CASL SCIENCE & TECHNOLOGY

Consortium for Advanced Simulation of Light Water Reactors

ITM/DNS for High Volume Fraction Bubbly Flow Regimes: Simulations for Closure Development

An essential part of developing a closed form set of equations (closures) for prediction of two-phase flow with computational fluid dynamics (CFD) is understanding how the bubbles generated by boiling interact. An accurate prediction of moderator and fuel performance once boiling has begun is needed to simulate CASL Challenge Problems related to boiling water reactors (BWRs), departure from nucleate boiling (DNB) behavior in pressurized water reactors (PWRs), loss of coolant accidents (LOCAs), and other scenarios where two-phase flow is present. NCSU researchers Fang and Bolotnov and UND researchers Lu and Tryggvason are breaking new ground in developing insights to this complex flow phenomena.

Previous direct numerical simulation (DNS) studies of multiphase flows have been in excellent agreement with experimentally based correlations. Bubbly flows (as opposed to boiling flows) lend themselves relatively well to small scale direct numerical simulations. They are therefore a natural starting point for investigations using DNS data to build closure models for the large scale flow fields within a nuclear reactor.

When implemented in CFD, DNS solves the Navier–Stokes equations numerically without the use of a turbulence model; this requires that the whole range of spatial and temporal scales of the turbulence must be resolved. One key to the DNS of multiphase flows is the accurate prediction of the phenomena taking place at the interface separating the phases—that is, at the surface of the bubbles. Use of direct interface tracking methods (ITM) uses a set of single-phase conservation equations, known as the one-fluid formulation, where the differences in material properties and surface tension are accounted for by solving a convection equation. Thus, DNS with ITM provides researchers with a small scale analytical method to examine two-phase flow dynamics.

To examine the merging and breakup of bubbles in churn-turbulent gas-liquid flows, the CASL thermal-hydraulics team has conducted a series of simulations of bubbles in a channel containing turbulent flow using a Front Tracking code (FTC3D) and a Level Set based code (PHASTA). The bubble interfaces separating the gas and the liquid is shown in Figure 1 for five different points in time for four variants of surface tension.

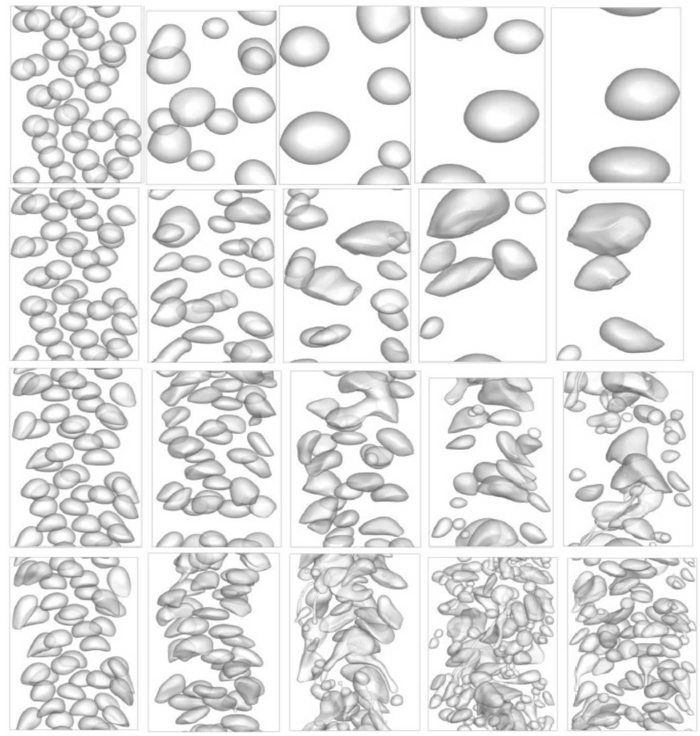

Figure 1 DNS IMS for five variants of surface tension at the same void fraction: (a) 0.08 (b) 0.01 (c) 0.004 (d) 0.002. The frames have been selected to illustrate how the flow evolves and are not evenly spaced in time.

As the bubbles move through the flow, the nearly spherical high surface tension bubbles tend to move to the channel wall, leading to rapid coalescence, and re-introduction to the free stream as the bubble reaches a critical size. Thus, when the surface tension is sufficiently high, the bubbles continuously merge to form larger and larger bubbles, until most of the gas is coalesced into one large bubble. As the bubbles in the high surface tension case coalesce they move to the center of the channel and since the bubble becomes ellipsoidal, it tends to block the channel and thus slow down the flow.

For lower surface tension bubbles, initially the bubbles merge into larger bubbles, but as they are deformed they also start to

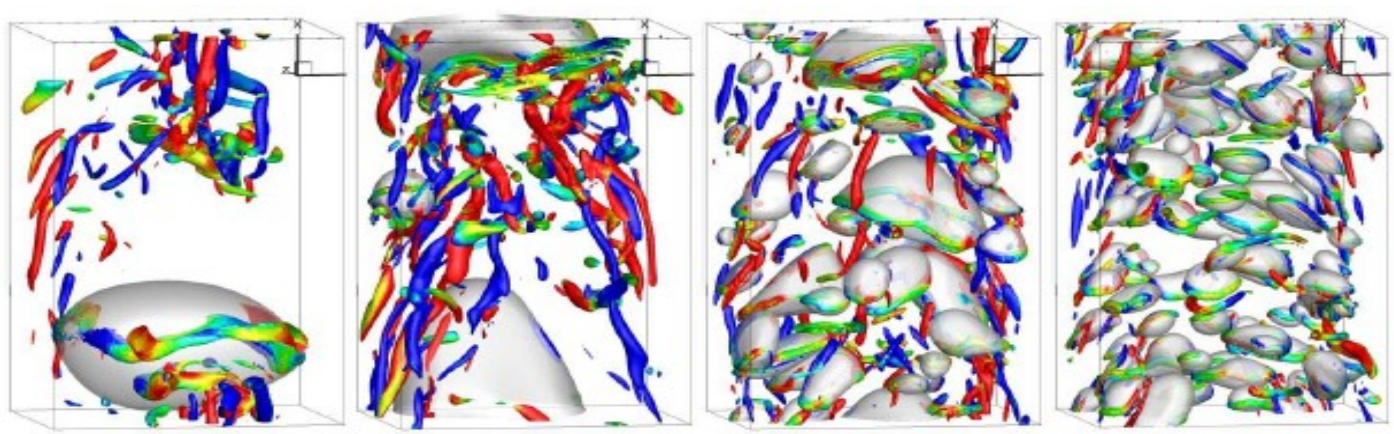

Figure 2 Bubbles with vertical structures observed late in the simulation for Figure 1 cases (a), (b), (c) and (d).

break up. The more deformable, low surface tension bubbles are not pushed to the wall. Because the lower surface tension bubbles are more deformable, the larger coalesced bubbles create less of a flow blockage than high tension bubbles.

For intermediate surface tension, the behavior is significantly more complex and includes the formation of long gas filaments. The study collected a wide variety of calculated parameters, including average void fraction, average vertical velocity, streaming stresses, lateral gas flux, volumetric flow rates, average wall shear stress, total interface area, projected surface area, etc, for various points in time in the simulation.

An example of the vorticity field is shown in Figure 2 at a time point late in the simulation, after the bubbles have coalesced (and broken up, in the lower surface tension cases); red and blue indicate vortices aligned with the flow, with rotation of the opposite sign, and green/yellowish vortices are perpendicular to the flow. While the initial velocity field of the simulation was turbulent, the bubble motion appears to quickly change the structure of the turbulence in major ways.

Table 1 Modeling Parameters for the PWR Subchannel Cases

Case	RE01	RE02
Domain sizes (mm)	40.5x12.6x12.6	
Rod radius (mm)	4.57	
Reynolds number resolved (Re_h)	29,079	80,774
Bulk resolution (mm)	8.11×10^{-2}	3.25×10^{-2}
Thickness of first B. L. (y^+=1) (mm)	8.11×10^{-3}	3.25×10^{-3}
Number of boundary Layers	13	13
Number of points	9,249,506	186,825,949
Number of elements	53,837,248	1,111,168,768
Number of computing cores used	8,192	131,072
Element per core	6,572	8,478

Case	RE01	RE02	Realistic PWR condition
Liquid/Gas Viscosities (Pa·s)	8.585×10^{-5}; 1.965×10^{-5}		
Liquid/Gas Densities (kg/m^3)	712.22; 46.17		
Mean velocity (m/s)	0.27	0.75	4.62
Reynolds number (Re_h)	29,079	80,774	452,500

Next, the research team created a single PWR subchannel domain for simulation. Both single-phase and two-phase turbulence were simulated for Reynolds numbers (Re) of 29,079 (53.8 million cell mesh, RE01) and 80,774 (1.11 billion cell mesh, RE02). The two cases were compared to investigate the influence of PWR geometry on the turbulent flow structures. Since the mesh size requirement for DNS decreases exponentially as Reynolds number increases, 80,774 was chosen to approach realistic PWR conditions while managing the computational resources required. Table 1 provides a listing of the model parameters for both cases.

Periodic boundary conditions are utilized to represent a much longer domain than is computationally feasible in the DNS with ITM approach. A single-phase turbulent velocity profile is first generated by placing a sphere blockage region at the domain center to create fluctuations. When large turbulence structures are observed the spherical barrier is removed. When the single phase turbulence achieved statistically steady state flow conditions, bubbles were introduced and the bubble motion and deformation were resolved using level-set interface tracking method. For this study, considering both computational cost and results reliability, 17 bubbles were used for the RE01 case and 262 bubbles for RE02. The bubble distribution and turbulence for RE02's 262 bubbles are shown in Figure 3 (the direction of mean flow is from left to right).

Both single- and two-phase subchannel simulations were performed at the Leadership Computing Facility (ALCF) located at the Argonne National Laboratory. The simulation results were visualized using the open-source software, ParaView. The void fraction and gas-liquid velocity profile from the two phase RE01 case are shown in Figure 4. In the region where the void fraction is higher than 0, the corresponding gas velocity is observed to be larger than liquid velocity because the bubbles are accelerated by the buoyancy force in the subchannel. When two-phase flows achieve statistically steady state conditions, the drag coefficient can be estimated based on the bubbly buoyancy force and bubble terminal velocity. Assuming steady state conditions and approximating the bubble relative velocity obtains a drag coefficient is close to the expected value.

In future CASL plans to apply a functional form of correlations distilled from these studies to be used in CFD and subchannel flow modeling. For more information, see L3:THM.CLS.P11.01. It is notable that this work received the International Data Corporation (IDC) HPC Innovation Excellence Award (November 2014).

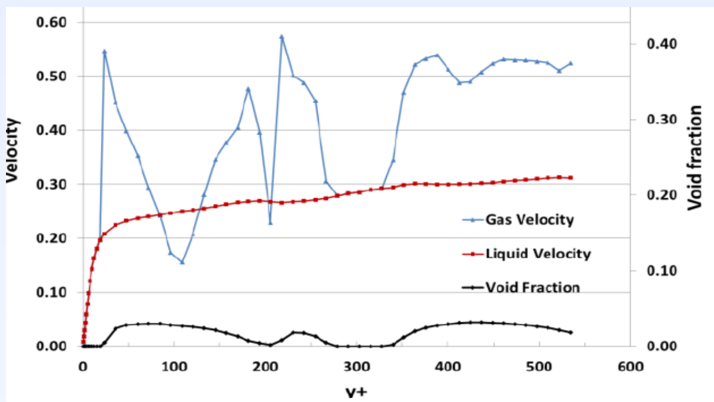

Figure 3 Void fraction and gas-liquid velocity profile from two-phase conditions, RE01

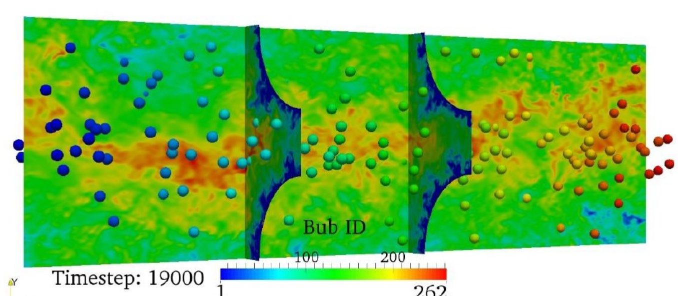

Figure 4 Distribution of 262 bubbles in the turbulent flow

References cited in this article:

[1] D. Lakehal, M. Meier, M. Fulgosi. Interface tracking towards the direct simulation of heat and mass transfer in multiphase flows. International Journal of Heat and Fluid Flow 23 (2002) 242–257.

[2] Lu, J., & Tryggvason, G. "Effect of bubble deformability in turbulent bubbly upflow in a vertical channel," Physics of Fluids (1994-Present), 20(4), pp. 040701 (2008).

[3] J. Fang, M. Rasquin, I.A. Bolotnov. "Interface Tracking Simulations of bubbly flows in the PWR relevant geometries," The 16th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-16), Hyatt Regency Chicago, Chicago, IL, USA, August 30-September 4, 2015.