Journal of Computational Physics 322 (2016) 33–51

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

An overset mesh approach for 3D mixed element high-order discretizations

Michael J. Brazell [a,*], Jayanarayanan Sitaraman [b], Dimitri J. Mavriplis [a]

[a] Department of Mechanical Engineering, University of Wyoming, Laramie, WY 82071-3295, USA
[b] Parallel Geometric Algorithms LLC, 814 S Mary Ave., Sunnyvale, CA 94087, USA

ARTICLE INFO

Article history:
Received 5 February 2016
Received in revised form 10 June 2016
Accepted 17 June 2016
Available online 22 June 2016

Keywords:
Overset
High-order
Discontinuous Galerkin
TIOGA

ABSTRACT

A parallel high-order Discontinuous Galerkin (DG) method is used to solve the compressible Navier–Stokes equations in an overset mesh framework. The DG solver has many capabilities including: *hp*-adaption, curved cells, support for hybrid, mixed-element meshes, and moving meshes. Combining these capabilities with overset grids allows the DG solver to be used in problems with bodies in relative motion and in a near-body off-body solver strategy. The overset implementation is constructed to preserve the design accuracy of the baseline DG discretization. Multiple simulations are carried out to validate the accuracy and performance of the overset DG solver. These simulations demonstrate the capability of the high-order DG solver to handle complex geometry and large scale parallel simulations in an overset framework.

1. Introduction

High-order methods are attractive because they provide higher accuracy with fewer degrees of freedom. Discontinuous Galerkin (DG) methods [1–22] have received particular attention; these methods combine the ideas of finite-element and finite-volume methods allowing for high-order approximations and geometric flexibility.

The goal of this work is to devise an accurate, efficient and robust three-dimensional high-order method based on DG discretizations [23] for simulating a wide variety of compressible flows in an overset grid framework. Although the DG solver can handle unstructured meshes there are some situations where even an unstructured solver could benefit from an overset grid framework. Bodies in relative motion such as helicopters or wind turbines are difficult to simulate with a single grid. Typically these simulations require mesh movement or re-meshing every time step. Overset grids solve this issue by allowing multiple grids to move relative to each other. Another advantage to overset grids is the ability to combine a near-body unstructured DG solver with an efficient cartesian mesh off-body solver, this can greatly increase the overall efficiency and capabilities of the solver. This has been demonstrated with success in [24], which combines a second-order accurate, unstructured, finite-volume solver [25] with an AMR finite difference solver [26,27]. Also, previous work by our group combined a hexahedral DG solver with a second-order, finite-volume, unstructured solver [28,29]. These approaches have two drawbacks: firstly they depend on low-order methods for the unstructured solvers and secondly many fringe cells are needed in the overlap region of the grids to maintain accuracy.

* Corresponding author.
E-mail address: mbrazell@uwyo.edu (M.J. Brazell).

http://dx.doi.org/10.1016/j.jcp.2016.06.031

To overcome these issues high-order methods with compact stencils can be used. Recently, DG has been used in an overset framework to solve the two-dimensional, Euler equations on abutting and overset grids [30]. Flux Reconstruction (FR) has been used to solve the two-dimensional, compressible Navier–Stokes equations on sliding meshes [31]. Streamline/upwind Petrov-Galerkin (SUPG) has been used to solve the two-dimensional, compressible, Navier–Stokes equations along with the Reynolds Averaged Navier–Stokes (RANS) equations on steady and unsteady moving mesh problems [32]. Spectral Element Method (SEM) has been used to solve the three-dimensional, incompressible Navier–Stokes equations on moving, overlapping, hexahedral grids [33]. This has demonstrated the feasibility and high level of accuracy that can be obtained using compact-stencil, high-order methods (DG, FR, SUPG, and SEM) in an overset framework.

In this paper, our previous work has been extended to the new DG solver using a newly developed high-order overset mesh framework called TIOGA (Topology Independent Overset Grid Assembler). The high-order, parallel DG solver [23] is capable of solving the compressible Navier–Stokes equations and the RANS equations closed by the Spalart–Allmaras turbulence model (negative-SA variant) [34,35]. It also can handle hybrid, mixed-element meshes (tetrahedra, pyramids, prisms, and hexahedra), moving meshes, curved elements, and incorporates both p-enrichment and h-refinement capabilities using non-conforming elements (hanging nodes). The novelty in this approach is the capability of using a 3D high-order method with mixed-elements, moving mesh, and turbulent flow in an overset framework [36]. This gives the ability to solve complicated relative motion problems at high-order and be combined with an efficient off-body solver.

In the following sections, the governing equations are described, followed by the DG discretization and its implementation for three-dimensional problems. The solution methodology is described next which discusses the strongly coupled, implicit, non-linear solver within the overset framework. Lastly, a variety of simulations are used to demonstrate the order of accuracy, temporal accuracy, implicit solver performance, and moving mesh accuracy of the overset solver. This is followed by two large scale simulations demonstrating the use of mixed element meshes and turbulent flow over a wing.

2. Governing equations

The Navier–Stokes equations govern the dynamics of compressible fluids and are given as:

$$\frac{\partial U_k}{\partial t} + \frac{\partial F_{ki}}{\partial x_i} = 0 \tag{1}$$

where they represent the conservation of mass, momentum, and energy. The solution vector U and flux F are defined as:

$$U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{bmatrix}, \quad F = \begin{bmatrix} \rho u & \rho v & \rho w \\ \rho u^2 + P - \tau_{11} & \rho uv - \tau_{12} & \rho uw - \tau_{13} \\ \rho uv - \tau_{21} & \rho v^2 + P - \tau_{22} & \rho vw - \tau_{23} \\ \rho uw - \tau_{31} & \rho vw - \tau_{32} & \rho w^2 + P - \tau_{33} \\ \rho uH - \tau_{1j}u_j + q_1 & \rho vH - \tau_{2j}u_j + q_2 & \rho wH - \tau_{3j}u_j + q_3 \end{bmatrix} \tag{2}$$

where ρ is the density, u, v, w are the velocity components in each spatial coordinate direction, P is the pressure, E is total internal energy, $H = E + P/\rho$ is the total enthalpy, τ is the viscous stress tensor, and q is the heat flux. The viscosity is a function of the temperature given by the Sutherland's formula. These equations are closed using the ideal gas equation of state:

$$\rho E = \frac{P}{\gamma - 1} + \frac{1}{2}\rho(u^2 + v^2 + w^2)$$

where $\gamma = 1.4$ is the ratio of specific heats. In all of the following, Einstein notation is used where the subscripts of i and j represent spatial dimensions and have a range of 1 to 3 and the index k varies over the number of variables. The arbitrary Eulerian–Lagrangian (ALE) formulation of the compressible Navier–Stokes equations is formed by adding the ALE flux terms:

$$F_{ALE} = -\begin{bmatrix} \rho\dot{x}_1 & \rho\dot{x}_2 & \rho\dot{x}_3 \\ \rho u\dot{x}_1 & \rho u\dot{x}_2 & \rho u\dot{x}_3 \\ \rho v\dot{x}_1 & \rho v\dot{x}_2 & \rho v\dot{x}_3 \\ \rho w\dot{x}_1 & \rho w\dot{x}_2 & \rho w\dot{x}_3 \\ \rho E\dot{x}_1 & \rho E\dot{x}_2 & \rho E\dot{x}_3 \end{bmatrix} \tag{3}$$

to the static flux F (defined previously in equation (3)), where $\dot{x}_i$ are the mesh movement terms in each coordinate direction. Currently only rigid, analytic mesh movements are implemented.