CFD Simulation of Subcooled Boiling Flow in Nuclear Fuel Bundle

W. K. In*, C. H. Shin* and C. Y. Lee*
Corresponding author: wkin@kaeri.re.kr

* Korea Atomic Energy Research Institute, Rep. of Korea

Abstract: A Computational Fluid Dynamics (CFD) analysis is performed to simulate the subcooled boiling flow in fuel bundles for Pressurized Water Reactor (PWR) and Boiling Water Reactor (BWR). The CFD simulation predicted the steady-state void distribution in the subchannels of the PWR and BWR fuel bundles. The CFD prediction shows a higher void fraction near the heated wall and migration of void in the subchannel gap region. The CFD prediction of void fraction for the PWR subchannel agrees with the measured ones within 10% for the low inlet subcooling. The CFD simulation for the BWR fuel bundle reproduced overall radial void distribution trend which shows less vapor in the central part of the bundle and more vapor in the periphery. However, the comparison of detailed subchannel void distribution shows a somewhat large discrepancy between the CFD and experimental results.

Keywords: Subcooled Boiling, Computational Fluid Dynamics, Fuel Bundle, Void Fraction, Subchannel.

1 Introduction

A subcooled boiling flow in nuclear fuel bundle is an important phenomenon in a nuclear reactor system for the safe and reliable operation. The nuclear fuel bundles for PWR and BWR consist of rod bundles with the coolant flowing axially through the subchannels formed between the fuel rods. OECD/NEA and US NRC organized the international benchmark programs [1, 2] for PWR Subchannel and Bundle Tests (PSBT) and BWR Full-size Fine-mesh Bundle Tests (BFBT). The void distribution benchmark provided measured void fraction data over a wide range of operating conditions in single subchannel and fuel bundle. This CFD study simulated the boiling flows in four different types of the single subchannel for the PSBT benchmark and in the BWR 8x8 fuel-rod bundle. The CFD predictions of the void distribution are compared with the PSBT and BFBT measurements.

2 Computational Multiphase Flow Model

The multiphase flow model used in this CFD analysis is the two-fluid model in which liquid(water) and vapor(steam) are considered as continuous and dispersed fluids, respectively. The two-fluid model uses the interfacial area per unit volume between the phases to model interfacial transfer of momentum, heat and mass. The interfacial momentum transfer rates included in this CFD simulation are drag force, lift force, wall lubrication force and turbulent dispersion force. Heat transfer across a phase boundary is predicted using an inter-phase heat transfer coefficient and an interfacial area. The inter-phase mass transfer is calculated depending on the liquid temperature, i.e., a bulk condensation or evaporation. A wall boiling model is also employed to simulate the bubble generation on a heated wall surface. The wall heat is assumed to be partitioned into three parts, i.e., convective, quenching and evaporative heat transfers.

3 Results and Discussion

The twenty-six test cases for the PSBT benchmark were simulated and their CFD results are compared with the measured ones. Figure 1 compares the CFD prediction of void distribution in the typical subchannel with the CT image. The predicted void contours show less vapour in the core region and high vapour in the gap region and the near-wall region, which agrees well with the CT measurements. The CFD predictions agree with the experimental data within 10% for the void fraction. The four test cases for the BFBT benchmark were analyzed to predict the void distribution in the 8x8 fuel bundle. The void distributions at the exit of the test bundle are compared in Fig. 2. The CFD simulation shows a reasonable radial void distribution trend predicting less vapor in the central region of the bundle and more vapor in the periphery.

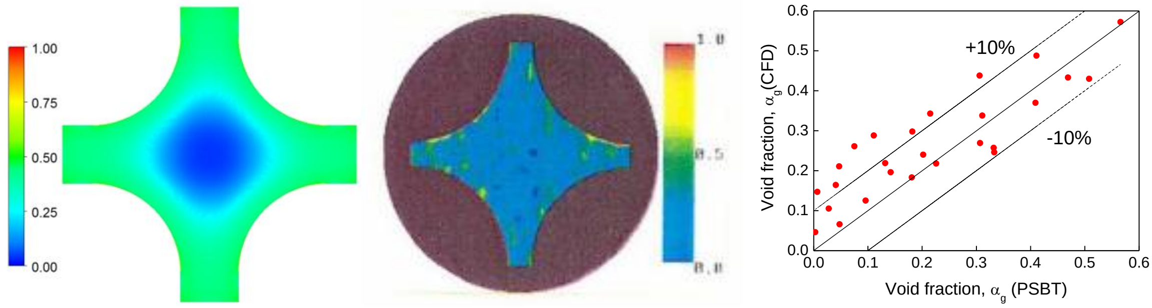

Figure 1: Void fraction comparison in the PSBT single subchannel.

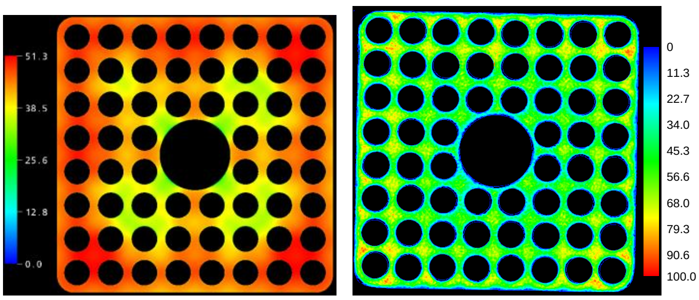

Figure 2: Void distribution in the 8x8 fuel bundle for the BFBT benchmark

4 Conclusion and Future Work

A CFD analysis is performed to simulate the subcooled boiling flow in the subchannels of PWR and BWR fuel bundles. The predicted void distribution in the subchannel shows less vapour in the core region and high vapour in the gap region and the near-wall region, which agrees well with the measurements. The CFD predictions for the single subchannel benchmark agree with the experimental data within 10% for the void fraction. The subchannel void distribution obtained from the CFD prediction appeared to be lower than the measured one and shows a small variation as the bundle-exit quality increases. A mechanitic model for bubble size and multiphase turbulence should be developed to improve the multiphase CFD accuracy in the future.

References

[1] A. Rubin, et al. *OECD/NRC benchmark based on NUPEC PWR subchannel and bundle tests (PSBT) – Vol. I*. NEA/NSC/Doc(2010). 2010.

[2] B. Neykov, et al. *NUPEC BWR full-size fine-mesh bundle test benchmark (PSBT) – Vol. I.* NEA/NSC/Doc(2005). 2005.