High-Order CENO Finite-Volume Scheme for Low-Speed Viscous Flows on Three-Dimensional Unstructured Mesh

Marc R.J. Charest*, Clinton P.T. Groth* and Pierre Q. Gauthier†

*University of Toronto Institute for Aerospace Studies
4925 Dufferin Street, Toronto, Ontario, Canada M3H 5T6

†Energy Engineering & Technology, Rolls-Royce Canada Limited
9245 Côte-de-Liesse, Dorval, Québec, Canada H9P 1A5

Abstract: High-order discretization techniques remain an active area of research in computational fluid dynamics (CFD) since they offer the potential to significantly reduce the computational costs necessary to obtain accurate predictions when compared to lower-order methods. In spite of the successes to date, efficient, universally-applicable, high-order discretizations remain somewhat illusive, especially for more arbitrary unstructured meshes and for incompressible/low-speed flows. A novel, high-order, central essentially non-oscillatory (CENO), cell-centered, finite-volume scheme is proposed for the solution of the conservation equations of viscous, incompressible flows on three-dimensional unstructured meshes. The proposed scheme is applied to incompressible form of the steady and unsteady Navier-Stokes equations using the pseudo-compressibility formulation and the resulting discretized equations are solved with an implicit Newton-Krylov algorithm. For unsteady flows, the temporal derivatives are discretized using the family of high-order backward difference formulas (BDF) and the resulting equations are solved via a dual-time stepping approach. The proposed finite-volume scheme for fully unstructured mesh is applied to several idealized viscous flow problems and demonstrated to provide both fast and accurate solutions of steady and unsteady viscous flows.

Keywords: Numerical Algorithms, Computational Fluid Dynamics, High-Order Methods, Incompressible Flows

Introduction

Computational fluid dynamics (CFD) has proven to be an important enabling technology in many areas of science and engineering. In spite of the relative maturity and widespread success of CFD in aerospace engineering, there is a variety of physically-complex flows which are still not well understood and are very challenging to predict by numerical methods. Such flows include, but are not limited to, multiphase, turbulent, and combusting flows encountered in propulsion systems (e.g., gas turbine engines and solid propellant rocket motors). These flows present numerical challenges as they generally involve a wide range of complicated physical/chemical phenomena and scales.

Many flows of engineering interest are incompressible or can be approximated as incompressible to a good degree of accuracy, i.e. low-speed flows. Incompressible flows can prove challenging to solve numerically because their governing equations are ill-conditioned as the partial derivative of density with respect to time vanishes. Various methods of solving the incompressible Navier-Stokes equations have been proposed and successfully used to overcome this ill-conditioning [1]. These include but are not limited to the pressure-Poisson, projection, vorticity-based, and pseudo-compressibility methods. The equations governing fully-compressible flows have also been successfully applied to incompressible and low-speed flows using preconditioning techniques [2]. The pseudo-compressible formulation [3, 4] is attractive because it is easily extended to three dimensions and applied in conjunction with high-order upwind finite-volume schemes [5].

High-order methods have the potential to significantly reduce the cost of modelling physically-complex flows, but this potential is challenging to fully realize. As such, the development of robust and accurate high-order methods remains an active area of research. Standard lower-order methods (i.e, methods up to second order) can exhibit excessive numerical dissipation for multi-dimensional problems and are often not practical for physically-complex flows. High-order methods offer improved numerical efficiency when accurate solution representations are sought since fewer computational cells are required to achieve a desired level of accuracy [6]. For hyperbolic conservation laws and/or compressible flow simulations, the main challenge involves obtaining accurate discretizations while ensuring that discontinuities and shocks are handled reliably and robustly [7]. High-order schemes for elliptic partial differential equations (PDEs) that govern diffusion processes should satisfy a maximum principle, even on stretched/distorted meshes, while remaining accurate [8]. In spite of many advances, there is still no consensus for a robust, efficient, and accurate scheme that fully deals with all of the aforementioned issues and is universally applicable to arbitrary meshes.

Ivan and Groth [9, 10] proposed a high-order Central Essentially Non Oscillatory (CENO), cell-centered, finite-volume scheme that was demonstrated to remain both accurate and robust in a variety of physically-complex flows. The CENO scheme is based on a hybrid solution reconstruction procedure that combines an unlimited high-order k-exact, least-squares reconstruction technique with a monotonicity preserving limited piecewise linear least-squares reconstruction algorithm. Fixed central stencils are used for both the unlimited high-order k-exact reconstruction and the limited piecewise linear reconstruction. Switching between the two reconstruction algorithms is determined by a solution smoothness indicator that indicates whether or not the solution is resolved on the computational mesh. This hybrid approach avoids the complexities associated with reconstruction on multiple stencils that other essentially non-oscillatory (ENO) and weighted ENO schemes can encounter. Originally developed for structured two-dimensional mesh, this scheme has been successfully extended to two- and three-dimensional unstructured mesh by McDonald et al. [11].

In this paper, the high-order CENO finite-volume scheme is extended to solve the equations governing incompressible, viscous, laminar flows on three-dimensional general unstructured mesh. For steady flows, the equations are solved using the pseudo-compressibility approach coupled with an implicit Newton-Krylov algorithm. The proposed scheme is extended to unsteady flows via a dual-time stepping approach. High-order temporal accuracy is achieved through the use of the backward difference formula (BDF) time-marching schemes. The resulting algorithm is applied to both steady and unsteady flows and analyzed in terms of accuracy, computational cost, and parallel performance. In particular, the spatial and temporal accuracy of solutions are examined and the influence of resolution on accuracy is assessed for several idealized flow problems. Both the steady flow over a flat plat and the unsteady decay of Taylor vortices were studied here. The ability of the scheme to rapidly and robustly obtain steady and unsteady solutions is demonstrated along with its ability to accurately represent solutions with smooth extrema yet robustly handle under-resolved and/or non-smooth solution content.

Preliminary results for the Taylor vortex decay obtained using the proposed algorithm are illustrated in Fig. 1. These solutions were obtained using $k = 2$ reconstruction and BDF2.

References

[1] D. Drikakis and W. Rider. *High-resolution methods for incompressible and low-speed flows*. Springer-Verlag, Berlin, Heidelberg, 2005.
[2] J. M. Weiss and W. A. Smith. *AIAA J.*, 33(11):2050–2057, 1995.
[3] A. J. Chorin. *J. Comput. Phys.*, 2(1):12–26, 1967.
[4] Dochan Kwak, James L. C. Chang, Samuel P. Shanks, and Sukumar R. Chakravarthy. *AIAA J.*, 24(3):390–396, 1986.
[5] Z. Qian and J. Zhang. *Int. J. Numer. Meth. Fluids*, 2011. Article in Press.
[6] S. Pirozzoli. *J. Comput. Phys.*, 219(2):489–497, 2006.
[7] A. Harten, B. Engquist, S. Osher, and S. R. Chakravarthy. *J. Comput. Phys.*, 71(2):231–303, 1987.
[8] W. J. Coirier and K. G. Powell. *AIAA J.*, 34(5):938–945, May 1996.
[9] L. Ivan and Clinton P. T. Groth. *AIAA paper 2007-4323*, 2007.
[10] L. Ivan and Clinton P. T. Groth. *AIAA paper 2011-0367*, 2011.
[11] Sean D. McDonald, Marc R. J. Charest, and Clinton P. T. Groth. *AIAA paper 2011-3854*, 2011.

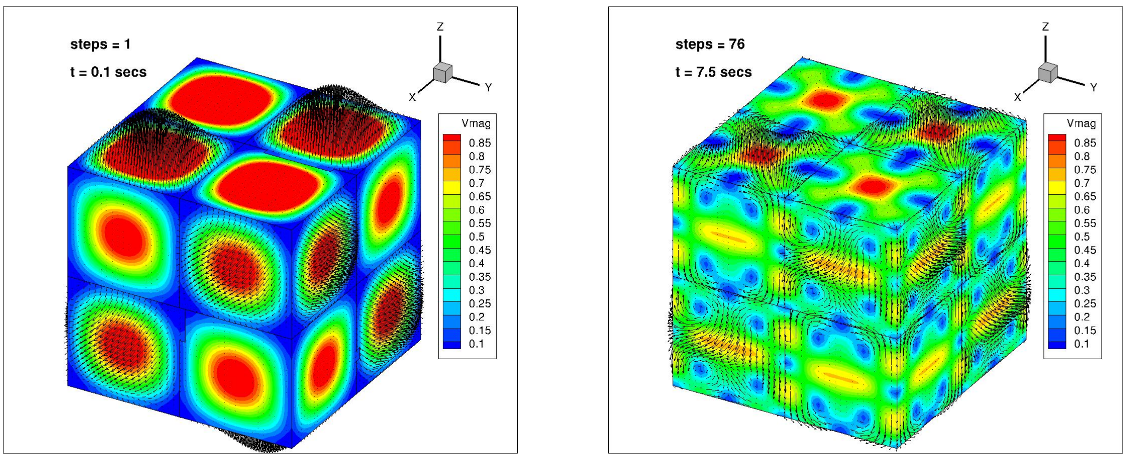

Fig. 1. Velocity magnitude and velocity vectors for the Taylor decaying vortices at two time levels.