14th World Congress on Computational Mechanics (WCCM XIV)
8th European Congress on Computational Methods in Applied Science and Engineering (ECCOMAS 2020)
July 19–24, 2019, Paris, France

COST-VS-ACCURACY IN COMPUTATIONAL FLUID DYNAMICS TRACK 600

ELIAS BALARAS*, **FRANCESCO CAPUANO**† **and LUIGI DE LUCA**†

* George Washington University
Washington DC, 20052, USA
balaras@gwu.edu

† Università di Napoli Federico II
Napoli, 80125, Italy
francesco.capuano@unina.it, deluca@unina.it

Key words: High-order methods, structured/unstructured meshes, immersed-boundary method, open-source software, direct/large eddy simulation

ABSTRACT

Utilizing computational fluid dynamics (CFD) for complex engineering problems requires the adoption of numerical algorithms that are simultaneously accurate, robust, flexible and, ultimately, efficient (i.e., with a high accuracy/cost ratio). The fulfilment of such a set of conflicting requirements continues to thrill the scientific community, especially when it comes to choosing the *best* method for a specific field of applications. On the one hand, industrial CFD simulations have been traditionally carried out using low-order finite-volume or finite-difference schemes, often in conjunction with the immersed boundary method (IBM) to cope with complex or moving boundaries. On the other hand, recent years have seen the development of high-order techniques, such as discontinuous Galerkin and spectral-element methods, that arguably combine high accuracy with geometric flexibility and ease of parallelization. Furthermore, the whole spectrum of methodologies is nowadays within reach of the community by the availability of open-source software (e.g., OpenFOAM, Gerris, Nek5000, PyFR, etc.). As a result, there is a growing interest in understanding and comparing the efficiency of various approaches/solvers.

This minisymposium (MS) brings together researchers from academia, government labs and industry in an effort to assess and explore the efficiency of a wide range of CFD techniques. The MS invites comparative studies regarding (but not limited to): unstructured vs Cartesian (IBM) techniques, low- vs high-order spatial/temporal methods, assessment of open-source solvers, efficiency of stabilization strategies (e.g., discrete conservation principles, filtering, etc.), adaptive mesh refinement strategies. Contributions focusing on novel architectures (e.g., GPU) and high-performance computing aspects are also welcomed. Example studies can be found in references [1, 2] below.

REFERENCES

[1] Vermeire, B.C., Witherden, F.D. and Vincent, P.E. On the utility of GPU accelerated high-order methods for unsteady flow simulations: A comparison with industry-standard tools. *J. Comput. Phys.* (2017) **334**:497-521.

[2] Capuano, F., Palumbo, A. and de Luca, L. Comparative study of spectral-element and finite-volume solvers for direct numerical simulation of synthetic jets. *Comput. Fluids* (2019) **179**:228-237.