SIMULATIONS OF TWO-PHASE FLOWS WITH A MULTIFIELD APPROACH

Solène Fleau[*,**], Stéphane Mimouni[*] and Stéphane Vincent[**]
[*]Electricité de France, R&D Division, 6 Quai Watier, Chatou, France
[**]Laboratoire de Modélisations et Simulations Multi-Echelle, Université Paris-Est Marne-La-Vallée UPEM, 5 boulevard Descartes, Champs sur Marne, 77454 Marne la Vallée Cedex 2, France

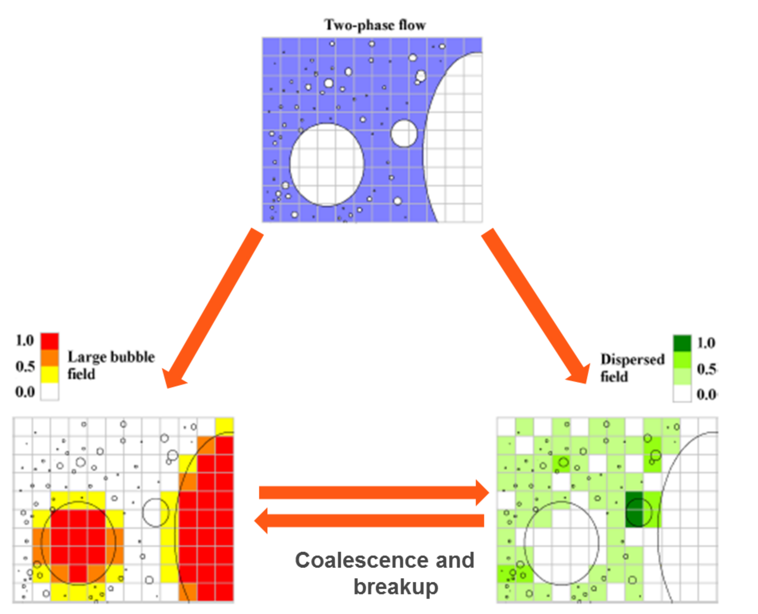
Context

Prediction of flow regimes remains a major limiting phenomenon for the analysis of operation and safety of both nuclear reactors and conventional thermal power systems. As an example, comprehension of flow behavior inside a steam generator is a crucial issue to prevent tube breaking caused by vibrations. To ensure an accurate simulation of these flows, the CMFD codes have to take up many challenges, among others the ability of dealing with a variety of inclusion sizes. The classical two-fluid model allows simulating small spherical inclusions but is not able to compute large deformable inclusions. Thus, a new approach, called the multifield approach, implemented in the CMFD code NEPTUNE_CFD is presented. Based on this two-fluid model, this approach includes an interface tracking method for large and deformable structures and takes into account turbulence effects and phase changes.

Multifield Approach within a two-fluid model

The classical two-fluid model has been developed for the simulation of small spherical inclusions. Thus, to simulate complex flows containing large interfaces, the method has to be adapted to deal with resolved interfaces. Therefore, we developed a multifield approach based on the two-fluid model:

- The small spherical structures are defined as a dispersed field evolving in a continuous carrier field, as usually done with a two-fluid model.
- The large deformable bubbles are considered as interfaces between two different fields.



Decomposition of a two-phase flow within the mutlifield approach, blue: continuous liquid field (subscript: cl), red: continuous gas field (subscript: cg) and green: dispersed gas field (subscript: dg).

Large Bubble Model

The interactions between the carrier field and the dispersed one are well known and the classical closure laws such as the lift and the drag forces have been widely validated. However, the accurate simulation of interactions between the two continuous fields within the two-fluid model requires specific treatments referred as the Large Bubble Model:

- Deformable interfaces → Surface tension model
- Two different velocities defined at the interface, one for each continuous field → Drag force law
- Interface smearing caused by the two-fluid model → Interface sharpening equation

Turbulence model

Large Eddy simulation (LES) has been investigated through *a priori* studies to identify the seven subgrid terms to be modeled, their order of magnitude and to compare different turbulence models. This study highlighted that standard models such as the Smagorinsky model were not able to reproduce negative values of turbulent viscosity close to interfaces.

Phase change and mass transfer

- Mass transfer term: $\Gamma_k = \dfrac{q_p + q_k}{H_p - H_k}$ for continuous fields at an interface with phase changes based on a specific heat transfer term: $q_k = \alpha_k \lambda_k \nabla T_k \cdot \nabla \alpha_k$
- Transition between the continuous gas field and the dispersed gas field (coalescence of small spherical inclusions, break up of large structures).

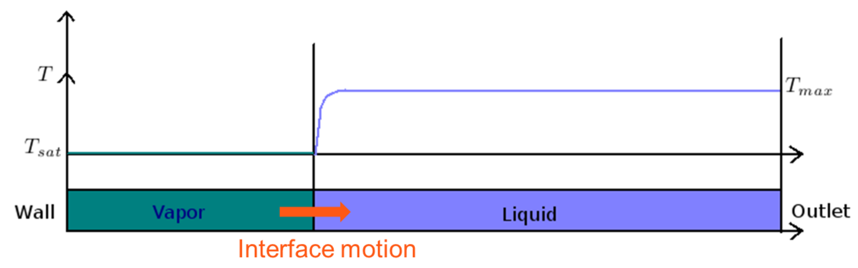
Simulation results

The multifield approach has been widely validated on a variety of test cases representing different physical phenomena, flow regimes and scales, from the analytical case to the industrial configuration.

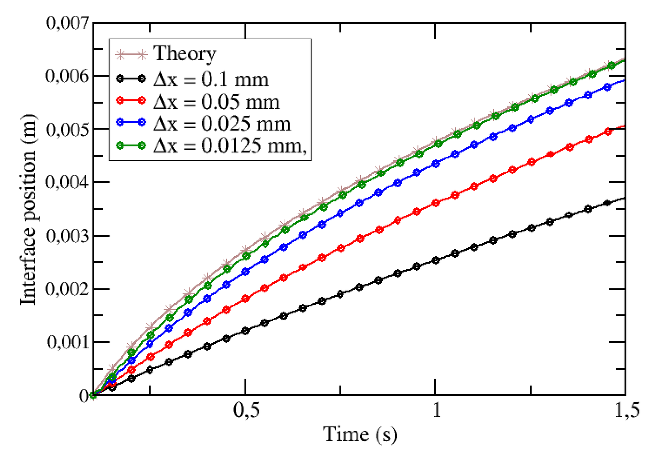
From analytical cases ...

Sucking problem:

- Phase change controls the interface motion
- Validation of the heat transfer term



Schematic view of the sucking problem, vapor is at the saturation temperature and water is superheated.

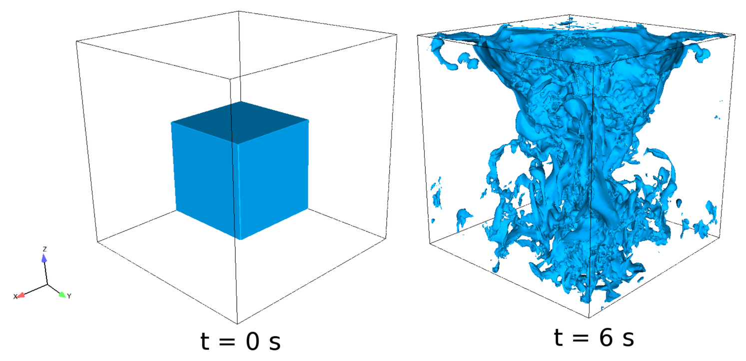


Evolution of the interface position for different mesh refinements.

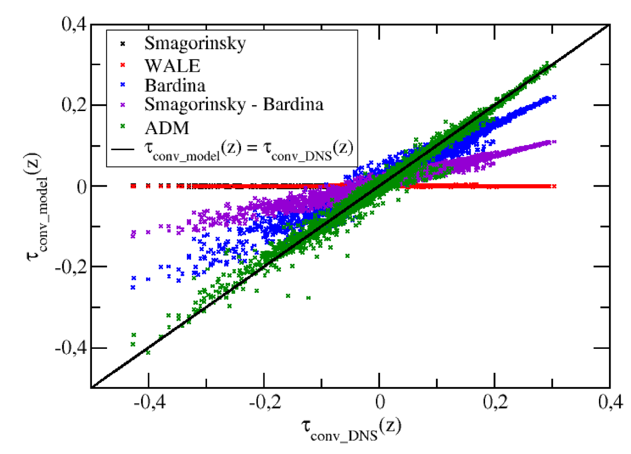
... Direct Numerical Simulations ...

Phase inversion benchmark:

- *A priori* LES study
- Comparison of turbulence models



Cube of oil (blue) evolving under gravity in water at different times.

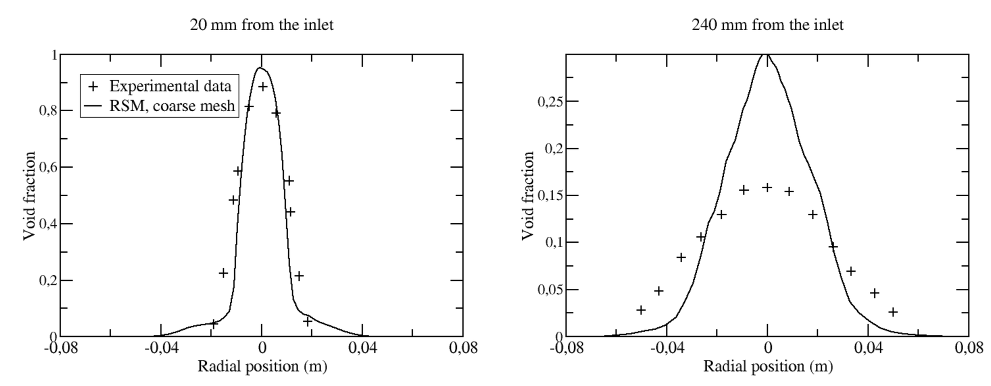


A priori LES study, comparison of the modelled convective subgrid term with the DNS term.

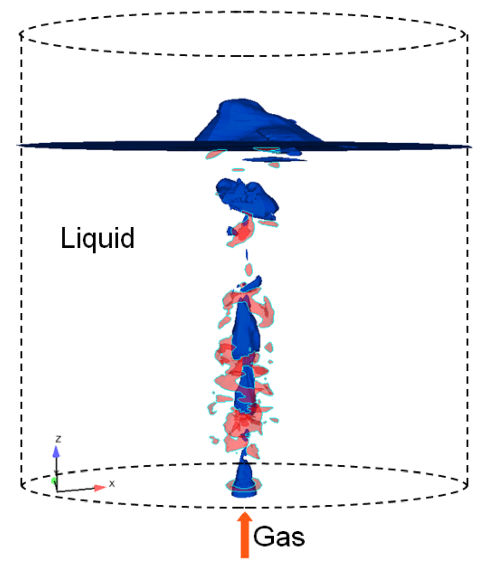
... Elementary validations ...

Castillejos' test case:

- Three fields test case
- Validation of the transition term between the gas fields



Averaged void fractions profiles at two different heights compared to the experimental results.

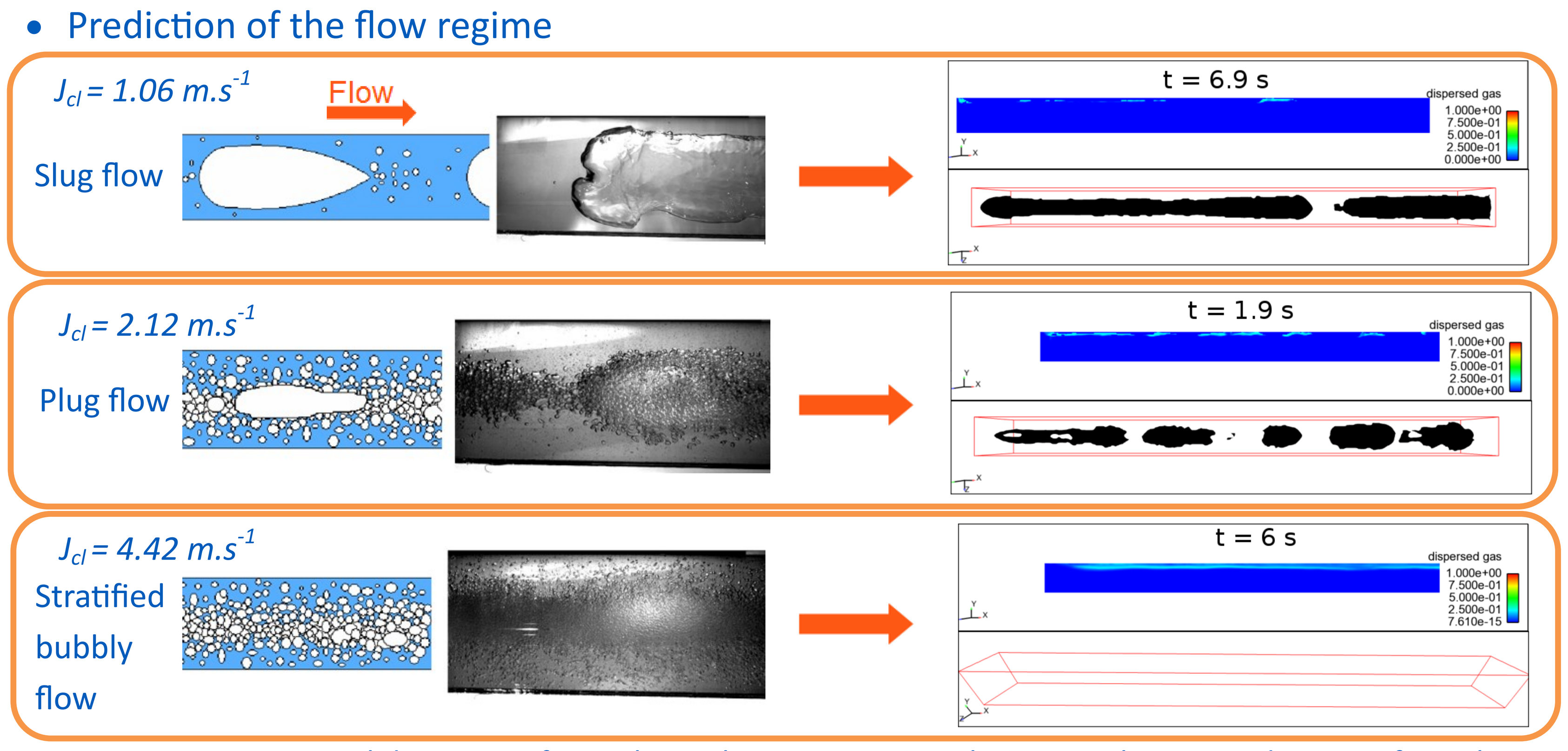


Gas plume at 1 s after the beginning of the injection, blue: isosurface of α_{cg}, red: isosurface of α_{dg}.

... To industrial configurations.

METERO test case:

- Air/water flow in a horizontal cylindrical pipe
- Prediction of the flow regime



Experimental data, view from above the test section, gas velocity J_G = 0.1273 m.s^{-1}.

Simulation results, top: side view of α_{dg}, bottom: view from above of the isosurface of α_{cg}.

[1] Denèfle R., Mimouni S., Caltagirone J.P., Vincent S., "Multifield hybrid approach for two-phase flow modeling – Part 1: Adiabatic flows", *Computers & Fluids.*, Vol. 113, pp. 106-111, 2015.
[2] Fleau S., Mimouni S., Mérigoux N. and Vincent S., "Validation of a multifield approach for the simulations of two-phase flows ", *Computational Thermal Sciences*, Vol. 7, Issue 5-6, pp. 441-457, 2015.
[3] Mimouni S., Fleau S. and Vincent S., "CFD calculations of flow pattern maps and LES of multiphase flows", *Nuclear Engineering and Design*, 2016.