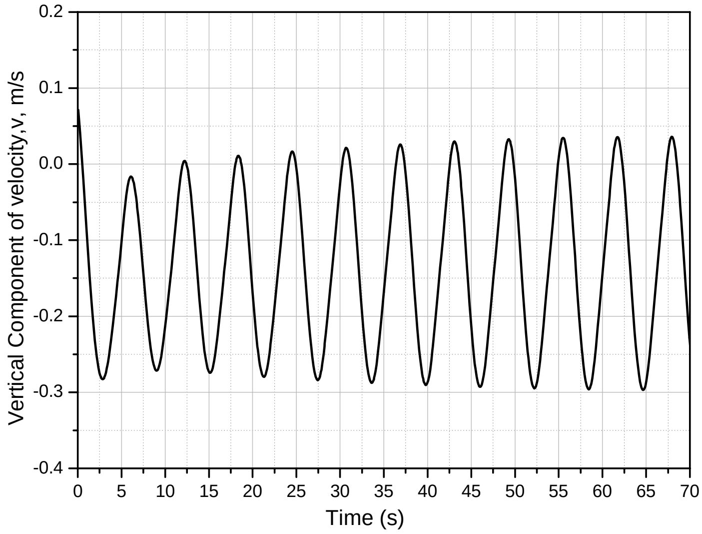

Fig. 4. Time History of vertical velocity component at point probe

4. Conclusion

This paper focuses on the unconfined flow around four cylinders in an in-line square configuration in the 2-D laminar regime for Re = 200 and L/D = 2.5 using a finite element based code COMSOL. It shows that the 2-D numerical simulation can replicate the basic characteristics of the flow across the four cylinders in an in-line square configuration. the recirculation zones from the upstream cylinders roll up into vortices and the vortices then impinge periodically on the downstream cylinder surfaces. The vortex shedding frequency i.e. Strouhal number obtained from the present study is in good agreement with the literature quoted value.

5. Reference

1. Behr, M., Hastreiter, D., Mittal, S., Tezduyar, T.E., "Incompressible flow past a circular cylinder: dependence of the computed flow field on the location of the lateral boundaries",. Computer Methods in Applied Mechanics and Engineering 123(1995)., 309–316.

2. COMSOL 4.0 User Guide, 2010.

3. Farrant, T., Tan, M., Price, W.G.,. "A cell boundary element method applied to laminar vortex-shedding from arrays of cylinders in various arrangements", Journal of Fluids and Structures 14(2000), 375–402.

4. K. Lam, W.Q. Gong, R.M.C. So,"Numerical simulation of cross-flow around four cylinders in an in-line square configuration", Journal of Fluids and Structures 24 (2008), 34–57

5. Tezduyar, T.E., Shih, R.,. "Numerical experiments on downstream boundary of flow past a cylinder", Journal of Engineering Mechanics117(1991), 854–871.