Numerical Simulations of Viscous Flows with Induced Moving Discs in a Periodic Lid-Driven Cavity

Decheng Wan
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China

ABSTRACT

This paper reports the results of direct numerical simulation of the motion of a two-dimensional neutrally buoyant circular cylinder in a periodic lid-driven cavity by using multigrid fictitious boundary method (MFBM) coupled with finite element method. The MFBM is based on a regular rectangular grid. The flow is computed by a finite element solver and the solid cylinder is allowed to move freely through the computational mesh which can be chosen independently from the cylinder of size and number. The interaction between the fluid and the cylinder is taken into account by the MFBM in which an explicit volume based calculation for the hydrodynamic forces is integrated. The main advantage of the MFBM is that the solid cylinder can move freely through the computational mesh for the fluid part which has not to change in time. Comparisons with experimental results indicate that the present numerical method can capture complex flow-cylinder interaction phenomena, and obtain the detailed results of trajectories, translation and rotation of the free neutrally buoyant cylinder in a lid-driven cavity flow.

KEY WORDS: Multigrid fictitious boundary method; FEM; Fluid-structure interaction; Moving discs; Periodic lid-driven cavity.

INTRODUCTION

Direct numerical simulation of solid-liquid flow is a difficult task since the domain occupied by the fluid is irregular and changes with motion of the particles. Also, the particles are advected by the fluid and exert forces at the fluid. The movement of both the interface and the solid particles is unknown in advance and must be determined as part of the solution, it can require a huge amount of time for the generation of a boundary-fitted mesh for each different position of the moving particles.

Generally speaking, there are two ways to solve this problem. The first is a generalized ALE standard Galerkin finite element method (Hu, et al, 2001, Maury 1996) in which both the fluid and particle equations of motion are incorporated into a single coupled variational equation. The hydrodynamic forces and torques on the particle are calculated in a very efficient way. The computational costs are practically independent of the number of body present in the computational domain. The FBM is based on an unstructured FEM background grid. The flow is computed by a multigrid finite element solver and the body is allowed to move freely through the computational mesh which can be chosen independently from the body of arbitrary shape, size and number. The same fixed grid is also used to represent the location of the body by imposing the velocities on the nodes covered by the body at any time. The new positions and the new velocities of the body are updated using Newton's law so that there is no need to remesh the domain. The interaction between the fluid and the body is taken into account by the FBM in which an explicit volume based calculation for the hydrodynamic forces is integrated. Based on the boundary conditions applied at the interface between the body and the fluid which can be seen as an additional constraint to the governing Navier-Stokes equations, the fluid domain can be extended into the whole domain which covers both fluid and body domains. It starts with a coarse mesh which may contain already many of the geometrical fine-scale details, and employs a (rough) boundary parameterization which sufficiently describes all large-scale structures with regard to the boundary conditions. Then, all fine-scale features are treated as interior.

We proposed another multigrid FEM-based explicit fictitious boundary method (MFBM) (Wan and Turek, 2006, Wan and Turek, 2007a, Wan and Turek, 2007b). In contrast to the implicit fictitious boundary approach, the explicit fictitious boundary approach is to solve fluid equations and body equations separately. The forces exerted on body are calculated in a very efficient way. The computational costs are practically independent of the number of body present in the computational domain. The FBM is based on an unstructured FEM background grid. The flow is computed by a multigrid finite element solver and the body is allowed to move freely through the computational mesh which can be chosen independently from the body of arbitrary shape, size and number. The same fixed grid is also used to represent the location of the body by imposing the velocities on the nodes covered by the body at any time. The new positions and the new velocities of the body are updated using Newton's law so that there is no need to remesh the domain. The interaction between the fluid and the body is taken into account by the FBM in which an explicit volume based calculation for the hydrodynamic forces is integrated. Based on the boundary conditions applied at the interface between the body and the fluid which can be seen as an additional constraint to the governing Navier-Stokes equations, the fluid domain can be extended into the whole domain which covers both fluid and body domains. It starts with a coarse mesh which may contain already many of the geometrical fine-scale details, and employs a (rough) boundary parameterization which sufficiently describes all large-scale structures with regard to the boundary conditions. Then, all fine-scale features are treated as interior.