Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **11**, No. 1, pp. 72-90

DOI: 10.4208/aamm.OA-2018-0167
February 2019

An Implicit Block LU-SGS Algorithm-Based Lattice Boltzmann Flux Solver for Simulation of Hypersonic Flows

Z. X. Meng[1], S. Z. Li[2,*], K. Peng[1] and W. H. Zhang[1]

[1] *College of Aerospace Science and Engineering, National University of Defense Technology, Changsha 410073, China*
[2] *National Innovation Institute of Defense Technology, Academy of Military Science, Beijing 100091, China*

Received 17 July 2018; Accepted (in revised version) 3 September 2018

Abstract. This paper proposes a stable and efficient implicit block Lower-Upper Symmetric-Gauss-Seidel (LU-SGS) algorithm-based lattice Boltzmann flux solver (LBFS) for simulation of hypersonic flows. In this method, the finite volume method (FVM) is applied to discretize the Navier-Stokes equations, and the LBFS is utilized to evaluate the numerical flux at the cell interface. In LBFS, the local solution of discrete velocity Boltzmann equation (DVBE) with the non-free parameter D1Q4 lattice Boltzmann model is adopted to reconstruct the inviscid flux across the cell interface, and the viscous flux is approximated by conventional smooth function approach. In order to improve the robustness and convergence rate of the simulation for hypersonic flows, especially for problems with complex geometry, the implicit block LU-SGS algorithm is introduced to solve resultant discrete governing equations. A double cone model at Mach number of $Ma=9.86$ is firstly simulated to validate the proposed scheme, and a hypersonic flight vehicle with wings and rudders at Mach number of $Ma=5.56$ is then calculated to extend the application in practical engineering problems. Numerical results show that the proposed scheme could offer a more accurate and effective prediction for hypersonic flows.

AMS subject classifications: 76M12

Key words: Hypersonic flows, lattice Boltzmann flux solver, implicit block LU-SGS, finite volume method.

1 Introduction

The computational fluid dynamics (CFD) plays an important role in solving flow field problems due to the prosperous development in numerical algorithms and computa-

*Corresponding author.
Emails: lishengze12@gmail.com (S. Z. Li), mengzhuxuan08@nudt.edu.cn (Z. X. Meng)

tional facility. CFD provides lower cost than wind tunnel experiment and higher accuracy than engineering approximation method, which brings it widely industrial application. The finite volume method (FVM) is one of the most popular numerical methods in CFD due to its good numerical conservation properties and suitability in solving flow problems with complex geometry [1–3]. The key of FVM is to construct a flux solver to evaluate the numerical flux at the cell interface. Godunov [4] simplified the Euler equations to a Riemann problem firstly in 1959 and proposed an accurate Riemann solver for calculating the inviscid numerical flux. Subsequently, varieties of approximate Riemann solvers [5–9] were developed to improve the efficiency of the original one. Among them, the upwind schemes such as Roe scheme [5], AUSM (Advection Upstream Splitting Method) scheme [7] and van Leer scheme [8] have been widely used in evaluating the numerical flux for simulating the compressible flows. However, some of them often exhibit a carbuncle phenomenon and induce numerical instability during the hypersonic simulation [10].

Boltzmann equation-based flux solver is another popular method for calculation of the numerical flux. Different from the conventional CFD approaches, which only compute the flux by numerical approximation, the Boltzmann equation-based schemes evaluate the flux through a local reconstruction with the solution for Boltzmann equation at the cell interface. This feature makes the computation of Boltzmann equation-based schemes be robustness and effectively prevent unphysical solutions. One of the representative Boltzmann equation-based flux solvers is the gas-kinetic scheme (GKS) [11–13], which calculates the numerical flux by the local solution of continuous Boltzmann equation. In the works of Xu and his coworkers [14–17], the local integral solution of Boltzmann equation with Maxwellian distribution function is utilized to reconstruct the numerical flux. To simplify the Maxwellian function-based GKS [14–17], the circular function-based GKS (CGKS) for two-dimensional cases [18–20] and the sphere function-based GKS (SGKS) for three-dimensional cases [21–23] have been developed recently by Shu and his coworkers. Another representative Boltzmann equation-based flux solver is the lattice Boltzmann flux solver (LBFS), which adopts the local solution of discrete velocity Boltzmann equation (DVBE) with lattice Boltzmann model to reconstruct the numerical flux at the cell interface [24–29]. By using the discrete model, the LBFS is more convenient for both mathematical derivation and coding. It has been proven that the LBFS can provide an accurate and efficient prediction for both incompressible and compressible flows [28,29]. Nowadays, more applications such as turbomachinery flows [30] and multi-component flows [31] are implemented by LBFS, which make it becoming a popular solver. However, the application of LBFS in hypersonic flows, especially for complex geometry, is still rarely studied. Therefore, a further endeavor is needed to extend the application of LBFS to hypersonic flows.

In the simulation of hypersonic flows, the extreme low-pressure area usually occurrs behind the object [32,33]. This phenomenon may slow down the convergence rate significantly or even produce a divergent result with negative density and pressure, especially for complex geometry. To overcome this defect, during the hypersonic simulation, the