Proceedings of the 7th GACM Colloquium on Computational Mechanics
for Young Scientists from Academia and Industry
October 11-13, 2017 in Stuttgart, Germany

Numerical Simulation of Flow with Volume Condensation during Accident in Containment of Nuclear Power Plant

Jing Zhang[1]*

Micro Abstract
One severe accident scenario is a leak in the primary circuit of a Pressurized Water Reactor (PWR), resulting in hydrogen and steam injection into the containment. Because of the influence of steam condensation on the gas mixing, the wall and volume condensation phenomena are of interest for the safety considerations. This presentation shows the simulation results of two-phase flow using the developed volume condensation model in the containment. Simulations were performed on the HPC systems.

[1] High Performance Computing Center Stuttgart, Stuttgart, Germany
* **Corresponding author**: jing.zhang@hlrs.de

Introduction

During a severe accident in a PWR, e.g. Loss of Coolant Accident (LOCA), significant amounts of hydrogen can be produced by a chemical reaction between steam and the zircaloy cladding ($Zr + 2H_2O \Longrightarrow ZrO_2 + 2H_2$). Through a break of surge line, steam and hydrogen can be released from the reactor coolant system in the primary loop into the containment, see Figure 1. Condensation can mostly occur on the walls inside the containment. If the steam partial pressure within the mixture is locally higher than the steam saturation pressure, the condensation could also happen within its volume, referred to as 'volume condensation'. Because steam condensation influences the hydrogen stratification, the change of temperature and pressure, as well as the flow condition in the containment, the condensation phenomena are of interest for the safety considerations. The wall condensation model is available in the CFD code ANSYS CFX. The aim of the present work is to develop and validate a phase exchange model for volume condensation in the presence of more than one non-condensable gas for a two-phase flow using ANSYS CFX. In order to validate the volume condensation model, a 3D computational grid is used, covering one half of an experiment vessel. In order to predict the local hydrogen behavior and temperature distribution within a real containment during a severe accident, a model containment is used, which was developed based on a German PWR.

1 Volume Condensation Model

In the present work the volume condensation model is developed for a two-phase flow in presence of more than one non-condensable gas, e.g. air and a light gas. In order to understand the volume condensation process between small droplets and the surrounding gas phase, the interactions are described between a single droplet and the steam-air-light gas mixture, as shown in Figure 2.
The liquid droplet is assumed to be spherical with a diameter d. The liquid droplet is in thermodynamic equilibrium at its surface and has a uniform temperature $\overline{T}^L$. The gas phase at the edge of the thermal boundary layer around the droplet has a temperature $\overline{T}^G$. The partial pressure of steam directly at the interface must be equal to the saturation pressure $p_{H_2O,sat}(\overline{T}^L)$ corresponding to the droplet temperature [3]. The molar fraction at the interface $y^G_{H_2O,sat}$ is assumed to be in equilibrium and is equal to the

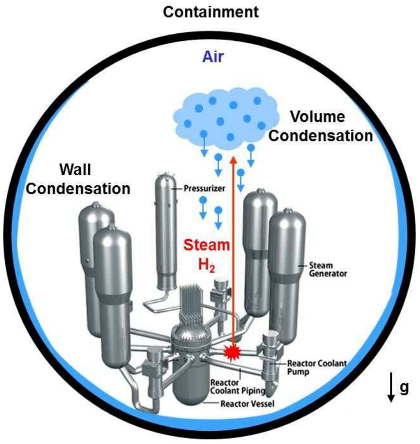

Figure 1. Condensation inside containment

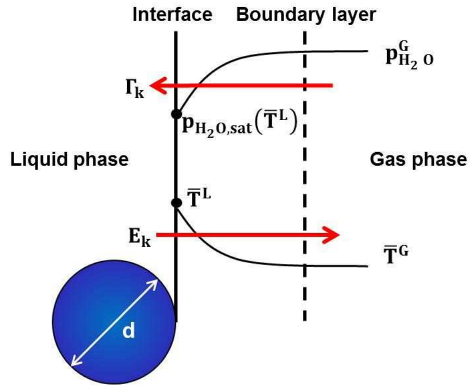

Figure 2. Pressure and temperature profiles for a condensing droplet

ratio of the saturation pressure to the static pressure p. The mass fraction at the interface $c^G_{H_2O,sat}$ is defined as follows:

$$c^G_{H_2O,sat} = y^G_{H_2O,sat}\frac{M_{H_2O}}{M_{G,sat}} = \frac{p_{H_2O,sat}(\overline{T}^L)}{p}\frac{M_{H_2O}}{M_{G,sat}}, \tag{1}$$

in which M_{H_2O} and $M_{G,sat}$ are the molar mass of water and of the saturated gas mixture. The gradient of concentration $(c^G_{H_2O} - c^G_{H_2O,sat})$ leads to mass transport across the diffusion boundary layer. If the steam mass fraction in the gas phase is higher than the steam saturation concentration, the volume condensation rate is calculated from the following simple expression:

$$\Gamma_G = \rho_G \beta A(c^G_{H_2O} - c^G_{H_2O,sat}) = -\Gamma_L, \tag{2}$$

where ρ_G is the density of gas mixture. The mass transfer coefficient β due to the flow around a sphere is given by the correlation of Ranz-Marshall [4] with Reynolds number Re and Schmidt number Sc.

$$\beta = \frac{D_{H_2O,G}}{d}(2 + 0.6Re^{\frac{1}{2}}Sc^{\frac{1}{3}}) \tag{3}$$

$D_{H_2O,G}$ is the diffusion coefficient of multicomponent gas mixture. The interface area density of all droplets is defined as follows:

$$A = \pi n d^2 = \frac{6\alpha_L}{d}. \tag{4}$$

Here n is the droplet number density (number of droplets per unit volume) and α_L is the volume fraction of droplets. During the volume condensation, the heat is extracted from droplets into the surrounding gas mixture. The energy source due to the release of latent heat Δh_{LG} is determined as:

$$E_G = -\Gamma_G \Delta h_{LG}, \quad E_L = 0. \tag{5}$$

2 Results of Validation Experiment

In order to validate the volume condensation model, an experiment was used, which was performed by Becker Technologies [1]. Two transient simulations of the experiment were carried out from 2700 s to 3100 s. The first calculation was a single-fluid simulation only using the wall condensation model, which was developed by ANSYS [5]. At the domain interface on the fluid side the wall condensation model was activated, in which condensation is modeled as a sink of mass and energy, but the liquid film is not modeled. The second calculation was a two-phase simulation including the wall and volume condensation model. In this calculation droplets are present in the vessel with a constant droplet diameter of 100 μm and an initial droplet volume fraction of 10^{-5}.
The right side of Figure 3 shows the temperature distribution, which was simulated with the wall and

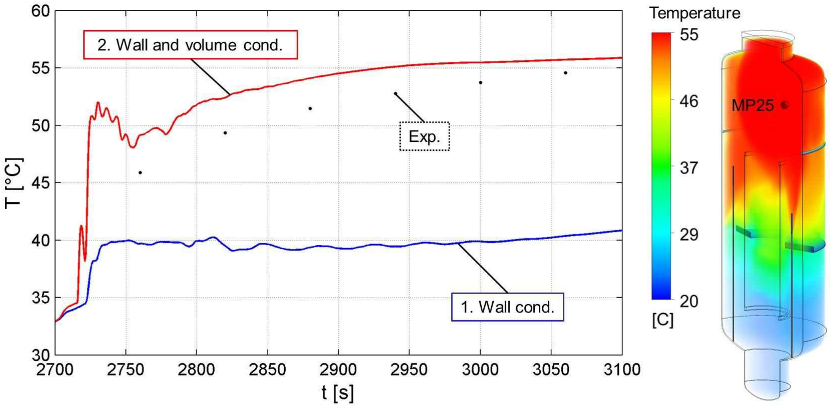

Figure 3. Left: comparison of calculated temperature with validation experiment at monitor point MP25; right: temperature distribution of simulation with wall and volume condensation at 3100 s

volume condensation model, at 3100 s. As shown in Figure 3 left, two calculated temperatures are compared with the experiment at the monitor point MP25. The hot steam injection leads to an increase of temperature. At the beginning, the fast heating of atmosphere by contact with hot steam contributes to the rapid rise in the temperature. The simulated temperature only with the wall condensation model is consistently underpredicted compared to the experiment and shows a poor agreement. The calculated temperature with the wall and volume condensation model agrees well with the experiment and is higher than only with the wall condensation model. Because the volume condensation happens directly in the upper region of the vessel, the released heat leads to a rapid temperature increase.

3 Results of Model Containment

In order to predict the local hydrogen behavior within a real containment during a severe accident, a PWR containment similar to the German design was used. A 3D geometry and its mesh were built including main rooms, free volumes and main metallic structures. A break is ca. 60 cm^2 and located at the lower part of steam generator tower as a horizontal jet. In order to reduce the computational effort, a single-phase simulation was carried out only with the wall condensation model from 448 s to 4110 s after the time of accident. The release profiles of hydrogen and steam were selected from [2].

The left side of Figure 4 shows the way of containment flow and the hydrogen distribution in the containment at 3470 s during the strong hydrogen injection. The horizontal flow released as a jet through a break into the lower part of steam generator tower. It travels upward as a plume driven by buoyancy. Only one burst disk, which is located in the steam generator tower on the injection side, is opened in the case of overpressure. Through the ruptured burst disk the flow moved towards the dome. After 3470 s the mass flow rate of hydrogen decreased strongly. As shown in Figure 4 right, at the end of the simulation (i.e. 4110 s), steam condensation on the cold walls locally increases the gas density and starts

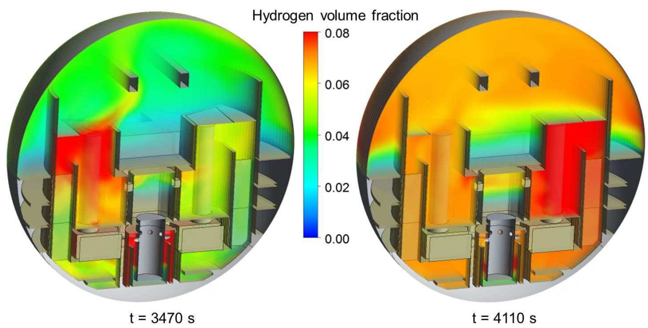

Figure 4. Hydrogen distribution in containment at 3470 s (left) and 4110 s (right)

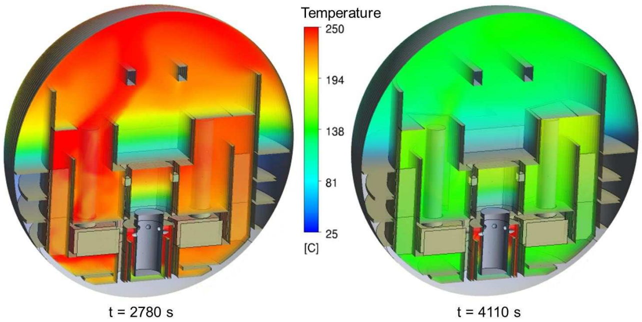

Figure 5. Temperature distribution in containment at 2780 s (left) and 4110 s (right)

a secondary downward-directed flow, and brings the hydrogen from the stratified region down into the lower part of containment. Because of the light gas of hydrogen, the hydrogen accumulates in the upper part of different rooms.
The released temperature profile has a strong peak at 2780 s. The left hand side of Figure 5 shows the temperature distribution in the containment at that time. The released hot gas flow of steam and hydrogen form a buoyant jet rising through the ruptured burst disk from one steam generator tower into the dome. The hot gas flow leads to a temperature increase inside the containment. After 2780 s the temperature of injected gas decreases consistently. The right side of Figure 5 shows a strong effect on cold walls and temperature decrease at the end of the calculation (i.e. 4110 s).

Conclusions

The physical formulation of newly developed volume condensation model was presented, which simulates the dispersed droplets and a continuous gas mixture in the presence of non-condensable gases based on the two-fluid model. Then, the volume condensation model was validated with an experiment. The two-phase simulation results using the wall and volume condensation model were compared with the single-fluid simulation solely using wall condensation model, as well as with the experiment from 2700 s to 3100 s. The transient temperature profile with the usage of wall and volume condensation model have shown good agreement with the experiment. Finally, a transient prediction was performed about the containment flow in a German PWR containment under the hypothetical, severe accident condition. To reduce the computational effort, a single-fluid simulation was carried out only using the wall condensation model from 448 s to 4110 s. The simulation has evaluated the time-dependent hydrogen and temperature distributions inside the containment.

References

[1] T. Kanzleiter and et al. *Final Report: Experimental Facility and Program for the Investigation of Open Questions on Fission Product Behaviour in the Containment, ThAI Phase II. Report No. 1501272*. Eschborn, Germany, 2007.

[2] S. Kelm, P. Broxtermann, S. Krajewski, and H.-J. Allelein. *Proposal for the Generic Containment Code-to-Code Comparison - run2. Internal Research Report*. Jülich, Germany, 2012.

[3] N. I. Kolev. *Multiphase Flow Dynamics 2: Thermal and Mechanical Interactions*. Multiphase Flow Dynamics. Springer Berlin Heidelberg, 2007.

[4] W. E. Ranz and W. R. Marshall. Evaporation from Drops Part I. *Chemical Engineering Progress*, Vol. 48(3):p. 141–146, 1952.

[5] G. Zschaeck, T. Frank, and A. D. Burns. CFD modelling and validation of wall condensation in the presence of non-condensable gases. *Nuclear Engineering and Design*, Vol. 279:p. 137–146, 2014.