On underresolved simulations of compressible turbulence using an entropy-bounded DG method: Solution stabilization, scheme optimization, and benchmark against a finite-volume solver

Yu Lv*, Peter C. Ma, Matthias Ihme

Department of Mechanical Engineering, Stanford University, Stanford, CA, 94305, USA

A R T I C L E   I N F O

Article history:
Received 13 May 2017
Revised 26 September 2017
Accepted 23 November 2017
Available online 24 November 2017

Keywords:
Discontinuous Galerkin method
Large-eddy simulation
Entropy-bounded
Turbulence
High-order methods

A B S T R A C T

This work focuses on assessing a set of discontinuous Galerkin (DG) discretization methods for high-fidelity simulations of compressible turbulence, considering a range of conditions that include very high turbulence Mach numbers. The sensitivities of numerical results to solution-stabilization techniques, viscous/inviscid flux formulations, hp-refinement, and operating conditions are investigated. Emphasis is placed on an entropy-bounded DG (EBDG) method that can genuinely enforce physical-realizability constraints for discrete solutions. The behaviors of EBDG solutions are characterized in a set of underresolved simulation settings. It is found that for turbulence at low and moderate Mach numbers, an increase in polynomial order (p-refinement) is more effective in capturing turbulence characteristics, while for highly compressible turbulence regimes, h-refinement becomes beneficial. The EBDG method is further benchmarked against an optimized finite-volume (FV) scheme that has second-order accuracy on generic meshes while retaining fourth-order accuracy on Cartesian meshes. As a result of this fact, the benchmark study shows that on regular hexahedral meshes FV scheme is less costly than the EBDG scheme for the same accuracy, while the EBDG method becomes significantly more efficient than the FV scheme on tetrahedral and irregular meshes.

© 2017 Elsevier Ltd. All rights reserved.

1. Introduction

Over recent years there has been an increasing interest in using high-order discontinuous Galerkin (DG) methods for high-fidelity simulations of turbulent flows, such as large-eddy simulations (LES) and direct numerical simulations (DNS). In particular, DG methods have shown potential for improving the predictive capability for LES/DNS applications. Carton de Wiart et al. [1] compared the performance of DG schemes with a non-dissipative finite-difference (FD) scheme for the prediction of inviscid vertical flows. They showed that their DG method outperforms the FD method in representing the turbulence spectrum at medium wavenumbers. Collis and Scott [2], Wei and Pollard [3], and Carton de Wiart et al. [4] applied DG schemes to turbulent channel flows. Wall-resolved simulations showed excellent agreement with well-established reference data from high-order FD and spectral methods. In the work of Collis and Scott [2], the benefit of using adaptive p-refinement to resolve near-wall flow structures was highlighted. The accuracy of DG schemes in underresolved calculations was assessed in the study of Gassner and Beck [5] by considering a Taylor–Green vortex case. It was found that simulations without explicitly adding sub-grid scale (SGS) models are comparable to those generated by low-order schemes with SGS-models. The same Taylor–Green case was also considered to examine the performance of other variations of DG schemes [6–9]. Beck et al. [10] and Carton de Wiart and Hillewaert [11] further explored implicit LES strategies for practical configurations by considering a separated flow over an airfoil. Compared to experimental data, their predictions for aerodynamic loads and separation locations were comparable with those obtained from FD/FV (finite-volume) schemes. The related research on developing new DNS/LES capabilities is also observable in the study of flux-reconstruction schemes [12–15].

Despite these encouraging results, DG methods have not yet been fully established for high-fidelity simulations. Several research questions remain open, especially in regard to applying DG methods to LES. Solution stabilization is one of the important issues. Previous studies [5,6,13] have exemplified the failure of underresolved DG simulations for turbulent flows. To improve the robustness of DG simulations, application of de-aliasing techniques, such as over-integration [5], can be helpful. However, the effectiveness of these treatments was mostly evaluated for
compressible flow regimes. For transonic or supersonic turbulence regimes, eddy motions induce dilatation effects, leading to the formation of shocklets, which can further challenge the robustness of DG calculations. Successful simulations of turbulent flows under these conditions require DG schemes that avoid the generation of unphysical numerical solutions. In this context, positivity-preserving [16,17] and entropy-bounded DG schemes [18], in which the solution is regularized using physical principles, represent attractive candidates to be applied to LES. Evaluating the merit of these methods for turbulent flow simulations, however, requires further assessment.

In addition to the stabilization of the numerical solution, another important question is related to the selection of the optimal algorithmic implementation. DG schemes, although based on the same variational form, can vary in different manners, such as by order of accuracy, representation of basis function, or the formulation of convective and viscous fluxes. Understanding the behavior of different schemes, especially in the context of LES, is still not adequate. In general, the order of accuracy (polynomial order) greatly alters the properties of DG schemes, subsequently influencing the behavior of the DG scheme in LES calculations [1,19]. In this regard, one interesting point is to examine if the high-order accuracy will benefit LES predictions, in which simulations are underresolved. DG schemes also vary in the form of Riemann solvers and viscous discretization schemes that are employed, and studies [4] found that Riemann solvers can affect the energy spectra predicted by DG-based LES. However, the optimal combination of Riemann solver and viscous discretization for DG-based LES calculations remains an open question.

To advance DG-based LES techniques, it is also useful to recognize the performance of DG methods and computational cost as compared to state-of-the-art numerical techniques. However, an objective evaluation of the efficiency of a method requires analyzing the computational accuracy, and an objective comparison of the computational cost should be done with respect to the same numerical error. Although previous studies [1,5] reported benchmark results, a comprehensive LES benchmark between DG and commonly used FV schemes, which considers a broader range of turbulence regimes and variations in mesh topologies, is still missing.

By addressing these aspects, the objective of this work is to assess the performance of high-order DG schemes in application to under-resolved turbulent flow calculations. In the context of LES, this topic is related to the so-called implicit LES methodology, in which no subgrid-scale closures are employed to represent unresolved scales of the governing equations. The subgrid-scale effects are implicitly represented by the inherent dissipation of the numerical scheme. The performance of LES is linked to the numerical resolution. Since the focus of this study is in utilizing DG schemes for high-performance computations, these investigations are concerned with relatively well resolved LES applications that are able to at least capture the inertial subrange. With this scope, it is expected that the order of accuracy of the DG scheme will play a key role in determining the performance of LES. Therefore, the focus will be placed on assessing an entropy-bounded DG (EBDG) scheme up to 4th-order polynomial representation (p = 4) that are more likely to be employed in practically relevant LES computations.

To assist further development of DG-based LES techniques, the specific objectives of this work are: i) to evaluate suitable solution stabilization techniques for underresolved DG calculations; ii) to understand the performance of different implementations of DG schemes in terms of polynomial order, and convective-viscous flux formulations; iii) to assess the status of DG methods through detailed comparisons against a state-of-the-art FV method.

The remainder of this manuscript is structured as follows. Section 2 introduces the governing equations, which is followed by providing details of the DG discretization in Section 3. After the problem configurations of the benchmark cases are discussed in Sections 4 and 5 focuses on examining different solution-stabilization techniques. Section 6 discusses the performance of different DG implementations with different Riemann solvers and viscous discretization schemes. Section 7 reports results of the EBDG solver versus a state-of-the-art FV solver. The manuscript finishes by summarizing the findings of this study.

2. Governing equations

The governing equations under consideration describe the conservation of mass, momentum and energy:

\[ \frac{\partial}{\partial t} \rho + \nabla \cdot (\rho u) = 0, \quad (1a) \]

\[ \frac{\partial}{\partial t} (\rho u) + \nabla \cdot (\rho uu + pI) = \nabla \cdot \tau, \quad (1b) \]

\[ \frac{\partial}{\partial t} (\rho e) + \nabla \cdot (\rho e u + p u) = -\nabla \cdot q + \nabla \cdot (\tau \cdot u), \quad (1c) \]

where \( \rho, u, p, e \) are density, velocity, pressure and specific total energy, respectively, and \( \mathbf{I} \) is the identity matrix. The viscous stress tensor \( \tau \) and the heat flux vector \( q \) are written as:

\[ \tau = -\frac{2}{3} \mu (\nabla u + (\nabla u)^T), \quad (2a) \]

\[ q = -\kappa \nabla T, \quad (2b) \]

in which \( \mu \) is the dynamic viscosity and \( \kappa \) is the heat conductivity. The system of equations, Eq. (1), is closed with the ideal gas relation:

\[ p = (\gamma - 1) \left( \rho e - \frac{1}{2} \rho u \cdot u \right), \quad (3) \]

where \( \gamma \) is the ratio of specific heat capacities.

3. DG discretization

This section introduces the DG discretization scheme. For notational clarity, Eq. (1) is written in the vector form as:

\[ \frac{\partial}{\partial t} U + \nabla \cdot F - \nabla \cdot Q = 0, \quad (4) \]

in which \( U \in \mathbb{R}^{N_e} \), \( F \in \mathbb{R}^{N_e \times N_f} \) and \( Q \in \mathbb{R}^{N_e \times N_q} \) refer to the vector of the solution variables, the convective fluxes and the viscous fluxes, respectively, with \( N_e \) being the number of solution variables and \( N_f \) being the number of spatial dimensions. Similar to common CFD methods, the DG discretization of Eq. (4) is based on a domain partition. We consider the problem to be posed on the domain \( \Omega \) with boundary \( \partial \Omega \). A mesh partition is defined as \( \Omega = \bigcup_{e=1}^{N_e} \Omega_e \), where \( N_e \) is the total number of elements and \( \Omega_e \) corresponds to a discrete element of this partition. The edge of element \( \Omega_e \) is defined as \( \partial \Omega_e \). On each element, \( \Omega_e \), any \( L^2 \)-function can be used to approximate the solution. Polynomial functions are typically used in this regard due to the well-known mathematical properties and the ease to carry out numerical integrations. Thus, we introduce the function space \( P^p \) to denote a set of polynomials with order not higher than \( p \). With this, the mathematical definition of a DG-space can be written as:

\[ \mathbb{V}_h^p = \{ \phi \in L^2(\Omega) : \phi \equiv \phi|_{\Omega_e} \in P^p, \forall \Omega_e \in \Omega \}, \quad (5) \]

where \( \phi_e \) is the basis function defined on \( \Omega_e \). With this, the element-wise solution \( U_e \) that approximates \( U \) takes the form:
\( U_p(x, t) = \sum_{i=1}^{N_p} \tilde{U}_{e,i}(t) \phi_{e,i}(x), \quad x \in \Omega_e, \)

where \( N_p \) is the number of basis functions and \( \tilde{U}_{e,i} \) refers to the coefficient associated with the \( i \)th basis function, \( \phi_{e,i} \). Since continuity across element interfaces is not required, we can denote the solution representation of the entire domain \( \Omega \) as,

\[ U \simeq U = \sum_{p=1}^{N_p} U_p, \]

where \( \simeq \) refers to a direct sum in function space. With this, \( U \in V_p \), \( U \in V_p^e \), which means that the global solution falls into the DG space defined by Eq. (5).

The semi-discrete form of the Navier–Stokes equations on the element \( \Omega_e \) is obtained by multiplying Eq. (4) with the test function \( \phi \) and integrating over \( \Omega_e \):

\[ \int_{\Omega_e} \phi \cdot (\partial_t U + \nabla \cdot F - \nabla \cdot Q) dx \]

\[ = \int_{\Omega_e} (\phi \cdot \partial_t U - \nabla \phi \cdot F) dx + \int_{\partial \Omega_e} \phi \cdot \hat{n} ds - \int_{\Omega_e} \phi \cdot \nabla \cdot Q dx, \]

\[ \approx \int_{\Omega_e} \phi \cdot \phi_x \cdot \tilde{U}(t) dV - \int_{\Omega_e} \nabla \phi \cdot F(U_{e}) dx + \int_{\partial \Omega_e} \phi \cdot \hat{n} ds - \int_{\Omega_e} \phi \cdot \nabla \cdot Q dx, \]

in which Eq. (8) utilize integration-by-parts, and Eq. (8c) is obtained by approximating \( U \) on element \( \Omega_e \) by \( U_e \). The last term on the right-hand-side of Eqs. (8b) and (8c), describing the viscous flux, requires further treatment, which will be addressed in the following paragraph. The outward-pointing normal vector with respect to the element \( \Omega_e \) is denoted by \( \hat{n} \) and \( F \) is the Riemann flux. It is important to note that the DG solution at the element boundary \( \partial \Omega_e \) is double-valued. Therefore, the superscripts + and - are introduced to distinguish the interior and exterior of the solution with respect to \( \partial \Omega_e \). By setting Eq. (8c) to zero, the semi-discretized DG form for the Navier–Stokes equations is obtained. The remaining issues are then the approximation of the viscous operator and the Riemann flux \( F \). Approximations of the Riemann fluxes have been examined extensively in the TV community [20]. In the present study, we will assess the numerical performance of different DG implementations associated with Rusnak [21], Roe [22], central and preconditioned Roe [23] fluxes.

Since the Navier–Stokes equations involve a set of diffusion terms, the first step in the formulation of a discretized form is to linearize \( Q \) with respect to \( \nabla U \). Using the index \( i \) to represent the first dimension of \( Q \), we denote \( \bar{Q}_i \) as the diffusion flux of the Navier–Stokes equations for the ith solution variable. \( Q_i \in \mathbb{R}^{N_e} \) and each component of \( Q_i \) can involve contributions from all spatial derivatives of the solution variables. Therefore, \( Q_i \) can be written as \( Q_i = \sum_{j=1}^{N_e} Q_{ij}^{(i)} \) (omitting Einstein summation convention), with \( Q_{ij}^{(i)} = D_i \nabla U_j \), where \( D_{ij} \in \mathbb{R}^{N_e \times N_e} \). In the following, we will provide the discretization for \( Q_{ij}^{(i)} \), and the discretization for \( Q_i \) directly following using the distributive property of addition. The bilinear form of \( Q_{ij}^{(i)} \) can then be derived as follows [24]:

\[ \int_{\Omega_e} \phi_i \cdot \nabla \cdot Q_{ij}^{(i)} dx = \int_{\Omega_e} \nabla U_j \cdot \nabla \phi_i dx + \int_{\partial \Omega_e} \phi_i Q_{ij}^{(i)} \cdot \hat{n} ds, \]

\[ = -\int_{\Omega_e} \nabla U_j \cdot \nabla \phi_i dx + \int_{\partial \Omega_e} \phi_i Q_{ij}^{(i)} \cdot \hat{n} ds, \]

\[ = \int_{\Omega_e} \nabla U_j \cdot \nabla \phi_i dx + \int_{\partial \Omega_e} \phi_i Q_{ij}^{(i)} \cdot \hat{n} ds. \]
[1.4–6,10,30]. However, except of the work by Johnsen et al. [31], previous investigations focused mostly on low Mach number conditions ($M_a ≲ 0.1$). To examine the performance of DG schemes in applications to compressible turbulence over a relatively broad range of conditions, the present study considers four cases with different Reynolds and Mach numbers. As listed in Table 1, these cases are described by the turbulence Reynolds number, $Re_{τ} = \frac{u'λ/\sqrt{c}}{v}$, and the turbulence Mach number, $Ma_{τ} = \frac{μ\sqrt{c}/(\gamma - 1) \rho_0}{\sqrt{c}}$, where $c$ is the speed of sound, $λ$ is the Taylor microscale, $u'$ is the root-mean-square (RMS) velocity, and $(\cdot)$ is the volumetric averaging operator. The associated eddy turnover time is defined as $τ = \frac{λ}{u'}$, and $τ$ is chosen to be 0.5 for all cases considered. A 3D periodic domain, $[-π, π]^{3}$, is employed. The specific heat ratio is set to a value of $\gamma = 1.4$, and the pressure and density are normalized so that the gas constant is unity. The molecular viscosity is calculated by considering the power law dependence on temperature, $\mu/\mu_0 = (T/T_0)^{0.76}$, where $\mu_0 = 0.0125$ Pa·s. The Prandtl number is set to be a constant with a value of 0.7.

Initial conditions for the current study are given by a divergence-free velocity field and constant thermodynamic quantities ($\rho_0 = 1$ and $\rho_0$ provided in Table 1). The energy spectrum for the initial field for $u'$ is prescribed as:

$$E_u(k) = 16\left(\frac{2}{\pi}\right)\frac{k^{4}}{k^{5}}\exp\left(-\frac{2k^{2}}{k^{5}}\right). \quad (12)$$

where $k_0$ is the most energetic wavenumber, which is related to the initial Taylor microscale through $k_0 = 2λ_0$. Based on this spectrum, a 3D velocity-field database is generated and used throughout this study. The velocity data is provided as supplementary materials. No spatial variation is introduced to the density and pressure. In Table 1, a combination of different $Re_{τ}$ and $Ma_{τ}$ is presented along with corresponding thermodynamic quantities and initial Kolmogorov length scale $λ_0$. It is noted that Case 3 is identical to the flow configuration considered in [31,32]. A more challenging case, Case 4, is also considered, in which the turbulence is in the supersonic regime. From this table, it can be seen that the initial pressure corresponding to Case 4 is very low, and enforcing physical realizability for numerical solutions becomes a major issue in this case. In regard to the initialization, for each DG calculation the pre-generated velocity data is directly projected onto the DG space using a least-square method. This approach is adequate for obtaining identical initial conditions for DG schemes with different polynomial bases, because the initial velocity field only contains larger scales that can be fully represented with the given resolution. Specific efforts are taken to make sure that the initial conditions for DG and FV solvers are consistent for all LES cases. The velocity fields for the FV simulations are not directly initialized using the databases; instead, each FV simulation uses the corresponding initial DG solution evaluated at the cell-center coordinates as the initial condition. This approach effectively guarantees the consistency of initial conditions between different schemes.

In terms of quantities of interests, the emphasis of the present study is directed towards examining: i) the representation of compressibility effects, and ii) the accuracy in predicting the decay rate of turbulence kinetic energy (TKE), which is given as:

$$E = \frac{1}{2} \int_{Ω} \rho u \cdot u \, dx. \quad (13)$$

In the following, the temporal decay rate of TKE is computed using a fourth-order finite difference approximation.

4.2. DG solver

In the present study, an EBDG solver is employed, which has been extensively validated for several flow configurations [18,33–36]. These simulations confirmed that the numerical solutions obtained by the solver are able to achieve $(p + 1)$th-order convergence on different types of meshes and sufficiently smooth flows. Convergence and validation study will not be repeated in the present study. This DG solver supports a variety of solution procedures with different polynomial bases, Riemann solvers and viscous discretization schemes. For the purpose of the present study, Lagrange polynomial bases are utilized to represent the numerical solution. Different element shapes, including hexahedra and tetrahedra, are considered and illustrated in Fig. 1. The number of polynomial bases per element is $N_p = (p + 1)^3$ on hexahedra and $N_p = (p + 1)(p + 2)(p + 3)/6$ on tetrahedra. Gauss quadrature with tensor product augmentation is employed for hexahedral elements, while quadrature on tetrahedral elements is conducted using the quadrature method proposed in [37]. The quadrature rules that can exactly integrate polynomials of order $2p + 1$ are applied for regular calculations. To keep a generic framework, we only consider the nodal DG implementation, in which equidistant nodes (on the reference element) are used for interpolation and the points specified by a given quadrature rule are used for numerical integration. The collocation-type approximation [38] is not investigated. The present DG solver employs an entropy-bounded scheme [18], which will be assessed as a technique for solution stabilization in Section 5. A standard fourth-order Runge–Kutta scheme is used for time-integration.

5. Solution-stabilization techniques

It has been recognized that robustness issues arise from high-order DG simulations with insufficient resolutions, even for incompressible flow conditions [6,13]. It is anticipated that for turbulence at transonic or supersonic regimes the robustness of DG-based LES techniques will become more severe, due to dilatational effects and the formation of shocklets. Therefore, the introduction of certain mechanisms to stabilize LES solutions is necessary from a pragmatic point of view. De-aliasing using over-integration was adopted in high-order DG solutions for LES, primarily for incompressible flow conditions [5,10]. A hierarchic limiting procedure was applied for simulations of turbulence at transonic conditions [7], but it was found that this implementation becomes rather involved for DG schemes with polynomial order higher than two. Artificial viscosity methods provide another treatment to stabilize DG solutions. Recent studies on Case 3 [19] showed that the introduction of artificial viscosity can effectively avoid the solution divergence for LES of turbulence at transonic conditions. However, the TKE prediction exhibits strong sensitivity to the magnitude of artificial viscosity, and additionally there is a lack of guideline for determining the magnitude of artificial viscosity. The present study places interests on assessing the EBDG scheme [18] for LES of turbulence. This scheme is a generalization of Zhang and Shu’s positivity-preserving DG schemes [16,17] to arbitrarily high-order discretizations and arbitrarily shaped elements. Compared to other solution-stabilization approaches mentioned above, the EBDG scheme provides mathematically provable stability for compressible flow calculations by preventing unphysical quantities. This
property makes it potentially useful for LES applications. The key idea of the EBDG scheme is to enforce entropy boundedness on subcell solutions. For this, an entropy bound is determined initially based on the flow conditions, \( s_b = \ln(c p_0/\rho_0^4) \), where \( p_0 \) and \( \rho_0 \) respectively, correspond to the pressure and density of the initial condition, and \( C \) is a user-input parameter. At the algorithmic level, the entropy of the subcell solution \( U_e \) in each element is monitored during time advancement. Once \( U_e \) produces entropy undershoots with respect to \( s_b \) (at quadrature points), a local and conservative scaling operator, initially proposed in [16,17], is utilized to correct the subcell solutions in troubled elements. The corrected subcell solution can be expressed as \( U_e = \bar{U}_e + \epsilon (U_e - \bar{U}_e) \). The correction is formulated as a problem that is to find the maximum of \( \epsilon \) such that the entropy of \( U_e \) is not lower than \( s_b \) (at quadrature points). This treatment ensures that the entropy values of the subcell solutions are bounded from below.

To examine the sensitivity of the solutions to \( s_b \), two different values of \( s_b \) are selected in the present investigation. This includes a restrictive bound with \( C = 0.9 \) and a less restrictive bound with \( C = 0.001 \). As for more details related to the EBDG scheme, the reader is referred to the original paper [18]. For comparison, DG solutions with overintegration as a de-aliasing treatment are also considered. Overintegration is performed using twice the amount of quadrature points along each spatial dimension. Due to the computational cost, further increase of quadrature points was found to make the calculations impractical. The procedure of overintegration technique was described in [5].

To examine the solution-stabilization technique, we consider Cases 3 and 4. These case are characterized by strong turbulence and compressibility effects and are expected to induce robustness issue in the DG solution. Simulations are performed using a set of EBDG schemes with different polynomial representations on a mesh with \( 64^3 \) DOF. The results for Cases 3 and 4 are provided in Figs. 2 and 3, respectively. The solutions of Case 3, including both the decay of TKE and density fluctuations, are insensitive to the value of \( s_b \). This behavior is observed for the considered EBDG schemes of different polynomial orders. The de-aliased DG calculations using overintegration are able to capture the first peak of the TKE decay that is triggered by strong dilatational effects; however, all calculations diverge at around \( t/\tau = 0.6 \). In contrast to Case 3, solutions of Case 4 exhibit some sensitivity to the value of \( s_b \). Comparatively speaking, the solutions corresponding to the less restrictive value of \( s_b \) are closer to the DNS reference solutions. The solutions obtained using the restrictive entropy bound tend to produce larger overpredictions in the decay rate, when the TKE is dominated by initial dilatational effects. Due to the stronger dilatation in Case 4 as compared to Case 3, the overintegrated DG calculations diverge at a much earlier time. It is evident that all
EBDG solutions slightly underpredict the density fluctuation. This can be explained since the EBDG scheme regularizes solutions by constraining the variations of thermodynamic quantities. Therefore, setting a more restrictive entropy bound, which is equivalent to setting a stronger constraint, enlarges the resulting underprediction for density fluctuation. Overall, for the flow configuration considered in this study, it is suggested to apply a relatively less restrictive entropy bound for the application of EBDG scheme. In the following investigation, \( s_0 = \ln(0.001 \rho_0/\rho_0^*) \), corresponding to \( C = 0.001 \), will be used for all EBDG calculations.

6. Different flux-function formulations

The discussion in this section focuses on the performance of different DG implementations. As introduced in Section 3, there are a number of options for Riemann and viscous flux formulations for DG schemes. These different implementations affect the accuracy of turbulence predictions. Therefore, it is necessary to examine the behavior of different schemes and select an implementation to achieve optimal accuracy and computational efficiency. We recognize that the difference between the behaviors of different implementations is only noticeable on considerably coarse meshes. Therefore, the discussion focuses on simulation results on \( 32^3 \) DOF resolution; further mesh refinement leads to identical results independent of the choice of a particular flux discretization. Several permutations of flux formulations are considered in this study, as listed in Table 2. For all simulations, an EBDGP3 scheme is employed to perform the assessment.

6.1. Choice of Riemann flux formulations

Fig. 4 presents simulation results for the decay rates of turbulence kinetic energy predicted with Setups 2–5 for Cases 1 and 3. The results show that the decay rate of TKE behaves distinctly different for both cases. Specifically, the decay rate for Case 1 exhibits a slow initial growth as the flow adapts to the initial condition, followed by a gradual decrease. In contrast, the decay rate for Case 3 shows very strong fluctuations at the early stage of the calculation due to dilatation. Because of the high Mach-number condition (\( Ma = 0.6 \)) of Case 3, the turbulence is strongly coupled with the compressibility of the fluid. The turbulence introduces strong dilatation in local fluid elements, triggering the formation of shocklets [32]. This physical aspect, as an additional dissipative mechanism, accelerates the decay of TKE [39], which is consistent with the first peak of the TKE decay observed in Case 3. Results from underresolved simulations also behave differently between Case 1 and Case 3. Specifically for Case 1, the TKE obtained for underresolved calculations decays faster than that of the DNS, which is most observable in the time interval of \( 0.1 \leq t/\tau \leq 1.0 \). This behavior is due to the numerical dissipation and the lack of resolution; similar observations were made by considering the Taylor–Green vortex problem [5,12]. In contrast to Case 1, the under-resolved calculations for Case 3 tend to underpredict the TKE decay at the early stage, \( t/\tau = 0.1 \sim 0.3 \). From these results, only marginal differences in the prediction of the TKE decay rate can be observed for different Riemann flux formulations for Cases 1 and 3.

Since this study is concerned with compressible turbulence, it is important that the numerical schemes can accurately represent the compressibility effects that are induced by turbulent flow motions. For this, density fluctuations are commonly considered as an indicator for assessing the accuracy [40–42]. Therefore, we examine the error of the root-mean-square (RMS) of density, \( \sqrt{\langle \rho'^2 \rangle} \), in order to assess the performance of different Riemann flux formulations. This error is evaluated as difference between the density RMS predicted for the underresolved setting and the density RMS of the corresponding DNS solution. For the low Mach number regime of Case 1, the density fluctuations are coupled with acoustic disturbances. Therefore, we observe very small density fluctuations through the simulation. The performance of different Riemann solvers in predicting the correct density RMS for Case 1 are shown in Fig. 5(a). It can be seen that Setup 2 (Rus-SIP) and Setup 3 (Roe-SIP) cause relatively large underpredictions at the late stage of the calculation, compared to Setup 4 (cent-SIP) and...
Fig. 4. Decay of TKE predicted by different Riemann solvers for (a) Case 1 (Re = 40 and Ma = 0.1) and (b) Case 3 (Re = 100 and Ma = 0.6) on a hexahedral mesh with 32^3 DOF using EBDGP3 scheme.

Fig. 5. Error of density fluctuations obtained by different Riemann solvers for (a) Case 1 (Re = 40 and Ma = 0.1) and (b) Case 3 (Re = 100 and Ma = 0.6) on a 32^3 DOF mesh using EBDGP3 scheme.

Setup 5 (pRoe-sIP). This behavior is consistent with the observations from previous studies of low-Mach Riemann solvers [43–45]. The conventional Riemann flux implementations (e.g. Roe and Rusanov formulations) often generate considerable numerical dissipation at low Mach number conditions, causing excessive damping of acoustic waves. In contrast, the numerical dissipation in the preconditioned-Roe solver is penalized at low Mach number conditions. As consequence, it is observed in Fig. 5(a) that Setup 5 with the preconditioned-Roe solver behaves very similarly to Setup 4 with a non-dissipative central flux formulation. As for Case 3 (see Fig. 12), the magnitude of the density fluctuations is considerably larger than that in Case 1 due to strong dilatation effects. With insufficient numerical resolution, Setups 2, 3 and 5 all underpredict the density RMS of Case 3 throughout the simulation, resulting in a weaker dilatation strength. The underprediction of density RMS at the very early stage, \( t/\tau = 0.1 \sim 0.3 \), coincides well with the underprediction of the TKE decay observed for the same time interval in Fig. 4(b). In terms of the performance of Riemann solvers, the Roe solver (Setup 3) and the preconditioned Roe solver (Setup 5) perform identically. The Rusanov scheme (Setup 2), regarded to be more dissipative, leads to a slightly lower level of density fluctuations, and overall reduces the discrepancies between the underresolved simulation and DNS. This observation requires further analysis. The error behavior for the density fluctuations is related to the density field produced by strong dilatation effects at the very early stage of Case 3 realization. With the added numerical dissipation from the Rusanov scheme, the resulting density field, along with the velocity field, could be physically more consistent with the DNS results (see the example [44]), which leads to slight improvements in the overall accuracy. However, this kind of advantage is case specific; as shown in Fig. 5, comparatively the Rusanov scheme performs better in Case 3 than in Case 1.

Another important criterion for selecting the optimal flux implementation is the computational efficiency. For the examined schemes (Setups 1 to 5), the main impact associated with the computational efficiency is the allowable time step size. The difference of the allowable time step size between setups is more pronounced in Case 1. It should be noted that the time step size does not only influence the stability but also the accuracy of the simulations. Therefore, we seek the largest time step size for a given solution accuracy. For each setup, multiple realizations are performed by varying the time step size \( \Delta t \), and evaluating the correspond-
ing errors of the predicted density fluctuation. This error is presented in Fig. 6 as a function of time-step size $\Delta t$. It is evident that conventional Riemann solvers used in Setups 2 and 3 significantly restrict the time-step size, due to the considerably larger spectral radius of the overall discretized operator. Applying less dissipative Riemann solvers, as done in Setups 4 and 5, can significantly improve the allowable time step size. It is also noted that the CFL condition proposed in the context of convection-dominated flow problems [46] does not effectively apply to this case. For a purely inviscid calculation, the Rusanov flux (Setup 1) retains the stability and accuracy up to a time-step size of $\Delta t/\tau = 2.6 \times 10^{-3}$. With the inclusion of the viscous term (Setup 2), the time-step size requires further reduction to around $1.7 \times 10^{-3}$, as shown in Fig. 6. In summary, it is found from this assessment that the preconditioned Riemann solver provides a better representation of density fluctuations while admitting relatively large time-steps.

6.2. Choice of viscous flux formulations

The performance of different schemes for the viscous-flux discretization are examined in this section. Fig. 7 shows the TKE decay rate obtained from underresolved simulations by Setups 5, 7 and 8, which are equipped with the same Riemann solver but different viscous flux formulations. It can be seen that these three setups do not yield noticeable differences in the predicted decay of TKE. However, by examining the results in Fig. 8, we find that the performance of the three setups differ in terms of predicted density fluctuations. In particular, SIP$_{\text{op}}$ tends to cause larger over-predictions of the magnitude of density fluctuations, compared to SIP and BR2 schemes. This behavior can be attributed to the lack of the viscous flux $\tilde{Q}$ for the density variable. Once the inter-element jump is triggered by convection, the dissipative effect on density has to be indirectly realized through momentum conservation. This indirect response, although seemingly more physical, introduces a lagging effect in the underresolved calculations. This hypothesis is further verified by removing the numerical stabilization from the convective flux, which leads to Setup 6 (cent-SIP$_{\text{op}}$). By comparing Setup 6 (cent-SIP$_{\text{op}}$) to Setup 7 (pRoe-SIP$_{\text{op}}$), it can be observed that errors for density RMS are amplified. Therefore, we can confirm that this error is genuinely related to the characteristics of the numerical stabilization terms. In contrast to SIP$_{\text{op}}$, the SIP scheme employed in Setup 5 is formulated with a scalar stabilization, which numerically introduces a direct response of the density field to inter-element solution jumps. This treatment appears to be more effective in capturing density fluctuations in underresolved calculations, at least for the cases examined in Fig. 8. The density fields obtained by Setups 5 and 7 are illustrated in Fig. 9. The density fluctuations, as well as the density jumps across element interfaces, are more apparent in the calculation by Setup 7 than by Setup 5.

As for the computational efficiency, it is found that the computational costs for the considered viscous flux formulations vary due to the required operations, although the allowable time step sizes are comparable. In particular, it was found that the BR2 scheme requires twice as much computational cost compared to SIP and SIP$_{\text{op}}$ schemes due to the required lifting operator for computing the stabilization terms. It is worth mentioning that this observation on computational cost is based on the specific implementation of our solver. In fact, on hexahedral elements, the BR2 scheme might become as fast as SIP by considering a collocation-type implementation [38] and tensor-product structures.

![Fig. 6. Comparison of predictive errors of density fluctuation as a function of time-step size for different setups and 64$^3$ DOF using EBDGP3 scheme for Case 1 (Re$_\infty = 40$ and Ma$_\infty = 0.1$) (legend: red - Rus-SIP (Setup 2); black - Roe-SIP (Setup 3); blue - cent-SIP (Setup 4); green - pRoe-SIP (Setup 5)). (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)](image)

![Fig. 7. Decay of TKE predicted by different viscous discretization schemes for (a) Case 1 (Re$_\infty = 40$ and Ma$_\infty = 0.1$) and (b) Case 3 (Re$_\infty = 100$ and Ma$_\infty = 0.6$) on a 32$^3$ DOF mesh using EBDGP3 scheme.](image)
Considering the accuracy and computational efficiency, in the following the SIP scheme is selected as viscous flux formulation. This choice is identical to that employed in previous studies [1,4]. Therefore, Setup 5 (pRoe-SIP) will be used in the benchmark study against a state-of-the-art FV solver. Based on the study in this section, it is concluded that i) the DG solution for TKE decay rate is insensitive to the choice of the inviscid and viscous flux discretization schemes; ii) the prediction of density fluctuations exhibits certain sensitivity to the choice of the viscous flux formulation; iii) Setup 5 (pRoe-SIP) provides better predictive accuracy with minimum computational cost. Based on this finding, Setup 5 will be employed for the subsequent benchmark study.

7. Benchmark against FV solver

7.1. Description of the CharlesX solver

CharlesX is a reconstruction-based unstructured FV solver, which has been extensively used for turbulent flow calculations [47–49]. Third-order polynomial reconstruction is utilized for the evaluation of the left- and right-biased face centroid values of the flow variables. A blending between central scheme and Riemann flux is performed based on the local grid quality. This flux computation procedure is formally second-order accurate and has maximum fourth-order accuracy on perfectly uniform Cartesian meshes with no numerical dissipation. This purely non-dissipative numerical procedure is employed for the calculations of Cases 1–3 on regular hexahedral meshes. For computations of shock-related problems (Case 4), CharlesX utilizes a sensor-based hybrid central-ENO scheme to minimize the numerical dissipation while stabilizing the simulation. For regions where shocks are present, a second-order ENO reconstruction is used on the left- and right-biased face values. A number of sensors are available for detecting shocks in CharlesX. For the current study, the “relative solution” (RS) sensor is adopted [50,51]. This sensor compares the difference between the left- and right-biased reconstructed face values of density and pressure to the minimum of the corresponding values in the two neighboring cells of the face. When this difference exceeds a certain fraction of the minimum cell value, the ENO reconstruction scheme will be used. For the following study, a value of 0.5 was found to be sufficiently effective. Details on the discretization of the viscous flux and gradient calculations can be found in Ham & Iaccarino [52] and Pecnik et al. [53]. The semi-discretized form of the governing equations is solved using a fully explicit, third-order Runge-Kutta time integration scheme.
7.2. Benchmark on regular hexahedral meshes

This section focuses on the comparison of LES solutions obtained on hexahedral meshes using the EBDG solver and CharlesX. LES calculations are performed with $32^3$ DOFs for the low Reynolds-number cases (Cases 1 and 2), and $64^3$ DOFs for the cases with higher Reynolds and Mach numbers (Cases 3 and 4). EBDG solutions of different polynomial orders are considered, and the number of elements are adjusted according to the polynomial order to match the number of DOFs. The decay of TKE for all cases are presented in Fig. 10. For Case 1, it is observed that all EBDG solutions tend to overpredict the initial rate of decay, followed by a slight underprediction after around $t/\tau = 1$. In contrast, LES solutions obtained from CharlesX exhibit the opposite behavior. It is worth noting that the benefit of using higher-order accurate EBDG schemes is evident for Case 1, and EBDGP4 predicts the decay of TKE closer to the DNS solutions compared to the other EBDG solutions of lower polynomial orders. This trend is directly attributed to the smooth flow characteristics at these low Mach and Reynolds number conditions. Different from the observation for Case 1, the decay of TKE is largely affected by dilatation effects in the other cases, and this is mostly noticeable at the early stage when the decay rate approaches its maximum. As for Cases 2 and 3, both solutions perform equally well in terms of capturing the initial peak of the decay rate. After the peak, discrepancies among different LES solutions become apparent as the flow field evolves further. An undershoot in the CharlesX prediction is observed for the decay rate of TKE between $t/\tau = 0.5$ and 1.5. In contrast to this, EBDG schemes, having inherent numerical dissipation, overpredict the decay rate of TKE during this time interval. EBDGP4 solution shows small discrepancies with the DNS solutions compared to other DG and FV results. It is interesting to note that the benefit of the high-order accuracy remains at the high turbulence conditions for Cases 2 and 3, which are characterized by considerable dilatation effects and the formation of shocklets. However, for Case 4, the dilatation effects are much stronger, and shocks form almost everywhere in the flow field at the early stage of the realization, as observed in Fig. 11. It was discussed in Section 5 that the solution stabilization plays an important role in this case. Solutions obtained with different stabilization methods show a relatively large scatter in representing the initial peak of the TKE decay. Comparatively, the CharlesX solution yields slight less overprediction than EBDG solutions. The EBDGP2 scheme provides improved predictions compared to EBDGP3/4 schemes.

Predictions for density fluctuations, between EBDG and FV schemes, are presented in Fig. 12. For the low Mach-number condition of Case 1, the thermodynamic variation is acoustic-controlled, and therefore the density signals exhibit strong fluctuations. It is found that high-order methods improve the representation of the fluctuating modes. LES solutions obtained by both EBDGP3 and
EBDGP4 schemes are in better agreement with DNS results for density RMS for all cases. EBDGP2 solution produces considerably larger errors. This is particularly at the early stage of the simulation. CharlesX solutions are comparable to results obtained with EBDGP3 and EBDGP4, but deteriorate slightly at the late stage of the simulation with small undershoots in the prediction. As for the LES calculations of Cases 2 and 3, high-order DG solutions, both from EBDG3 and EBDG4, shows less discrepancies with respect to the DNS reference. CharlesX tends to produce slightly larger density fluctuations at LES resolution. This can be attributed to the numerical error introduced in representing the initially strong dilatational flows using a non-dissipative scheme. For Case 4, the solution behavior in predicting density RMS is consistent with results of the decay of TKE. The sensor-based hybrid central-ENO scheme, employed in CharlesX, is more effective than EBDG schemes in capturing density fluctuations. Solutions obtained from EBDG show larger sensitivities to polynomial order. EBDGP2 outperforms other EBDG schemes, producing a solution that agrees with CharlesX. EBDG solutions with \( p > 2 \) introduce larger discrepancies to the DNS solution for this case. This behavior might be unique to EBDG schemes, and this observation is not necessarily generalizable to DG schemes that employ other stabilization techniques. However, the behavior of the EBDG scheme is generally consistent with the current understanding of the behavior of high-order schemes [12] in that polynomial refinement benefits in capturing

![Density field for Case 4 (Re_λ = 100, Ma_0 = 1.5) at t/τ = 0.6 obtained using EBDGP3 scheme.](image1)

![Comparison of predictions for density fluctuations for (a) Case 1 (Re_λ = 40, Ma_0 = 0.1); (b) Case 2 (Re_λ = 40, Ma_0 = 0.6); (c) Case 3 (Re_λ = 100, Ma_0 = 0.6); (d) Case 4 (Re_λ = 100, Ma_0 = 1.5).](image2)
smooth solutions while mesh refinement is more effective in resolving discontinuous solutions.

The velocity spectra of the resulting turbulence fields, calculated using different methods, are compared in Fig. 13. It can be seen that the solutions obtained by each individual method show a consistent behavior for all cases. Due to the inherent numerical dissipation, EBDG schemes result in consistent undershoots of energy, which is mostly visible near the cutoff wavenumber. An increase in polynomial order can clearly improve the representation of turbulence energetics for Cases 1–3. However, this trend does not translate to Case 4. Fundamentally different from the EBDG solutions, CharlesX results show significant energy pileup around the cutoff wavenumber. Besides this, the part of the spectra around intermediate wavenumbers is adversely affected, which is most prominent from Case 1 and Case 2. These issues have been recognized in previous LES calculations using non-dissipative schemes [54], and can be resolved by applying SGS models [40,55]. This conventional modeling approach, however, might not be suited to DG schemes due to the dissipative properties of the scheme itself.

7.3. Benchmark on tetrahedral meshes

Cases 1 and 3 are selected for the benchmark study on tetrahedral meshes. For comparison, computations are carried out with \(5 \times 44^3\) (\(\approx 75^3\)) DOFs for both the EBDG solver and CharlesX. This number of DOFs corresponds to \(5 \times 16^3\) DGP3 hexahedral elements and \(5 \times 21^3\) DGP2 elements, respectively. CharlesX’s purely non-dissipative scheme that was successfully employed on hexahedral meshes leads to certain stability issues on tetrahedral meshes. To achieve numerical stability, a blending of central and Riemann fluxes based on the local cell skewness is employed [50]. The blending factor is purely grid-based and can be evaluated prior to the simulation. For regular hexahedral meshes, studied in Section 7.2, the blending scheme reverts to the purely non-dissipative scheme. However, for the considered tetrahedral meshes, the blending procedure yields a blending factor of 0.075, which means that 7.5% of the flux evaluations at the faces are computed from a HLLC flux while the remaining portion is evaluated using a central flux.

The LES results obtained from EBDG solver and CharlesX are compared in Fig. 14. On tetrahedral meshes the accuracy of CharlesX reduces to second order. In contrast, EBDG solutions do not reveal apparent sensitivity to the mesh topology. Given the same resolution (\(5 \times 44^3\)), the FV results are significantly less accurate than the EBDG solutions. Further mesh refinement is conducted using CharlesX, showing that two more levels of refinement are required for the FV scheme to obtain comparable predictions for TKE and spectra. However, the prediction for density fluctuations from CharlesX is still not as accurate as the EBDG predictions.
for Case 1 with $Re = 40$ and $Ma = 0.1$, confirming the sensitivity of density fluctuations to numerical errors. The slow convergence also emphasizes the difficulty in discretizing viscous terms with the FV scheme on unstructured meshes [56]. In contrast, the discretization operator for viscous-diffusive terms in EBDG, which is built completely on a weak formulation, provides more advantages in this regard. To illustrate the quality of solutions on different mesh topologies, the vorticity fields obtained from both EBDG solver and CharlesX are illustrated in Fig. 15. From this figure, it is evident that the accuracy of the FV solution deteriorates, with substantial loss of small scales, while the EBDG prediction remains insensitive to the mesh topology.

7.4. Estimation of computational cost

The previous sections focused on the assessment of the accuracy of FV and DG solutions. This section complements this analysis by comparing computational cost between the DG scheme and the FV implementation in CharlesX. It is recognized that there are several aspects that affect computational efficiency, including com-
puting architecture, programming language, parallel environment, and extent of solver optimization. As such, having direct comparisons between solvers does not objectively imply the fact of performance difference between different numerical schemes. In fact, several examples show that solvers, built on either DG [57] or FV [58] schemes, are able to scale on massively parallel computing environments. Here, we provide an estimate of FLOP demands of the examined numerical schemes, following the analysis performed in [32], rather than determining which scheme (or solver) is more efficient. Time-step size is another influence factor for computational cost. In this regard, we find that CFL-constraint proposed for linear convection [46] is not sufficiently reliable for turbulence simulations due to the nonlinearities of the fluid-flow solutions. To obtain a sharp estimate for the time-step size for this problem is nontrivial. To this end, we employ a direct approach to determine the maximum allowable time-step sizes for both DG and FV cases, and it is shown that the FV solver (CharlesX) allows a 10–15% larger time-step size than the DG solver. The difference is not very notable compared with the difference in FLOP demands, which will be given in the following.

A summary of FLOP counts for the DG implementation is listed in Table 3. As discussed in the context of Eqs. (8) and (9), there are five terms to be calculated for each residual evaluation of the DG discretization: two volumetric terms and three surface terms. Since the DG solver is implemented based on numerical quadrature, the number of quadrature points for different element types and polynomial orders are given in Table 3. The implementations for the different terms of the same type are grouped together for enabling variable reuse and reducing FLOPs. For each group, it typically requires solution interpolation ($N_qN_v$ DOF FLOPs), gradient interpolation ($3N_qN_v$ DOF FLOPs) onto quadrature points, and projection of the spatial residual onto local DG space ($3N_qN_v$ DOF FLOPs for volumetric terms and $N_qN_v$ DOF FLOPs for edge terms, as indicated in Eq. (8). The counts are doubled on the edges since both the left and right sides should be taken into account. There are some remaining FLOPs, as shown in Table 3, which have to be done with the evaluation of nonlinear fluxes and the multiplication of the fluxes by quadrature weights. Apparently, the FLOPs cost for different terms are different. As shown in Table 3, the evaluation of the diffusion terms requires an order of magnitude more FLOPs than that of the convective terms. This is partially attributed to the generality of the DG diffusion discretization, which involves several matrix-vector multiplications. By utilizing collocation-type treatment [38], a more efficient implementation can be achieved [59]; however, the treatment might be restricted to certain element types and polynomial orders. Nevertheless, the FLOPs estimate provided here is based on the specific implementation of the DG solver. Compared to the DG solver, the im-

![Fig. 15. Vorticity magnitude obtained using the DG and CharlesX FV solvers for Case 3 (Re$_x$ = 100, Ma$_x$ = 0.6) on both hexahedral mesh with $64^4$ DOF and tetrahedral mesh with $5 \times 44^4$ DOF.](image-url)
plementation in CharlesX is more efficient. The volumetric terms do not exist in the FV discretization and viscous terms only require 36 gradient evaluations, each of which utilizes pre-processed mesh information and only involves one vector-vector multiplication. Further, since the evaluation of the viscous terms in CharlesX is tailored to the specific form of governing equations, it also leads to large savings in operation counts over the DG solver. With this, the FLOPs-per-cell count of CharlesX is 2440, which is only about one quarter of the FLOPs required for the DG3 implementation, as presented in Table 3. Although some FLOPs might be saved in the DG implementation, for example, by customizing the viscous operators for specific equations, it is anticipated that in general a factor of three or four is a reasonable estimate of the difference in operating counts between DG solver and FV solver for the same number of DOFs.

To examine the performance from the accuracy-vs-cost point of view, we report the mean TKE error as a function of FLOPs for different mesh resolutions in Fig. 16 for Case 3. This figure shows that on regular hexahedral meshes CharlesX is able to achieve the same accuracy as the DG solver using less FLOPs. However, on tetrahedral meshes, the DG solver shows significant advantages over the CharlesX for the same number of DOF. As presented in Section 7.3, CharlesX predictions are still not as accurate as the EBDG results even after two levels of mesh refinement, which already introduces a factor of $4^3 = 64$ increase in cost.

In summary, to achieve the same accuracy, on regular hexahedral meshes, CharlesX only requires one quarter of FLOPs of the DG solver, while on tetrahedral meshes CharlesX requires one order of magnitude more FLOPs than the DG solver, to achieve the same accuracy.

8. Findings and conclusions

In this study, an entropy-bounded DG scheme as a numerical technique for large-eddy simulations is assessed by considering decaying isotropic turbulence. This assessment concerns the suitability of the entropy-bounding treatment for capturing turbulence, the effects of Riemann and viscous flux formulations on LES solutions, and the performance of the EBDG schemes with different polynomial orders. A benchmark study of the EBDG solver against a state-of-the-art FV solver is carried out to identify the status and potential merit of EBDG methods as LES technique. The main findings are summarized as follows:

- As a solution stabilization technique, entropy-bounding is shown to be more effective than overintegration to enforce numerical stability, especially for compressible turbulent flows at higher Reynolds- and Mach numbers. The entropy bound in the EBDG scheme does not introduce noticeable effects on the solution representation of turbulence characteristics at conditions of low and moderate compressibility. For conditions involving supersonic turbulence considered in this study, it is found that the application of a less restrictive entropy bound results in improved accuracy.

- From the benchmark tests on hexahedral meshes, it is found that i) an increase in the polynomial order of the EBDG scheme improves the accuracy of LES predictions at low and moderate turbulence Mach numbers; ii) at supersonic turbulence conditions, h-refinement is more effective over polynomial refinement in capturing turbulence; iii) non-dissipative FV predictions for the decay rate of TKE are comparable to those of higher-order EBDG schemes (EBDG3 and EBDG4) for flow conditions at low and moderate turbulence Mach numbers, while prone to produce appreciable errors for representing fluctuations of thermodynamic quantities.

- From the benchmark test on tetrahedral meshes, it is found that i) DG solutions retain a similar accuracy as on hexahedral meshes; ii) FV solvers suffer from a significant deterioration in solution accuracy. With the same number of DOF, the FV solution is considerably less accurate due to the deterioration of the order of accuracy (reduction from 4th-order accuracy on regular hexahedral meshes to 2nd-order accuracy on tetrahedral meshes). In order to achieve comparable accuracy, the FV method requires at least two additional levels of mesh refinement.

- FLOP demands for a DG solver and CharlesX are estimated. For the same DOF, the DG solver requires approximately three times more FLOPs as the FV solver given the specific implementation considered in this study. From a cost-accuracy viewpoint, the FV solver holds advantages over the DG solver on regular hexahedral meshes; however, if the geometric complex-
ity requires the discretization with unstructured and tetrahedral meshes, high-order DG methods become more beneficial over FV methods.

Acknowledgments

This work was supported by an Early Career Faculty grant from NASA’s Space Technology Research Grants Program. Resources supporting this work were provided by the NASA High-End Computing (HEC) Program through the NASA Advanced Supercomputing (NAS) Division at Ames Research Center.

Appendix A. Consistency of DNS results

This section reports on the consistency between the DNS results obtained from the EBDG solver and CharlesX. For the DG solution, an EBDGP3 scheme is selected to retain the same order of accuracy as the FV scheme of the CharlesX solver. Regular hexahedral meshes are utilized and the mesh resolutions are chosen according to the estimate of the Kolmogorov scales listed in Table 1. DNS for all four cases are carried out, and for each case one additional calculation with a refined mesh is performed to confirm solution convergence. The solutions are shown in Fig. A.17. There is very good agreement between predictions obtained from both solvers for TKE, density fluctuation, dilatation and velocity spectrum. It is found that the dilatation for Case 4 is not fully converged with the mesh resolution estimated by the corresponding Kolmogorov scale. This is due to the formation of shocklets, which leads to the generation of smaller scales in the turbulent field. All other quantities presented in Fig. A.17 are converged. For the purpose of numerical assessment, this set of solutions are adequate to serve as a reference for the LES studies on coarser meshes.

Appendix B. Sensitivity of entropy bound

From Section 5, it is found that the LES solutions obtained using the EBDG scheme exhibit certain sensitivities to the magnitude of the imposed entropy bound \( s_0 \). This discussion is here supplemented to examine the effect of the magnitude of \( s_0 \) on the LES solution. For the cases presented in Figs. 2 and 3, we sample the percentage of troubled elements (referred to elements in which the scaling limiter of EBDG scheme is applied) and the amount of truncated TKE through the scaling limiter over the whole computational domain. These two quantities are presented as a function of simulation time in Figs. B.18 and B.19, corresponding to Case 3 and Case 4, respectively. As we can see, a restrictive entropy bound \( s_0 = \ln 0.9 p_0 / \rho_0^{2/3} \) tends to identify a larger number of troubled elements and truncates more TKE for both cases, compared to a relaxing bound \( s_0 = \ln 0.001 p_0 / \rho_0^{2/3} \). For Case 3, the number of troubled elements is less than 1.5% throughout the simulation for any of the considered setups, and the overall impact of applying the limiter is small. Therefore, the solutions of Case 3, as exhibited in Fig. 2,
are insensitive to the magnitude of $s_b$, although the scaling lim-
its with different values of $s_b$ perform differently. In contrast to
Case 3, the LES solutions for Case 4 exhibit a stronger sensitivity
to the magnitude of $s_b$. At the maximum, nearly 70% of the ele-
ments are identified as troubled elements with the restrictive en-
tropy bound. With the application of a less restrictive bound,
the number of troubled elements reduces to around 30%. In terms of
the performance of the EBDG scheme with different polynomial or-
ders, the scaling limiter for the EBDGP2 scheme tends to truncate
less TKE in Case 4, which could explain the better performance of
EBDGP2 observed in Fig. 3.

Appendix C. Data of initial conditions

Initial conditions for different mesh resolution used in the cur-
cent study are available for downloading at http://web.stanford.
.edu/group/ihmegroup/archive/LvMalhme_CF2017/initial_condition.
tar.gz.

References


cedings of the Summer Program, Center for Turbulence Research. Stanford

[3] Wei L, Pollard A. Direct numerical simulation of compressible turbulent channel

[4] de Wiart CC, Hillewaert K, Bricteux L, Winckelmans G. Implicit LES of free and


[7] Johnsen E, Nair A. A simple method to improve the accuracy of advection in
discontinuous Galerkin methods for Navier-Stokes simulations. 2014. AIAA Pa-
er 2014-1276.

[8] Liu X, Xia Y, Luo H, Xuan L. A comparative study of Rosenbrock-type and
implicit Runge-Kutta time integration for discontinuous Galerkin method for


order discontinuous Galerkin spectral element methods for transitional and

SD7003 using a high order discontinuous Galerkin method. Seventh in-
ternational conference on computational fluid dynamics (ICCFD7), Big Island,
Hawaii; 2012.


