Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **8**, No. 6, pp. 887-910

DOI: 10.4208/aamm.2015.m1172
December 2016

A Hybrid Lattice Boltzmann Flux Solver for Simulation of Viscous Compressible Flows

L. M. Yang[1,2], C. Shu[3,*] and J. Wu[2]

[1] *State Key Laboratory of Aerodynamics, China Aerodynamics Research and Development Center, Mianyang 621000, China*
[2] *Department of Aerodynamics, College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, Yudao Street, Nanjing 210016, China*
[3] *Department of Mechanical Engineering, National University of Singapore, 10 Kent Ridge Crescent, Singapore 119260, Singapore*

Received 26 June 2015; Accepted (in revised version) 8 September 2015

Abstract. In this paper, a hybrid lattice Boltzmann flux solver (LBFS) is proposed for simulation of viscous compressible flows. In the solver, the finite volume method is applied to solve the Navier-Stokes equations. Different from conventional Navier-Stokes solvers, in this work, the inviscid flux across the cell interface is evaluated by local reconstruction of solution using one-dimensional lattice Boltzmann model, while the viscous flux is still approximated by conventional smooth function approximation. The present work overcomes the two major drawbacks of existing LBFS [28–31], which is used for simulation of inviscid flows. The first one is its ability to simulate viscous flows by including evaluation of viscous flux. The second one is its ability to effectively capture both strong shock waves and thin boundary layers through introduction of a switch function for evaluation of inviscid flux, which takes a value close to zero in the boundary layer and one around the strong shock wave. Numerical experiments demonstrate that the present solver can accurately and effectively simulate hypersonic viscous flows.

AMS subject classifications: 76M12

Key words: Lattice Boltzmann flux solver, hybrid, 1D lattice Boltzmann model, switch function.

1 Introduction

As fast development of computer hardware and numerical approaches, computational fluid dynamics (CFD) plays a more and more important role in industrial applications. The CFD is to apply a numerical method to solve governing equations (Navier-Stokes

*Corresponding author.
Email: mpeshuc@nus.edu.sg (C. Shu), yangliming_2011@126.com (L. M. Yang)

equations) on the computer. Currently, there are a number of numerical methods available [1–17]. Among them, the finite volume method (FVM) [7] is widely used. The application of FVM is in line with application of physical conservation laws to a control volume. Thus, its major advantage is to keep numerical conservation of physical quantities. It is also flexible and suitable for solving problems with complex geometry. The key issue in applying FVM is to develop an appropriate numerical scheme, which is also known as flux solver, for evaluation of inviscid and viscous fluxes at cell interface.

In the conventional application of FVM to simulate viscous compressible flows, evaluation of inviscid flux and viscous flux is made by different ways. The viscous flux is evaluated by using a smooth function approximation such as polynomial approximation, while inviscid flux is computed by using various upwind schemes such as Roe scheme [10], van Leer scheme [11], and AUSM (Advection Upstream Splitting Method) scheme [12]. Roe scheme is widely used in simulation of compressible viscous flows due to its high accuracy for boundary layers and good resolution for shock waves. However, for the simulation of hypersonic flows, the Roe scheme often exhibits carbuncle phenomenon and induces numerical instability [13]. Van Leer scheme has a good performance in solving Euler equations, but may smear out the solution of boundary layers and also lead to inaccurate stagnation and wall temperature for simulation of viscous compressible flows [14].

An alternative approach for evaluation of fluxes is the Boltzmann equation-based flux solver, which is also known as gas kinetic scheme. Different from the conventional CFD approaches, the Boltzmann equation-based schemes evaluate fluxes by local reconstruction of solution for the Boltzmann equation at a cell interface. This kind of scheme can be used to simulate both incompressible and compressible flows. Kinetic Flux Vector Splitting (KFVS) scheme [15] and gas-kinetic Bhatnagar-Gross-Krook (BGK) scheme [16,17] are the commonly-used Boltzmann equation-based flux solvers. Basically, for the KFVS scheme, the collisionless Boltzmann equation is solved in the gas evolution stage, and the collision process is controlled by a numerical time step. As a result, the KFVS scheme usually gives poorer results than those obtained from Roe scheme [10] and AUSM scheme [12]. Unlike KFVS scheme, the gas-kinetic BGK scheme considers the particle collisions during the gas evolution stage by the BGK model. As a consequence, the dissipation in the streaming process is controlled by the collision time rather than by the numerical time step. Numerical results showed that the gas-kinetic BGK scheme can accurately simulate both inviscid and viscous flows [17, 18]. On the other hand, as far as we know, most of existing gas kinetic schemes are based on the Maxwellian distribution function [15–18]. Due to complexity of the Maxwellian function, these schemes are usually more complicated and less efficient than the traditional numerical schemes [10–12].

Recently, lattice Boltzmann method (LBM) [19, 20] receives more and more attention due to its kinetic nature, simplicity, and easy implementation. However, the conventional LBM is only limited to the simulation of incompressible flows since its equilibrium distribution function is approximated from truncated Taylor series expansion of Maxwellian function in terms of Mach number. To simulate compressible flows by LBM, one has to