Numerical simulation of cross-flow around four cylinders by Local Domain Free Discretization-Immersed Boundary Method

***Y.L.Wu[1], C.Shu[2], and H.Ding[3]**

[1]Llord's Register Global Technology Centre, 1 Bukit Batok Street 22 #03-02, Singapore 659592.
[2]Department of Mechanical Engineering,, National University of Singapore, 10 Kent Ridge Crescent, Singapore 119260
[3]Department of Modern Mechanics, University of Science and Technology of China, Hefei, China

*Corresponding author: yanling.wu@LR.ORG

Abstract

In this paper, a hybrid of Local Domain Free Discretization and Immersed Boundary Method (termed as LDFD-IBM), is applied to simulate the incompressible flow over four circular cylinders in an in-line square configuration. LDFD-IBM belongs to the family of "Cartesian mesh methods", which means the complication of mesh generation is avoided for the problems with complex geometries. A Stencil Adaptive Mesh Refinement (SAMR) is also adopted to improve the computational efficiency. Instantaneous flow patterns and other quantitative information from the numerical simulation agree well with the available data from literatures.

Keywords: Flow over cylinders, Cartesian mesh method, LDFD-IBM, Local Domain-Free Discretization, Immersed Boundary Method, Stencil Adaptive Mesh Refinement

Introduction

Cross-flow of fluid around a group of cylinders has practical importance in engineering applications, such as offshore oil and gas pipelines. From the viewpoint of traditional numerical method (such as Finite Difference Method, FDM) in Computational Fluid Dynamics (CFD), the mesh generation in the flow domain for this kind of problem is obviously not a trivial task.

To solve this kind of problems involving with complex geometries, non-conforming mesh methods seem to be a good choice. Local Domain-Free Discretization (LDFD) (Shu and Wu, 2006) and Immersed Boundary Method (IBM) (Peskin,1977) are among of them.

LDFD is inspired from the analytical method. Consider a partial differential equation (PDE) on an irregular domain. The PDE is discretized at all mesh points inside the solution domain (referred here as to *interior points*), but the spatial discretization at an interior mesh point may involve some points outside the solution domain (referred here as to *exterior points*). The functional values at those exterior points can be approximated using the values at the interior points nearby by a local extrapolation scheme. However, it is found that the extrapolation in LDFD can cause errors and bring in numerical instabilities.

The basic idea of IBM is that the enforcement of the boundary to the surrounding fluids is through a body force appeared in the governing equation. The major advantage of IBM is its simplicity and easy implementation in dealing with flows with complex geometries. However, it should be indicated that the conventional IBM suffers two major drawbacks: one is to allow for the penetration of fluid flows into the immersed bodies; the other is the low order accuracy of presenting the immersed boundary.

Recently, Wu et al (2012) proposed LDFD-IBM, which is a delicate combination of LDFD method and IBM, and enjoys the merits of both methods. For example, the penetration of streamlines into solid objects in the conventional IBM, due to inaccurate satisfaction of no-slip boundary conditions, can be avoided by using the LDFD method. On the other hand, the treatment of boundary condition for pressure at the solid boundary in the LDFD method, is no longer necessary after introducing IBM.

In general, LDFD-IBM can be applied to any kind of mesh configuration. When the LDFD-IBM is applied on the Cartesian mesh, it can also be considered as a kind of Cartesian mesh solver. It is well known that solving these flow problems on a uniform Cartesian mesh usually needs very expensive computation. Therefore, from the viewpoint of computational efficiency, adaptive mesh refinement (AMR) is a desirable technique for Cartesian mesh solvers. In this paper, the recently proposed stencil mesh refinement (Ding and Shu, 2006) technique is introduced into LDFD-IBM.

Solution of Incompressible N-S Equations by LDFD-IBM

In this work, LDFD-IBM is used to simulate incompressible N-S equations. The governing equations are solved at all the mesh nodes (referred as the Eulerian points in conventional IBM) in the computational domain, regardless they are inside the solid bodies or inside the fluid field. The velocity boundary conditions are accurately enforced at the intersection points of the mesh lines and the boundary (referred as the Lagrangian points in the conventional IBM). The functional values at the intersection points are used to determine the solution at the interior dependent points via an interpolation scheme with second- or higher order of accuracy.

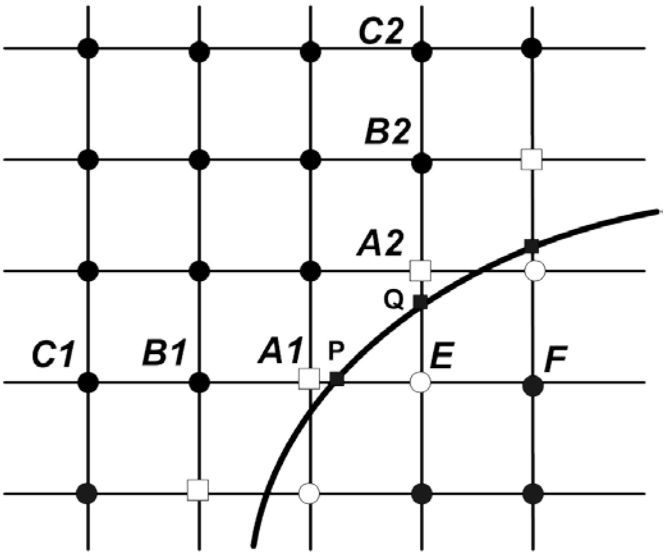


Figure 1 Classification of points in LDFD-IBM

For the application of LDFD-IBM to problems with complex geometry, let us consider a solid boundary immersed in a fixed Cartesian mesh as shown in Fig. 1. The mesh points take one of the following three statuses:

1) interior dependent points (the mesh nodes which are inside the fluid domain just adjacent to the immersed boundary, represented by symbol "□" in Figure 1);
2) exterior dependent points (the mesh nodes which are inside the solid domain just adjacent to the immersed boundary, represented by symbol "○" in Figure 1);
3) all other mesh nodes (represented by symbol "●" in Figure 1).

Obviously, A1 and A2 are interior dependent points. In present LDFD-IBM method, the functional values at interior dependent points are approximated by interpolation. For example, the velocity at A1 can be evaluated by a second order polynomial along the x direction, which involves three points P, B1, C1 as shown in Fig. 1. Here, B1 and C1 are the interior mesh points, and point P is the intersection point of the horizontal mesh line with the immersed boundary (represented by symbol "■") , where the velocity of immersed boundary is assigned. Point E is the exterior dependent node. In the original LDFD method, the functional values at exterior dependent points are obtained by extrapolation. However, in this work, since E is inside the solid body (outside flow domain), its velocity is simply assigned to the wall velocity

Clearly, the present approach combines the advantages of conventional IBM and LDFD method in the sense that:

(1) Since the pressure on all the Eulerian nodes is obtained by solving the pressure Poisson equation, the treatment of Neumann boundary condition at the Lagrangian points for pressure is no longer needed;
(2) There is no need to calculate the restoration force F on the Lagrangian points, and thus no need to distribute the restoring force F to the Eulerian nodes. The boundary effect is considered by updating the velocity at interior and exterior dependent points. Velocity boundary conditions are enforced accurately; As a consequence, the flow penetration is avoided.
(3) An interpolation scheme rather than extrapolation scheme is adopted to obtain the approximate solution at the dependent points, which makes the computation more stable.

Application of LDFD-IBM to Simulate Flow over Four Circular Cylinders

Flow past cylinder arrays can be found in many engineering applications, such as offshore structures, heat exchangers. The complexity of flow separation and free shear layer interference generated by the cylinder arrays have been studied by many researchers (Farrant et al, 2000; Lam et al, 2010). Flows over four equally spaced cylinders of equal diameter are computed by LDFD-IBM in this section. The configuration is shown in Figure 2, in which G is denoted as the minimum distance between the centers of 4 cylinders.

1) Two-dimensional study

The computation was performed at Re=200, based on one cylinder diameter and the spacing of cylinders G/D=3.0.

The computational domain is taken as 50D x 40D. Uniform Cartesian mesh of 201×161 is taken as the initial mesh. 8 levels of mesh refinement by stencil mesh refinement technique (Ding and Shu, 2006) are performed around the cylinders, and the final mesh contains 97121 nodes.

Figure 3 shows the instantaneous vorticity contours. According to Farrant et al (2000), the vortices formed in between the cylinders are weaker than those on the outside. This is confirmed by our results as shown in Figure 3.

Figure 4 shows the lift and drag coefficients history. From Fig. 4, it is found that the flow patterns around all cylinders are periodic with same frequencies, which suggests that the shedding is synchronized. It has been found that the present results have a good agreement with those of Farrant et al (2000).

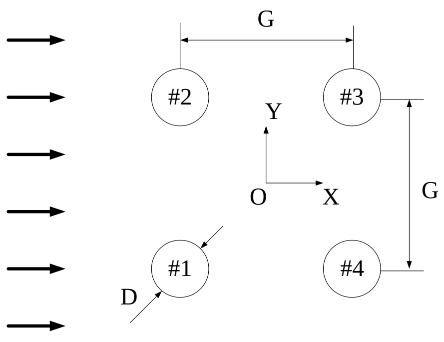


Figure 2. Configuration of flow past four equi-spaced cylinders

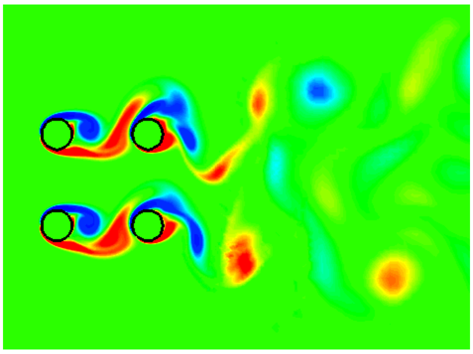


Figure 3. Instantaneous vorticity contours for 2D flow at Re=200 and G/D=3.0

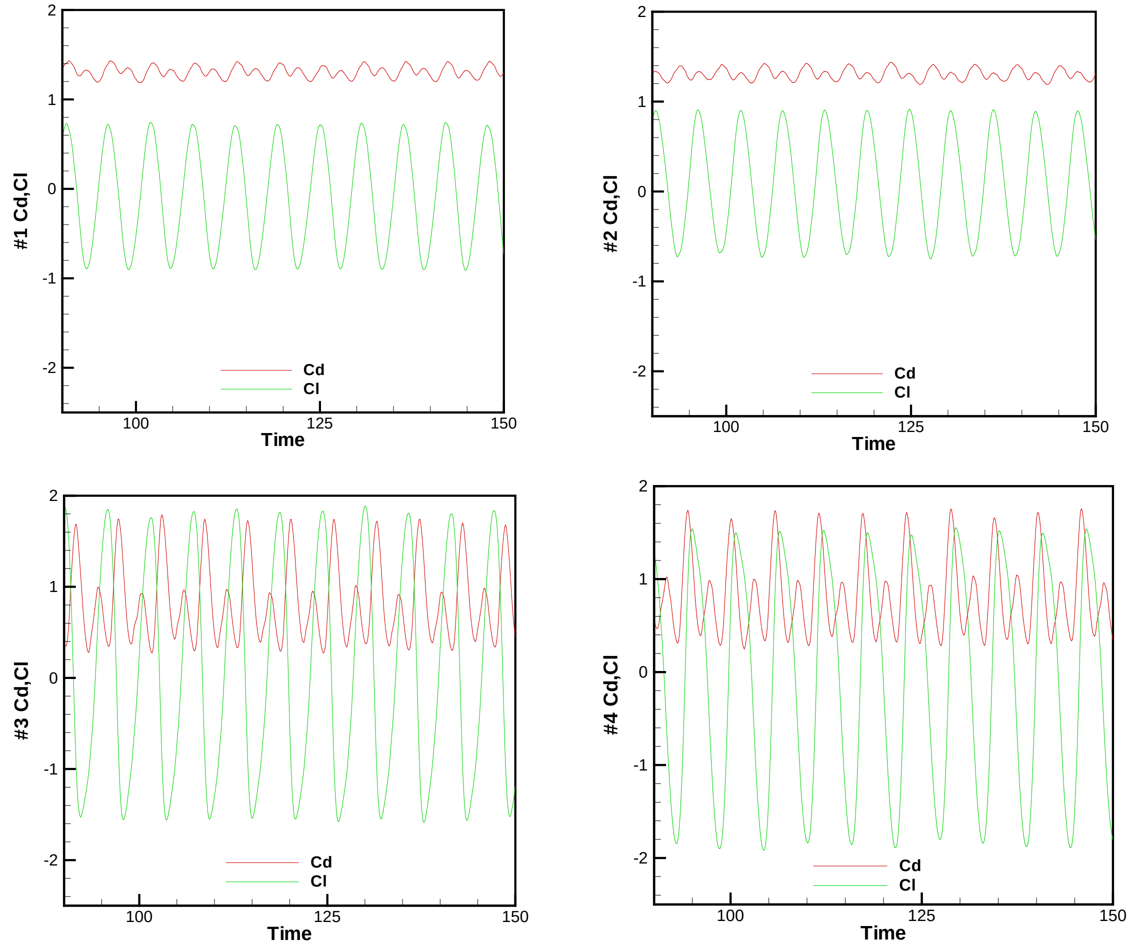


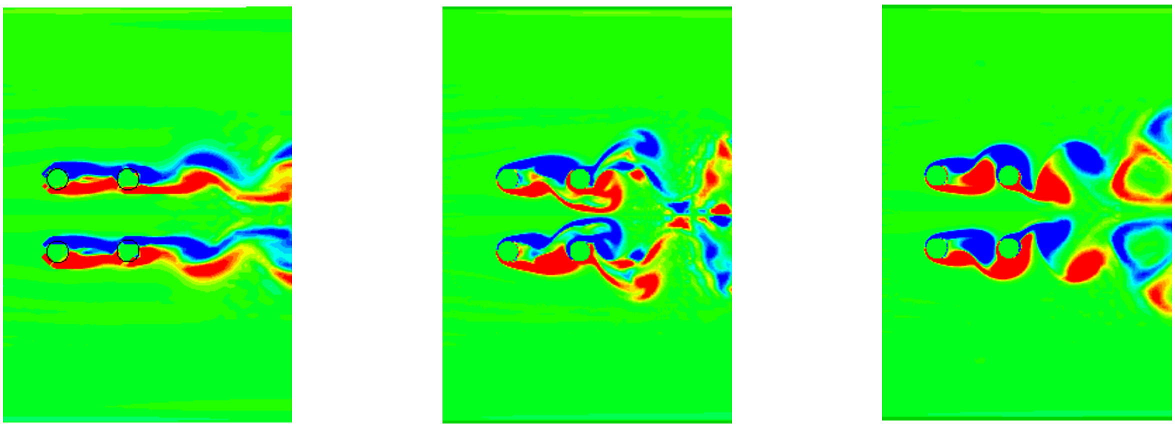
Figure 4 Drag and lift coefficients C_D and C_L: the curves correspond to cylinders #1 to #4,

2) Three-dimensional study

The LDFD-IBM can be easily extended to three-dimensional simulation of flows with curved geometry. The numerical discretization and the treatment of boundary condition are very similar to the two-dimensional case, except that one additional coordinate Z needs to be considered.

The computations were performed at Re=200, based on one cylinder diameter and the spacing of cylinders G/D=3.5, H/D=12. The two end wall is taken as the boundary condition along z direction.

The computational domain is a rectangular box with 32D×20D×11D in x, y, z directions, respectively. The mesh strategy in 3D is to use the two-dimensional stencil AMR technique to refine the mesh in the X-Y plane, and use a uniform mesh in the Z-direction. The initial mesh on the x-y plane is 161×101, after refined around cylinder by 8 levels, final mesh on the x-y plane has 46791 nodes. There are 41 nodes uniformly distributed along the z direction.



(a) z/H=0.0625 (b) z/H=0.25 (c) z/H=0.5

Figure 5 Instantaneous vorticity contours for 3D flow at Re=200 and G/D=3.5, H/D=12.

Figure 5 shows the instantaneous spanwise vorticity component ω_z at three different spanwise positions (z/H=0.0625 (near end wall), z/H=0.25,z/H=0.5 (mid-span of the cylinders)) for G/D=3.5 with cylinder length H=12D. In general, due to the end wall effect, with additional stronger streamwise vorticity and transverse vorticity generated along the spanwise direction of the cylinders, the 3-D vortex structure distributions are complex. It is found that the change of flow pattern at different spanwise positions of the cylinders is successfully captured by LDFD-IBM.

Conclusions

LDFD-IBM is the combination of LDFD and Immersed Boundary method (IBM), as well as their merits. In this paper, LDFD-IBM is applied to predict the 2D and 3D cross-flow over four cylinders in an in-line square configuration. Numerical results show that LDFD-IBM is a promising method to simulate flow problems with complex geometries accurately and easily.

References

Ding H. and Shu C. (2006): A Stencil Adaptive Algorithm for Finite Difference Solution of Incompressible Viscous Flows, *J. Comput. Phys.*, 214, pp.397-420.

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

Lam K, Zou L. (2010), Three-dimensional numerical simulations of cross-flow around four cylinders in an in-line square configuration, *Journal of Fluids and Structures*, 26, pp.482-502.

Peskin C.S. (1977): Numerical analysis of blood flow in the heart, *J.Comput.Phys*.25, pp 220

Wu Y.L., Shu C, and Ding H. (2012): Simulation of Incompressible Viscous Flows by Local DFD-Immersed Boundary Method, *Advances in Applied Mathematics and Mechanics*, 4 (3), pp. 311-324.