Available online at www.sciencedirect.com
ScienceDirect
Comput. Methods Appl. Mech. Engrg. 299 (2016) 245–282
Computer methods in applied mechanics and engineering
www.elsevier.com/locate/cma

A discontinuous Galerkin approach for high-resolution simulations of three-dimensional flows

Konstantinos T. Panourgias[a], John A. Ekaterinaris[b,*]

[a] *Department of Mechanical Engineering and Aeronautics, University of Patras, Greece*
[b] *Aerospace Engineering, Embry–Riddle Aeronautical University, Daytona Beach, FL, United States*

Received 3 March 2015; received in revised form 21 October 2015; accepted 22 October 2015
Available online 3 November 2015

Abstract

High order discontinuous Galerkin (DG) discretizations possess features making them attractive for high-resolution computations in three-dimensional flows that include strong discontinuities and embedded complex flow features. A key element, which could make the DG method more suitable for computations of these time-dependent flows in complex domains, is application of limiting procedures that ensure sharp and accurate capturing of discontinuities for unstructured mixed-type meshes. A unified limiting procedure for DG discretizations in unstructured three-dimensional meshes is developed. A total variation bounded (TVB) limiter is applied in the computational space for the characteristic variables. The performance of the unified limiting approach is shown for different element types employed in mixed-type meshes and for a number of standard inviscid flow test problems including strong shocks to demonstrate the potential of the method. Other alternatives, such as hierarchical three-dimensional limiting with the proposed limiting approach and TVD limiting, are developed and demonstrated. Furthermore, increased order of expansion and adaptive mesh refinement is introduced in the context of it h/p-adaptivity in order to locally enhance resolution for three-dimensional flow simulations that include discontinuities and embedded complex flow features.

Keywords: High-order methods; Discontinuous Galerkin; Limiting; h/p-adaptive methods; Unstructured meshes

1. Introduction

Developments of high-order numerical methods for unstructured meshes could offer significant advantages for the simulation of complex high-speed flows, compressible turbulence [1], and high-speed combustion [2] in non-trivial geometries of interest to practical applications. The discontinuous Galerkin (DG) [3–6], the spectral volume (SV) [5,7], and the spectral difference (SD) [5,8,9] methods have shown promise for high-resolution computations of complex flows because they have a compact stencil, and retain the design order of accuracy even for meshes of moderate quality that would often result from grid generation over complex three-dimensional configurations. The

* Corresponding author.
E-mail addresses: panourgias@iacm.forth.gr (K.T. Panourgias), ekaterij@erau.edu (J.A. Ekaterinaris).

http://dx.doi.org/10.1016/j.cma.2015.10.016
0045-7825/

potential applications of these methods for high-resolution simulations of practical flow problems can further be enhanced with the use of mixed-type meshes and solution adaptive schemes.

Despite the advantages that high-order methods possess compared to traditional second order accurate finite-volume methods, and uniformly high-order, finite-difference methods, such as ENO [10] and WENO [11], the higher computational cost prevents their more widespread use in practical applications. Solution adaptive refinement strategies of *h*-, *p*-, or *h/p*-type can reduce computing time for high-resolution simulations of complex flows without compromising numerical accuracy. Use of *p*-type refinement is well suited for time dependent flows, such as vortical flows for example [12], because the order of accuracy can be selectively increased in regions of high vorticity or density gradients. Furthermore in order to ensure accuracy in the near wall region, the true curvature of the geometry must be taken into account. For inviscid flow computations, the simple procedure suggested by Krivodonova and Berger [13] may be employed. For viscous flow computations, layers of curved elements must be constructed in the near wall region [14]. Currently, only few mesh generation packages could provide higher-order elements. In order to exploit the full potential of high-order finite element methods, the idea of isogeometric analysis [15] recently applied for fluid dynamic simulations in complex geometries [16,17] must be employed. Therefore the next necessary logical step should be to employ NURBS representation of both the geometry (mesh) and the expansion functions [18] of the DG discretizations in order to ensure accuracy enhancements through adaptive mesh refinement and selective increase of the order of expansion in the unified, high-fidelity framework of the isogeometric analysis.

The *h/p* finite element framework [19], which can be combined with the isogeometric analysis [15], has been adopted for the implementation of DG finite element [12] method. The key elements of this framework that are exploited in this work, are the hierarchical bases, and the collapsed coordinate transformations. Utilization of these features allows to perform all required operations for the implementation of the DG method for the standard cubical element. Modal, hierarchical bases are constructed in the computational space over a unit cube using tensor products and transformed back to the physical space of hexahedra, prisms, or tetrahedra using a collapsed coordinates system [19]. Modal bases facilitate mixed element implementation and adaptive mesh refinement with non-conforming faces compared with nodal bases [20,21] and they are better suited for *h/p*-type adaptive refinement. The geometry of the computational domain is simple, therefore, all evaluations of numerical fluxes, surface and volume integrals, required in the implementation of the DG method, are carried out in the computational domain. For the cubical elements of the computational domain, appropriate number of quadrature points are used to integrate exactly polynomials of the maximum degree required for both linear and higher-order elements. Use of the resulting methodology for mixed-type unstructured meshes is straightforward. Furthermore, application of *p*- and *h*-type adaptivity of the solution based on computed flow features is facilitated because the required interpolations at the element faces can be obtained directly. In the computational examples, simple *p*-type adaptive schemes based on the gradient of the computed density, or vorticity field are employed. For *p*-type refinement, in regions of high gradients the order of the polynomial expansion is raised to a preset desired level and then progressively drops to the lowest order of accuracy ($\mathbb{P}_1$ expansion) that is used for the computation of the free stream where no gradients exist. The dynamic adaptive mesh refinement strategy [22] is discussed next.

High-resolution computations of shock dominated flows, utilize Riemann solvers at the cell interfaces and the discontinuities are typically captured within one or two cells the most [23]. For time dependent computations of three-dimensional flows, sharp resolution of discontinuities through the use of small cells for the entire flow field is not practical even for second order accurate ($\mathbb{P}_1$ expansion) computations. In these cases, use of adaptive mesh refinement (AMR) is necessary because it offers significant savings in computational time without compromising solution accuracy. The idea of using AMR [24–30], is not new and has been employed in order to enhance resolution of complex flow features or discontinuities and capture them in regions of fine mesh not necessarily aligned with the flow features once elements with non-conformal faces are used. Mesh adaptivity is continuously pursued both in the isogeometric [31] and in the unstructured mesh context [32]. However as it has been recently pointed out [33], use of adaptivity, even if it offers the potential for enhanced accuracy at a reduced computational cost, has not seen widespread use in fluid dynamics. The DG method is particularly suitable for local mesh refinement and use of elements with non-conforming faces. It will be demonstrated in the results section that highly localized AMR, alone or combined with *p*-type refinement, could produce high-resolution computations of three-dimensional flows with significant reductions in computational cost.

Limiting approaches for DG discretizations [34], and [35–42] have been applied in the physical domain and do not have a straightforward extension for three-dimensional discretizations in mixed-type unstructured meshes.