

A Fourth-Order Gas-Kinetic CPR Method for the Navier-Stokes Equations on Unstructured Meshes

Chao Zhang and Qibing Li
Corresponding author: lqb@tsinghua.edu.cn

AML, Department of Engineering Mechanics, Tsinghua University, Beijing 100084, China.

Abstract: A fourth-order accurate gas-kinetic CPR method is developed for the Navier-Stokes equations on triangular meshes. Different from the previous single-stage third-order gas-kinetic CPR method, the current method adopts the second-order gas-kinetic flux solver, which is simpler and less expensive, to reduce the computational cost of flux. Meanwhile, the temporal accuracy is improved by using the two-stage temporal discretization, which is more efficient than the four-stage Runge-Kutta method usually adopted in existing fourth-order CPR. Numerical tests are presented to demonstrate the accuracy and efficiency of the current method.

Keywords: Gas-Kinetic Scheme, Correction Procedure via Reconstruction, Two-stage Temporal Discretization.

1 Introduction

During the past decades, high-order CFD methods have attracted many researchers due to their advantages on accuracy and efficiency. There are many popular high-order methods, such as the Discontinuous Galerkin (DG) method, the $P_N P_M$ and so on. In recent years, the correction procedure via reconstruction (CPR) method has been developed rapidly, which provides a unified framework for many high-order methods [1]. The CPR method has been shown to be very competitive in terms of accuracy and efficiency, compared to the original DG method. Most of high-order methods mainly concentrate on the high-order reconstruction. As for the flux evolution, the Riemann solution of the Euler equations is the foundation of compressible flux solvers. However, developing a genuinely multidimensional Riemann solution is very difficult. For solving viscous flows, the viscous terms in the Navier-Stokes equations require special treatment.

Alternatively, the gas-kinetic scheme (GKS) offers a different way to recover the N-S solutions, which has shown good performance in a wide range of flow problems. Based on the local integral solution of the BGK model, the time evolving kinetic flux function can be constructed, in which the inviscid and viscous terms are coupled and obtained simultaneously. Through the high-order expansion of the gas distribution function, a high-order gas evolution model has been constructed successfully, which provides more abundant information to describe the flux evolution process [2]. Accordingly, a series of high-order gas-kinetic schemes have been developed. Recently, a third-order gas-kinetic CPR method, denoted as CPR-GKS(P2), has been developed for the Navier-Stokes equations on unstructured meshes [3]. It combines the high-efficient CPR framework with the third-order single-stage time stepping gas-kinetic flux solver, which shows high accuracy and efficiency in many typical flow problems. However, the third-order gas-kinetic flux solver is more complicated and time-consuming than its second-order counterpart.

In this study the fourth-order CPR framework is adopted to enhance the spatial accuracy. The second-order time-dependent gas-kinetic flux solver [4] is adopted for less computational cost, combined with the two-stage time-stepping method [5, 6] to improve the temporal accuracy. In comparison with the four-stage Runge-Kutta time-stepping method, the temporal discretization is more efficient due to less intermediate stages. The developed scheme, denoted as CPR-GKS(P3) is then validated through typical numerical tests.

2 Fourth-Order Gas-Kinetic CPR Method

2.1 CPR framework

By dividing the computational domain into N non-overlapping triangular cells Ω_i , a solution polynomial Q_i of degree k is constructed in each cell by using the Lagrange interpolation based on a set of solution points (SPs). For fourth-order CPR, i.e., $k = 3$, 10 solution points are needed in each cell. Additionally, to compute the common flux, $k + 1$ flux points (FPs) are set at each cell interface. For efficiency, the solution points are chosen the same as the flux points, as shown in Fig.1.

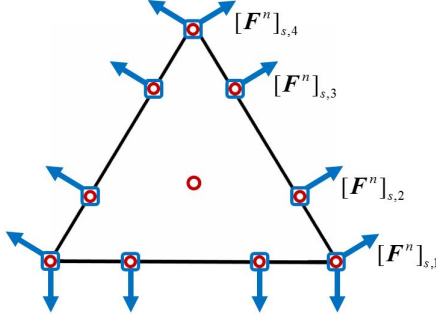


Figure 1. Solution points (circles) and flux points (squares) for $k=3$.

Then the CPR framework can be expressed as

$$\frac{\partial Q_{i,j}}{\partial t} + \Pi_j \left(\nabla \cdot \vec{F}(Q_i) \right) + \frac{1}{|\Omega_i|} \sum_{s \in \partial\Omega_i} \sum_{l=1}^4 \alpha_{j,s,l} [F^n]_{s,l} |\Gamma|_s = 0, \quad (1)$$

where $Q_{i,j}$ is the conservative variable at each solution point, $j = 1 \sim 10$. The second term is the projection of the flux divergence onto P^k . The last one is the correction term where $[F^n]_{s,l}$ is the normal flux difference at each flux point, i.e., $[F^n_{com}(Q_i, Q_{i+}, \vec{n}) - F^n(Q_i)]_{s,l}$. $\alpha_{j,s,l}$ is the lifting coefficient, $|\Omega_i|$ is the area of Ω_i , $|\Gamma|_s$ is the length of triangular edge. More details of the flux divergence term and correction term can be found in [1].

2.2 Gas-kinetic scheme

In the present study, the flux in the CPR framework is computed by the gas-kinetic flux solver. For completeness, the gas-kinetic scheme is briefly reviewed. For convenience, the summation convention is adopted in the following, where $\vec{x} = (x_1, x_2) = (x, y)$, $\vec{u} = (u_1, u_2) = (u, v)$, macroscopic velocity vector $\vec{U} = (U_1, U_2) = (U, V)$. The scheme is based on the BGK equation,

$$\frac{\partial f}{\partial t} + u_i \frac{\partial f}{\partial x_i} = \frac{g - f}{\tau}, \quad (2)$$

where f is the gas distribution function, u_i is the particle velocity, and $\tau = \mu/p$ is the particle collision time related to the viscosity and p is the pressure. The equilibrium state g is the Maxwellian distribution

$$g = \rho \left(\frac{\lambda}{\pi} \right)^{\frac{K+2}{2}} e^{-\lambda(|\vec{u} - \vec{U}|^2 + \xi^2)}, \quad (3)$$

where ρ is the density. λ is equal to $1/(2RT)$, where R is the gas constant and T is the temperature. Since mass, momentum, and energy are conserved during particle collisions, the collision term in Eq.(2) satisfies the compatibility condition

$$\int \frac{g - f}{\tau} \psi d\Xi = 0, \quad (4)$$

where ψ is the the vector of moments

$$\psi = \left(1, \vec{u}, \frac{1}{2} (|\vec{u}|^2 + \xi^2) \right)^T, \quad (5)$$

and $d\Xi = du_1 du_2 d\xi_1 d\xi_2 \cdots d\xi_K$ is the element of the phase space and $\xi^2 = \xi_1^2 + \xi_2^2 + \dots + \xi_K^2$. The total number of degrees of freedom K is equal to $(5 - 3\gamma)/(\gamma - 1) + 1$ for two-dimensional flow, in which γ is the specific heat ratio.

The relations between macroscopic variables and the distribution function are

$$\mathbf{Q} = \int f \psi d\Xi, \quad \vec{F} = \int \vec{u} f \psi d\Xi. \quad (6)$$

Taking moments of BGK equation, the N-S equations can be recovered based on the first-order Chapman-Enskog expansion

$$f_{NS} = g - \tau \left(\frac{\partial g}{\partial t} + u_i \frac{\partial g}{\partial x_i} \right). \quad (7)$$

In order to compute the flux at each solution point and flux point, the gas distribution function is constructed locally at these points. Since the flow distribution inside each cell is continuous, Eq.(7) can be directly adopted for computing the flux at each solution point (SP). The corresponding gas distribution function can be expressed as

$$f_{SP}(t, \vec{u}, \xi) = g_0 (1 - \tau a_i u_i + (t - \tau) A). \quad (8)$$

By introducing the notation

$$\langle \cdots \rangle = \int g_0(\cdots) \psi d\Xi, \quad (9)$$

the coefficients a_i and A can be determined by the spatial derivatives of conservative variables and the compatibility condition [2]

$$\langle a_i \rangle = \frac{\partial \mathbf{Q}}{\partial x_i} \Big|_{\vec{x}} \rightarrow a_i, \quad \langle a_i u_i + A \rangle = 0 \rightarrow A. \quad (10)$$

To compute the common flux at each flux point (FP) where the discontinuity exists, the gas distribution function is constructed based on the local analytical solution of BGK equation,

$$f(\vec{x}, t, \vec{u}, \xi) = \frac{1}{\tau} \int_0^t g(\vec{x}', t', \vec{u}, \xi) e^{-(t-t')/\tau} dt' + e^{-t/\tau} f_0(\vec{x} - \vec{u}t, 0, \vec{u}, \xi), \quad (11)$$

where f_0 is the piece-wise discontinuous initial distribution function at the beginning of each time step ($t = 0$), $\vec{x}' = \vec{x} - \vec{u}(t - t')$ is the particle trajectory. Note that the construction is performed in a local coordinate system $\vec{\tilde{x}} = (\tilde{x}_1, \tilde{x}_2)$, in which the $\tilde{x}_1$ -axis is perpendicular to the cell interface. Based on Eq.(11), the second-order time-dependent gas distribution function can be expressed as

$$\begin{aligned} f_{FP}(t, \vec{u}, \xi) = & g_0 \left(1 - e^{-t/\tau} \right) + g_0 \left((t + \tau) e^{-t/\tau} - \tau \right) a_i \tilde{u}_i + g_0 \left(t - \tau + \tau e^{-t/\tau} \right) A \\ & + e^{-t/\tau} g_0^R \left(1 - (\tau + t) a_i^R \tilde{u}_i - \tau A^R \right) H(u) \\ & + e^{-t/\tau} g_0^L \left(1 - (\tau + t) a_i^L \tilde{u}_i - \tau A^L \right) (1 - H(u)), \end{aligned} \quad (12)$$

where $H(u)$ is the Heaviside function. The equilibrium states g_0^L, g_0^R correspond to the flow states at the left and right sides of the cell interface. The coefficients a_i, A, a_i^L, A^L and a_i^R, A^R come from derivatives of g_0, g_0^L and g_0^R respectively, which can be determined by Eq.(10) as well. Besides, the equilibrium state g_0 is

determined by the compatibility condition

$$\int g_0 \psi d\Xi = \tilde{Q}_0 = \int_{u_1 > 0} g_0^L \psi d\Xi + \int_{u_1 < 0} g_0^R \psi d\Xi, \quad (13)$$

where $\tilde{Q}_0$ are the conservative variables corresponding the equilibrium state g_0 . In order to fully determine g_0 , a continuous flow distribution across the cell interface is constructed by using the least-square method, with the stencil shown in Fig.2.

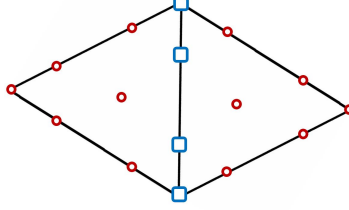


Figure 2. The stencil for constructing the equilibrium state in the local coordinate.

2.3 Two-stage temporal discretization

Based on the above gas-kinetic flux solver, the solution at each solution point inside a cell can be updated by adopting the following two-stage temporal discretization,

$$\begin{aligned} Q_{i,j}^* &= Q_{i,j}^n - \Pi_j \left(\nabla \cdot \left(\int_{t_n}^{t_*} \vec{F}(Q_i^n, t) dt \right) \right) - \frac{1}{|\Omega_i|} \sum_{s \in \partial\Omega_i} \sum_l \alpha_{j,s,l} \left(\int_{t_n}^{t_*} \hat{F}(Q_i^n, t) dt \right)_{s,l} |\Gamma|_s, \\ Q_{i,j}^{n+1} &= Q_{i,j}^n - \Pi_j \left(\nabla \cdot \vec{\mathbb{F}} \right) - \frac{1}{|\Omega_i|} \sum_{s \in \partial\Omega_i} \sum_l \alpha_{j,s,l} \left(\hat{\mathbb{F}} \right)_{s,l} |\Gamma|_s, \end{aligned} \quad (14)$$

where $t_* = t_n + \Delta t/2$ is the intermediate stage, $\hat{F}(Q_i^n, t)$ indicates the normal flux difference in Eq.(1), and $\vec{\mathbb{F}}$ can be expressed as

$$\vec{\mathbb{F}} = \frac{8}{3} \int_{t_n}^{t_*} \vec{F}(Q_i^n, t) dt - \frac{1}{3} \int_{t_n}^{t_{n+1}} \vec{F}(Q_i^n, t) dt + \frac{4}{3} \int_{t_*}^{t_* + \Delta t} \vec{F}(Q_i^*, t) dt - \frac{8}{3} \int_{t_*}^{t_{n+1}} \vec{F}(Q_i^*, t) dt, \quad (15)$$

and $\hat{\mathbb{F}}$ has the same form as Eq.(15) by replacing $\vec{F}$ with $\hat{F}$.

It is not easy to construct a fourth-order gas-kinetic flux solver with one-step time discretization as the corresponding gas distribution function is complicated and the computational cost is far more expensive. Fortunately, with the above two-step time discretization, the second-order gas-kinetic flux solver can be used, which only takes about one-sixth the computational cost of its third-order counterpart. Therefore, although the current method computes the flux pointwisely and has two stages, the total cost of flux is at the same level as the single-stage third-order gas-kinetic CPR, however, both spatial and temporal accuracy are improved.

Furthermore, the current temporal discretization is more efficient than the fourth-order Runge-Kutta method due to less intermediate stages. As a result, although the gas-kinetic flux solver is more expensive than traditional flux solvers, the current scheme can achieve nearly the same efficiency as the original fourth-order CPR. What's more, when simulating high-speed compressible viscous flows, the slope limiting procedure is necessary to capture flow discontinuities, which takes a large part of computational time. Therefore, it can be expected that the current scheme can be more efficient than the original CPR in compressible viscous flows.

3 Numerical Results

3.1 Compressible Couette flow

In order to verify the accuracy of the current scheme in viscous flows, the compressible Couette flow is simulated. The traditional CPR is also tested by using the HLLC and BR2 scheme with the fourth-stage Runge-Kutta method, denoted as CPR-BR2(P3). The lower wall with $y = 0$ is stationary and adiabatic. The upper wall at $y = 2$ is moving with a constant speed U_1 and temperature $T_1 = 1$. There is an analytical solution for this steady flow, i.e.,

$$\begin{aligned} \left(1 + \text{Pr} \frac{\gamma - 1}{3} \text{Ma}^2\right) \frac{y}{2H} &= \frac{U}{U_1} + \text{Pr} \frac{\gamma - 1}{2} \text{Ma}^2 \left(\frac{U}{U_1} - \frac{1}{3} \left(\frac{U}{U_1} \right)^3 \right), \\ V &= 0, \quad \frac{T}{T_1} = 1 + \text{Pr} \frac{\gamma - 1}{2} \text{Ma}^2 \left(1 - \left(\frac{U}{U_1} \right)^2 \right), \quad p = 1/\gamma, \end{aligned} \quad (16)$$

where the Prandtl number is $\text{Pr} = 1$, the Mach number is $\text{Ma} = 0.5$. The viscosity is determined by the linear law $\mu = \mu_1 T/T_1$. The Reynolds number is $\text{Re} = 500$. The CFL number to compute time steps is set as 0.1 in all test cases.

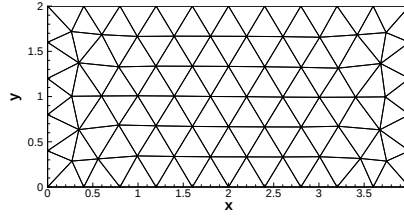


Figure 3. Computational mesh for compressible Couette flow.

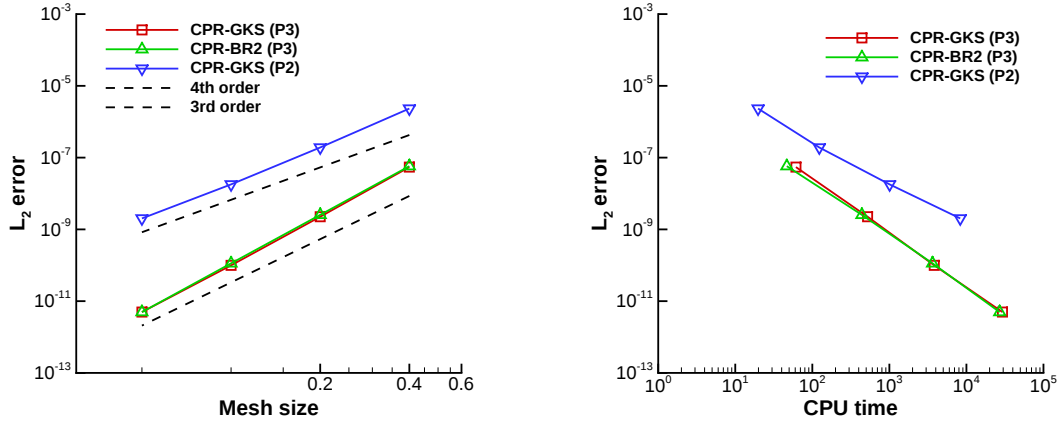


Figure 4. Temperature error vs. mesh size (left) and CPU time (right) for the compressible Couette flow.

The coarsest mesh used in this case is shown in Fig.3. The temperature error vs. mesh size and CPU time are presented in figure 4, in which the CPU time is the computational cost to achieve a steady state when the temperature residual less than 10^{-15} . It shows that CPR-GKS(P3) can achieve the fourth-order accuracy. To achieve the same level of absolute error, CPR-GKS(P3) is more efficient than CPR-GKS(P2) and has nearly the same efficiency as CPR-BR2(P3), demonstrating the high accuracy and efficiency of the current scheme. Nevertheless, further tests are necessary, especially for supersonic flows, to evaluate the

robustness of the current scheme. It can be anticipated that, with less intermediate stages, CPR-GKS(P3) can be more efficient than the CPR-BR2(P3) in supersonic flows when the slope limiting is included which has a considerable computational cost.

3.2 Lid-driven cavity flow

The second case is the lid-driven cavity flow problem. The top boundary of the unit square $[0, 1] \times [0, 1]$ is a moving plate at a speed $U_1 = 1$. The Mach number is set as $Ma = 0.15$. The Reynolds number is $Re = 1000$. The non-slip and isothermal boundary conditions are imposed on all boundaries with temperature $T_w = 1$. The initial flow is stationary with density $\rho_0 = 1$ and tempture $T_0 = T_w$. As presented in Fig.5, the mesh contains 200 elements with the minimum mesh size $h = 0.01$.

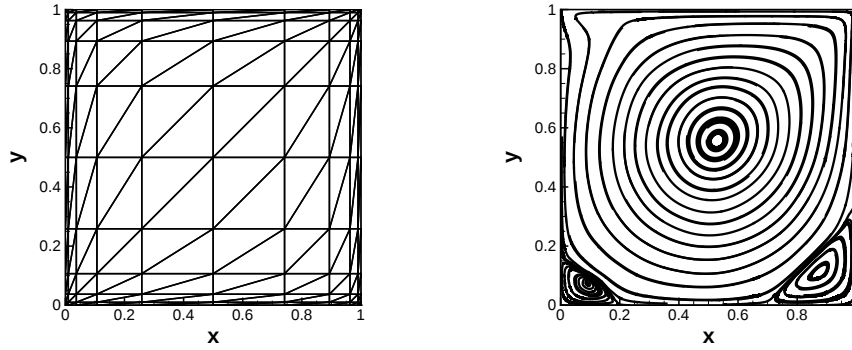


Figure 5. Computational mesh (left) and streamlines (right) in lid-driven cavity flow.

The streamlines are also shown in Fig.5, in which the flow structure, including the primary and secondary vortices, are well captured. The results of U -velocities along the vertical centerline and V -velocities along the horizontal centerline are shown in Fig.6. It can be observed that numerical results obtained by both CPR-GKS (P3) and CPR-BR2 (P3) match very well with the benchmark data. CPR-GKS (P3) is more accurate than CPR-GKS (P2) with such a coarse mesh, which also demonstrates the high accuracy and efficiency of the current scheme.

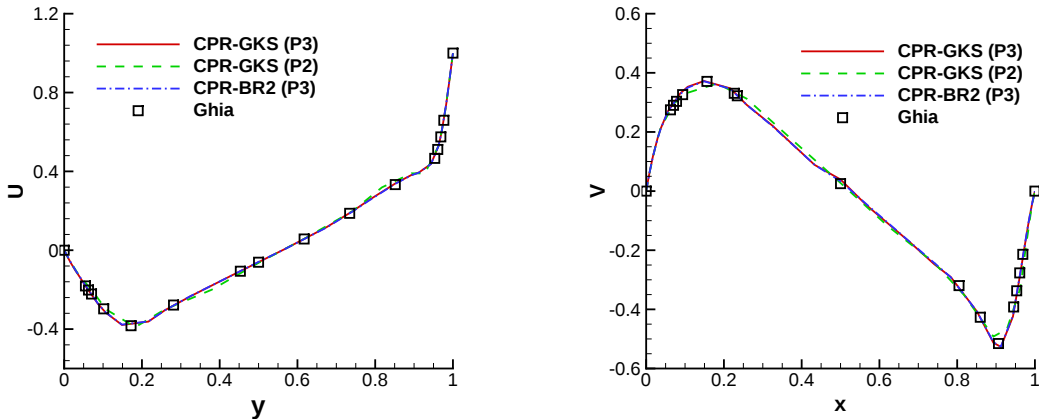


Figure 6. U -velocities along the vertical centerline and V -velocities along the horizontal centerline with $Re=1000$ in the lid-driven cavity flow.

4 Conclusion

In this study, based on the second-order gas-kinetic flux solver and the two-stage temporal discretization, a fourth-order gas-kinetic CPR method is developed for the Navier-Stokes equations on unstructured meshes. Compared with the previous third-order gas-kinetic CPR method, the second-order gas-kinetic flux solver is adopted to reduce the computational cost of flux evaluations, while the temporal accuracy can be improved by using the two-stage temporal discretization. Numerical tests show that the current scheme is more efficient than its third-order counterpart and achieves nearly the same efficiency as the original CPR method. Further investigations is required to solve high-speed flows where a more complicated reconstruction with slope limiters is needed to capture flow discontinuities, which takes much more CPU time. Accordingly, it can be expected that, with less intermediate stages, the current scheme can be more efficient than the original CPR method with the four-stage Runge-Kutta method.

5 Acknowledgments

This work is supported by the NSFC (11672158), National Key Basic Research and Development Program (2014CB744100) and Special Program for Applied Research on Super Computation of the NSFC-Guangdong Joint Fund (the second phase) (U1501501).

References

- [1] Z.J. Wang and H.Y. Gao. A unifying lifting collocation penalty formulation including the discontinuous Galerkin, spectral volume/difference methods for conservation laws on mixed grids. *J. Comput. Phys.*, 228:8161–8186, 2009.
- [2] Q.B. Li, K. Xu, and S. Fu. A high-order gas-kinetic Navier-Stokes flow solver. *J. Comput. Phys.*, 229:6715–6731, 2010.
- [3] C. Zhang, Q.B. Li, S. Fu, and Z.J. Wang. A third-order gas-kinetic CPR method for the Euler and Navier-Stokes equations on triangular meshes. *J. Comput. Phys.*, 363:329–353, 2018.
- [4] K. Xu. A gas-kinetic BGK scheme for the Navier-Stokes equations and its connection with artificial dissipation and Godunov method. *J. Comput. Phys.*, 171:289–335, 2001.
- [5] J. Li and Z. Du. A two-stage fourth order time-accurate discretization for Lax-Wendroff type flow solvers, I: hyperbolic conservation laws. *SIAM J. Sci. Comput.*, 38:A3046–A3069., 2016.
- [6] L. Pan, K. Xu, Q.B. Li, and J. Li. An efficient and accurate two-stage fourth-order gas-kinetic scheme for the Euler and Navier-Stokes equations. *J. Comput. Phys.*, 326:197–221, 2016.