Seventh International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
9-11 December 2009

RECENT ADVANCES IN ANSYS CFD MULTIPHASE FLOW MODEL DEVELOPMENT, VALIDATION AND APPLICATION

Th. FRANK

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

ABSTRACT

Many flow regimes in engineering applications, chemical and process technology, power generation and 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 disperse particles, bubbles or droplets. In gas-liquid flows the disperse phase consists of either inert fluid in bubbles/liquid droplets or the multiphase flows become further complicated by strong heat and mass transfer between the phases (boiling, condensation, either in the bulk flow or on solid walls), chemical reactions and often by varying flow regimes and flow morphology, e.g. ranging from disperse bubbly flows, churn turbulent flow, slug flow, annular flow to mist/droplet flow. By today the CFD simulation 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 the named fields of engineering applications.

The presentation will focus on recent advances in the development of multiphase flow models in the ANSYS CFD software packages ANSYS CFX and ANSYS Fluent. An overview about the established multiphase flow model hierarchy in both software packages will be given and common grounds, differences and special focus of the two software packages will be highlighted. Recent advances and application of multiphase flow modeling will be discussed by focusing on a limited number of applications of Eulerian and Lagrangian multiphase flow modeling capabilities in ANSYS CFX and ANSYS Fluent. 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 and engineering applications. Discussed MPF applications include:

- Modeling and simulation of polydisperse bubbly flows using a discrete bubble size class model with inhomogeneous velocity field assumption regarding bubbles of different size. Investigated testcases include 3d model validation against experimental data for adiabatic air-water flows and 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.
- Gas-liquid wall-heated flows with nucleate subcooled boiling. 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 and including conjugate heat transfer prediction as available in ANSYS CFX 12.0.
- Investigation of free surface flows with homogeneous/inhomogenous VOF and surface tracking techniques. Shown applications include simplified validation testcases as well as technical applications like the prediction of slug flow in long pipelines and flow through slug catchers in the oil & gas industry.
- Lagrangian particle tracking methods. The presentation will focus on spray simulation with emphasis on flow simulation for fuel injection process into internal combustion engines (ICE) including in-nozzle cavitation, primary and secondary spray breakup and droplet-wall interaction modeling. Further application of a newly developed model (DDPM – Dense Discrete Particle Model in ANSYS Fluent) focuses on the prediction of dense gas-particle flows with particle concentrations up to the maximum packing limit, e.g. fluidized beds.

Finally the presentation will give an outlook on future directions of ANSYS CFD multiphase flow model development. Further investigations are aimed to the improvement of convergence robustness and simulation accuracy for steam-water flow regimes, two-phase turbulence modeling, validation of two-phase flow models for flow conditions with high temperature and pressure including condensation and evaporation processes, the further improvement of free surface flow modeling and the Dense Discrete Particle Model (DDPM) in application to fluidized beds and other dense gas-particle flows.