Three-dimensional high-order least square-based finite difference-finite volume method on unstructured grids

Cite as: Phys. Fluids 32, 123604 (2020); https://doi.org/10.1063/5.0032089
Submitted: 06 October 2020 . Accepted: 17 November 2020 . Published Online: 07 December 2020

Y. Y. Liu (刘阳阳), L. M. Yang (杨锂铭), C. Shu (舒昌), and H. W. Zhang (张黄伟)
Three-dimensional high-order least square-based finite difference-finite volume method on unstructured grids

Y. Y. Liu (刘阳阳), L. M. Yang (杨铭铭), C. Shu (舒昌), and H. W. Zhang (张黄伟)

AFFILIATIONS
Department of Mechanical Engineering, National University of Singapore, 10 Kent Ridge Crescent, Singapore, 119260

Note: This paper is part of the Special Issue on the Lattice Boltzmann Method.
Author to whom correspondence should be addressed: mpeshuc@nus.edu.sg

ABSTRACT
The least square-based finite difference-finite volume (LSFD-FV) method has been developed and successfully applied to solve various two-dimensional flow problems with a high-order of accuracy. In this paper, the extension of the LSFD-FV method to the three-dimensional (3D) case on unstructured grids is presented. Different from other existing high-order methods, the LSFD-FV method combines the good features of the least square-based finite difference (LSFD) scheme for derivative approximation and the finite volume (FV) discretization for conservation of physical laws. Within each control cell, a high-order polynomial resultant from a Taylor series expansion is used to approximate the solution in the FV discretization of governing equations, where the derivatives are approximated by the LSFD scheme. As a result, the mesh-free nature of the LSFD scheme endows the LSFD-FV method with the flexibility on unstructured grids. Additionally, the straightforward algorithm of the LSFD scheme and the direct utilization of the Taylor series polynomial make the LSFD-FV method user-friendly and easy to implement. Furthermore, the inviscid and viscous fluxes are simultaneously evaluated by lattice Boltzmann and gas kinetic flux solvers in this work, which avoids additional and complicated treatment for the viscous discretization. Accuracy studies on unstructured hexahedral and tetrahedral grids validate the third-order of accuracy of the 3D LSFD-FV method. Various 3D incompressible and compressible numerical experiments are also presented. The results show that the proposed method enjoys the higher accuracy than the conventional second-order method on the same mesh. When the same accuracy is achieved, the present high-order method has superior computational efficiency to the second-order counterpart.

Published under license by AIP Publishing. https://doi.org/10.1063/5.0032089

I. INTRODUCTION
The past decades have witnessed the tremendous advances of computational fluid dynamics (CFD). Plenteous numerical methods have been developed for solving various flow problems in industrial applications. Among them, the high-order methods enjoy increasing popularity due to their higher accuracy and lower dissipation than the conventional second-order methods. Their superior performance is especially demonstrated in studying aeroacoustics, vortex-dominant flows, and turbulent flows. Generally, it is easier to develop a high-order numerical method on structured grids than on unstructured grids. The high-order finite difference (FD), essentially non-oscillatory (ENO), and weighted essentially non-oscillatory (WENO) methods are good examples developed on structured grids. However, the structured grid-based methods alone are not easy for engineering applications and some techniques would be required for complex geometry problems. For example, Ref. 7 applied the immersed boundary method with the Runge–Kutta discontinuous Galerkin method to solve two-dimensional flows with complex geometry on structured grids. For complex configurations in the three-dimensional (3D) case, it is usually difficult and time-consuming to generate structured grids, whereas unstructured grids can be generated with ease. As a result, great flexibility of the unstructured grids in handling complex geometries stimulates the
development of unstructured high-order methods. In this work, we also focus on developing a 3D high-order method on unstructured grids.

Over the last two decades, intensive efforts have been made to develop unstructured high-order methods. Typical examples in the literature include finite volume (FV) methods,\textsuperscript{20–23} discontinuous Galerkin (DG) methods,\textsuperscript{24–27} spectral volume (SV)\textsuperscript{28–30} methods, spectral difference (SD)\textsuperscript{31–33} methods, etc. These methods differ in terms of the high-order spatial discretization and thus possess distinct characteristics. A detailed review of them can be found in Ref. 28. Among the high-order methods mentioned above, the high-order FV methods draw much attention. Due to the popularization of the FV discretization and the well-developed numerical techniques (e.g., shock capturing technique and time marching strategy) on unstructured grids, the present high-order method is developed in the FV framework.

When considering a numerical method for the practical 3D engineering problems, the complexity of the high-order spatial discretization is significantly important. It determines whether the newly developed high-order method would be widely used or not. Recently, Liu and his co-workers proposed a straightforward and user-friendly high-order FV method on unstructured grids in two dimensions, called least square-based finite difference-finite volume (LSFD-FV) method.\textsuperscript{34} The LSFD-FV method directly applies a Taylor series expansion within each control cell to approximate the solution with the high-order accuracy. The unknown coefficients in the approximation polynomial, i.e., the spatial derivatives, are determined by the mesh-free least square-based finite difference (LSFD) scheme using the solutions at centers of the current cell and its neighboring cells. The LSFD method\textsuperscript{35,36} is a simple and mesh-free method for derivative approximation at randomly distributed points. Thus, it is an ideal approach to approximate derivatives on unstructured grids. However, due to the nature of the FD discretization, its numerical discretization cannot be guaranteed to be conservative. The FV discretization, unlike the FD method, can remain conservative at the cell and the global level. The LSFD-FV method embeds the LSFD scheme into the framework of the FV method, which provides a simple and effective high-order solver on unstructured mesh. Various two-dimensional applications\textsuperscript{37,38} of the LSFD-FV method, such as the incompressible isothermal and thermal flows, and the compressible inviscid and viscous flows, have been made. It is shown that the high-order LSFD-FV method achieves superior accuracy and computational efficiency compared with the k-exact method of the same accuracy order and the second-order method. Therefore, in the present study, the LSFD-FV method is further extended to solve the 3D incompressible and compressible flow problems on unstructured grids.

Apart from the spatial discretization, the flux evaluation also plays a vital role in the high-order solver. In the literature, many high-order methods use the Riemann solver and average approach to evaluate the inviscid and viscous fluxes, respectively. However, for viscous fluxes, the average approach, which takes a simple arithmetic mean of the gradients from the left and right cells of the interface, is inconsistent. It does not consider a possible jump of solutions and may result in a wrong solution.\textsuperscript{39} Numerous studies have been conducted for the proper treatment of viscous fluxes, such as the interior penalty approach, the local DG approach,\textsuperscript{40} and the second Bassi–Rebay scheme.\textsuperscript{41–43} Nevertheless, how to reduce the computational cost or achieve the good convergence is still an open problem. In this work, for viscous flow problems, the viscous and inviscid fluxes are evaluated simultaneously, and different flux solvers are used for the incompressible flow and compressible flow to achieve better efficiency. Particularly, the lattice Boltzmann flux solver (LBFS)\textsuperscript{44} is adopted for incompressible isothermal and thermal flows, and the discrete gas-kinetic flux solver (DGKFS)\textsuperscript{45} is applied for simulation of compressible viscous flows. These two flux solvers, respectively, use the local solution of the lattice Boltzmann equation and continuous Boltzmann equation to construct the numerical flux at the cell interface. By using such a physical treatment for the evaluation of the numerical flux, the present 3D LSFD-FV method seems to be competitive for simulating viscous flow problems.

A series of incompressible and compressible numerical tests are conducted on unstructured grids to assess the accuracy, efficiency, and robustness of the present 3D high-order LSFD-FV method. Numerical results demonstrate that this high-order method has the third-order of accuracy and it outperforms the conventional low-order methods in terms of the accuracy and computational efficiency.

\section{II. THREE-DIMENSIONAL HIGH-ORDER LEAST SQUARE-BASED FINITE DIFFERENCE-FINITE VOLUME (LSFD-FV) METHOD}

\subsection{A. Governing equations and high-order finite volume discretization}

N–S equations are considered for a general case and discretized by the FV method, where the conservative variables are defined at cell centers. In the 3D case, the semi-discrete form of the N–S equations discretized by the cell-centered FV method can be cast as

\begin{equation}
\frac{d}{dt} (\int_{\Omega_i} U d\Omega_i) = -\sum_{j=1}^{N_f} \sum_{p=1}^{nGQP} (F_{i,p}GQP) w_{GQP} + \int_{\Omega_i} Q d\Omega_i, \tag{1}
\end{equation}

with

\begin{equation}
U = [\rho, \rho u_x, \rho u_y, \rho u_z, \rho E]^T, \tag{2}
\end{equation}

where $U$ is the vector of conservative variables and $F_n$ represents the flux vector in the Cartesian coordinate system. Note that the Gaussian quadrature has been applied for the high-order accurate computation of the integral of fluxes at the cell interface. $N_f$ is the number of cell faces of the control volume $\Omega_i$, $nGQP$, $A$, and $w_{GQP}$ are the same as those in Ref. 29. $\rho$, $u = (u_x, u_y, u_z)$, and $E$ are, respectively, the density, velocity, and total energy of the fluid flow. $E = e + (u_x^2 + u_y^2 + u_z^2)/2$, where $e = p[(y - 1)/\gamma]$ is the internal energy of the mean flow. $p$ is the pressure, and $\gamma$ is the specific heat ratio. The source term $Q$, which comprises all volume sources due to body sources and volumetric heating, can be calculated directly from the conservative variables at cell centers.

In order to obtain higher-order numerical solutions of flow variables, the present high-order LSFD-FV method applies a Taylor series expansion at the cell center with the designed order of accuracy. For instance, the following polynomial approximates the solution variable at the location $(x, y, z)$ with the third-order of accuracy,
\[ U(x, y, z) = U_i + \frac{\partial U}{\partial x} (x - x_i) + \frac{\partial U}{\partial y} (y - y_i) + \frac{\partial U}{\partial z} (z - z_i) \]
\[ + \frac{\partial^2 U}{\partial x^2} \frac{(x - x_i)^2}{2} + \frac{\partial^2 U}{\partial y^2} \frac{(y - y_i)^2}{2} + \frac{\partial^2 U}{\partial z^2} \frac{(z - z_i)^2}{2} \]
\[ + \frac{\partial^2 U}{\partial x \partial y} (x - x_i)(y - y_i) + \frac{\partial^2 U}{\partial x \partial z} (x - x_i)(z - z_i) \]
\[ + \frac{\partial^2 U}{\partial y \partial z} (y - y_i)(z - z_i), \]

where \( U \) is the solution in the whole flow domain. Note that Eq. (4) involves the solution \( (x, y) \), \( (x, z) \), \( (y, z) \) expected. In this work, Eq. (1) is rewritten as
\[ W_i = \sum_{k=1}^{N} C_{i,j} (U_j - U_i), \]

where \( C_{i,j} \) represent the volume integral of the distance relations with respect to the cell center and they could be diverse for different cells in the whole flow domain. Note that Eq. (4) involves the solution \( U_i \) at the cell center and its \( N_i \) unknown spatial derivatives \( dU_i \). As shown in Eq. (5), \( N_i \) is 9 when the third-order of accuracy is expected. In this work, \( dU_i \) are approximated by the mesh-free LSFD method \(^1\) by using the solutions at the centers of current cell \( i \) and its \( N \) neighboring cells, i.e.,
\[ dU_i^T = \left[ \begin{array}{cccccc}
\frac{\partial U}{\partial x} & \frac{\partial U}{\partial y} & \frac{\partial U}{\partial z} & \frac{\partial^2 U}{\partial x^2} & \frac{\partial^2 U}{\partial y^2} & \frac{\partial^2 U}{\partial z^2} \\
& & & \frac{\partial^2 U}{\partial x \partial y} & \frac{\partial^2 U}{\partial x \partial z} & \frac{\partial^2 U}{\partial y \partial z}
\end{array} \right]. \]

with \( \Omega \) representing the solution vector composed by all control cells and \( M \) is a sparse coefficient matrix formed by coefficients on the left-hand side of Eq. (7). To solve the resultant ordinary differential equations efficiently, the implicit time marching techniques are adopted.

### B. Derivative approximation by least square-based finite difference (LSFD) scheme

In this work, we apply the LSFD method \(^1\) to approximate the spatial derivatives in Eq. (3). Since the number of unknowns is 9, the same number of equations is required to construct a well-posed system. Therefore, Eq. (3) is applied to 9 neighboring cells and a linear system can be obtained as follows:
\[ \hat{S} \Delta U = \Delta \hat{U}, \]

where \( \Delta U \) is the discrete form of the right-hand side of Eq. (1). In this work, the numerical fluxes in \( \hat{R}_i \) are evaluated by the specific

\[ \hat{S} = \left[ \begin{array}{cccccccc}
\Delta x_1 & \Delta y_1 & \Delta z_1 & \frac{\Delta x_1^2}{2} & \frac{\Delta y_1^2}{2} & \frac{\Delta z_1^2}{2} & \Delta x_1 \Delta y_1 & \Delta x_1 \Delta z_1 & \Delta y_1 \Delta z_1 \\
\vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\
\Delta x_9 & \Delta y_9 & \Delta z_9 & \frac{\Delta x_9^2}{2} & \frac{\Delta y_9^2}{2} & \frac{\Delta z_9^2}{2} & \Delta x_9 \Delta y_9 & \Delta x_9 \Delta z_9 & \Delta y_9 \Delta z_9
\end{array} \right], \]

\[ \Delta \hat{U} = \left[ \begin{array}{c}
U_{i1} - U_i - U_{i2} - U_i - \ldots - U_{i9} - U_i
\end{array} \right]. \]

where the reference point \((x_i, y_i, z_i)\) is the cell centroid of \( \Omega_i \). Based on this high-order Taylor polynomial, the solution at the quadrature points along the cell interface is interpolated with the third-order of accuracy. This provides the basis for high-order accurate evaluation of numerical fluxes. When Eq. (3) is substituted into the volume integral in Eq. (1), the following equation can be obtained,
\[ \int_{\Omega_i} U(x, y) d\Omega = \Omega_i U_i + \frac{\partial U}{\partial x} \frac{(x - x_i)^2}{2} + \frac{\partial U}{\partial y} \frac{(y - y_i)^2}{2} + \frac{\partial U}{\partial z} \frac{(z - z_i)^2}{2} + \frac{\partial^2 U}{\partial x \partial y} (x - x_i)(y - y_i) + \frac{\partial^2 U}{\partial x \partial z} (x - x_i)(z - z_i) + \frac{\partial^2 U}{\partial y \partial z} (y - y_i)(z - z_i), \]

due to unstructured cell distribution for a general case. To overcome this difficulty, the least square optimization and local scaling technique are used. In addition, the distance-related weighting function

\[ \hat{R}_i = \sum_{k=1}^{N} W_{k,j} (U_j - U_i), \]

where \( W_{k,j} \) will be presented in Sec. II B. With Eqs. (4) and (6), Eq. (1) is rewritten as
\[ \left( \Omega_i - \sum_{k=1}^{9} \Delta R_i^2 \right) \frac{\partial U_i}{\partial t} + \sum_{k=1}^{9} \Delta R_i^2 \frac{\partial U_j}{\partial t} = -R_i, \]

with \( \hat{S} \) and \( \Delta \hat{U} \) given by

\[ \hat{S} = \left[ \begin{array}{cccccc}
\Delta x_1 & \Delta y_1 & \Delta z_1 & \frac{\Delta x_1^2}{2} & \frac{\Delta y_1^2}{2} & \frac{\Delta z_1^2}{2} \\
\vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\
\Delta x_9 & \Delta y_9 & \Delta z_9 & \frac{\Delta x_9^2}{2} & \frac{\Delta y_9^2}{2} & \frac{\Delta z_9^2}{2}
\end{array} \right], \]

\[ \Delta \hat{U} = \left[ \begin{array}{c}
U_{i1} - U_i - U_{i2} - U_i - \ldots - U_{i9} - U_i
\end{array} \right]. \]
is also introduced to reduce the influence of data farther from the reference point. Since the basic idea and procedures of such techniques have been given in Refs. 29 and 30, the details are omitted here. Finally, the derivative vector \( dU \) can be calculated by

\[
dU = W \Delta U,
\]

where \( W \) is a \( 9 \times N \) dimensional weighting coefficient matrix and it is uniquely determined by the coordinates of mesh points (centers of the current cell and its neighboring cells). In practical implementation, \( W \) is calculated once and stored for problems with the fixed computational mesh. This saves computational effort.

C. Evaluation of numerical flux

Subsection II B addresses the derivative approximation in the high-order LSFD-FV method. After obtaining the various derivatives in Eq. (3), the flow variables at the cell interface can be interpolated with the third-order of accuracy. Based on this, accurate evaluation of the numerical flux at the cell interface is discussed in this subsection. In this work, three kinds of flux solvers are applied to compute the flow physically. For incompressible isothermal and thermal flows, the LBFS is adopted and it can evaluate the inviscid and viscous fluxes simultaneously by local reconstruction of lattice Boltzmann solution. For compressible viscous flows, the compressible LBFS \(^{19,41}\) based on the non-free parameter D1Q4 model is used to compute the inviscid flux at the cell interface. In the simulation of compressible viscous flows, the DGKFS is applied for simultaneous evaluation of the inviscid and viscous fluxes by the local solution of the continuous Boltzmann equation. These flux solvers have been applied in many problems \(^{12,34}\) and have shown good features, such as excellent computational efficiency and easy implementation. Therefore, with applying them for specific problems, the present high-order solver would present a competitive performance in simulating various incompressible and compressible flows.

For simplicity, the development and derivation of the above flux solvers are briefly reviewed. Special attention is paid to their implementation in the present solver and effect on the overall accuracy. To apply these flux solvers in the present 3D unstructured LSFD-FV method with ease, a local coordinate system is introduced at the cell interface. In such a local coordinate system, the directions are the outward normal direction and two tangential directions of the cell interface, which are, respectively, denoted by the subscripts “1,” “2,” and “3.” Details of these three flux solvers in the local coordinate system are shown below.

First, the LBFS for both isothermal and thermal incompressible flows is introduced. Following the work of Wang et al., the N–S equations can be recovered through multiscale Chapman-Enskog expansion analysis to the thermal lattice Boltzmann equation in the low Mach number limit, which gives the interface flux

\[
F = (F_1, F_2, F_3, F_4, F_5)^T
\]

in the local coordinate system as follows:

\[
F_1 = \sum_{a=0}^{Nd} (e_a) f_{eq}^a, \\
F_2 = \sum_{a=0}^{Nd} (e_a) (e_a) [f_{eq}^a + (1 - 1/2\tau_e) f_{eq}^{neq}], \\
F_3 = \sum_{a=0}^{Nd} (e_a) (e_a) [f_{eq}^a + (1 - 1/2\tau_v) f_{eq}^{neq}], \\
F_4 = \sum_{a=0}^{Nd} (e_a) (e_a) [f_{eq}^a + (1 - 1/2\tau_v) f_{eq}^{neq}], \\
F_5 = \sum_{a=0}^{Nd} (e_a) [g_{eq}^a + (1 - 1/2\tau_v) g_{eq}^{neq}].
\]

where \( e_a \) is the lattice velocity vector, and the single relaxation parameters \( \tau_e \) and \( \tau_v \) are, respectively, related to the dynamic viscosity and thermal diffusivity. \( f_{eq}^a \) denotes the equilibrium density distribution function along the \( a \) direction, and \( f_{eq}^{neq} \) is the corresponding non-equilibrium distribution function. \( Nd + 1 \) is the number of discrete particle velocities in the lattice Boltzmann model. In the present study, the commonly used D3Q15 lattice model \((Nd + 1 = 15)\) is applied and the corresponding \( f_{eq}^a \) and \( g_{eq}^a \) are referred to Ref. 38, \( \tau_e \) and \( \tau_v \) in Eq. (12) are determined from the kinematic viscosity \( \nu \) and thermal diffusivity \( \kappa \) by

\[
\nu = (\tau_e - 1) c_s^2 \delta_i, \\
\kappa = \frac{5(\tau_e - 1/2) c_s^2 \delta_i}{3},
\]

where \( \delta_i \) denotes the streaming step time, which equals the lattice spacing, and \( c_s \) is the sound speed. The non-equilibrium distribution functions \( f_{eq}^{neq} \) and \( g_{eq}^{neq} \) are approximated by

\[
f_{eq}^{neq}(r, t) = -\tau_e \left[ f_{eq}^a(r, t) - f_{eq}^a (r - e_a \delta_i, t - \delta_t) \right] + O(\delta_t^2), \\
g_{eq}^{neq}(r, t) = -\tau_e \left[ g_{eq}^a(r, t) - g_{eq}^a (r - e_a \delta_i, t - \delta_t) \right] + O(\delta_t^2),
\]

where \( r \) is the physical location of the Gaussian quadrature point along the cell interface and \( t \) is the time. \( f_{eq}^a(r, t) \) and \( f_{eq}^{neq}(r - e_a \delta_i, t - \delta_t) \) are the equilibrium density distribution functions at \( r \) and its surrounding nodes \( r - e_a \delta_i \), respectively. \( g_{eq}^a(r, t) \) and \( g_{eq}^{neq} (r - e_a \delta_i, t - \delta_t) \) are the equilibrium internal energy distribution functions at the corresponding locations.

To calculate \( f_{eq}^{neq}(r, t) \) and \( g_{eq}^{neq}(r, t) \), the flow variables at the location \( r - e_a \delta_i \) are interpolated first by using Eq. (3) with the third-order of accuracy based on the information at cell centroids \( r_i \), i.e.,

\[
U(r - e_a \delta_i) = \begin{cases} 
U(r_i) + \nabla U(r_i) \Delta X_i + \frac{1}{2} \Delta X_i^T \nabla H(X_i) \Delta X_i + O(\Delta X^2), & (r - e_a \delta_i) \in \Omega_i, \\
U(r_j) + \nabla U(r_j) \Delta X_j + \frac{1}{2} \Delta X_j^T \nabla H(X_j) \Delta X_j + O(\Delta X^2), & (r - e_a \delta_i) \in \Omega_j,
\end{cases}
\]
with
\[
\Delta X_k = \left( (r - e_d \delta_i - r_k), (r - e_d \delta_i - r_k), (r - e_d \delta_i - r_k) \right)^T, \\
k = i \text{ or } j.
\]
\[
\nabla U \text{ and } H \text{ are given by}
\]
\[
\nabla U = \left( \frac{\partial U}{\partial x}, \frac{\partial U}{\partial y}, \frac{\partial U}{\partial z} \right), \quad H = \begin{bmatrix}
\frac{\partial^2 U}{\partial x^2} & \frac{\partial^2 U}{\partial x \partial y} & \frac{\partial^2 U}{\partial x \partial z} \\
\frac{\partial^2 U}{\partial y \partial x} & \frac{\partial^2 U}{\partial y^2} & \frac{\partial^2 U}{\partial y \partial z} \\
\frac{\partial^2 U}{\partial z \partial x} & \frac{\partial^2 U}{\partial z \partial y} & \frac{\partial^2 U}{\partial z^2}
\end{bmatrix},
\]
where the derivatives are approximated by the LSFD method as introduced in Subsection 11 B. Once the required flow variables are available, all the equilibrium density and internal energy distribution functions in Eqs. (15) and (16) can be computed following the derivations in Ref. 38. In this way, \( f_{w}^{m}(r, t) \) and \( g_{w}^{m}(r, t) \) can be, respectively, calculated, and then, the flux vector \( F = (F_1, F_2, F_3, F_4, F_5)^T \) in Eq. (12) can be obtained.

As stated previously, the whole flux \( F \) is computed in the local coordinate system by the LBFS. However, in the practical calculation, the flux \( F_i \) in Eq. (1) is defined in the global coordinate system. Thus, the transformation as the following form should be adopted,
\[
F_k = \left(F_1, F_2n_{1x} + F_3n_{2x} + F_4n_{3x} + F_5n_{1y} + F_3n_{2y} + F_4n_{3y}, \\
F_2n_{2z} + F_3n_{2y} + F_4n_{3y}, F_5 + F_4n_{3x} + F_3n_{2x} + F_2n_{2z} \right)^T,
\]
where \( n_1 = (n_{1x}, n_{1y}, n_{1z}) \) is the unit vector in direction “1” of the local coordinate system. \( n_2 = (n_{2x}, n_{2y}, n_{2z}) \) and \( n_3 = (n_{3x}, n_{3y}, n_{3z}) \) are, respectively, the unit vectors in directions “2” and “3.”

Second, the non-free parameter D1Q4 compressible LBFS is introduced to simulate incompressible flows in the present high-order method. For simplicity, it is hereinafter referred to as the D1Q4 model. In this model, the discrete lattice velocities contain 4 equilibrium distribution functions \( g_i, i = 1, 4 \), and 2 lattice velocities \( d_1 \) and \( d_2 \), \( g_i \) are determined from the moment relationships corresponding to Euler equations, and the lattice velocities are computed from two appended high-order moment relationships. The expressions of \( g_i, d_1, \) and \( d_2 \) can refer to Refs. 39 and 40. Following the work of Yang et al., \( \xi \) is the dimensionless relaxation time and \( \xi \) is the lattice velocity in the \( i \) direction. Generally, \( \tau_0 = \tanh[10(4p_1 - p_0)/p_L] \), where \( p_L \) is the hyperbolic tangent function, and \( p_L \) and \( p_0 \) are the pressure at the left and right sides of the cell interface. When strong shock waves are considered, a switch function is introduced and the details can be found in Ref. 40. \( \phi_a \) represents the moment vector, which can be written as
\[
\phi_a = \left( \xi n_{1x} + u_{tx}, \xi n_{1y} + u_{ty}, \xi n_{1z} + u_{tz}, \left( \xi^2 + |u|^2 \right)/2 + \xi \right)^T,
\]
where \( \xi = [1 - (y - 1/2)] \) refers to the potential energy of particles and the tangential velocity vector is \( u_i = (u_{tx}, u_{ty}, u_{tz}) = u - u_n n_1 \). It is clear in Eq. (21) that the total inviscid flux consists of the fluxes attributed to the equilibrium distribution functions at the cell interface \( g_i(r, t) \) and at the surrounding points of the cell interface \( g_i(r - \xi \delta_i, t - \delta_i) \). We can calculate \( g_i(r - \xi \delta_i, t - \delta_i) \) according to the location of \( r - \xi \delta_i \) as follows:
\[
g_i(r - \xi \delta_i, t - \delta_i) = \left( \frac{1}{g_i}, \quad i = 1, 3, \\
\frac{1}{g_i}, \quad i = 2, 4, \right)
\]
where \( g_{c_1} \) and \( g_{c_2} \) denote the equilibrium distribution functions at the left and right sides of the cell interface, which are calculated from the corresponding conservative variables based on the expressions of \( g_i \). In the present high-order method, these conservative variables are interpolated by the way like Eq. (17) with the third-order of accuracy. If there exits the discontinuity, the shock-capturing technique would be used during the interpolation process. The conservative variables at the cell interface are then computed by
\[
U(r, t) = \sum_{i=1}^{13} \phi_{c_1}^{i} + \sum_{i=1}^{14} \phi_{c_2}^{i}.
\]
With Eq. (24), the primitive variables \( \rho, u_1, u_2, u_3, p \) at the cell interface used to compute \( g_i(r, t) \) can be obtained. Once \( g_i(r, t) \) and \( g_i(r - \xi \delta_i, t - \delta_i) \) are both available, the flux \( F_i^L \) and \( F_i^H \) can be computed explicitly via the following equations:
\[
F_i^c = \left[ \\
\begin{array}{c}
\rho u_1 \\
\rho u_1 u_1 + p \end{array}
\right],
\]
\[
F_i^c = \left[ \\
\begin{array}{c}
\rho u_1 u_2 + p u_1 \\
\rho u_1 u_3 + p u_1 \\
\rho u_1 + p u_2 / 2 + e \end{array}
\right] / 2,
\]
\[
F_i^c = \sum_{i=1}^{13} \xi \phi_{c_1}^{i} + \sum_{i=1}^{14} \xi \phi_{c_2}^{i}.
\]

The whole flux \( F \) in the local coordinate system is finally computed by Eq. (21). As stated previously, the flux \( F_n \) in Eq. (1), which is defined in the global coordinate system, can be obtained via Eq. (20).

Finally, the DGKFS\textsuperscript{11} adopted in this high-order LSFD-FV method for compressible viscous flows is presented. In the DGKFS, the D3Q8 discrete velocity model is applied. It defines eight discrete points on a spherical surface with the radius \( c \) and center \( u = (u_1, u_2, u_3) \). Since the moment integrations to recover N–S equations by the continuum Boltzmann equation can be accurately replaced with the integral quadrature, the conservative variables and numerical fluxes can be computed by the weighted summation of distribution functions with the D3Q8 model. The formulations of the equilibrium distribution functions \( g_0 \) and discrete velocities \( \xi \) are given in Ref. 41. With the D3Q8 model, the fluxes at the cell interface can be computed by the DGKFS as the following numerical integrations:
\[
\left[ F_1, F_2, F_3, F_4, F_5 \right]^T = \sum_{i=1}^{8} \xi \phi_{c}^{i} \phi_{c} \Delta S_i.
\]
where $\Delta S = \pi/2$ and $\phi_q$ stands for the moment vector given by
\[
\phi_q = \left( 1, \xi_{q,1}, \xi_{q,2}, \xi_{q,3}, \frac{1}{2} \| \xi_q \|^2 + \epsilon_{p,q} \right)^T.
\] (28)

The distribution function $f_q$ at the cell interface, i.e., $f_q(r, t)$, can be expressed as
\[
f_q(r, t) = g_q(r, t) + \gamma \left[ g_q(r - \xi_q \delta_i, t - \delta_i) - g_q(r, t) \right] + O(\delta_i^2).
\] (29)

where $g_q(r, t)$ and $g_q(r - \xi_q \delta_i, t - \delta_i)$ are, respectively, the equilibrium distribution functions at the cell interface and at the discrete points on the sphere. For convenience, “$(r, t)$” and “$(r - \xi_q \delta_i, t - \delta_i)$” are, respectively, noted by superscripts “face” and “sph.” Substituting Eq. (29) into Eq. (27), the flux vector $F$ can be obtained by

\[
F = \sum_{q=1}^8 \phi_{q,1} f_{q,1} \Delta S + \rho \sum_{q=1}^8 \phi_{q,2} f_{q,2} \Delta S - \sum_{q=1}^8 \phi_{q,3} f_{q,3} \Delta S.
\] (30)

Clearly, the flux $F$ contains two parts: one is the flux $F^I$ attributed to the equilibrium distribution functions and moments at the cell interface and the other is the flux $F^II$ attributed to the equilibrium distribution functions and moments on the spherical surface. The conservative variables at the discrete points on the spherical surface $\rho^{\text{sph}}$, $\phi^{\text{sph}}$ are computed like the way in Eq. (17) with $\Delta S$.

$\delta_i = \left\{ \left( \frac{1}{2} - \frac{1}{2 \sqrt{3}} \right) h_{\text{min}}, \text{LBFS} \right\} \left( \min (u_1, u_2, u_3) + \epsilon, \text{DGKFS} \right)$, (34)

where $h_{\text{min}}$ is the minimum edge length of the left and right cells of the interface for a hexahedral mesh or the radius of the inscribed sphere in the left and right cells of the interface for a tetrahedral mesh.

III. NUMERICAL RESULTS AND DISCUSSIONS

In this section, the accuracy, computational efficiency, and robustness of the developed high-order LSFD-FV method are assessed by simulating a series of 3D incompressible and compressible benchmark cases. All simulations are done on a PC with 2.30 GHz CPU.

A. Advection of density perturbation problem for accuracy test

First, the advection of density perturbation for the 3D flow is presented to validate the order of accuracy. The initial condition of this problem is set as follows:

\[
\rho(x, y, z) = 1 + 0.2 \sin(\pi (x + y + z)),
\]

\[
u(x, y, z) = 1, w(x, y, z) = 1, \rho(x, y, z) = 1.
\] (35)

The exact solution under periodic boundary conditions is

\[
\rho(x, y, z, t) = 1 + 0.2 \sin(\pi (x + y + z - t)),
\]

\[
u(x, y, z, t) = 1, w(x, y, z, t) = 1, \rho(x, y, z, t) = 1.
\] (36)

Numerical tests are conducted on the computational domain of $[0, 2] \times [0, 2] \times [0, 2]$. The uniform hexahedral grids with the
mesh size from \( h = 1/10 \) to 1/30 and the unstructured tetrahedral grids with almost equivalent mesh size from \( h = 1/5 \) to 1/15 are used. Results at \( t = 2 \) are extracted. The density contours computed by the high-order LSFD-FV method on uniform hexahedral and tetrahedral grids with \( h = 1/10 \) are, respectively, presented in Figs. 1 and 2, in which good agreements with the analytical solutions are observed. For quantitative comparison, the \( L_2 \) errors and convergence orders on the hexahedral and tetrahedral meshes are, respectively, given in Tables I and II. These results show that the expected accuracy can be achieved for the developed high-order LSFD-FV method in the 3D case. Moreover, the high-order LSFD-FV method has smaller relative errors than the second-order one, which indicates the higher accuracy of the high-order method on the same mesh. This conclusion can also be visualized through the change curves in Fig. 3.

TABLE I. Accuracy test of the LSFD-FV method on hexahedral grids for the 3D advection of density perturbation.

<table>
<thead>
<tr>
<th>Schemes</th>
<th>Grid size</th>
<th>( L_2 ) errors</th>
<th>Order</th>
</tr>
</thead>
<tbody>
<tr>
<td>2nd order LSFD-FV</td>
<td>1/10</td>
<td>( 2.29 \times 10^{-2} )</td>
<td>2.054</td>
</tr>
<tr>
<td></td>
<td>1/15</td>
<td>( 9.94 \times 10^{-3} )</td>
<td>2.006</td>
</tr>
<tr>
<td></td>
<td>1/20</td>
<td>( 5.58 \times 10^{-3} )</td>
<td>2.006</td>
</tr>
<tr>
<td></td>
<td>1/30</td>
<td>( 2.49 \times 10^{-3} )</td>
<td>1.991</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>1/10</td>
<td>( 6.98 \times 10^{-3} )</td>
<td>3.006</td>
</tr>
<tr>
<td></td>
<td>1/15</td>
<td>( 2.06 \times 10^{-3} )</td>
<td>3.006</td>
</tr>
<tr>
<td></td>
<td>1/20</td>
<td>( 8.61 \times 10^{-4} )</td>
<td>3.041</td>
</tr>
<tr>
<td></td>
<td>1/30</td>
<td>( 2.51 \times 10^{-4} )</td>
<td>3.037</td>
</tr>
</tbody>
</table>

TABLE II. Accuracy test of the LSFD-FV method on tetrahedral grids for the 3D advection of density perturbation.

<table>
<thead>
<tr>
<th>Schemes</th>
<th>Grid size</th>
<th>( L_2 ) errors</th>
<th>Order</th>
</tr>
</thead>
<tbody>
<tr>
<td>2nd order LSFD-FV</td>
<td>1/5</td>
<td>( 1.79 \times 10^{-2} )</td>
<td>2.430</td>
</tr>
<tr>
<td></td>
<td>1/10</td>
<td>( 3.32 \times 10^{-3} )</td>
<td>2.430</td>
</tr>
<tr>
<td></td>
<td>1/15</td>
<td>( 1.41 \times 10^{-3} )</td>
<td>2.101</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>1/5</td>
<td>( 1.39 \times 10^{-2} )</td>
<td>3.443</td>
</tr>
<tr>
<td></td>
<td>1/10</td>
<td>( 1.95 \times 10^{-3} )</td>
<td>2.837</td>
</tr>
<tr>
<td></td>
<td>1/15</td>
<td>( 4.83 \times 10^{-4} )</td>
<td>3.443</td>
</tr>
</tbody>
</table>
B. Lid-driven cavity flow

The lid-driven cavity flow in a cube is then simulated to demonstrate the ability of the present high-order FV method with the LBFS in modeling 3D incompressible viscous flow problems. The physical configuration of this problem is common and can be found in many previous studies.\cite{15,46–48} In this case, the flow is dominated by the Reynolds number defined as $Re = \rho u_0 L / \mu$, where $u_0$ denotes the velocity of the lid, $L$ is the length of the cubic cavity, and $\mu$ refers to the dynamic viscosity. The flow parameters applied in the simulation are given as $\rho = 1$, $u_0 = 0.1$, and $L = 1$. Initially, the dynamic viscosity can be computed from the Reynolds number, and then, the single relaxation parameter can be obtained. Three cases of $Re = 100$, 400, and 1000 are considered here. The $40 \times 40 \times 40$ non-uniform mesh shown in Fig. 5 is used for the simulations at $Re = 100$ and 400. The $60 \times 60 \times 60$ uniform mesh is used for the case at $Re = 1000$. 

**FIG. 3.** $L_2$ error of density on hexahedral mesh (left) and tetrahedral mesh (right) for different schemes.

**FIG. 4.** Efficiency comparison on hexahedral mesh (left) and tetrahedral mesh (right) for different schemes.

efficiency than the second-order one when the same accuracy of the solution is pursued.

**FIG. 5.** Mesh used for the three-dimensional lid-driven cavity flow.
FIG. 6. Three-dimensional lid-driven cavity flow: comparison of the \( u \)-velocity (left) and \( w \)-velocity (right) profiles along vertical and horizontal central lines at the \( x-z \) plane with \( y = 0 \) at (a) \( Re = 100 \), (b) \( Re = 400 \), and (c) \( Re = 1000 \).
The results obtained by the third-order LSFD-FV method are extracted, and the results of Shu et al. using the global method of differential quadrature are taken as the benchmark data. In addition, for comparison purposes, the second-order method is also used to simulate the three cases on the same mesh. Figure 6 shows the computed $w$-velocity profiles along the horizontal centerline and $u$-velocity profiles along the vertical centerline at the $x-z$ plane with $y = 0.5$ for different Reynolds numbers. Clearly, on the coarse grids, the results obtained by the third-order LSFD-FV method match well with the benchmark data and they are in better agreement with the reference data than those of the second-order method. Moreover, the streamlines calculated by the third-order LSFD-FV method at the $y-z$ plane with $x = 0.5$, $x-z$ plane with $y = 0.5$, and $x-y$ plane with $z = 0.5$ at three different Reynolds numbers are

![Streamlines](image)

**FIG. 7.** Three-dimensional lid-driven cavity flow: the streamlines at the $y-z$ plane with $x = 0.5$, $x-z$ plane with $y = 0.5$, and $x-y$ plane with $z = 0.5$ (from left to right) for (a) $Re = 100$, (b) $Re = 400$, and (c) $Re = 1000$. 
presented in Fig. 7. It is observed that the flow patterns agree fairly well with those reported in Refs. 15 and 38. Thus, it can be concluded that the present high-order LSFD-FV method can simulate the 3D incompressible viscous flows reliably with the high-order accuracy.

C. Natural convection in a cubic cavity

The natural convection in a cubic cavity serves as a classic benchmark problem in validating the performance of numerical methods for simulation of the 3D thermal flows. In this problem, two vertical walls at $x = 0$ and $x = L$, respectively, have a cool temperature of $T_0 = 0$ and a hot temperature of $T_1 = 1$. Other four walls are adiabatic. The no-slip boundary conditions are applied on all walls. The flow pattern of this test example is governed by two dimensionless parameters, i.e., the Prandtl number $Pr$ and the Rayleigh number $Ra$. Here, the Rayleigh number $Ra$ is defined as

$$Ra = \frac{g\beta(T_1 - T_0)L^3}{\nu \kappa} = \frac{V_c^2 L^2}{\nu \kappa},$$

(37)

where $V_c = \sqrt{g\beta L \Delta T}$ is the characteristic thermal velocity constrained by the low Mach number limit, $\nu$ denotes the kinematic viscosity, $g$ denotes the gravity acceleration, $\beta$ is the thermal expansion coefficient, and $L$ is the characteristic length of the cubic

![Temperature contours](image_url)
cavity. The effect of the temperature field on the flow field is considered by the buoyancy force, which is exerted on the momentum equation as a source term \( Q = [0, 0, -\rho g \beta (T - T_m), 0]^T \) according to the Boussinesq approximation. \( T_m \) is the average temperature of the flow field. For this test case, we set \( Pr = 0.71 \) and \( V_c = 0.1 \). Four cases of the Rayleigh number \( Ra = 10^3, 10^4, 10^5, \) and \( 10^6 \) are simulated. All simulations are conducted on a hexahedral mesh with \( 40 \times 30 \times 30 \) cells by the high-order LSFD-FV method with the LBFS for flux evaluation.

The computed isotherms on the \( y = 0.5 \) plane at various Rayleigh numbers of \( 10^3 \text{–} 10^6 \) are visualized in Fig. 8. Denser isotherms squeezed close to the isothermal walls are observed in the case at high Rayleigh numbers, which agrees well with those obtained by Wang et al. using the second-order FV method with the LBFS on uniform \( 80 \times 80 \times 80 \) hexahedral grids and Chen et al. using the simplified lattice Boltzmann method on non-uniform \( 80 \times 40 \times 80 \) hexahedral grids. These successful predictions of abrupt flow changes by coarse uniform meshes show that the developed high-order LSFD-FV method enjoys higher accuracy than the conventional second-order method. Furthermore, Fig. 9 compares the obtained \( u \)—velocity along the vertical centerline and \( w \)—velocity along the horizontal centerline on the \( y = 0.5 \) plane with the results of Yang et al. computed by a second-order simple gas-kinetic scheme. Clearly, good agreements are achieved.

![Graphs showing isotherms at different Rayleigh numbers](image_url)
For quantitative comparison, the mean Nusselt number on the $y = 0.5$ plane at the hot wall and the overall Nusselt number at the hot wall defined as follows are compared,

$$Nu_{\text{mean}}(y) = \int_0^1 \frac{\partial T(y,z)}{\partial x} \bigg|_{x=1} dz,$$  

$$Nu_{\text{overall}} = \int_0^1 Nu_{\text{mean}}(y) dy.$$  

The detailed comparisons with the reference data reported in the literature\textsuperscript{38,50,51} are shown in Table III, where the maximum relative error of the high-order method is found to be within 1.6%. Such a good accordance validates the correctness of the high-order LSFD-FV method in the 3D scenario. In addition, it can be seen from Table III that the high-order LSFD-FV method outperforms the second-order method in terms of the accuracy due to the smaller relative error. Moreover, the comparisons of computational efficiency and memory cost between the high- and second-order methods are presented in Table IV. It can be observed that more than 4 times CPU time and larger virtual memory are required by the second-order method to achieve comparable results of the high-order method. This observation indicates that the high-order LSFD-FV method has better computational efficiency.

### D. Flow around ONERA M6 wing

To further quantitatively assess the performance of the present high-order LSFD-FV method in simulating the 3D compressible flow problems with complex configurations, the case of transonic flow around the ONERA M6 wing is tested.

#### TABLE III. Comparison of representative field properties along the symmetric plane at $y = 0.5$ for 3D natural convection at four different Rayleigh numbers: $Ra = 10^3$, $10^4$, $10^5$, and $10^6$. Boldface values denote those computed by the LSFD-FV methods.

<table>
<thead>
<tr>
<th></th>
<th>$Ra = 10^3$</th>
<th>$Ra = 10^4$</th>
<th>$Ra = 10^5$</th>
<th>$Ra = 10^6$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_{\text{max}}$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wang et al.\textsuperscript{38}</td>
<td>0.132</td>
<td>0.200</td>
<td>0.142</td>
<td>0.082</td>
</tr>
<tr>
<td>Yang et al.\textsuperscript{50}</td>
<td>0.132</td>
<td>0.198</td>
<td>0.142</td>
<td>0.081</td>
</tr>
<tr>
<td>Fusegi et al.\textsuperscript{51}</td>
<td>0.131</td>
<td>0.201</td>
<td>0.147</td>
<td>0.084</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>0.132</td>
<td>0.198</td>
<td>0.140</td>
<td>0.080</td>
</tr>
<tr>
<td>2nd order LSFD-FV</td>
<td>0.136</td>
<td>0.202</td>
<td>0.148</td>
<td>0.085</td>
</tr>
<tr>
<td>$Z$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wang et al.\textsuperscript{38}</td>
<td>0.187</td>
<td>0.176</td>
<td>0.147</td>
<td>0.146</td>
</tr>
<tr>
<td>Yang et al.\textsuperscript{50}</td>
<td>0.179</td>
<td>0.179</td>
<td>0.142</td>
<td>0.142</td>
</tr>
<tr>
<td>Fusegi et al.\textsuperscript{51}</td>
<td>0.200</td>
<td>0.183</td>
<td>0.145</td>
<td>0.144</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>0.179</td>
<td>0.174</td>
<td>0.147</td>
<td>0.141</td>
</tr>
<tr>
<td>2nd order LSFD-FV</td>
<td>0.167</td>
<td>0.174</td>
<td>0.133</td>
<td>0.141</td>
</tr>
<tr>
<td>$w_{\text{max}}$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wang et al.\textsuperscript{38}</td>
<td>0.133</td>
<td>0.221</td>
<td>0.244</td>
<td>0.253</td>
</tr>
<tr>
<td>Yang et al.\textsuperscript{50}</td>
<td>0.133</td>
<td>0.221</td>
<td>0.245</td>
<td>0.255</td>
</tr>
<tr>
<td>Fusegi et al.\textsuperscript{51}</td>
<td>0.132</td>
<td>0.225</td>
<td>0.247</td>
<td>0.259</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>0.133</td>
<td>0.221</td>
<td>0.244</td>
<td>0.261</td>
</tr>
<tr>
<td>2nd order LSFD-FV</td>
<td>0.138</td>
<td>0.228</td>
<td>0.266</td>
<td>0.280</td>
</tr>
<tr>
<td>$x$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wang et al.\textsuperscript{38}</td>
<td>0.829</td>
<td>0.885</td>
<td>0.932</td>
<td>0.968</td>
</tr>
<tr>
<td>Yang et al.\textsuperscript{50}</td>
<td>0.821</td>
<td>0.883</td>
<td>0.932</td>
<td>0.957</td>
</tr>
<tr>
<td>Fusegi et al.\textsuperscript{51}</td>
<td>0.833</td>
<td>0.883</td>
<td>0.935</td>
<td>0.967</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>0.821</td>
<td>0.882</td>
<td>0.940</td>
<td>0.960</td>
</tr>
<tr>
<td>2nd order LSFD-FV</td>
<td>0.821</td>
<td>0.882</td>
<td>0.940</td>
<td>0.960</td>
</tr>
<tr>
<td>$Nu_{\text{mean}}$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wang et al.\textsuperscript{38}</td>
<td>1.092</td>
<td>2.289</td>
<td>4.622</td>
<td>8.921</td>
</tr>
<tr>
<td>Yang et al.\textsuperscript{50}</td>
<td>1.088</td>
<td>2.251</td>
<td>4.606</td>
<td>8.771</td>
</tr>
<tr>
<td>Fusegi et al.\textsuperscript{51}</td>
<td>1.105</td>
<td>2.302</td>
<td>4.646</td>
<td>9.012</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>1.088</td>
<td>2.251</td>
<td>4.585</td>
<td>8.639</td>
</tr>
<tr>
<td>2nd order LSFD-FV</td>
<td>1.099</td>
<td>2.359</td>
<td>5.076</td>
<td>10.163</td>
</tr>
<tr>
<td>$Nu_{\text{overall}}$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wang et al.\textsuperscript{38}</td>
<td>1.071</td>
<td>2.062</td>
<td>4.344</td>
<td>8.684</td>
</tr>
<tr>
<td>Yang et al.\textsuperscript{50}</td>
<td>1.071</td>
<td>2.055</td>
<td>4.331</td>
<td>8.540</td>
</tr>
<tr>
<td>Fusegi et al.\textsuperscript{51}</td>
<td>1.085</td>
<td>2.085</td>
<td>4.361</td>
<td>8.770</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>1.071</td>
<td>2.056</td>
<td>4.319</td>
<td>8.432</td>
</tr>
<tr>
<td>2nd order LSFD-FV</td>
<td>1.080</td>
<td>2.155</td>
<td>4.798</td>
<td>9.965</td>
</tr>
</tbody>
</table>
TABLE IV. Comparison of computational cost of the third-order and second-order LSFD-FV methods for natural convection in a cubic cavity at $Ra = 10^4$.

<table>
<thead>
<tr>
<th>Methods</th>
<th>Third order LSFD-FV</th>
<th>Second order LSFD-FV</th>
<th>Second order LSFD-FV</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grids</td>
<td>$40 \times 30 \times 30$</td>
<td>$40 \times 30 \times 30$</td>
<td>$100 \times 100 \times 100$</td>
</tr>
<tr>
<td>$N_t^{mean}$</td>
<td>2.251</td>
<td>2.359</td>
<td>2.289</td>
</tr>
<tr>
<td>$N_t^{overall}$</td>
<td>2.056</td>
<td>2.155</td>
<td>2.090</td>
</tr>
<tr>
<td>Virtual memory (Megabytes)</td>
<td>431.5</td>
<td>286.8</td>
<td>1898.1</td>
</tr>
<tr>
<td>Iteration steps</td>
<td>8544</td>
<td>8617</td>
<td>20929</td>
</tr>
<tr>
<td>Computational time (s)</td>
<td>26 505.9</td>
<td>7697.9</td>
<td>115 629.5</td>
</tr>
</tbody>
</table>

Simulation, the free-stream Mach number is 0.84 and the angle of attack is 3.06°. The computational mesh contains 347,718 tetrahedral cells, where 13,236 triangular cells are set on the surfaces of the wing as shown in Fig. 10.

Figure 11 depicts the pressure contours obtained by the present high-order LSFD-FV method. The “λ” shape shock wave on the upper surface of the wing can be clearly seen, which agrees well with the results reported by Yang et al.52 Figure 12 shows the pressure coefficient distributions at selected spanwise locations computed by the high-order LSFD-FV method. The experimental results53 are also presented in Fig. 12 for comparison. The results of the high-order LSFD-FV method quantitatively agree well with the experimental data. Overall, the above results confirm the capability of the present high-order FV method for simulation of non-smooth compressible inviscid flows with complex geometry.

E. Steady subsonic viscous flow around a sphere

To test the performance of the developed high-order LSFD-FV method in simulating the 3D compressible viscous flows with curved boundaries, the problem of the steady viscous flow around a sphere54 is solved. The Reynolds number is set to be 118 based on the spherical diameter, and the inlet Mach number is 0.2535, which is in line with the experimental condition55 and the computational condition.54,56 The spherical diameter is taken as 1, and the computational domain is a cylinder whose diameter is 10 and height is 25. This computational domain is discretized by 576,631 tetrahedrons, and there are 5574 triangular cells on the walls. The mesh near the sphere is refined, and the corresponding mesh size is set as $h = 0.04$.

Figure 12 shows the computational mesh. Since the flow field in this simulation is smooth, no shock-capturing technique is required. The computations are conducted using the second- and high-order methods with the DGKFS. Figure 14 presents the computed Mach number contours and streamlines near the wake region by the high-order LSFD-FV method. It can be confirmed that the obtained Mach number contours, streamlines, and the size of the separation zone are in good agreement with both the experimental data and the numerical results in the literature shown in Fig. 15. To quantitatively validate the correctness of the computed results, the computed drag coefficients,

![FIG. 10. Unstructured mesh on the walls and symmetry boundary of the ONERA M6 wing.](image_url)

![FIG. 11. Pressure contours obtained by the high-order LSFD-FV method for flow around the ONERA M6 wing.](image_url)
FIG. 12. Comparison of pressure coefficient distributions on the ONERA M6 wing at the (a) 20%, (b) 44%, (c) 65%, (d) 80%, (e) 90%, and (f) 95% spanwise stations.
FIG. 13. Computational mesh for the subsonic viscous flow around a sphere. Top: entire mesh; bottom: mesh around the sphere.

predicted separation angle $\theta_s$ and wake length $L_s$ by the second- and high-order methods are compared with the data reported in Refs. 27, 54, 56, and 57. The details are tabulated in Table V, where the results of the high-order scheme agree better with the reference data than the second-order one. This observation validates the correctness and robustness of the high-order LSFD-FV method in solving the 3D compressible viscous flow with curved boundaries on unstructured mesh.

F. Steady supersonic viscous flow around a sphere

The supersonic flow around a sphere is further tested to assess the capacity of the current method for capturing the shock wave. In
Table V. Comparison of separation angle $\theta_s$, wake length $L_s$, and drag coefficient $C_d$ for the steady subsonic viscous flow around a sphere. Boldface values denote those computed by the LSFD-FV methods.

<table>
<thead>
<tr>
<th>$Re$</th>
<th>$C_d$</th>
<th>$\theta_s$ (deg)</th>
<th>$L_s/D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>118</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Haga et al. 54</td>
<td>. . .</td>
<td>123.6</td>
<td>. . .</td>
</tr>
<tr>
<td>Cheng et al. 56</td>
<td>1.0162</td>
<td>123.7</td>
<td>0.96</td>
</tr>
<tr>
<td>Castonguay 57</td>
<td>1.0162</td>
<td>123.6</td>
<td>. .</td>
</tr>
<tr>
<td>Sun et al. 52</td>
<td>. . .</td>
<td>123.6</td>
<td>1.04</td>
</tr>
<tr>
<td>2nd order LSFD-FV</td>
<td>1.0245</td>
<td>129.3</td>
<td>0.94</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>1.0156</td>
<td>123.7</td>
<td>0.96</td>
</tr>
</tbody>
</table>

the simulation, the Reynolds number based on the spherical diameter and the inlet Mach number are 300 and 2, respectively. Under these conditions, as discussed in Ref. 58, the flow field becomes steady and axisymmetric.

Figure 16 presents the flow domain used in the simulation. It is centered around the sphere whose diameter $D$ is 1. The far field boundary is at 24.8$D$. There are 362,880 hexahedrons to discretize the flow domain. Near the sphere, the computational grid is refined with a mesh size of $h = 0.023$. No-slip and far field boundary conditions are, respectively, enforced on the sphere and the far field boundaries. In Fig. 17, the streamlines and Mach number contours near the wake region obtained by the high-order LSFD-FV method are depicted. Clearly, stable and axisymmetric flow structures are obtained. To quantitatively assess the results, Table VI compares some characteristic parameters computed by the present high-order method, i.e., the drag coefficient $C_d$, shock stand-off distance $L_{ssd}$, separation angle $\theta_s$, and wake length $L_s$, with those in the literature. 58–60 It is observed that the results are in good agreement. These outcomes well validate the capability of the current method for capturing the shock waves in supersonic flow problems.

Table VI. Comparison of drag coefficient $C_d$, shock stand-off distance $L_{ssd}$, separation angle $\theta_s$, and wake length $L_s$ for the steady supersonic viscous flow around a sphere. Boldface values denote those computed by the LSFD-FV methods.

<table>
<thead>
<tr>
<th>$Re$</th>
<th>$C_d$</th>
<th>$L_{ssd}$</th>
<th>$\theta_s$ (deg)</th>
<th>$L_s/D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>300</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nagata et al. 59</td>
<td>1.387</td>
<td>0.20</td>
<td>150.0</td>
<td>0.36</td>
</tr>
<tr>
<td>Riahi et al. 59</td>
<td>1.41</td>
<td>0.20</td>
<td>. . .</td>
<td>. .</td>
</tr>
<tr>
<td>Nagata et al. 55</td>
<td>1.452</td>
<td>. . .</td>
<td>148.6</td>
<td>0.39</td>
</tr>
<tr>
<td>3rd order LSFD-FV</td>
<td>1.395</td>
<td>0.21</td>
<td>149.7</td>
<td>0.36</td>
</tr>
</tbody>
</table>
IV. CONCLUSIONS

In this paper, a 3D high-order least square-based finite difference-finite volume method on unstructured grids is proposed for incompressible and compressible flows. This 3D LSFD-FV method directly applies a Taylor series expansion within each control cell as the solution approximation polynomial. The unknown derivatives are approximated by the mesh-free LSFD scheme. Due to such straightforward and simple approaches of solution approximation and derivative approximation, the resultant LSFD-FV method is user-friendly and easy to implement. Various flux solvers are adopted to evaluate the fluxes at the cell interface. Especially for simulations of incompressible and compressible viscous flows, the LBFS and GKSs are, respectively, applied. In this process, the inviscid and viscous fluxes are simultaneously computed, which avoids additional treatments for the viscous discretization. This makes the LSFD-FV method competitive in simulating viscous flow problems. Due to the application of the mesh-free LSFD method and flux solvers at each cell interface, the LSFD-FV method is flexible for solving problems with complex geometries on unstructured grids.

The performance of the 3D high-order LSFD-FV method is examined systematically by representative incompressible and compressible benchmark cases. First, the third-order of accuracy is validated on both hexahedral and tetrahedral grids. Second, the numerical results of the high-order LSFD-FV method agree better with the benchmark data than the conventional low-order method on the same grids. Additionally, the present high-order method outperforms the low-order one in terms of computational efficiency. For instance, for natural convection in a cubic cavity at $Ra = 10^5$, the high-order LSFD-FV method takes only about 22.9% computational time and 22.7% virtual memory of the low-order method to get converged results. Furthermore, the present high-order method successfully simulates the compressible flows with complex geometries on relatively coarse tetrahedral grids, which demonstrates its capability of handling unstructured mesh. Given the performance stated above, this 3D high-order LSFD-FV method looks promising in practical problems of engineering interest.

DATA AVAILABILITY

The data that support the findings of this study are available within the article.

REFERENCES


