Computers & Fluids 100 (2014) 13–29

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Discontinuous Galerkin and Petrov Galerkin methods for compressible viscous flows

Li Wang *, W. Kyle Anderson, J. Taylor Erwin, Sagar Kapadia

SimCenter: National Center for Computational Engineering, University of Tennessee at Chattanooga, 701 East M.L. King Boulevard, Chattanooga, TN 37403, USA

ARTICLE INFO

Article history:
Received 8 September 2013
Received in revised form 4 April 2014
Accepted 30 April 2014
Available online 14 May 2014

Keywords:
Discontinuous Galerkin methods
Petrov Galerkin methods
Spalart and Allmaras turbulence model
Linear elasticity
Mesh movement
Curved elements
Steady and unsteady viscous flows
Turbulent flows

ABSTRACT

In this paper, high-order finite-element discretizations consisting of discontinuous Galerkin (DG) and streamline upwind/Petrov Galerkin (SUPG) methods are investigated and developed for solutions of two- and three-dimensional compressible viscous flows. Both approaches treat the discretized system fully implicitly to obtain steady state solutions or to drive unsteady problems at each time step. A modified Spalart and Allmaras (SA) turbulence model is implemented and is discretized to an order of accuracy consistent with the main Reynolds Averaged Navier–Stokes (RANS) equations using the present high-order finite-element methods. To accurately represent the real geometry configurations for viscous flows, high-order curved boundary meshes are generated via a Computational Analysis PRogramming Interface (CAPRI), while the interior meshes are deformed subsequently through a linear elasticity solver. The mesh movement procedure effectively avoids the generation of invalid elements that can occur due to the projection of curved physical boundaries and thus allows high-aspect-ratio curved elements in viscous boundary layers. Several numerical examples, including large-eddy simulations of viscous flow over a three-dimensional circular cylinder and turbulent flows over a NACA 4412 airfoil and a high-lift multi-element airfoil at high angles of attack, are considered to show the capability of the present high-order finite-element schemes in capturing typical viscous effects such as flow separation and to compare the accuracy between the high-order DG and SUPG discretization methods.

1. Introduction

High-order discretization methods have gained increasing popularity in the past decade for a wide variety of fluid dynamics applications [1–7]. More and more physically realistic problems can be tackled through the use of such methods to obtain high accuracy solutions, while reducing the need for extremely high-resolution meshes that are often required by lower-order methods. Moreover, a great deal of effort has been devoted to the versatility, robustness and efficiency of high-order flow solvers, including adaptive mesh refinement techniques [8–12], solution limiting and shock capturing methods [13–15,9,16], hybrid methodologies and multigrid solution strategies [2,17–19]. To this end, this work is concentrated on the development of high-order discretization methods, consisting of both discontinuous Galerkin [1–3,6,18,9,20] and streamline upwind/Petrov Galerkin [21–24] discretizations, to further expand the capability of high-order schemes in solving a wide range of viscous flow problems for complex geometries. Research efforts are also placed on qualitative comparisons of the solution accuracy between the high-order DG and SUPG methods. Moreover, applications of the present methods for studying the flows around bluff bodies at sub-critical Reynolds numbers and simulations of turbulent flows are considered.

The application of high-order methods to turbulent flows has become an active research topic in the computational fluid dynamics (CFD) community. It is known that high-order schemes are well-suited for problems with smooth solutions, in which the order of solution accuracy can be generally achieved. However, as the high-order methods are applied to turbulent flows using, for example, the Spalart–Allmaras (SA) turbulence model [25], robustness becomes a challenge for high-order methods because the turbulence working variable decreases abruptly at the edge of turbulent and laminar regions, thus leading to unavoidable oscillations and often negative values, which subsequently causes the solver to fail. Recent work [5,26] has investigated a modified SA turbulence model to avoid the stability issues caused by negative turbulence working variable. In the present work, the modified SA turbulence model is utilized and is fully coupled with the main conservation equations, and furthermore, consistent high-order finite-element

* Corresponding author. Tel.: +1 4234255627.
E-mail addresses: Li-Wang@utc.edu (L. Wang), Kyle-Anderson@utc.edu (W.K. Anderson), j.taylor.e@gmail.com (J.T. Erwin), Sagar-Kapadia@utc.edu (S. Kapadia).

http://dx.doi.org/10.1016/j.compfluid.2014.04.035
0045-7930/

discretizations, including the present DG and SUPG methods are applied to the full system of equations.

In addition, the use of high-order curved boundary elements is important for high-order schemes to deliver an overall accurate solution [27,28]. Poor representation of the actual geometry can lead to excessive production of artificial entropy along the geometry surface, thus degrading the solution accuracy [24]. This is especially the case in the context of high-order methods because relatively coarse meshes are generally utilized where surface nodes or cells can span a large portion of the surface. On the other hand, more attention is needed when high Reynolds number flows are involved. To allow the use of high-aspect-ratio elements in a thin boundary layer, curvilinear interior elements are required in response to the projection of curved boundary faces and edges into the interior mesh. In this context, the current work makes use of a CAPRI [29] mesh parameterization tool [30] for evaluating the true positions of additional surface quadrature points [31] on arbitrary three-dimensional configurations, thus enabling higher-order (greater than linear) representations to enhance the surface grid definitions. A linear elasticity mesh movement strategy is solved subsequently to prevent the generation of negative-Jacobian elements.

For viscous or turbulent flow problems, the viscous flux terms are numerically treated using an explicit symmetric interior penalty (SIP) method [3,19] for the present DG discretizations. The SIP method is simultaneously applied to the modified one-equation SA turbulence model to handle the second derivatives of the turbulent working variable arising from the model. An advantage of the SIP method is that it does not require introduction of any auxiliary variables, and moreover, the scheme maintains a compact element-based stencil, which simplifies the linearization of the discretized system for developing implicit methods. In the SUPG methods, on the other hand, the solution is assumed to be continuous across the computational domain. Therefore, the surface fluxes cancel at elemental interfaces (different from the DG methods) and are only carried out over the domain boundaries. However, due to lack of dissipation, stabilization becomes a key component in the Petrov Galerkin finite element method. This is achieved through the addition of a stabilization term which acts only in the streamline direction [32]. A consistent and stabilized SUPG formulation is thus constructed by modifying the Galerkin weighting function with the additional streamline-upwind stabilization term and applying the modified weighting function to the entire system of equations. A noticeable difference between the SUPG and DG methods is that, due to the fact that the solution modes or nodes in the SUPG method are stored only once at each edge or face [24], the former scheme requires fewer degrees of freedom than the latter scheme for the same order of discretizations. For time-dependent problems, a standard second-order backward difference formula (BDF2) is considered. The implicit system is solved using an approximate Newton method, where the linear system is solved via an element Gauss–Seidel scheme or a preconditioned GMRES approach with ILU (k) preconditioning [33,34]. A multigrid approach [18,35] is also implemented in the DG flow solver to accelerate the solution convergence.

An outline of the paper is as follows. In Section 2 the governing equations are introduced, including the modified Spalart and Allmaras turbulence model. Section 3 describes consistent discretizations of the full system of equations using discontinuous Galerkin and streamline upwind/Petrov Galerkin discretizations as well as an implicit time-integration scheme. Section 4 briefly reviews the integration of CAPRI with the mesh movement strategy used in the current work. Several numerical examples are presented in Section 5 to demonstrate the performance of the current high-order schemes in capturing complex flow structures for three-dimensional viscous flow and turbulent boundary layer separation. Finally, Section 6 summarizes the conclusions and discusses the future work.

2. Governing equations

The compressible Reynolds Averaged Navier–Stokes equations coupled with the modified one-equation Spalart–Allmaras turbulence model [5,6,26] can be written in the following conservative form:

$$\frac{\partial \mathbf{U}(\mathbf{x},t)}{\partial t} + \nabla\cdot(\mathbf{F}_e(\mathbf{U}) - \mathbf{F}_v(\mathbf{U},\nabla\mathbf{U})) = \mathbf{S}(\mathbf{U},\nabla\mathbf{U}) \quad \text{in} \quad \Omega \tag{1}$$

where Ω is a bounded domain. The vector of conservative flow variables $\mathbf{U}$, the inviscid and viscous Cartesian flux vectors, $\mathbf{F}_e$ and $\mathbf{F}_v$, are defined by:

$$\mathbf{U}=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ \rho E\\ \rho\tilde{v}\end{bmatrix}, \mathbf{F}_e^x=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ \rho uw\\ (\rho E+p)u\\ \rho u\tilde{v}\end{bmatrix}, \mathbf{F}_e^y=\begin{bmatrix}\rho v\\ \rho uv\\ \rho v^2+p\\ \rho vw\\ (\rho E+p)v\\ \rho v\tilde{v}\end{bmatrix}, \mathbf{F}_e^z=\begin{bmatrix}\rho w\\ \rho uw\\ \rho vw\\ \rho w^2+p\\ (\rho E+p)w\\ \rho w\tilde{v}\end{bmatrix} \tag{2}$$

$$\mathbf{F}_v^x=\begin{bmatrix}0\\ \tau_{xx}\\ \tau_{xy}\\ \tau_{xz}\\ u\tau_{xx}+v\tau_{xy}+w\tau_{xz}+\kappa\frac{\partial T}{\partial x}\\ \frac{1}{\sigma}\mu(1+\psi)\frac{\partial\tilde{v}}{\partial x}\end{bmatrix}, \mathbf{F}_v^y=\begin{bmatrix}0\\ \tau_{xy}\\ \tau_{yy}\\ \tau_{yz}\\ u\tau_{xy}+v\tau_{yy}+w\tau_{yz}+\kappa\frac{\partial T}{\partial y}\\ \frac{1}{\sigma}\mu(1+\psi)\frac{\partial\tilde{v}}{\partial y}\end{bmatrix},$$

$$\mathbf{F}_v^z=\begin{bmatrix}0\\ \tau_{xz}\\ \tau_{yz}\\ \tau_{zz}\\ u\tau_{xz}+v\tau_{yz}+w\tau_{zz}+\kappa\frac{\partial T}{\partial z}\\ \frac{1}{\sigma}\mu(1+\psi)\frac{\partial\tilde{v}}{\partial z}\end{bmatrix} \tag{3}$$

where the notations ρ , p, and E denote the fluid density, pressure and specific total energy per unit mass, respectively. $\mathbf{u}=(u,v,w)$ represents the Cartesian velocity vector and $\tilde{v}$ represents the turbulence working variable in the modified SA model. The pressure p is determined by the equation of state for an ideal gas,

$$p=(\gamma-1)\left(\rho E-\frac{1}{2}\rho(u^2+v^2+w^2)\right) \tag{4}$$

where γ is defined as the ratio of specific heats, which is 1.4 for air. τ represents the fluid viscous stress tensor and is defined, for a Newtonian fluid, as,

$$\tau_{ij}=(\mu+\mu_T)\left(\frac{\partial\mathbf{u}_i}{\partial\mathbf{x}_j}+\frac{\partial\mathbf{u}_j}{\partial\mathbf{x}_i}-\frac{2}{3}\frac{\partial\mathbf{u}_k}{\partial\mathbf{x}_k}\delta_{ij}\right) \tag{5}$$

where δ_{ij} is the Kronecker delta and subscripts i, j, k refer to the Cartesian coordinate components for $\mathbf{x}=(x,y,z)$. μ refers to the fluid dynamic viscosity and is obtained via the Sutherland's law. μ_T denotes the turbulence eddy viscosity, which is obtained by:

$$\mu_T=\begin{cases}\rho\tilde{v}f_{v1} & \text{if } \tilde{v}\geqslant 0\\ 0 & \text{if } \tilde{v}<0\end{cases} \tag{6}$$

The source term, $\mathbf{S}$, in Eq. (1) is given by $\mathbf{S}=[0,0,0,0,0,S_T]^T$, where the components for the continuity, momentum and energy equations are zero. The source term corresponding to the turbulence model equation takes the following form [5,6]: