Journal of Computational Physics 300 (2015) 492–519

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Explicit formulations of gas-kinetic flux solver for simulation of incompressible and compressible viscous flows

Y. Sun [a], C. Shu [a,*], C.J. Teo [a], Y. Wang [a], L.M. Yang [b]

[a] Department of Mechanical Engineering, National University of Singapore, 10 Kent Ridge Crescent, Singapore 119260, Singapore
[b] Department of Aerodynamics, College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing 210016, China

ARTICLE INFO

Article history:
Received 12 December 2014
Received in revised form 22 April 2015
Accepted 29 July 2015
Available online 3 August 2015

Keywords:
Gas-kinetic flux solver
Maxwellian distribution function
Viscous flow
Streaming time

ABSTRACT

In this paper, a gas-kinetic flux solver (GKFS) is presented for the simulation of incompressible and compressible viscous flows. In this solver, the finite volume method is applied to discretize the Navier–Stokes equations. The inviscid and viscous fluxes at the interface are obtained simultaneously via the gas-kinetic scheme, which locally reconstruct the solution for the continuous Boltzmann equation. Different from the conventional gas-kinetic BGK scheme [1], a simple way is presented in this work to evaluate the non-equilibrium distribution function, which is calculated by the difference of equilibrium distribution functions at the cell interface and its surrounding points. As a consequence, explicit formulations for computing the conservative flow variables and fluxes are simply derived. In particular, three specific schemes are proposed and validated via several incompressible and compressible test examples. Numerical results show that all three schemes can provide accurate numerical results for incompressible flows. On the other hand, Scheme III is much more stable and consistent in simulation of compressible flows.

1. Introduction

In conventional computational fluid dynamics (CFD), Euler equations are usually used to simulate inviscid flows while Navier–Stokes equations are commonly used for simulation of viscous flows. Various numerical methods have been developed to solve these macroscopic governing equations [2–4], among which the finite volume method (FVM) is one of the most popular approaches. In the FVM, the discrete forms of governing equations usually involve the conservative variables at cell centers and numerical fluxes at cell interfaces. In the solution process, the numerical fluxes at the cell interfaces are needed to be constructed from the conservative variables at the cell centers.

Currently, there are three major approaches to evaluate the numerical fluxes in CFD. The first one is based on the smooth function approximation. This is a mathematical approach. In this method, a smooth function, such as polynomial approximation [5], is applied to approximate the solution. The related coefficients in the polynomial approximation are determined by collocation method. However, this solver is not applicable to problems with discontinuities. To resolve discontinuity problems, the Riemann solver and approximate Riemann solvers, which can be considered as the second one, are extensively used in the last few decades. Godunov [6] firstly presented the first-order upwind scheme for the hyperbolic conservation laws in 1959. After that, numerous approximate Riemann solvers were developed [7–12]. In general, the approximate Riemann solvers can only evaluate the inviscid flux and it is still challenging to directly solve the viscous compressible flows.

* Corresponding author.
E-mail address: mpeshuc@nus.edu.sg (C. Shu).

http://dx.doi.org/10.1016/j.jcp.2015.07.060
0021-9991/

One of the popular ways is to treat the convective terms and dissipative terms separately, which means that the inviscid flux is evaluated by the approximate Riemann solvers and the viscous flux is calculated by the smooth function approximation.

Another type of flux solver is called the gas-kinetic scheme, which locally applies the Boltzmann equation to evaluate inviscid and viscous fluxes simultaneously. There are two common types of gas-kinetic schemes: the kinetic flux vector scheme (KFVS) and gas-kinetic Bhatnagar–Gross–Krook (BGK) scheme. In KFVS, the collision term vanishes in the Boltzmann equation. There are two stages in KFVS: free transport and collision. In the free transport stage, the collisionless Boltzmann equation is solved to calculate the flux at the interface. In the collision stage, the artificial collisions are added in the calculation of initial Maxwellian distribution at the beginning of next time step. Owing to the large dissipation introduced, the KFVS performs stably for strong shock waves and expansion waves [13]. However, it is demonstrated in [1] that the KFVS scheme usually gives more diffusive results than the Godunov or flux difference splitting (FDS) scheme because the numerical dissipation in KFVS is proportional to the mesh size. Thus, the KFVS is not able to give accurate NS solutions except for cases in which the physical viscosity is much larger than the numerical viscosity. Some of representative researches on the KFVS include Pullin [14], Perthame [15], Mandal and Deshpande [16], and Chou and Baganoff [17].

The gas-kinetic BGK scheme was firstly proposed by Prendergast and Xu [18] in 1993 and then developed gradually afterwards [1,19]. In this method, the BGK collision model is adopted in the gas evaluation stage to obtain the numerical fluxes across the interface. As a consequence, the dissipation in the transport can be controlled by a real collision time, which is a function of dynamic viscosity and pressure. In contrast to conventional upwind schemes, the gas-kinetic BGK scheme computes the inviscid and viscous fluxes simultaneously from the solution of Boltzmann equation with collision term. In the work of Xu [1], it has been shown that the gas-kinetic BGK scheme is able to generate a stable and crisp shock transition in the discontinuous region with a delicate dissipative mechanism. At the same time, an accurate Navier–Stokes solution can be obtained in the smooth region. Moreover, it is demonstrated that the entropy condition is always satisfied in the gas-kinetic BGK scheme and the "carbuncle phenomenon" is avoided for hypersonic flow simulations [20]. Owing to the advantages of gas-kinetic BGK scheme, it has attracted more and more attention and has been applied to various flow problems [21–26]. Recently, Xu and Huang [27] proposed a unified gas-kinetic scheme which can perform both continuum and rarefied flow computations with discretized particle velocity space.

In spite of these advantages, the gas-kinetic BGK scheme also suffers from some drawbacks. It is usually more complicated and inefficient than conventional CFD schemes. To be more specific, numerous terms and coefficients associated with non-equilibrium distribution functions should be calculated to yield the numerical fluxes at each interface and each time step. Moreover, it is an arduous task to get the explicit expression for the numerical flux. Owing to the complexity and low computational efficiency, the gas-kinetic BGK scheme is a bit difficult for new users to implement. In the literature, several works have been done to simplify this scheme. Chae et al. [28] abandoned a time evolution term in the integral solution of BGK model and they claimed that the computational efficiency and convergence were improved. May et al. [29] proposed two modifications to the conventional gas-kinetic BGK scheme. At first, they proposed a new formulation to the calculation of the initial non-equilibrium terms in the consideration of relaxation state. A new time derivative was also introduced to reduce the CPU time. In the work of Tang [30], the spatial derivative of the equilibrium distribution function across the interface is assumed to be continuous rather than piecewise linear used in the conventional scheme. All the time-related terms were not considered in the calculation of flux. There are some other efforts to improve the accuracy of gas-kinetic BGK scheme, including DG method [31,32], Runge–Kutta method [33] and so on.

Most of the above modifications are to improve the original gas-kinetic BGK scheme, where the non-equilibrium distribution function is approximated by a low order polynomial in terms of time, physical space and phase velocity space. In this way, many terms associated with phase velocity, space coordinate and time should be considered. The simplification in these terms may add more uncertainty to the derivation of the gas-kinetic BGK scheme. This motivates the present work. We aim to simplify the original gas-kinetic BGK scheme while keep its intrinsic advantages at the same time.

In the present work, the non-equilibrium distribution function will be approximated by a simple way. At any cell interface, the distribution function can be expressed as $f = f^{eq} + f^{neq}$, where f^{eq} is the equilibrium distribution function and f^{neq} is the non-equilibrium distribution function. The equilibrium distribution function f^{eq} can be simply calculated by the conservative variables using Maxwellian distribution function. According to the compatibility condition, the non-equilibrium distribution function f^{neq} has no contribution to the conservative variables [34], but it has effect on the numerical fluxes. With Chapman–Enskog expansion analysis, to recover Navier–Stokes equations, f^{neq} can be approximated by $-\tau(g_t + ug_x + vg_y)$, where τ is the collision time, g is the equilibrium distribution function f^{eq}, and subscripts represent the derivative. In the conventional gas-kinetic schemes, f^{neq} is approximated by a low order polynomial and g, which involves many coefficients. It is indeed that determination of those coefficients and their implementation into the gas-kinetic scheme make the solver be complicated. Recently, it was found that f^{neq} can be approximated by the difference of the equilibrium distribution functions at the cell interface and its surrounding points with a small streaming step. As the equilibrium distribution function at the surrounding points can be easily given by the conservative variables at cell centers through interpolation, the equilibrium distribution function at the cell interface can be obtained from a streaming process at surrounding points. Thus, f^{neq} can be explicitly calculated. Then, the numerical fluxes can be computed without involving any coefficients. The above idea was first applied in the development of lattice Boltzmann flux solver (LBFS) [35–38]. It was shown that LBFS can be accurately and efficiently applied to solve many fluid flows. However, due to intrinsic limitation of lattice Boltzmann method, LBFS is limited to the simulation of incompressible flows. To simulate compressible flows, Yang et al. [13,39] extended the idea to the circular function-based gas-kinetic scheme. In this scheme, all the particles are as-