A multi-dimensional, all Mach-number, arbitrary high order space-time discontinuous Galerkin method for the compressible Navier-Stokes equations on staggered unstructured meshes

M.Tavelli and M.Dumbser

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

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 scheme is based on the general ideas proposed in [1] on staggered Cartesian meshes where a second order method for the compressible Navier-Stokes equations with a general equation of state was introduced. Regarding the high order extension, we follow the ideas introduced in [2]. While the discrete pressure is defined on the primal grid, the discrete velocity field and the density are defined on a face-based dual grid. Formal substitution of the discrete momentum equation into the energy equation yields a linear system for the only unknown the pressure. Here the equation of state is assumed linear with respect to the pressure and the enthalpy is taken explicitly and then updated using a simple Picard procedure. Thanks to the use of staggered grid, the system is a sparse block five-points system for three dimensional problems and it is a four-points system for the two dimensional case. Furthermore, for high order in space, low order in time, the system is observed to be symmetric and positive definite. This allows to use fast linear solvers such as the 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 a significant saving 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. This new numerical method is limited by a CFL condition due to an explicit treatment of the nonlinear convective term. However, this condition is based on the fluid velocity and not on the sound speed. This make the method particularly interesting for low Mach number flows. Finally, a simple combination of artificial viscosity and MOOD technique allows to simulate also shock wavers and high Mach number flows. In this work we will present several numerical tests on a large range of Mach number flows and several academic numerical test problems up to polynomial degrees $p = 4$.

REFERENCES

[1] M. Dumbser and V. Casulli, A conservative, weakly nonlinear semi-implicit finite volume scheme for the compressible Navier-Stokes equations with general equation of state. *Applied Mathematics and Computation*, Vol. **272**, pp. 479–497, (2016).

[2] M. Tavelli and M. Dumbser, A staggered, space-time discontinuous Galerkin method for the three-dimensional incompressible Navier-Stokes equations on unstructured tetrahedral meshes. *Journal of Computational Physics*, Vol. **319**, pp. 294–323, (2016).