Validation of an OpenFOAM gas distribution solver for containment analysis

Jörn Stewering, Berthold Schramm
Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH

In the case of a severe accident in a nuclear power plant hydrogen could be produced in the superheated reactor core. The hydrogen could leak into the containment and a hydrogen deflagration could happen. An example for such an event was the Fukushima accident in 2011. For the analysis of such accidents different phenomena in the reactor containment have to be covered such as the distribution of steam and hydrogen, steam condensation, thermal radiation, conjugated heat transfer, hydrogen recombination and deflagration.

Computational Fluid Dynamics (CFD) application for such analysis is becoming more relevant. CFD is a method for the numerical solution of flow problems and OpenFOAM is an open-source CFD software package. The models needed to simulate the different containment phenomena with OpenFOAM are currently being tested and validated or improved and further developed by a network of different German research institutes and universities.

Current aims of the work at GRS are the implementation of necessary models in OpenFOAM (e.g. for diffusion, buoyancy turbulence, wall and bulk condensation, passive autocatalytic recombiner (PAR)), the validation of all models with test cases and different experiments and the comparison to experimental data and commercial CFD tools (e.g. ANSYS CFX). The validation cases include the participation in blind benchmarks and open post-test analyses.

This paper discusses three different validation cases. One experiment for gas distribution (Panda H2P1_10 / Erosion of a helium-rich layer by a vertical jet), one experiment for steam condensation (THAI TH-24 / Natural convection and wall condensation) and one experiment for PAR performance (THAI HR-43 / PAR in counter current flow). The simulation results are compared to the experimental data. In general, a good agreement to experimental data was observed.