Massively parallel direct numerical simulations of bubbly flows

L. Silva*, **Z. Hassani***,†, **T. Coupez*** **and H. Digonnet***

* Ecole Centrale de Nantes
High Performance Computing Institute
1 rue de la Noë, 44321 Nantes Cedex 3, France
e-mail: luisa.rocha-da-silva@ec-nantes.fr - web page: https://ici.ec-nantes.fr

† Arcelor Mittal
Global Research and Development, Maizires Process
Voie Romaine, BP30320, 57283 Maizières-lès-Metz Cedex, France
e-mail: zineb.hassani@arcelormittal.com - Web page: http://www.arcelormittal.com

ABSTRACT

Gas bubbles are widely present in manufacturing processes, either as a consequence of the processing conditions or as an added element that will promote better final properties. For example, in steel-manufacturing, argon bubbles are injected in the molten steel to help mixing and produce an homogeneous chemical composition, or to limit nozzle clogging arising from the inclusions transported with the liquid. Efficiency of bubble addition or creation is often dependent on the bubble size.

In this paper, computational flow dynamics of a representative system composed of a Newtonian fluid and numerous gas bubbles is studied. A two phase modeling approach is considered, the two fluids being treated as a continuous phase in an inhomogeneous Navier Stokes solver. The interface between phases is defined by a level set like method and the motion and deformation is obtained by a transport equation. To perform such simulations, a finite element code (ICItech) is used, where exists a monolithic adaptive finite element solver. They also require massively parallel implementations of the following tools: resolution of very large linear/non-linear solver for the Navier-Stokes and bubble advection equations, anisotropic mesh adaptation on unstructured distributed meshes, automatic generation of samples with a very large number of bubbles. An important feature is the high level of the surface tension value that may considered which requires an accurate prediction of the capillary force, achieved through an implicit treatment of this term in the solver.

To validate our approach, the case of a single bubble rising in a liquid matrix is considered, at different Bond and Morton numbers (to represent physics of multiphase systems like air/water, argon/steel...). Then, scalability of the code is tested on computational domains with several thousands of bubbles, for fixed material properties and initial size distribution, to see how well is it able to predict expected phenomena like clustering, bubble deformation, coalescence or break-up,... for different flow boundary conditions. To run full test cases of bubbles evolution, up to 8 Ki cores are used.

REFERENCES

[1] Coupez, T., Silva, L. and Hachem, E. Implicit boundary and adaptive anisotropic meshing, *New Challenges in Grid Generation and Adaptivity for Scientific Computing*, (2015) **5**, 1–18.

[2] Kosseifi, N., Hachem, E., Silva, L., Boyer, S.A.E., Massoni, E. and Coupez, T. Numerical simulation of boiling during the quenching process, *10e colloque national en calcul des structures* (2011).

[3] Digonnet, H., Coupez, T., Laure, P. and Silva, L. Massively parallel anisotropic mesh adaptation, *Int. J. High Perf. Comp. Appl.*, (2017) accepted.