PFEM based solvers implemented in the OpenFOAM® suite

Juan M. Gimenez [*†], Horacio J. Aguerre [*o], Norberto M. Nigro [*†], Sergio R. Idelsohn [*†+#]

[*] Centro de Investigación de Métodos Computacionales (CIMEC CONICET/UNL), Santa Fe, Argentina

[†] Facultad de Ingeniería y Ciencias Hídricas, Universidad Nacional del Litoral, Santa Fe, Argentina

[+] Institució Catalana de Recerca i Estudis Avançats (ICREA), Barcelona, Spain

[#] Centre Internacional de Mètodes Numèrics en Enginyeria (CIMNE), Barcelona, Spain

[o] Universidad Tecnológica Nacional - Facultad Regional Concepción del Uruguay, (UTN-FRCU), Concepción del Uruguay, Argentina

ABSTRACT

The latest version of the Particle Finite Element Method (PFEM-2) [1] is a numerical method based on the Lagrangian formulation of the equations which presents advantages in terms of robustness and efficiency over classical Eulerian methodologies especially when convection plays an important role [2]. Previous publications demonstrated its ability to achieve solutions with a good compromise between accuracy and efficiency, in both, academic [3,4] and real engineering problems [5] where very complex geometries and operating conditions require very large computations. This work affords the next step which consists in opening the implementation of this methodology in order to extend the number of users along the world.

In this context, the suite OpenFOAM® is the platform selected where including the developments. This free open source code is one of three world's most used CFD softwares and the library is structured such as resulting simple to develop reliable and efficient computational continuum-mechanics algorithms [6]. This selected platform implements the Finite Volume Method (FVM) which discretizes the space in a different way from the Finite Element Method (FEM), from which the PFEM-2 is based. This fact makes necessary the development of FVM-based interpolations and projection operators in order to communicate the fields between the background fixed-mesh and the cloud of particles. A deep discussion of these new strategies is included in this work.

When stable, monotonous and accurate convective solutions are searched, the High Resolution Schemes (HRS) as TVD and NVD are the native OpenFOAM® best discretization schemes among the availables. However, in these schemes is necessary to previously know the features of the solution due to their compressive/non-compressive behavior. In this context, employing particles for convection, as in PFEM-2, gives several advantages as enlarging time-step without distorting any kind of solution shape [7]. Another important advantage of the Lagrangian approach is that convection becomes independent of mesh quality avoiding typical FVM induced errors as skewness. Benchmark tests are proposed to verify these facts: accuracy of results is evaluated and special focus on computing times and parallel efficiency is done in order to demonstrate the improvement that offers PFEM-2 over the currently available CFD tools.

REFERENCES

[1] Idelsohn S., Nigro N., Gimenez J., Rossi R., Marti J., "A fast and accurate method to solve the incompressible navier-stokes equations", Engineering Computations , 30-Iss:2:197–222 (2013).

[2] Idelsohn S., Oñate E., Nigro N., Becker P. and Gimenez J., "Lagrangian versus eulerian integration errors", Computer Methods in Applied Mechanics and Engineering, 293(0):191 – 206, 2015
[3] Gimenez J., González L. "An extended validation of the last generation of particle finite element method for free surface flows", J Comput Phys , 284(0):186–205 (2015).
[4] Gimenez J., Nigro N, Idelsohn S. and Oñate E.. "Surface tension problems solved with the particle finite element method using large time-steps". Computers Fluids, 141:90-104 (2016).
[5] Gimenez, J., Ramajo, D., Márquez Damián, S. et al. Comp. Part. Mech. (2016). doi:10.1007/s40571-016-0135-2
[6] H. Weller, G. Tabor, H. Jasak and C. Fureby, "A tensorial approach to computational continuum mechanics using object-oriented techniques", Computers in Physics, 12, 620-631 (1998)
[7] Gimenez J, "Enlarging time-step for solving one and two-phase flows with the Particle Finite Element Method", Ph.D. Thesis, Universidad Nacional del Litoral, Argentina (2015)