Available online at www.sciencedirect.com
ScienceDirect
Journal of Computational Physics 218 (2006) 510–548
JOURNAL OF COMPUTATIONAL PHYSICS
www.elsevier.com/locate/jcp

A generalized finite-difference (GFD) ALE scheme for incompressible flows around moving solid bodies on hybrid meshfree–Cartesian grids

C.S. Chew [a], K.S. Yeo [b,*], C. Shu [b]

[a] The Institute of High Performance Computing (IHPC), CFD Division, 1 Science Park Road, #01-01, The Capricorn, Singapore Science Park II, Singapore 117528, Singapore
[b] Department of Mechanical Engineering, National University of Singapore, 9 Engineering Drive 1, Singapore 117576, Singapore

Received 9 March 2005; received in revised form 6 January 2006; accepted 22 February 2006
Available online 18 April 2006

Abstract

A scheme using the mesh-free generalized finite differencing (GFD) on flows past moving bodies is proposed. The aim is to devise a method to simulate flow past an immersed moving body that avoids the intensive remeshing of the computational domain and minimizes data interpolation associated with the established computational fluid methodologies; as such procedures are time consuming and are a significant source of error in flow simulation. In the present scheme, the moving body is embedded and enveloped by a cloud of mesh-free nodes, which convects with the motion of the body against a background of Cartesian nodes. The generalized finite-difference (GFD) method with weighted least squares (WLS) approximation is used to discretize the two-dimensional viscous incompressible Navier–Stokes equations at the mesh-free nodes, while standard finite-difference approximations are applied elsewhere. The convecting motion of the mesh-free nodes is treated by the Arbitrary Lagrangian–Eulerian (ALE) formulation of the flow equations, which are solved by a second-order Crank–Nicolson based projection method. The proposed numerical scheme was tested on a number of problems including the decaying-vortex flow, external flows past moving bodies and body-driven flows in enclosures.

Keywords: Moving body; Convecting nodes; Meshless method; Projection method; Generalized finite difference; Arbitrary Lagrangian–Eulerian; Navier–Stokes equations

1. Introduction

Computational fluid dynamics (CFD) is traditionally practised using mesh-based methods like the finite element (FE), finite volume (FV) and finite difference (FD), and they are widely accepted as the mainstream solution tools. While the mesh structure allows the FE and FV methods to handle complex geometries effectively, the same structure becomes a limitation when one has to alter the meshing to accommodate a

* Corresponding author. Tel.: +65 6516 2246; fax: +65 6779 1459.
E-mail address: mpeyeoks@nus.edu.sg (K.S. Yeo).

0021-9991/$ - see front matter
doi:10.1016/j.jcp.2006.02.025

continuously changing computational domain. Problems of this nature are becoming common today, particularly with the growing preoccupation of engineers with biologically motivated problems, where large boundary motions are frequently encountered. Besides appearing cumbersome and inelegant, extensive remeshing and data interpolation in these methods can add to computational costs and solution inaccuracies. The traditional finite-difference method is highly effective when coordinate transformations are available to simplify the computational geometries. Unlike FE and FV methods, however, the transformation approach cannot be extended to complex geometries.

In recent decades, a new class of what is termed mesh-free or meshless numerical methods has flourished. This is due in part to their ability to alleviate some of the shortcomings of the conventional procedures. As opposed to mesh-based methods, they do not require pre-specified connectivity between the computational nodes or grid points. There is no need to identify the element or volume edges and surfaces. These methods are thus better able to represent irregular boundaries and negotiate complex spaces with ease and flexibility. Mesh-free methods are known to originate from the works of astrophysicists. The smooth particle hydrodynamics (SPH) method [1] is used to simulate astrophysical phenomena such as exploding stars and dust clouds. This method belongs to the class of particle methods [2] which adopts kernel approximations to obtain the Lagrangian flow field. Belytschko et al. [3] provided a detailed description of the broad spectrum of mesh-free methods that have been developed, including the diffuse element method (DEM) [4], the element-free Galerkin method (EFG) [5], the reproducing-kernel particle method (RKPM) [6], the partition of unity method (PUM, or PUFEM) [7], hp clouds [8], finite point method (FPM) [9–11] and the generalized finite-difference method (GFD) [12–15]. These methods have been applied to various problems in solid and fluid mechanics.

Particle methods like SPH and the vortex particle method are Lagrangian in nature and avoid grid-based numerical dissipation, which causes the flow to be more viscous. However, these 'particles' require good distribution and redistribution, and their Lagrangian nature makes this difficult in flows with extensive boundary motions. DEM and EFG belong to the type of mesh-free methods that are integral by nature. In this sense, they are not strictly 'free from mesh', and may suffer from the same deficiencies as the FE methods in moving body problems. The truly mesh-free methods are the non-integral types in hp clouds, FPM and GFD. These methods adopt mesh-free computational nodes that may not necessarily be Lagrangian in nature, and such nodes may be more suited for handling flows with moving boundaries.

As mentioned earlier, there has been growing interest in the modeling of moving boundary problems, which is aided not least by the continuous increase in computing power and corresponding reduction in the cost of computation. In this paper, we shall concern ourselves with the development of a hybrid mesh-free computational scheme for the simulation of flows past and flows driven by moving solid bodies. We believe the advantages of mesh-free methods can be harnessed to solve this class of problem. Before we proceed with the details of the new scheme, we shall first present a broad overview of the various methods that have been devised over the years to deal with the simulation of moving boundary/body problems.

Methods for solving moving boundary or interface problems fall under two main categories: those that employ boundary-fitted grids (also termed body-conformal grids) and those that employ non-boundary-fitted grids. A method is boundary-fitted when the boundaries or interfaces coincide with the computational nodes or grid points. The nodes/grid points then clearly demarcate the interface, and morph and move with the interface during motion. Boundary conditions are applied directly on the interface, and the motion of the interface is explicitly tracked. The two common boundary-fitted methods are composite grids [16,17], which are normally applied with finite differencing, and FV or FE schemes, which are frequently implemented with the arbitrary Lagrangian–Eulerian (ALE) formulation [18].

Composite grids with generalized coordinates and transformed governing equations are commonly used to embed solid objects within flows. A separate set of coordinate frame and grid points (the sub-mesh) are generated for each embedded object. Local sub-meshing is usually performed using an orthogonal or algebraic grid generator. Communication between the main and the local meshes is usually done through extensive interpolation of data between the meshes, typically at the outer boundaries of the mesh systems. Apart from interpolation, the sub-meshes are also restricted in their geometric complexity due to the use of generalized coordinates. The second boundary-fitted method involves the use of unstructured grids with FE or FV methodologies to solve for the flow in arbitrary Lagrangian–Eulerian (ALE) formulation [19,20]. This is an elegant approach in which the solution is derived based on a mixed treatment of grid points in either the Lagrangian