Development of lattice Boltzmann flux solver for simulation of hypersonic flow past flight vehicles

Zhuxuan Meng1,3, Liming Yang1, Chang Shu2, Fan Hu1, Donghui Wang1 and Weihua Zhang1

1College of Aerospace Science and Engineering, National University of Defense Technology, Changsha, 410073, China
2Department of Mechanical Engineering, National University of Singapore, 10 Kent Ridge Crescent, Singapore 119260, Singapore
3Email: ml3687361976@163.com

Abstract. A stable Lattice Boltzmann Flux Solver (LBFS) is proposed for simulation of complex hypersonic flows in this paper. In LBFS, the finite volume method is applied to solve the Navier-Stokes equations. One-dimensional Lattice Boltzmann (D1Q4) model is applied to reconstruct the inviscid flux across the cell interface, while the viscous flux is solved by central difference method. A switch function is applied to control the numerical dissipation to enhance computational robustness. Also a local block grid refinement method based on the flow conditions is applied to improve calculation stability. A double ellipse model is studied to validate the accuracy and feasibility of the proposed scheme. Simulation results show that the method presented in this paper is able to calculate hypersonic flow field and the strong shock wave interaction can be well described.

1. Introduction

Hypersonic vehicle has been studied as a hotspot for more than 70 years since 1950s. It is popular because of its fine properties, such as fast arriving, able to adapt flying environment, reusable and etc. However, hypersonic flows are more difficult to simulate than low speed flows because [1]: (1) strong shock effect can produce fierce compression behind the shock wave, (2) strong viscosity effect has influence on the whole flows, (3) low density effect in high altitude will result in continuous medium assumption no longer appropriate. Hence, traditional CFD methods are not suitable for high altitude rare atmosphere because they are derived from the continuity hypothesis. Lattice Boltzmann Flux Solver is an ideal approach for simulating rarefied flows of high altitude rely on its well performance in computing ordinary flows [2-8].

Lattice Boltzmann Flux Solver (LBFS) is developed from Boltzmann equation-based method, including DVBE [9-11] (discrete velocity Boltzmann equation) and gas-kinetic schemes [12-15]. Gas-kinetic schemes are more effective than DVBE. However, both of them are less efficient and more complicated than conventional Navier-Stokes methods. LBFS is more efficient and easy to apply than DVBE and gas-kinetic method due to only 1-D Lattice Boltzmann model is applied (the inviscid flux at cell interface is reconstructed by local solution of 1-D compressible Lattice Boltzmann model) and macroscopic governing equations are N-S equations (discretized by FVM). Due to 1-D model is applied along normal direction at cell interface, the tangential effect cannot be computed properly. From Chapman-Enskog expansion analysis [16], the inviscid flux can be fully determined by
equilibrium part, and the non-equilibrium part which contributes to the viscous flux can be treated as numerical dissipation. For compressible viscous flows, especially for hypersonic flows, the numerical dissipation should be controlled. Also, physical values such as pressure in leeward side can be extremely low in hypersonic flow. Hence it will be easy to get a negative value in calculation process, which leads to a divergent result.

In this work, a stable LBFS framework is proposed for simulation of 2-D compressible hypersonic viscous flows, in which shock waves interaction occurs to make the flow field more complex and difficult to compute. The inviscid flux is calculated by LBFS and viscous flux is computed by smooth function approximation [17-18]. Multi block grids and local grid refinement technique are applied to ensure high grid quality to improve calculation stability. Also a switch function is applied to control the numerical dissipation. Besides, the implicit LU-SGS method is used to speed up the convergent rate. In the end, a double ellipse model in Ma=5.0 flow field is computed to validate the developed LBFS. The results are compared with experimental data, Van Leer and Roe schemes to validate the accuracy and feasibility proposed in this article.

2. Methodology
In LBFS, Navier-Stokes equations are solved in macroscopic scale and local solution of Lattice Boltzmann equations are applied to reconstruct inviscid flux solver at the cell interface. It has been proved that particle potential energy is independent from lattice velocity [6]. Lattice velocity would be decided by higher momentum conservation relation in this work instead of artificial selection in traditional method.

The integral of fluxes in N-S equations can be discretized by FVM and the approximated summation form can be written as

$$\frac{dW_I}{dt} = -\frac{1}{\Omega_I} \sum_{I'} (F_{i'} - F_i)S_i \tag{1}$$

in which I represents the control volume index, $\Omega_I$ is the volume, $N_I$ is number of faces of control volume I and $S_I$ is the area of the interface in this volume. Here, the inviscid flux $F_i$ in equation (1) will be solved by LBFS with non-free parameter D1Q4 model [4] and viscous flux $F_i$ will be solved by central difference method.

2.1. Non-free parameter D1Q4 lattice Boltzmann model
The distribution of discrete lattice velocities for D1Q4 model is shown in figure 1. This model contains 4 equilibrium distribution functions $f_i^{eq}$, $f_2^{eq}$, $f_3^{eq}$, $f_4^{eq}$ and 2 lattice velocities $d_1$, $d_2$ (shown as equation (2)). The derivation process details are shown in references [4].

![Figure 1. Distribution of discrete lattice velocities for D1Q4 model.](image)

$$f_i^{eq} = \rho \left( -d_i^2 d_1^2 u + d_i u^2 + d_i^2 c^2 + u^3 + 3uc^2 \right), \quad f_2^{eq} = \rho \left( -d_2^2 d_1^2 u + d_2 u^2 + d_2^2 c^2 + u^3 - 3uc^2 \right)$$

$$f_3^{eq} = \rho \left( d_1^2 d_2^2 u - d_1 u^2 - d_2 c^2 - u^3 - 3uc^2 \right), \quad f_4^{eq} = \rho \left( d_1^2 d_2^2 u - d_1 u^2 - d_2 c^2 + u^3 + 3uc^2 \right) \tag{2}$$

$$d_1 = \sqrt{u^2 + 3c^2 - \sqrt{4u^2 c^2 + 6c^4}}, \quad d_2 = \sqrt{u^2 + 3c^2 + \sqrt{4u^2 c^2 + 6c^4}}$$
in which \( c = \sqrt{\frac{Dp}{\rho}} \) represents particular velocity of particles and \( D \) is space dimension. It has been proved by Yang [4] that physical conservation laws (equation (3)) can revivify the N-S equations by applying relations in equation (2). \( \xi_i \) means particle velocity in \( i \)-direction, for example \( \xi_1 = d_1, \xi_2 = -d_1, \xi_3 = d_2 \) and \( \xi_4 = -d_2 \). For higher dimensional problems, the D1Q4 model should be applied along the normal direction of cell interface [4] shown in figure 2 (2D case). The normal velocity \( U_n \) and tangential velocity \( U_t = (u_{ix}, u_{iy}, u_{iz}) = \mathbf{u} - U_n \mathbf{n} \) in figure 2 will replace \( u \) in equation (3).

Finally, we will get \( u = U_n n_x + u_{ix} \).

\[
\rho = \sum_{i=1}^{4} f_{i0}^\rho, \quad \rho u = \sum_{i=1}^{4} f_{i0}^\rho \xi_i \rho u^2 + \rho c^2 = \sum_{i=1}^{4} f_{i0}^\rho \xi_i \rho u^2 + 3\rho pc^2 = \sum_{i=1}^{4} f_{i0}^\rho \xi_i \rho \xi_i
\]  

(3)

\[
\begin{aligned}
\mathbf{F}_t &= U_t \\
\mathbf{U}_n &= \mathbf{u} \\
\mathbf{n} &= \text{interface}
\end{aligned}
\]

Figure 2. Application of 1D model to 2D case.

2.2. Inviscid flux model

Suppose that cell interface is located at \( x_{c,j+1/2} = 0 \), then the inviscid flux at interface \( \mathbf{F}_c \) is decided by normal velocity and can be written as equation (4). In which the moments \( \varphi_a = \left[ 1, \xi_1^2 / 2 + \epsilon_p \right] \) at \( f_i(0, t) \) is the distribution function at cell interface. Generally, \( f_i(0, t) \) is summation of equilibrium part \( f_i^{eq}(0, t) \) and non-equilibrium part \( f_i^{non}(0, t) \).

\[
\mathbf{F}_c = \left[ \begin{array}{c}
\rho U_n \\
\rho U_n^2 + p \\
(\rho U_n^2 / 2 + \epsilon_p) + p \rho U_n
\end{array} \right] = \sum_{i=1}^{4} \xi_i \varphi_a f_i(0, t)
\]

(4)

To recover N-S equations by Boltzmann equation from Chapman-Enskog analysis [16, 19-22], the non-equilibrium part \( f_i^{non}(0, t) = -\tau \left( \frac{\partial f_i}{\partial t} + \xi_i \frac{\partial f_i}{\partial x} \right) \).

At the cell interface the equilibrium distribution function is \( f_i(0, t) = f_i^{eq}(0, t) \). \( f_i(-\xi_i \partial t, t - \delta t) \) is equilibrium distribution function at surrounding point of the cell interface. Then we can get \( f_i(0, t) = f_i^{eq}(0, t) - \tau_0 \left[ f_i(0, t) - f_i(-\xi_i \partial t, t - \delta t) \right] \). in which \( \tau_0 = \tau / \delta t \) is the dimensionless collision time and \( \delta t \) is the streaming time step and represents the physical viscous of N-S equations. The contribution of non-equilibrium part is always treated as numerical dissipation in LBFS. Therefore, \( \tau_0 \) can be regarded as the weight of numerical dissipation. Here we define a switch function as \( \tau_0 = \max \left( \tau^e, \tau^n \right) \). In which \( \tau^e = \max_{j=1,...,N_a} \left\{ \tau_j \right\}, \tau^n = \max_{j=1,...,N_a} \left\{ \tau_j \right\} \) and \( \tau_j = \tanh \left( \frac{p_p - p_j}{p_p + p_j} \right) \), in which \( C \) is an amplification factor and ranges from 1 to 100, we take it as 10 here. \( N_a \), \( N_b \) and \( p^l, p^r \) are the number of control volume and pressure value on left and right side of cell interface separately.

Total inviscid flux at cell interface considering tangential velocity contribution is shown as following.

\[
\mathbf{F}_c = \mathbf{F}_c^l + \tau_0 (\mathbf{F}_c^{\ll} - \mathbf{F}_c^l)
\]

(5)
in which \( F_{i}^{I} \) represents contribution of equilibrium distribution function \( f_{i}^{eq}(0,t) \) at cell interface and \( F_{i}^{II} \) means equilibrium distribution function \( f_{i}(-\xi, \delta t, t-\delta t) \) at surrounding point of the cell interface.

Supposing that a local Riemann problem is formed at cell interface, which is shown in figure 3. Hence, equilibrium distribution function \( f_{i}(-\xi, \delta t, t-\delta t) \) at surrounding point of cell interface can be decided by the location of \(-\xi, \delta t\), which is shown as equation. (6).

\[
f_{i}(-\xi, \delta t, t-\delta t) = \begin{cases}
  f_{i}^{L} & i = 1, 3 \\
  f_{i}^{R} & i = 2, 4
\end{cases}
\]  

(6)

Then \( F_{i}^{I} \) and \( F_{i}^{II} \) can be written as following by applying variables at cell interface, in which \( f_{i} = f_{i}(-\xi, \delta t, t-\delta t) \).

\[
F_{i}^{I} = \begin{bmatrix}
\rho U_{n} \\
(\rho U_{n}^{2} + p) n_{x} + \rho U_{x} U_{n} \\
(\rho U_{n}^{2} + p) n_{y} + \rho U_{y} U_{n} \\
(\rho (U_{n}^{2} / 2 + e) + p) U_{n} + \rho U_{n} U_{n}^{2}/2
\end{bmatrix}
\]

\[
F_{i}^{II} = \begin{bmatrix}
\sum_{i=1}^{4} \xi_{i} f_{i} \\
\sum_{i=1}^{4} \xi_{i} \xi_{i} f_{i} \cdot n_{x} + \sum_{i=1}^{4} \xi_{i} \xi_{i} f_{i} \cdot U_{n}^{H} \\
\sum_{i=1}^{4} \xi_{i} \xi_{i} f_{i} \cdot n_{y} + \sum_{i=1}^{4} \xi_{i} \xi_{i} f_{i} \cdot U_{n}^{H} \\
\sum_{i=1}^{4} \xi_{i} \xi_{i} \xi_{i} f_{i} \cdot n_{x} + \sum_{i=1}^{4} \xi_{i} \xi_{i} \xi_{i} f_{i} \cdot U_{n}^{H} \\
\sum_{i=1}^{4} \xi_{i} \xi_{i} \xi_{i} \xi_{i} f_{i} \cdot U_{n}^{H} / 2
\end{bmatrix}
\]  

(7)

In this work, central difference scheme is applied to solve the viscous flux and details are shown in reference [23-24]. Multi block grids are applied to refine local grid quality. Grids local refinement method is used for near wall surface and block interface. Besides, a constraint judgement is added in our code to avoid negative value in calculating process. Also implicit LU-SGS method [25] is applied to speed up rate of convergent and keep robustness. The solid wall is treated as adiabatic wall and outer flow field around solid body is treated as far field.

**Figure 3.** Streaming process based on D1Q4 model at cell interface.  
**Figure 4.** Measurements of double ellipse wind tunnel model (mm).

### 3. Numerical simulation

Double ellipse model is used to verify the present solver. The shape of double ellipse model is shown in figure 4. The outline is driven by two ellipse functions. equation. (8) characterizes the horizontal ellipse and equation. (9) is the vertical one [26]. The free stream has a pressure of \( P_{w} = 1.5 \text{MPa} \), temperature of \( T_{w} = 370 \text{K} \), per meter Reynolds number of \( \text{Re}_{m} = 3.15 \times 10^{7} \) and Mach number of \( \text{Ma}_{m} = 5.0 \).
\[
\left( \frac{x}{157.9} \right)^2 + \left( \frac{y}{39.47} \right)^2 = 1
\]

\[
\left( \frac{x}{92.11} \right)^2 + \left( \frac{y}{65.79} \right)^2 = 1
\]

Figure 5 and figure 6 show pressure and Mach number distribution at -5° angle of attack obtained by present solver in this article. Figure 7 and figure 8 are contours at 0° angle of attack. Figure 9 and figure 10 are at 5° angle of attack. It can be seen clearly that LBFS has the ability to capture strong shock waves and the interaction phenomenon exactly.

Figure 11 shows comparison of pressure coefficient distribution on double ellipse upper surface at -5° angle of attack computed by present LBFS, Roe scheme, Van Leer scheme and experimental data [26]. It can be seen that the result of present solver has better accordance with experimental data than Van Leer scheme, as well as Roe scheme. For results on the lower surface (figure 12) at the same angle of attack, all numerical methods have same tendency. Similar conclusions can be obtained according to figure 13-figure 16, LBFS performs better than Van Leer scheme and well agreed with experimental data. Therefore, the solver presented in this article shows both high computational accuracy and numerical stability. Hence, LBFS shows the potential of future industrial application in hypersonic vehicle research and design.

![Figure 5](image1.png) **Figure 5.** Pressure contours at -5° angle of attack.

![Figure 6](image2.png) **Figure 6.** Mach number contours at -5° angle of attack.

![Figure 7](image3.png) **Figure 7.** Pressure contours at 0° angle of attack.

![Figure 8](image4.png) **Figure 8.** Mach number contours at 0° angle of attack.

![Figure 9](image5.png) **Figure 9.** Pressure contours at 5° angle of attack.

![Figure 10](image6.png) **Figure 10.** Mach number contours at 5° angle of attack.
4. Conclusions
This paper presents a stable Lattice Boltzmann flux solver for simulation of 2-D hypersonic complex flows. It is a new realization in computing hypersonic flows with shock waves interaction. Multi block and region grids refinement technique provide high quality grids. The switch function helps control
numerical dissipation and improve computational stability. Besides, the implicit LU-SGS method is used to speed up the convergent speed.

Numerical simulation of a double ellipse model in $Ma=5.0$ is used to validate the developed solver. The results show a good computational accuracy and robustness which can be applied to the industrial hypersonic vehicle research and design. Further, some new models will be tested in the future.

References

[1] Anderson J D 1989 Hypersonic and High Temperature Gas Dynamics McGraw-Hill
[2] Yang L M, Shu C and Wu J 2016 A hybrid lattice Boltzmann flux solver for simulation of viscous compressible flows Adv. Appl. Math. Mech. 8(6) 887-910
[3] Yang L M, Shu C and Wu J 2015 A three-dimensional explicit sphere function-based gas-kinetic flux solver for simulation of inviscid compressible flows Journal of Computational Physics 295 322-339
[4] Yang L M, Shu C and Wu J 2016 Extension of lattice Boltzmann flux solver for simulation of 3D viscous compressible flows Computers and Mathematics with Application 71 2069-2081
[5] Yang L M, Shu C, Wu J, Zhao N and Lu Z L 2013 J. Comput. Phys. 255 540-557
[6] Yang L M, Shu C and Wu J 2012 Adv. Appl. Math. Mech. 4 454-472
[7] Yang L M, Shu C and Wu J 2013 Comput. Fluids. 79 190-199
[8] Yang L M, Shu C and Wu J 2014 Transactions of Nanjing University of Aeronautics and Astronautics 31 1-15
[9] Kataoka T and Tsutahara M 2004 Phys. Rev. E. 69 R035701
[10] Qu K, Shu C and Chew Y T 2007 Phys. Rev. E 75 036706
[11] Li Q, He Y L, Wang Y and Tao W Q 2007 Phys. Rev. E 76 056705
[12] D Chae, C Kim and O H Rho 2000 J. Comput. Phys. 158 1-27
[13] Xu K 2001 J. Comput. Phys. 171 289-335
[14] Jiang J and Qian Y H 2012 Comput. Fluids 66 21-28
[15] Chen S Z, Xu K, Lee C B and Cai Q D 2012 J. Comput. Phys. 231 6643-6664
[16] Guo Z L and Shu C 2013 Lattice Boltzmann method and its applications in engineering (World Scientific Publishing)
[17] R C Swanson and R Radespiel 1991 Cell centered and cell vertex multigrid schemes for the Navier-Stokes equations AIAA J. 29 697-703
[18] J Blazek 2001 Computation Fluid Dynamics: Principle and Application Elsevier
[19] R Benzi, S Succi and M Vergassola 1992 The lattice Boltzmann equation: theory and application Physics Report
[20] P L Bhatnagar, E P Gross and M Krook 1954 Phys. Rev. 94 511-525
[21] Xu K and He X Y 2003 J. Comput. Phys. 190 100-117
[22] Yang L M, Shu C and Wu J 2016 Adv. Appl. Math. Mech. 8(6) 887-910
[23] Yang L M, Shu C and Wu J 2015 J. Comput. Phys. 295 322-339
[24] R C Swanson and R Radespiel 1991 AIAA Journal, 29 697-703
[25] Li X L, Fu D X and, Ma Y W 2010 Acta Mech Sin 26 795-806
[26] Zhuang F G, Zhang H X, Li C X and Wang J 2007 Flow properties of typical hypersonic vehicles National Defense Industry Press (Chinese edition)