Validation of multiphase flow modeling in ANSYS CFD

Th. Frank[(1)], C. Lifante[(1)], A.D. Burns[(2)]

[(1)]ANSYS Germany GmbH, Staudenfeldweg 12, D-83624 Otterfing, Germany
Phone: +49 (8024) 9054 76, Fax: +49 (8024) 9054 17
Thomas.Frank@ansys.com

[(2)] School of Process Material and Environmental Engineering, CFD Centre,
University of Leeds, LS2 9JT, UK

Abstract

Many flow regimes in applications of nuclear reactor thermal hydraulics and engineering as well as in Nuclear Reactor Safety (NRS) Research are characterized by multiphase flows, where one of the phases is continuous and the other phase(s) consist(s) of gas or vapour of the liquid phase with varying flow morphologies ranging from disperse bubbly flows, churn turbulent flow, slug flow, annular flow to droplet flow. By today CFD for multiphase flows with varying flow morphologies is still a topic of ongoing research, where the challenges are in the characterization of different flow regimes from local flow properties, the corresponding modeling of model closure correlations for mass, momentum and heat transfer between phases and in the higher computational demands of resulting model formulations. Nevertheless reliable and accurate multiphase flow models have been developed for a broader range of multiphase flow applications in nuclear reactor engineering and NRS applications.

The presentation will focus on a limited number of applications of Eulerian multiphase flow modeling capabilities in ANSYS CFX with a focus on nuclear reactor thermal hydraulics and directions of ongoing model development effort. Furthermore ANSYS is undertaking permanent effort in CFD model validation and takes part in national and international research collaborations. Multiphase flow model application and assessment of model reliability and accuracy will be shown and discussed on a number of selected test cases. Model validation and comparison to experimental data will be shown for a testcase of strong steam condensation in subcooled water in the vertical DN200 pipe of the FZD TOPFLOW test facility applying newly developed model extension for the inhomogeneous MUSIG model approach for polydisperse bubbly flow simulation. Furthermore validation results are shown for the Lee at al. wall boiling experiment under nearly atmospheric pressure conditions as published on the ICONE-16 conference applying the RPI wall boiling model with some submodel modifications as available in ANSYS CFX 12.0.

Finally the presentation will give an outlook on future directions of CFX multiphase flow model development. Further investigations are aimed to the improvement of simulation accuracy for steam-water flow regimes, two-phase turbulence modeling and the validation of two-phase flow models for flow conditions with high temperature and pressure including condensation and evaporation processes, e.g. flows through fuel assemblies of PWR's including wall boiling.