ARTICLE IN PRESS

Computers and Mathematics with Applications ▮ (▮▮▮▮) ▮▮▮–▮▮▮

Contents lists available at ScienceDirect

Computers and Mathematics with Applications

journal homepage: www.elsevier.com/locate/camwa

An implicit gas-kinetic scheme for turbulent flow on unstructured hybrid mesh

Dongxin Pan, Chengwen Zhong *, Congshan Zhuo

National Key Laboratory of Science and Technology on Aerodynamic Design and Research, Northwestern Polytechnical University, Xi'an, Shaanxi 710072, China

ARTICLE INFO

Article history:
Received 17 September 2017
Received in revised form 19 December 2017
Accepted 11 February 2018
Available online xxxx

Keywords:
Implicit method
Gas-kinetic scheme
LU-SGS scheme
Spalart–Allmaras model
Unstructured hybrid mesh
Turbulent flow

ABSTRACT

In this study, an implicit scheme for the gas-kinetic scheme (GKS) on the unstructured hybrid mesh is proposed. The Spalart–Allmaras (SA) one equation turbulence model is incorporated into the implicit gas-kinetic scheme (IGKS) to predict the effects of turbulence. The implicit macroscopic governing equations are constructed and solved by the matrix-free lower-upper symmetric-Gauss–Seidel (LU-SGS) method. To reduce the number of cells and computational cost, the hybrid mesh is applied. A modified non-manifold hybrid mesh data(NHMD) is used for both unstructured hybrid mesh and uniform grid. Numerical investigations are performed on different 2D laminar and turbulent flows. The convergence property and the computational efficiency of the present IGKS method are investigated. Much better performance is obtained compared with the standard explicit gas-kinetic scheme. Also, our numerical results are found to be in good agreement with experiment data and other numerical solutions, demonstrating the good applicability and high efficiency of the present IGKS for the simulations of laminar and turbulent flows.

1. Introduction

Over the past half-century, the gas-kinetic theory has been extensively studied for incompressible and compressible fluid flows. As an alternative method based on the Bhatnagar–Gross–Krook (BGK) collision model, the gas kinetic scheme (GKS) proposed by Xu and Prendergast [1] has been rapidly developed in the last two decades and has attracted an increasing amount of attention from the CFD community. Through new modifications as reported in the literature [2], the revised GKS scheme is of a certain advantage of resolving dissipative structure, especially in capturing shock waves. In the traditional Navier–Stokes (NS) solvers, the gas is assumed to stay in two equilibrium states on both sides of the shock wave and the shock wave appears as a discontinuity; that is to say, the physical dissipation in a cell size is replaced by numerical one. In the flux reconstruction approach, an additional artificial dissipation is also introduced by NS solver due to application of upwind scheme and/or central difference method. To remove the spurious dissipation, a general non-equilibrium state is considered for constructing the initial gas distribution function at each time step and the equilibrium state at a cell interface utilizing the Chapman–Enskog expansion [2]. In the complex flow simulations, the GKS can give us more real description from the aspect of physical evolution than traditional NS solvers.

In practice, as reported by May et al. [3], the flux evaluation in GKS is slightly more complicated compared with that generally used in the other finite volume method (FVM) solvers, which means that GKS takes more computational cost than other FVM solvers under the same situation. So, in practical applications, acceleration methods for GKS must be developed. To

* Corresponding author.
E-mail addresses: chungou@mail.nwpu.edu.cn (D. Pan), zhongcw@nwpu.edu.cn (C. Zhong), zhuocs@nwpu.edu.cn (C. Zhuo).

https://doi.org/10.1016/j.camwa.2018.02.032

reduce the computational time, the parallel implementation is a good strategy. Ilgaz and Tuncer [4] investigated the parallel implementation of the gas-kinetic BGK scheme on unstructured grids by using the domain decomposition method. Kumar et al. [5] proposed a parallelized WENO-enhanced GKS for the three-dimensional (3D) direct simulations of compressible transitional and turbulent flows.

On the other hand, for the steady flow simulations concerned in this study, using implicit scheme is an efficient way to accelerate convergency and reduce computational cost. Chit et al. [6] applied an approximate factorization and alternating direction-implicit (AF-ADI) method for the GKS simulations of the inviscid compressible flow on structured grids, in which an improved algorithm was achieved with fast convergence and the capability for using a large Courant–Friedrichs–Lewy (CFL) number, and this implicit method exhibited better results than the traditional explicit method. Recently, Li et al. [7] presented an implicit gas-kinetic method with the matrix-free lower-upper Symmetric Gauss–Seidel (LU-SGS) time marching implicit scheme to simulate the hypersonic inviscid flows on unstructured mesh, in which the implicit CFL number was determined by decreasing residual and the good robustness of this kind of implicit gas-kinetic scheme (IGKS) was validated in their works. However, the convergent behavior was not shown with the residual curve in their works. As mentioned above, several kinds of IGKS have been applied for compressible flow simulations [6,7]. For unsteady flows, a dual time-stepping strategy of gas-kinetic scheme is proposed by Li et al. [8]. In their works, fluid flows from laminar to turbulent and from incompressible to compressible are accurately simulated with better efficiency than previous method. However, there were few studies on implementing IGKS on unstructured hybrid grids for the compressible turbulent flows around/in complex geometries. This study focus on presenting an IGKS with the LU-SGS scheme on unstructured hybrid grids for complex turbulent flows, and the convergent behavior of IGKS is also investigated in detail.

For the simulations of compressible flow around/in complex geometries, the unstructured hybrid grid becomes a promising choice, due to its advantage in balancing the accuracy and the computational cost. For example, for flow around complex realistic configuration, the body fitted grid can be used to resolve the boundary layer region, while the unstructured grid with a suitable growth rate can be applied to fill all other computational domain. Besides, the grid refinement and coarsening are quite easy to be implemented [9], compared to the structured or block-structured grid technique which relies on regular connectivity of quadrilateral or hexahedral cells. In problems with complex configurations, high-order finite volume method under unstructured grids can also obtain more elaborate and precise results [10]. In compressible cases, limiter for unstructured grid is easy to implement [11]. As we known, the complexity of hybrid grids, including elements, edges, nodes, and connectivity, needs an efficient mesh data structure to reduce the extra computational costs. In this study, the extended non-manifold hybrid mesh data (NHMD) [12,13] is employed to build an accessible library for mesh.

Most flows encountered in engineering applications are of turbulent nature. For the compressible turbulent flow simulations [14], compared to the direct numerical simulation (DNS) and the large eddy simulation (LES), the turbulence models require coarse grid resolution and have been validated to be suitable for the turbulent flow simulations in engineering. In particular, the one-equation Spalart–Allmaras (SA) model [15] became quite popular because of its satisfactory results for a wide range of flow problems and its reliable numerical properties. On the basis of the explicit GKS, we have successfully carried out the simulation of compressible turbulent flows with shock waves on unstructured meshes, in which the SA turbulence model [13] and SST turbulence model [8] are incorporated to include the effect of turbulence. In this study, the SA turbulence model is also selected on account of its good accurate and stability under relatively coarse grid near the wall, as well as its low computational cost. Other choices, such as the two-equation models, might also be good candidates, which will be developed in the future.

In this paper, an IGKS coupled with SA turbulence model is proposed for the incompressible and compressible flow simulations on unstructured hybrid mesh, and several flow simulations including the lid-driven cavity flow, the laminar and the turbulent flow around a flat plate, the turbulent flow around a multi-element airfoil, as well as the turbulent transonic flow over a NACA0012 airfoil are performed here.

The rest of the paper is organized as follows. The basic GKS proposed by Xu and Prendergast [1], the matrix-free LU-SGS scheme and the implementation of IGKS, the expanded NHMD structure and the coupled SA turbulence model, as well as four different boundary conditions are described in Section 2. Then, the numerical validations and several turbulent flow cases are carried out to show the accuracy and reliability of the present IGKS–NHMD method in Section 3. Finally, some remarks concluded from this study are grouped in Section 4.

2. Implicit method for gas-kinetic scheme

2.1. Gas-kinetic scheme

The two-dimensional (2D) gas-kinetic scheme based on BGK model is written as [1]

$$\frac{\partial f}{\partial t} + u\frac{\partial f}{\partial x} + v\frac{\partial f}{\partial y} = \frac{g-f}{\tau}, \tag{1}$$

where f and g are the gas distribution functions, which are the functions of space (x, y), particle velocity $\vec{u} = (u, v)$, time t, and internal variable ξ. τ is the particle collision time. g is the Maxwell distribution function which has the following form

$$g = \rho\left(\frac{\lambda}{\pi}\right)^{\frac{K+2}{2}} e^{-\lambda\left((u-U)^2+(v-V)^2+\xi^2\right)}, \tag{2}$$

Please cite this article in press as: D. Pan, et al., An implicit gas-kinetic scheme for turbulent flow on unstructured hybrid mesh, Computers and Mathematics with Applications (2018), https://doi.org/10.1016/j.camwa.2018.02.032.