Journal of Computational Physics 326 (2016) 862–877

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Cartesian grid method for gas kinetic scheme on irregular geometries

Songze Chen [a,*], Kun Xu [b], Zhihui Li [c]

[a] Huazhong University of Science and Technology, State Key Laboratory of Coal Combustion, Luoyu Road 1037, Wuhan, China
[b] The Hong Kong University of Science and Technology, Clear Water Bay, Kowloon, Hong Kong, China
[c] Hypervelocity Aerodynamics Institute, China Aerodynamics Research and Development Center, Mianyang 621000, China

A R T I C L E   I N F O

Article history:
Received 15 February 2015
Received in revised form 1 May 2016
Accepted 9 September 2016
Available online 20 September 2016

Keywords:
Gas kinetic scheme
Cartesian grid
Viscous flow
Supersonic flow

A B S T R A C T

A Cartesian grid method combined with a simplified gas kinetic scheme is presented for subsonic and supersonic viscous flow simulation on complex geometries. Under the Cartesian mesh, the boundaries are represented by a set of direction-oriented boundary points, and the computational grid points are classified into four different categories, the fluid point, the solid point, the drop point, and the interpolation point. A constrained weighted least square method is employed to evaluate the physical quantities at the interpolation points. Different boundary conditions, including isothermal boundary, adiabatic boundary, and Euler slip boundary, are presented by different interpolation strategies. We adopt a simplified gas kinetic scheme as the flux solver for both subsonic and supersonic flow computations. The methodology of constructing a simplified kinetic flux function can be extended to other flow systems. A few numerical examples are used to validate the Cartesian grid method and the simplified flux solver. The reconstruction scheme for recovering the boundary conditions of compressible viscous and heat conducting flow with a Cartesian mesh can provide a smooth distribution of physical quantities at solid boundary, and present an accurate solution for the flow study with complex geometry.

1. Introduction

Solving flow problem on conformal mesh or unstructured mesh is the most practical strategy in industry applications. However, the grid generation on complex geometry is time consuming and requires sophisticated technique. In this sense, the grid generation on complex computational domain is still the bottle neck of practical applications. Thereby, the Cartesian grid method arises, in which the flow region is discretized by a Cartesian grid regardless of the shapes of objects inside the flow region. The obvious advantage of this method over the conventional conformal approach is that the computational mesh is easily generated when different geometries are considered. Cartesian grid methods free the researchers and engineers from the burdensome grid generation, but introduce two new problems about boundary treatment.

The first problem with the Cartesian grid is about how to represent the boundary. It depends on the boundary property. For the multi-fluid and multi-phase flows, where the boundaries or the interfaces are deformable, the volume-of-fluid (VOF) method [1] and the phase-field approach [2] are the most popular approaches. The boundary is reconstructed from an

* Corresponding author.
E-mail address: jacksongze@hust.edu.cn (S. Chen).

http://dx.doi.org/10.1016/j.jcp.2016.09.018
0021-9991/

Nomenclature

a	Expansion coefficient of the spatial derivative of g
F	Macroscopic fluxes for density, momentum, and energy
n	Normal direction of a surface
t	Tangential direction of a surface
U	Macroscopic velocity
u	Particle velocity
W	Conserved macroscopic variables, including density, momentum, and energy
ρ	Gas density
τ	Relaxation time of kinetic equation
f	Particle velocity distribution function
g	Equilibrium state of particle velocity distribution function
K	Equivalent internal degree of freedom which is 1 for monatomic gas, and 3 for diatomic gas in two dimensional flow
M	The number of the boundary points which are chosen to be the constraint
N	The number of the fluid points in the boundary interpolation stencil
P	Interpolation polynomial
p	Gas pressure
R	Gas constant
T	Gas temperature
w	Weight function
Ma	Mach number
Re	Reynolds number

auxiliary function, say, a marker function. Or, the boundary is captured by solving a partial differential equation for the phase field. These approaches are efficient for boundary deformation and splitting. But obviously, this kind of approaches is less accurate than the Lagrangian representation of the boundary. The latter approach takes the boundary as a sharp interface either explicitly tracking as curves [3] or as level sets [4]. Sharp interface is desirable for high Reynolds number flows where the boundary layer plays an important role.

The second problem is how to impose the boundary condition. Following the classification by Mittal and Iaccarino [5], there are two different categories. The first one is the continuous forcing approach; the second one is the discrete forcing approach. The continuous forcing approach directly modifies the governing equation by adding a forcing term to take the boundary effect into account [6,7]. An ideal force term is represented by a Dirac delta function. Since the boundary cuts the grid line at arbitrary location, the forcing should be distributed over a band of cells around the boundary point. This approach results in a diffusive boundary. The discrete forcing approach imposes the boundary condition on the numerical solution directly at discrete level. Mittal and Iaccarino [5] further subdivided the second category into "Indirect BC Imposition" and "Direct BC Imposition". The former one employs a forcing term which is determined from a priori estimation of flow field at discrete level [8]. The external force is explicitly computed in advance or solved by an implicit method to guarantee the no-slip boundary condition [9,10]. However, the artificial forcing procedure diffuses the flow field around the boundary. For this reason, the "Direct BC Imposition" approaches retain the boundary as a sharp interface with no spreading. This can usually be accomplished by modifying the computational stencil near the boundary to directly impose the boundary condition. This kind of approach is always referred to as Cartesian grid method which is the main focus of this study.

Berger and LeVeque [11] presented a Cartesian grid algorithm with adaptive refinement to compute flows around arbitrary geometries by solving the Euler equations. They treated the intersection between the grid line and the boundaries as a grid point and performed conventional finite difference scheme with amendments of boundary fluxes. However, the small cell instability was observed. Pember et al. [12] proposed a corrector applied to cells at the boundary to redistribute flow quantities in order to maintain the conservation, therefore, avoid time step restrictions arising from small cells. In these methods, the amendments or the correctors depend on the specific physical system, thereby, the numerical scheme is difficult for further extension.

Ye et al. [13] and Udaykumar et al. [14] proposed a sharp interface Cartesian grid method for solving the incompressible Navier–Stokes equations with complex moving boundaries. The boundaries cut the mesh, then form some trapezoidal cells. If the trapezoidal cell is very small, then it merges with its neighboring cell, and forms an irregular control volume. The finite volume method can be applied on the merging control volume. Then the small cell problem is circumvented. They proposed an interpolation procedure to construct the flow field near the boundary, so that the boundary condition is satisfied. Tullio et al. [15] and Palma et al. [16] combines the method for solving the three-dimensional preconditioned Navier–Stokes equations for compressible flows with an immersed boundary approach, to provide a Cartesian-grid method for computing complex flows over a wide range of the Mach number. Moreover, a flexible local grid refinement technique is employed to achieve high resolution near the immersed body and in other high-flow-gradient regions.

Forrer and Jeltsch [17] proposed a ghost cell method to apply the boundary condition on Cartesian grid for inviscid compressible Euler equations. The flow quantities at ghost cells which locate inside the boundary are set by their images in the fluid side across the boundary. As a result, the grid points near the boundary also preserve a complete control volume. Thus the small cell restriction is removed. Hereafter, this kind of method is referred to as the boundary interpolation method. In recent years, the boundary interpolation method receives more and more attentions. Udaykumar et al. [3] proposed a finite-difference formulation to track solid-liquid boundaries on a fixed underlying grid. The interface is not of finite thickness