Available online at www.sciencedirect.com
ScienceDirect
Journal of Computational Physics 222 (2007) 155–175
JOURNAL OF COMPUTATIONAL PHYSICS
www.elsevier.com/locate/jcp

A unified moving grid gas-kinetic method in Eulerian space for viscous flow computation

Changqiu Jin, Kun Xu *

Department of Mathematics, Hong Kong University of Science and Technology, Clear Water Bay, Kowloon, Hong Kong, China

Received 6 March 2006; received in revised form 8 July 2006; accepted 19 July 2006
Available online 1 September 2006

Abstract

Under a generalized coordinate transformation with arbitrary grid velocity, the gas-kinetic BGK equation is reformulated in a moving frame of reference. Then, a unified conservative gas-kinetic scheme is developed for the viscous flow computation in the moving grid system in the Eulerian space. Due to the coupling between the grid velocity and the overall solution algorithm, the Eulerian and Lagrangian methods become two limiting cases in the current gas-kinetic method. A fully conservative formulation can be obtained even in the Lagrangian limit. The moving grid method extends the applicable regime of the gas-kinetic scheme to the flows with free surface or moving boundaries, such as dam break problem and airfoil oscillations. In order to further increase the robustness of the moving grid method, similar to the arbitrary Lagrangian–Eulerian (ALE) method, a conservative adaptive grid technique is also implemented in the current method to redistribute the mesh concentration to the rapid variational flow region and remedy the distorted moving mesh due to the coupling between grid velocity and fluid speed. Many numerical examples from incompressible flow to the supersonic shock interaction are presented. The test cases verify the accuracy and robustness of the unified moving grid gas-kinetic method.

MSC: 65M06; 76P05; 76T05

Keywords: Unified coordinate system; Moving grid; Gas-kinetic scheme; Navier–Stokes equations

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 fluid particles. 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

* Corresponding author. Tel.: +852 2358 7440; fax: +852 2358 1643.
E-mail addresses: majchq@ust.hk (C. Jin), makxu@ust.hk (K. Xu).

0021-9991/$ - see front matter
doi:10.1016/j.jcp.2006.07.015

required [17]. This research will help significantly the industry, such as the aerodynamic shape optimization studies and the detonative chemical reactive flow computation.

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 [9]. 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, 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. Unfortunately, this process requires interpolations of geometry and flow variables [14]. In aerospace engineering, in order to re-distribute the boundary deformation dynamically into the whole computational domain a spring network approach has been usually used to determine the motion of the mesh point, such as those around a deforming airfoil [2,16,23]. Here, a smoothing global operator is applied in maintaining grid smoothness and grid angles. This process is always associated with iterative methods resembling an elliptic grid generator. With a general transformation between the physical (t, x, y) and the computational space (λ, ξ, η), the Navier–Stokes equations can be written in a conservative form [7]. Many numerical schemes have been developed based on the above formulation for the Navier–Stokes equations directly, such as in the cases of fluid-structure interaction and fluid induced vibration. Instead of constructing an exact Riemann solver, an efficient approximate Riemann solution has been obtained [6], where the grid velocity is explicitly used in the wave decomposition. Even without using conservative governing equations explicitly, many moving mesh methods for incompressible Navier–Stokes equations, hyperbolic system, or chemical reactive flow, have also been developed with detailed consideration of numerical cell deformation [20,4,1].

Recently, a successful moving mesh method for inviscid Euler equations has been developed by Hui et al. [10] on the target of crisp capturing of slip line. 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 is that the fluid equations and geometric evolution equations are written in a combined system, which is different from the fluid equations alone [7]. Furthermore, due to the coupling of the fluid and geometric system, for the first time the multi-dimensional Lagrangian gas dynamic equations have been written in a conservative form. As a consequence, theoretically it has been shown that the multi-dimensional Lagrangian system is only weakly hyperbolic. Numerically, in the unified coordinate system the fluid and geometric variables can be updated simultaneously. In order to overcome the disadvantage in the Lagrangian method, in the unified coordinate system the grid velocity is set to be $h\mathbf{q}$, where $\mathbf{q} = (U, V)$ is the fluid velocity and h is a parameter which is to be determined by conditions, such as the mesh alignment with the slip surface, or keeping grid angle during the mesh movement. Therefore, the grid velocity can be changed locally according to the value of h. In a recent paper [11], the grid velocity has been further generalized to (hU, kV), where h and k are two parameters to be determined. The great achievement of the unified coordinate method is that the numerical diffusion across the slip line is reduced to a minimum 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 with the fluid velocity, will distort the mesh eventually and stop the computation. Also, in order to capture the slip line, the unified method is mainly focusing on the solution of the Euler equations. For the viscous flow, the equations, see Appendix, become much more complicated in a unified coordinate system.