Asociación Argentina
de Mecánica Computacional

Mecánica Computacional Vol XXXVI, págs. 9-9 (resumen)
José G. Etse, Bibiana M. Luccioni, Martín A. Pucheta, Mario A. Storti (Eds.)
San Miguel de Tucumán, 6-9 Noviembre 2018

A PARTICLE-BASED METHOD FOR THE SOLUTION OF TURBULENT FLUID FLOWS

Sergio R. Idelsohn[a,b], Norberto M. Nigro[b], Axel Larreteguy[c], Juan M. Gimenez[b], and Pavel Ryzhakov[a]

[a] *CIMNE, Barcelona, Spain, sergio@cimne.upc.edu, https://www.upc.edu/ca*

[b] *CIMEC, Santa Fe, Argentina*

[c] *UADE, Buenos Aires, Argentina.*

Keywords: DNS, Lagrangian formulations, multi-scale problems, homogenization.

Abstract. Although the Navier-Stokes equations are equivalently applicable to both laminar and turbulent fluid flows, the current computing power generally precludes employing fine meshes that would allow to simulate turbulent flows without introducing empirical approximations. The prediction power of the models is therefore restricted to problems within the margins of the selected empirical approximation. Solving a problem without such approximations on a mesh sufficiently fine so as to represent the whole expected range of eddy sizes is known as "Direct Numerical Simulation" (DNS). Taking into account that many fluid flow problems of industrial interest are indeed turbulent, it is worthwhile to continue improving the models so that they fit more and more with the physics of the problem. Simulating a CFD problem in a given domain with a "fine enough" DNS mesh introduces an unmanageable number of unknowns for current computers. The project we are working on involves modelling turbulent flow in a DNS fashion, i.e. without any additional turbulence model. However, we strive to develop an approach considerably more computationally efficient than a classical DNS. The basic idea is as follows. The total solution is splitted in two parts: a macro solution obtained on a coarse mesh, and a micro solution obtained on a fine mesh that can be subdivided into many equally-shaped small domains, the so-called RVEs (for "Representative Volume Elements"). These RVEs, in principle, can be solved individually and independently from each other. The RVEs may even be previously solved off-line for different time- dependant loads. The micro and macro problems are coupled iteratively. Another important aspect of the solution is the use of a particle-based fluid formulation at the macro level to move the fine mesh. Then, the RVEs will take care not only of the transport of the velocity and pressure, but also of the transport of the micro-eddies that may appear in some RVEs, thus convecting the turbulent energy. This is a very important feature in turbulent flows where turbulence is produced in some high gradient regions and is then convected to other regions with different gradients. This new technique open the door to the simulation of active fluids for which none of the current turbulent models would give acceptable results. This presentation is a work in progress, but the results obtained so far are encouraging.