Proceedings of CHT-15
ICHMT International Symposium on Advances in Computational Heat Transfer

May 25-29, 2015, Rutgers University, Piscataway, USA

CHT-15-014

SIMULATIONS OF TWO-PHASE FLOWS WITH A MULTIFIELD APPROACH

Solène Fleau[*,§], Stéphane Mimouni[*], Nicolas Mérigoux[*], Stéphane Vincent[**]
[*]Electricité de France, R&D Division, Chatou, France.
[**]Laboratoire de Modélisations et Simulations Multi-Echelle, Université Paris-Est, France.
[§]Correspondence author. Email: solene.fleau@edf.fr

ABSTRACT

Two-phase flows are present in nuclear power plants at normal conditions in condenser and steam generator and may appear during various hypothetical accidents such as boiling crisis. Therefore, important investigations are carried out to understand these complex flows to ensure safety of the nuclear reactors and the thermal power systems.

In NEPTUNE_CFD code, the choice is made to model these bubbly flows with a hybrid approach, in which the gas phase is separated into two fields with different closure laws. Thus, the distorted bubbles are simulated with an interface locating method whereas the small and spherical bubbles are modelled through a dispersed approach within the two-fluid model. Coupling terms are implemented in isothermal conditions in order to allow mass exchanges between the two fields. With this method, the behaviour of the large distorted bubbles is predicted with a good accuracy and at the same time, the CPU consumption is limited by modelling the small scales.

The flow motion is followed using the two-fluid model of Ishii extended to n-phase. The three balance equations, describing the mass, momentum and energy conservation, are solved for each field. The dispersed gas field is modelled using validated closure laws presented in Mimouni et al. [2010, 2011]. Concerning the located gas field, motion equation is closed by a local surface tension and an internal wall law. This law is applied at the interface to enforce the equality between the velocities of the two continuous fields. Moreover, an artificial compression step is run until convergence to calculate accurately the local interface curvature and the normal vector.

In this paper, the three field approach is being generalized to free surfaces, considered as continuous gas phase with zero curvature. A new internal wall law is also investigated to take into account the variation of viscosity. Different verification test cases are proposed: simulations of a static air bubble, the Thorpe experiment [*e.g.*, Thorpe 1969], in which Kelvin-Helmholtz instabilities are observed, and the Rayleigh Taylor instabilities. This new approach shows reasonable convergence of the circularity and the intensity of spurious velocities with the mesh refinement (see Figure 1). The interface position in the simulation of the Rayleigh Taylor instabilities (see Figure 2(a)) and the interface velocity for the Thorpe experiment simulation (see Figure 2(b)) compare also quite well with the theoretical expectations.

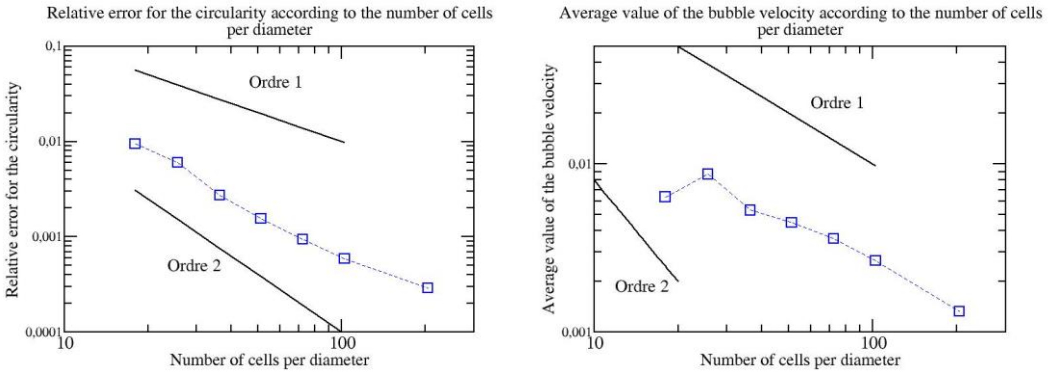
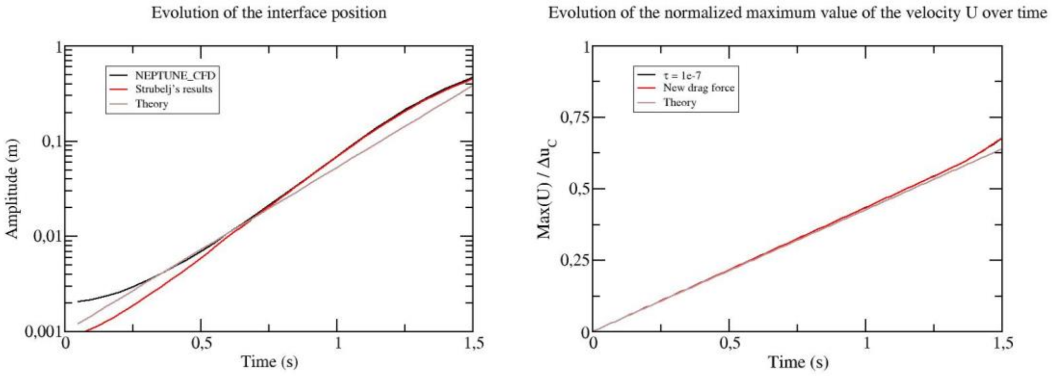


Figure 1. Effect of the mesh refinement on the relative error for the circularity and the spurious velocities induced by the simulation of a static air bubble.



(a) (b)

Figure 2. Comparison between the two drag forces at the beginning of the simulations in terms of (a) interface position for the Rayleigh-Taylor instabilities and (b) maximum interface velocity for the Thorpe experiment.

REFERENCES

Mimouni, S.et al. [2010], A second order turbulence model based on a reynolds stress approach for two-phase boiling flow and application to fuel assembly analysis, *Nuclear Engineering and Design*, Vol. 240, pp 2225-22232.

Mimouni, S. et al. [2010], Combined evaluation of second order turbulence model and polydispersion model for two-phase boiling flow and application to fuel assembly analysis, *Nuclear Engineering and Design*, Vol. 241, Issue 11, pp 4523-4536.

Thorpe, S. [1969], Experiments on the instability of stratified shear flows: immiscible fluids, *J Fluid Mech.*, Vol. 39, pp 25–48.