Nuclear Engineering and Design 328 (2018) 359–371

Contents lists available at ScienceDirect

Nuclear Engineering and Design

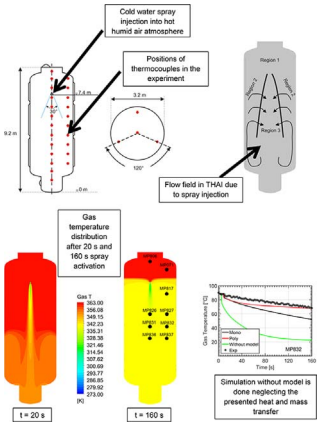
journal homepage: www.elsevier.com/locate/nucengdes

CFD simulation of spray cooling in the model containment THAI

C. Kaltenbach*, E. Laurien

University of Stuttgart, Institute of Nuclear Energy and Energy Systems (IKE), Pfaffenwaldring 31, 70569 Stuttgart, Germany

GRAPHICAL ABSTRACT



ARTICLE INFO

Keywords:
Spray
Droplets
THAI
Two-phase flow
Spray cooling

ABSTRACT

CFD (Computational Fluid Dynamics) simulation of spray is a challenging task in the field of nuclear reactor safety. In the current publication a CFD model for spray cooling is presented, which is able to predict heat and mass transfer between cold droplets and a hot humid air gas atmosphere. The model, which is implemented via user defined functions in the commercial CFD code ANSYS CFX 16.1, enables the simulation of spray cooling physics with an Euler-Euler two-fluid approach. A comparison of simulations with mono- and polydisperse spray configurations shows the relevance to consider droplet size distributions within spray. For validation, the pressure and temperature transients of the experiment THAI HD-31-SE (Thermal-hydraulics, Hydrogen, Aerosols and Iodine) are used. During the experiment, a cold spray is injected into a hot humid air gas atmosphere, which leads to a cooling effect inside of the model containment. A full three dimensional geometrical mesh of THAI is used for all simulations. Simulation results indicate a good agreement with experimental data for the polydisperse spray configuration.

1. Introduction

In case of a leak in the primary circuit due to a severe accident in a PWR (Pressurized Water Reactor) containment, hot water vapor is released into the containment atmosphere and leads to a pressure and temperature rise. A spray nozzle system in the upper part of the containment is an effective method to reduce the stress on containment walls and prevent its damage. For example, such spray systems contributes to the integrity of the EPR containment (European Pressurized Water Reactor), see Kessler et al. (2014). When cold spray is injected,

* Corresponding author.
E-mail address: christian.kaltenbach@ike.uni-stuttgart.de (C. Kaltenbach).

https://doi.org/10.1016/j.nucengdes.2017.12.030
Received 15 August 2017; Received in revised form 7 December 2017; Accepted 19 December 2017

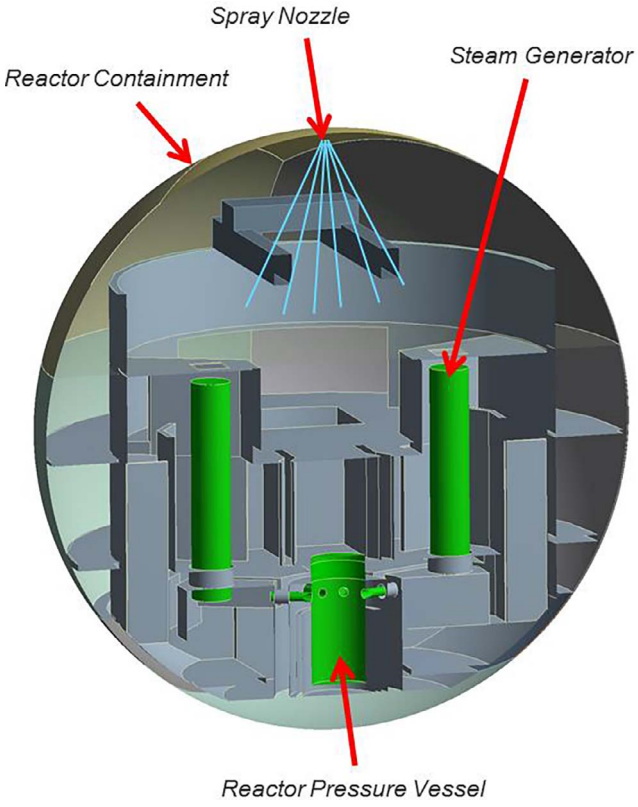


Fig. 1. Model containment with schematically presented spray system.

water vapor condenses on cold droplets. A convective heat flux to the cold droplet surface consumes energy and cools down the heated containment atmosphere. Fig. 1 shows schematically a PWR spray system. This is a simplified illustration, indeed a various number of spray nozzles are distributed in rings in the upper containment part.

Considering the large volume of a PWR containment, thermo-hydraulic phenomena are experimentally investigated in a scientific scope in model containments like TOSQAN (Porcheron et al., 2007), MISTRA (Studer et al., 2007), (OECD NEA, 2012), THAI (Gupta et al., 2014) and PANDA (OECD NEA, 2012). Experimental work is done to produce data for validation of numerical codes. Lumped-Parameter (LP) codes and Computational Fluid Dynamics (CFD) methods are numerical approaches, which are similarly accepted in nuclear research. Compared to LP codes, which are used to simulate long and complete accident scenarios, CFD methods resolve additionally momentum equations and therefore the flow field, see Houkema et al. (2008). Commercial CFD codes normally do not have models for heat and mass transfer implemented, so models have to be developed and must be introduced by user functions into the codes. In the following, a brief overview will be given on work which was done concerning spray droplet flows with heat and mass transfer and CFD methods.

Babić et al. (2008) modeled wall and volume condensation with a single phase approach in CFD. Droplets and gas are summarized as a single homogenous fluid and share the same velocity field. The heat and mass transfer between droplets and gas is considered via heat and mass transfer correlations and not on basic physical laws. A single phase approach is not appropriate for spray applications due to the assumption of a negligible small Stokes number and the assumption of one velocity field. This is confirmed by Ding et al. (2017). They compared spray simulations with single phase and two-fluid approaches. In order to take into account different velocity fields for gas and droplets in a two-fluid description, Babić et al. (2009) numerically investigate spray injection with heat and mass transfer into a hot gas atmosphere in TOSQAN. For their CFD simulations, they used a two-fluid approach with a Lagrangian description for droplets and an Eulerian for the gas phase. Despite an insufficient resolution and the assumption of a rotationally symmetric mesh of TOSQAN (2D), they got an acceptable agreement with experimental data. Filippov et al. (2016) used an Euler-Lagrange two-fluid approach with user implemented models for evaporation and condensation of droplets to investigate the influence of spray on gas stratification and gas mixing behavior in PANDA PE1 and PE2. Droplets are assumed spherical with a constant diameter of 550 μm for a full cone nozzle in PE1 respectively 260 μm for a hollow cone nozzle in PE2. An Euler-Euler approach was used for modeling of volume condensation by Zhang and Laurien (2014) in THAI. Gas was characterized continuous and droplets are described continuous disperse with a small monosized droplet diameter of 150 μm. In this work, the developed condensation model was combined with wall condensation modeled by ANSYS (Zschaeck et al., 2014). Condensation and evaporation of spray in TOSQAN was numerically investigated by Mimouni et al. (2010). An Euler-Euler two-fluid approach was used with a monodisperse spray configuration and droplet diameters up to 200 μm. Malet and Huang (2015) compare a monodisperse with a polydisperse spray configuration and with experimental data gained in TOSQAN. They note a better agreement with the experiment for polydisperse sprays and point out the importance of modeling a spray droplet distribution.

To enable the simulation of spray cooling in the commercial CFD code ANSYS CFX 16.1, a heat and mass transfer model for spray was developed and implemented in CFX via user defined functions. The basic Euler-Euler two-fluid approach is briefly introduced and a detailed phenomenological and mathematical description for the spray cooling model is done. The phenomenological discussion on spray cooling is focused on different gas-droplet temperature configurations and the balance of water vapor on the droplet surface and in the bulk flow. Simulations of the experiment THAI HD-31-SE are carried out with the state of the art monodisperse spray approach and also with a polydisperse configuration. In the polydisperse case, different velocity fields for different droplet sizes are applied. Droplet diameters vary from 139 μm to 1243 μm. To point out the necessity of modeling spray cooling, a simulation only using convective heat transfer between droplets and gas is compared to mono- and polydisperse simulations applying additionally the developed model for spray cooling. In this simulation the heat amount due to mass transfer is neglected, only the convective heat transfer produces the pressure and temperature decrease. Due to the geometrical and physical complexity, all simulations are carried out on the supercomputer CRAY XC40 (Hazel Hen) at the High Performance Computing Center Stuttgart (HLRS). An overview over the computational resources rounds off the results.

2. Numerical approach

2.1. Euler-Euler two-fluid model

The present model and simulations are based on the Euler-Euler two-fluid approach, which was introduced by Ishii and Mishima (1984) and Ishii and Hibiki (2006). Every physical condition in this approach is described by one single phase (e.g. liquid or gas). In an Euler-Euler multiphase flow, all participating phases are continuous and are interpenetrating each other all over the flow domain. Each phase is determined with the volume fraction α_k. Index k represents the phase and can stand for liquid (L) or gas (G). The continuous gas phase in our simulations is a mixture of a noncondensable gas (air) and water vapor. Air is considered as a mixture of 79% nitrogen and 21% oxygen. Droplets consist of pure water and are described continuous. Interactions between phases (mass-, momentum- and energy transfer etc.) can be taken into account with source terms in the conservation equations. All of the presented equations assume droplets in spherical shape.

Below, the conservation equations are presented. The mass conservation equation is described by