Journal of Computational Physics 341 (2017) 341–376

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A pressure-based semi-implicit space–time discontinuous Galerkin method on staggered unstructured meshes for the solution of the compressible Navier–Stokes equations at all Mach numbers

Maurizio Tavelli, Michael Dumbser *

Department of Civil, Environmental and Mechanical Engineering, University of Trento, Via Mesiano 77, I-38123 Trento, Italy

ARTICLE INFO

Article history:
Received 28 December 2016
Accepted 13 March 2017
Available online 29 March 2017

Keywords:
Pressure-based semi-implicit space–time discontinuous Galerkin scheme
Staggered unstructured meshes
High order of accuracy in space and time
Compressible Navier–Stokes equations
All Mach number flows

ABSTRACT

We propose a new arbitrary high order accurate semi-implicit space–time discontinuous Galerkin (DG) method for the solution of the two and three dimensional compressible Euler and Navier–Stokes equations on *staggered* unstructured curved meshes. The method is pressure-based and semi-implicit and is able to deal with all Mach number flows.
The new DG scheme extends the seminal ideas outlined in [1], where a second order semi-implicit finite volume method for the solution of the compressible Navier–Stokes equations with a general equation of state was introduced on staggered Cartesian grids. Regarding the high order extension we follow [2], where a staggered space–time DG scheme for the incompressible Navier–Stokes equations was presented. In our scheme, the discrete pressure is defined on the primal grid, while the discrete velocity field and the density are defined on a face-based staggered dual grid. Then, the mass conservation equation, as well as the nonlinear convective terms in the momentum equation and the transport of kinetic energy in the energy equation are discretized explicitly, while the pressure terms appearing in the momentum and energy equation are discretized implicitly. Formal substitution of the discrete momentum equation into the total energy conservation equation yields a linear system for only one unknown, namely the *scalar* pressure. Here the equation of state is assumed linear with respect to the pressure. The enthalpy and the kinetic energy are taken explicitly and are then updated using a simple Picard procedure. Thanks to the use of a staggered grid, the final pressure system is a very sparse block five-point system for three dimensional problems and it is a block four-point system in the two dimensional case. Furthermore, for high order in space and piecewise constant polynomials in time, the system is observed to be symmetric and positive definite. This allows to use fast linear solvers such as the conjugate gradient (CG) method. In addition, all the volume and surface integrals needed by the scheme depend only on the geometry and the polynomial degree of the basis and test functions and can therefore be precomputed and stored in a preprocessing stage. This leads to significant savings in terms of computational effort for the time evolution part. In this way also the extension to a fully curved isoparametric approach becomes natural and affects only the preprocessing step. The viscous terms and the heat flux are also discretized making use of the staggered grid by defining the viscous stress tensor and the heat flux vector on the dual grid, which corresponds to the use of a lifting operator, but on the dual grid. The time step of our new numerical method is limited by a CFL condition based only on the fluid velocity and not on the sound speed. This makes the method particularly interesting for low Mach number flows. Finally, a very simple

* Corresponding author.
E-mail addresses: m.tavelli@unitn.it (M. Tavelli), michael.dumbser@unitn.it (M. Dumbser).

http://dx.doi.org/10.1016/j.jcp.2017.03.030

combination of artificial viscosity and the *a posteriori* MOOD technique allows to deal with shock waves and thus permits also to simulate high Mach number flows. We show computational results for a large set of two and three-dimensional benchmark problems, including both low and high Mach number flows and using polynomial approximation degrees up to $p = 4$.

1. Introduction

Computational fluid mechanics is a very important field for a wide set of applications that ranges from aerospace and mechanical engineering, energy production at the aid of gas, wind and water turbines over geophysical flows in oceans, rivers, lakes as well as atmospheric flows to blood flow in the human cardiovascular system. Although the field of application is extremely large, there exists one universally accepted mathematical model of governing equations that can describe fluid flow in all the above mentioned circumstances. It can be derived from the conservation of mass, momentum and total energy and is given by the well-known compressible Navier–Stokes equations. In their complete form they can describe a wide range of phenomena, including also the effects of momentum transport via molecular viscosity and heat conduction. The compressible Navier–Stokes equations also comprehend several simplified sub systems, like the compressible Euler equations in the inviscid case, or the incompressible Navier–Stokes equations as the zero Mach number limit, where the Mach number is defined as usual by the ratio between the fluid velocity and the sound speed, see e.g. [3–5]. Furthermore, from the incompressible Navier–Stokes equations the so-called shallow water equations can be derived by integration over the depth and by assuming a hydrostatic pressure. In this sense the applications of the compressible Navier–Stokes equations split into two main classes: low Mach number flows, typical for geophysical, environmental and biological applications, and high Mach number flows that are typical of industrial applications such as in aerospace and mechanical engineering. For the high Mach number case, the families of explicit *density-based* upwind finite difference and Godunov-type finite volume schemes are very popular, see for example [6–15]. Due to the elliptic behavior of the pressure in the incompressible limit, the use of purely explicit schemes introduces a very severe restriction on the maximum time step for low Mach number flows, since the CFL condition of explicit methods includes also the sound speed. This explains why semi-implicit *pressure-based* schemes are more popular in this class of applications. In the past several semi-implicit numerical schemes have been developed for the incompressible case, see [16–24] and have been recently extended also to the new family of staggered semi-implicit discontinuous Galerkin schemes in [25–27,2]. Regarding semi-implicit schemes for the compressible case on staggered and collocated grids we refer the reader for example to the work presented in [28–36]. For implicit DG schemes applied to the compressible Euler and Navier–Stokes equations on collocated grids see [37–45].

Very recently, a new weakly nonlinear semi-implicit finite volume scheme for the solution of the compressible Navier–Stokes and Euler equations with general equation of state (EOS) was presented by Dumbser and Casulli in [1]. The aim of the present paper is to extend those ideas to higher order of accuracy in space and time and thus to develop a novel pressure-based semi-implicit staggered DG scheme for the solution of the compressible Navier–Stokes and Euler equations in multiple space dimensions that is globally and locally conservative for mass momentum and total energy and that involves a CFL time step restriction that is only based on the local flow velocity and not on the sound speed. Furthermore, we derive the method on a general unstructured curved staggered grid in order to fit also complex geometries. While the pressure is defined on a main tetrahedral (respectively triangular) grid, the density and the velocity fields are defined on a face-based staggered dual grid (respectively edge-based dual grid). The formal substitution of the discrete momentum equation into the energy equation leads to a linear system for only one single unknown, namely the scalar pressure. The discrete form of the equations looks very similar to the high order staggered DG scheme proposed in [2] for the incompressible Navier–Stokes equations. Numerical evidence shows that the good properties of the resulting linear system for the pressure can be maintained also in the compressible framework, while in the incompressible case there is a rigorous analysis available, see [26,27,46]. The use of a staggered mesh as well as the semi-implicit solution strategy applied to the resulting discrete equations makes our new scheme totally different from the space–time DG schemes presented in [47–50].

High Mach number flows typically lead to shock waves where unlimited high order numerical schemes produce spurious oscillations – the well-known Gibbs phenomenon – which can also lead to unphysical quantities, i.e. negative values for pressure or density. It is then necessary to introduce a limiter in order to overcome the problem. Several types of limiting procedures have been introduced in the past, such as WENO limiters [51–53], slope and moment limiting [54–57] and artificial viscosity (AV), see [47,37,58]. The concept of AV was already introduced in the 1950ies by Von Neumann and Richtmyer [59], in order to deal with shock waves and high Mach number flows.

A completely new way of limiting high order DG schemes was very recently developed by Dumbser et al. in [60–62], where a novel *a posteriori* sub-cell finite volume limiter was used to suppress spurious oscillations of the DG polynomials in the vicinity of shock waves and other discontinuities. Originally, the idea to use an *a posteriori* approach was introduced by Clain, Diot and Loubère [63–66] in the finite volume context with the so-called Multi-dimensional Optimal Order Detection (MOOD) method. For alternative *a priori* subcell DG limiters, see the work of Sonntag and Munz [67,68] and others [69–71].