Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2011,23(4):476-482
DOI: 10.1016/S1001-6058(10)60138-2
www.sciencedirect.com/science/journal/10016058

NUMERICAL METHOD FOR MULTI-BODY FLUID INTERACTION BASED ON IMMERSED BOUNDARY METHOD*

MING Ping-jian, ZHANG Wen-ping
School of Power and Energy Engineering, Harbin Engineering University, Harbin 150001, China,
E-mail: pingjianming@hrbeu.edu.cn

(Received February 12, 2011, Revised April 7, 2011)

Abstract: A Cartesian grid based on Immersed Boundary Method (IBM), proposed by the present authors, is extended to unstructured grids. The advantages of IBM and Body Fitted Grid (BFG) are taken to enhance the computation efficiency of the fluid structure interaction in a complex domain. There are many methods to generate the BFG, among which the unstructured grid method is the most popular. The concept of Volume Of Solid (VOS) is used to deal with the multi rigid body and fluid interaction. Each body surface is represented by a set of points which can be traced in an anti-clockwise order with the solid area on the left side of surface. An efficient Lagrange point tracking algorithm on the fixed grid is applied to search the moving boundary grid points. This method is verified by low Reynolds number flows in the range from $Re = 100$ to 1 000 in the cavity with a moving lid. The results are in a good agreement with experimental data in literature. Finally, the flow past two moving cylinders is simulated to test the capability of the method.

Key words: fluid-structure interaction, Immersed Boundary Method (IBM), Volume Of Solid (VOS), unstructured grids

Introduction

The multi-body and fluid interaction can be widely found in marine and ocean engineering, and is a very important issue in Computation Fluid Dynamics (CFD). Numerical methods were developed as a complementary to experimental and analytical studies, including the Lagrangian moving mesh method[1] and the Chimera grid method[2,3]. The Lagrangian moving mesh is very inefficient or even impossible to simulate large amplitude motions, for which a remeshing procedure[4] is needed to deal with problems related with the low mesh quality. The Chimera grid method can deal with large amplitude moving body problems, but it is very expensive for the interpolation of variable values between moving grids and background grids. Recently, an immersed boundary method was proposed to simulate the fluid flow and rigid body interaction on fixed meshes. The immersed boundary method was originally proposed by Peskin in 1972 to analyze the blood flow in a human heart model. In his simulation, only a simple Cartesian mesh system was used, with grids not necessarily conforming to the geometry of the heart[5]. The immersed boundary method is put into a mathematical formulation and the effects of solid boundaries are reflected in forces in the Navier-Stokes equations. There comes the so-called "Immersed Boundary (IB)" method, which gradually becomes a basic concept that the body is simply 'immersed' in the background mesh without necessarily fitting the underlying mesh points. Many modified IBMs were proposed, which are quite different from the most conventional CFD approaches[6]. More details of IBM history can be found in literature[7].

The Immersed Boundary Method (IBM) is originally implemented in the Cartesian grid, with an obvious drawback that the grid number is increased drastically with the increase of the Reynolds number[8]. In some situations, a great number of extra outside grids are required in the Cartesian grid, for example, in the lobe pump as shown in Fig.1(a). In order to reduce the extra grid number, IBM is extended to curvilinear grids to generate a body fitted grid through a coordinate transform[9]. In this article, the immersed boundary method is further improved, and then is used for unstructured grids as shown in Fig.1(b). The scope

* **Biography:** MING Ping-jian (1980-), Male, Ph. D., Lecturer