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

MULTIGRID FICTITIOUS BOUNDARY METHOD FOR VORTEX-INDUCED MOTION OF CYLINDERS ON MOVING GRIDS

D.C. Wan

State Key Laboratory of Ocean Engineering
School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University
Huashan Road 1954, Shanghai 200030, China
Email: dcwan@sjtu.edu.cn

Key Words: *Fluid-Structure Interaction, Multigrid Fictitious Boundary Method, ALE, Moving Mesh Method.*

ABSTRACT

The coupling between the unsteady wake of cylinders in a cross-flow and vortex-induced motion of the cylinders is important in structural, offshore and thermal power engineering applications. In some cases, this has to be taken into account in their design as a potential cause of fatigue damage, such as marine risers for exploration drilling and production in greater deep water areas [1-2]. Experimental work has been unable to capture all of the characteristics of vortex-induced motion of the cylinders. Some major features of the near wake vortex shedding and vortex-induced motion of the cylinders have been thus qualitatively and quantitatively described using analytical and numerical methods. Increasing computational resources have made possible the direct numerical simulation of incompressible viscous flows around a fixed, forced or free structure profile. This has provided more detailed flow field analysis to compare with experimental observations and previous simplified method.

In this paper, multigrid fictitious boundary method (MFBM) [3-5] coupled with arbitrary Lagrangian-Eulerian (ALE) and moving mesh techniques is presented to direct numerical simulation of vortex-induced motion of one or several cylinders under an incoming incompressible viscous flow. The flow is computed by a special ALE formulation of Navier-Stokes equations with a multigrid finite element solver. The interaction between the fluid and the cylinders is taken into account by the MFBM in which an explicit volume based calculation for the forces is integrated. The solid cylinders are allowed to move freely through the computational mesh which is adaptively aligned by a special mesh deformation method (*r*-type moving mesh method) such that the accuracy for dealing with the interaction between the fluid and the cylinders is highly improved. The moving mesh is created from an equidistant cartesian mesh in which the topology is preserved and only the grid spacing is changed such that the grid points are concentrated near the surfaces of the solid cylinders. Only the

solution of additional linear Poisson problems in every time step is required for generating the moving mesh, which means that the additional work is significantly less than the main fluid-solid part. The main advantage of this methodology is that it allows the numerical treatment on a structured mesh on a simple shape auxiliary domain containing the actual one, independent of the actual boundary of the moving cylinders, allowing therefore the use of fast solvers. Since the size of computation and data structure of the moving meshes are fixed, which enables the proposed method is much easier to incorporate into most CFD codes without the need for the changing of system matrix structures and special interpolation procedures.

In this paper, numerical examples of vortex-induced motion of one, two and four 2D circular cylinders (both tandem and side-by-side arrangements of the cylinders are considered) in a channel viscous flows are computed to illustrate the efficiency of the presented method for numerical solution of vortex-induced motion of cylinders, respectively. The presented method can well simulate the cylinder oscillations it causes, and unsteay flows caused by the oscillating cylinders. Both quantitative and qualitative comparisons with published experimental and numerical data are made, showing excellent agreement. Comparisons with experimental results indicate that the present numerical method can capture complex interference and flow-structure interaction phenomena. Numerical results are also presented to demonstrate wake galloping effects, in which a cylinder in the wake of another experiences large flow-induced vibration over a wide range of flow velocities.

REFERENCES

[1] H.M. Blackburn and R.D. Henderson, "A study of two-dimensional flow past an oscillating cylinder", *J. Fluid Mech.*, Vol. **385**, pp. 255-286, (1999).

[2] R. Glowinski, T.W. Pan, T.I. Hesla, D.D. Joseph and J. Periaux, "A fictitious domain approach to the direct numerical simulation of incompressible viscous flow past moving rigid bodies: Application to particulate flow", *J. Comput. Phys.*, Vol. **169**, pp. 363-426, (2001).

[3] D.C. Wan and S. Turek, "Fictitious boundary and moving mesh methods for the numerical simulation of rigid particulate flows", *J. Comput. Phys.*, Vol. **222**, pp. 28-56, (2007).

[4] D.C. Wan and S. Turek, "An efficient multigrid-FEM method for the simulation of liquid-solid two phase flows", *J. for Comput. and Appl. Math.*, Vol. **203**, pp. 561-580, (2007).

[5] D.C. Wan and S. Turek, "Direct numerical simulation of particulate flow via multigrid FEM techniques and the fictitious boundary method", *Int. J. Numer. Meth. Fluids*, Vol. **51**, pp. 531-566, (2006).