Numerical Simulation of Flat Plate Boundary Layer with High-Order Gas-Kinetic Scheme

Qibing Li[1], Kun Xu[2] & Song Fu[1]

[1]Department of Engineering Mechanics, Tsinghua University, Beijing 100084, China
[2]Department of Mathematics, Hong Kong University of Science and Technology, Hong Kong
lqb@tsinghua.edu.cn

1. Code description

The high-order multidimensional gas-kinetic BGK scheme [1,2] is used, which is a finite volume method and the fluxes at a cell interface are computed based on the approximate solution of BGK equation to approach the NS equations. In the obtained velocity distribution function, the movement of particles in any direction is allowed, which guarantees the present scheme is truly multidimensional. Furthermore, the distribution function is a combination of Maxwellian distributions, and only one integral point at a cell interface is required to achieve the third-order accuracy in both space and time, which means low computational cost of the present method. Additionally, it is an explicit scheme thus has good parallel capability.

2. Case summary

We try case C1.4, the laminar boundary layer on a flat plate ($M_\infty = 0.5, L = 1, \mathrm{Re}_L = 10^6, U_\infty = 1$), and follow the requirement to calculate the converged solutions. The limiter in the scheme is shut off for this case. The L_2 norm of the density residual is adopted to monitor convergence. The simulation is started from a uniform free stream everywhere and steady state is assumed when the initial residual is dropped by 8 orders of magnitude. Intel Xeon CPU, E5520, is used and the work units for 100 residual evaluations with 250,000 DOFs are 26.4.

3. Meshes

The computation is done with the structured meshes similar to those provided on the website of the workshop, named as a1, a2, a3, a4 and a5 (corresponding to mesh files from a1-125-2s.msh to a5-125-2s.msh). To study the effect of domain size, two additional meshes, a3s and a3l are tried, with half and double L_H and L_V, respectively. The predicted drag coefficients Cd for a3, a3s and a3l are 1.31328E-3, 1.31353E-3 and 1.31287E-3 respectively.

4. Results

Figure 1 shows the density residual and drag coefficient variations with work units for all the meshes. Good convergence to steady state is achieved in the present computations. The converged velocity distributions calculated with mesh a3 are presented in Fig. 2, from which it can be observed that the velocity profiles, especially for the transverse ones can be well captured, even with only four cells inside the boundary layer. The wall friction coefficients and the corresponding drag coefficients also show good grid convergence (see Fig. 3). Detailed data can be found in table 1. The third-order accuracy is verified through the variation of drag coefficient errors with grid refinement.

It should be noted that for the present explicit method, the computational cost is a bit high. However, if combined with acceleration techniques, the performance of high-order gas-kinetic scheme can be greatly improved. Here a preliminary study is carried out. The LU-SGS and local time step methods are tried, and the corresponding results are presented in Fig. 4 and table 2. The cost to achieve the steady state (with initial density residual dropped by 10 orders of magnitude) is decreased by more than two orders of magnitude. Further study to improve the present scheme is under consideration.

Acknowledgments

This work is supported by National Natural Science Foundation of China (Project No. 10932005 and 11172154).

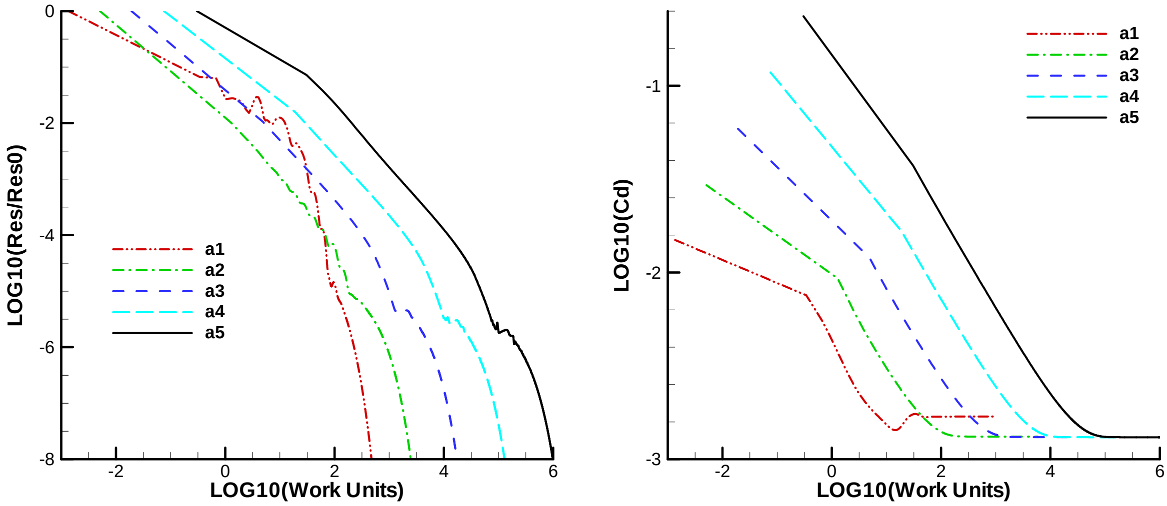

Fig. 1 Convergence histories for the computation with different meshes

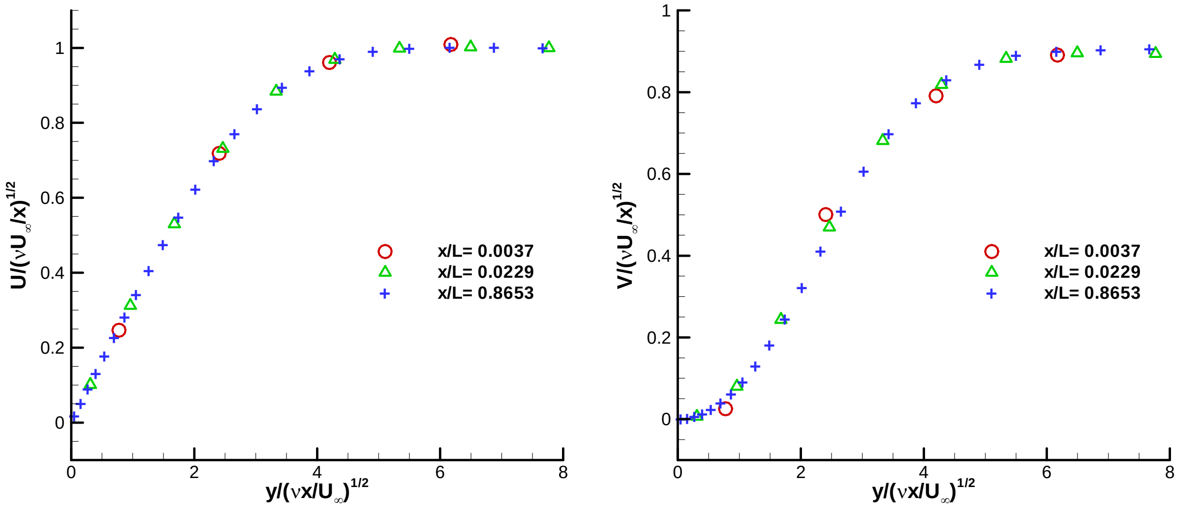

Fig. 2 Streamwise and transverse velocity profiles at different streamwise locations (a3 mesh)

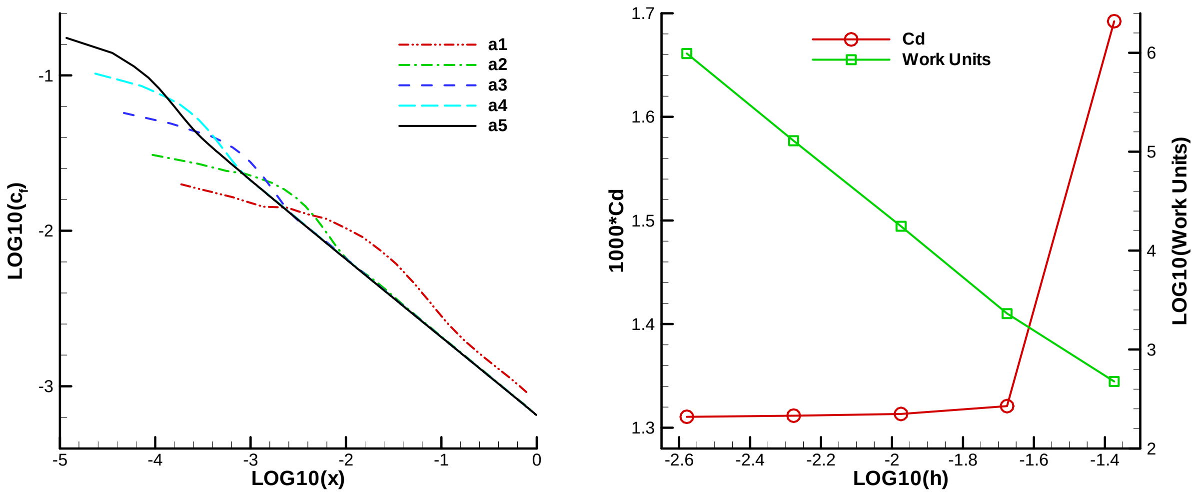

Fig. 3 Wall friction coefficients, drag coefficients and computational costs for grid refinement

Table 1　Computational parameters and results

Mesh	ncells	nDOFs/eqn.	h	Cd	Cd Error	Work Units
a1	560	560	4.23E-2	1.69227E-3	3.82E-4	4.75E2
a2	2240	2240	2.11E-2	1.32086E-3	1.03E-5	2.31E3
a3	8960	8960	1.06E-2	1.31328E-3	2.74E-6	1.73E4
a4	35840	35840	5.28E-3	1.31159E-3	1.05E-6	1.29E5
a5	143360	143360	2.64E-3	1.31054E-3	---	9.83E5

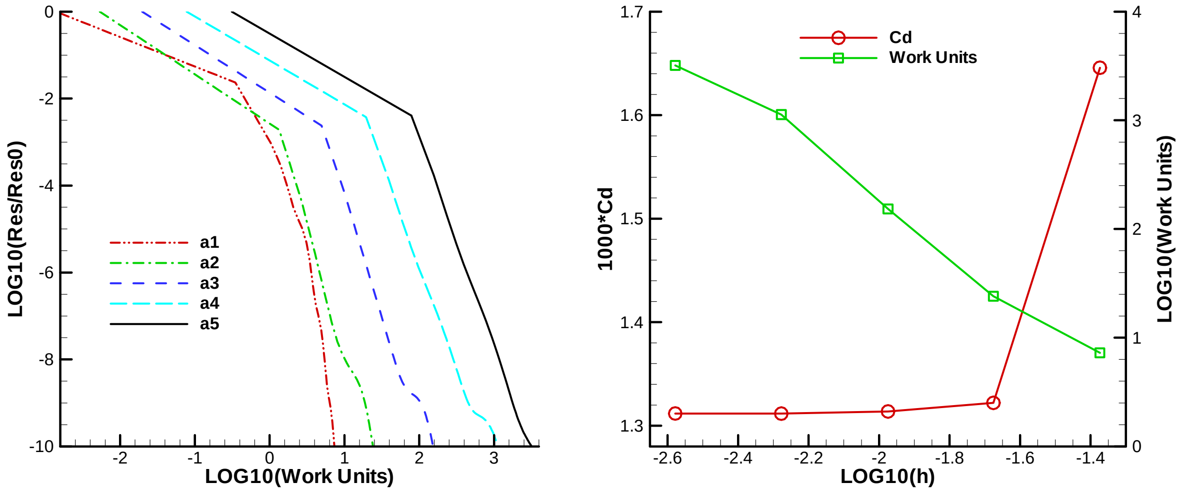

Fig. 4　Convergence histories, drag coefficients and computational costs for grid refinement with implicit method

Table 2　Computational parameters and predicted results with implicit method

Mesh	Cd	Cd Error	Work Units
a1	1.645775E-3	3.34E-4	7.26E0
a2	1.322095E-3	1.03E-5	2.41E1
a3	1.313742E-3	1.97E-6	1.53E2
a4	1.311686E-3	0.82E-7	1.13E3
a5	1.311768E-3	---	3.20E3

References

[1] Q.B. Li, K. Xu, S. Fu, A new high-order multidimensional scheme, *Computational Fluid Dynamics 2010: Proceedings of the Sixth International Conference on Computational Fluid Dynamics*, ICCFD6, St Petersburg, Russia.

[2] Q.B. Li, K. Xu, S. Fu, A high-order gas-kinetic Navier-Stokes flow solver, *J. Comput. Phys.* **229**, 6715-6731 (2010).