Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **2**, No. 6, pp. 746-762

DOI: 10.4208/aamm.09-m09112
December 2010

A Three Dimensional Gas-Kinetic Scheme with Moving Mesh for Low-Speed Viscous Flow Computations

Changqiu Jin[1,*], Kun Xu[2] and Songze Chen[3]

[1] *Institute of Applied Physics and Computational Mathematics, Beijing 100088, China*
[2] *Mathematics Department, Hong Kong University of Science and Technology, Clear Water Bay, Kowloon, Hong Kong*
[3] *College of Engineering, Peking University, Beijing 100871, China*

Received 20 December 2009; Accepted (in revised version) 9 May 2010
Available online 26 August 2010

Abstract. The paper introduces the gas-kinetic scheme for three-dimensional (3D) flow simulation. First, under a unified coordinate transformation, the 3D gas-kinetic BGK equation is transformed into a computational space with arbitrary mesh moving velocity. Second, based on the Chapman-Enskog expansion of the kinetic equation, a local solution of gas distribution function is constructed and used in a finite volume scheme. As a result, a Navier-Stokes flow solver is developed for the low speed flow computation with dynamical mesh movement. Several test cases are used to validate the 3D gas-kinetic method. The first example is a 3D cavity flow with up-moving boundary at Reynolds number 3200, where the periodic solutions are compared with the experimental measurements. Then, the flow evolution inside a rotating 3D cavity is simulated with the moving mesh method, where the solution differences between 2D and 3D simulation are explicitly presented. Finally, the scheme is applied to the falling plate study, where the unsteady plate tumbling motion inside water tank has been studied and compared with the experimental measurements.

AMS subject classifications: 76D05

Key words: Gas-Kinetic scheme, moving mesh, low-speed viscous flow, freely moving body.

---

1 Introduction

There are two different coordinate system for description of fluid motion: the Eulerian one describes fluid motion at fixed locations, and the Lagrangian one follows

*Corresponding author.
URL: `http://www.math.ust.hk/~makxu/`
Email: jin_changqiu@iapcm.ac.cn (C. Q. Jin), makxu@ust.hk (K. Xu), jacksongze@tom.com (S. Z. Chen)

fluid elements. Considerable progress has been made over the past two decades on developing computational fluid dynamics (CFD) methods based on the above two coordinates system. As the unsteady flow calculations with moving boundaries and interfaces become important, such as found in the flutter simulation of wings, turbomachinery blades, and multiphase flow, the development of fast and reliable methods for dynamically deforming computational domain is required [16].

There are many moving mesh methods in the literature. One example is the static mesh movement method, where the new mesh is generated at each time step according to certain monitor function and the flow variables are interpolated into the newly generated mesh. Then, the flow update through the cell interface fluxes is done on a static mesh. In order to increase the accuracy, the mesh can be properly adapted [5,8]. Another example is the dynamical one, where the mesh is moving according to certain velocity. At the same time, the fluid variables are updated inside each moving control volume within a time step. The second method is mostly used to track the interface location [14], to account for changes in the interface topology, and to resolve small-scale structure at singular point. The most famous one for this dynamical mesh moving method is the Lagrangian method. Through the research in the past decades, it has been well recognized that the Lagrangian method is always associated with the mesh tangling once the fluid velocity is used as the mesh moving velocity. In order to avoid severe mesh distortion in the Lagrangian method, many techniques have been developed. The widely used one at present time is the Arbitrary Lagrangian-Eulerian (ALE) technique, which uses continuous re-zoning and re-mapping from Lagrangian to the Eulerian grid. This process requires interpolations of geometry and flow variables once the computational grid is getting too distorted [13].

Recently, a successful moving mesh method for inviscid Euler equations has been developed by Hui et al. on the target of crisp capturing of slip line [9]. In this unified coordinate method, with a prescribed grid velocity, the inviscid flow equations are written in a conservative form in the computational domain (λ, ξ, η), as well as the geometric conservation laws which control the mesh deformation. The most distinguishable merit in the unified coordinate method [9] is that the fluid equations and geometric evolution equations are written in a combined system, which is different from the fluid equations alone [5, 10]. Furthermore, due to the coupling of the fluid and geometric system, for the first time the multidimensional Lagrangian gas dynamic equations have been written in a conservative form. As a consequence, theoretically it has been shown that the multidimensional Lagrangian system is only weakly hyperbolic. The distinguishable achievement of the unified coordinate method is that the numerical diffusion across the slip line can be reduced to a minimum level with the crisp capturing of contact discontinuity. However, in the complicated flow movement, in order to avoid the severe mesh distortion, the constraints, such as keeping mesh orthogonality and grid angles, have to be used in the unified coordinate system. As a result, in most cases, the constraint automatically enforces the mesh velocity being zero, such as in the case of gas implosion inside a square. Otherwise, for flow problems with circulations, any mesh movement method, once the grid speed is coupled