CFD modeling for reactor coolant system natural circulation applied with a tube simplification technique

Dae Kyung Choi [a], Kukhee Lim [b], Yong Jin Cho [b], Choengryul Choi [a*]
[a]*ELSOLTEC, 1401-2 U-Tower BD., 184, Jungbu-daero, Giheung-gu, Yongin-si, Gyeonggi-do, 17095, Korea*
[b]*Korea Institute of Nuclear Safety, 62 Gwahak-ro, Yuseong-gu, Daejeon, 34142, Korea*
[*]*Corresponding author: crchoi@elsoltec.com*

1. Introduction

The rupturing of steam generator tubes leads to severe accidents in nuclear power plants. It causes radioactive materials to leak into the secondary system and release outside the reactor containment region. When a power loss accident occurs at a nuclear power plant, the reactor coolant pump operation is interrupted. As a result, a loop seal is formed between the rear end of the steam generator and the reactor coolant pump, causing the natural circulation between the reactor and steam generator. This natural circulation may cause an increase in temperature at the steam generator tube, leading to the rupturing of the tube [1]. Therefore, it is essential to develop a technique to evaluate whether the natural circulation within a steam generator can cause temperature-induced steam generator tube rupture. This study provides a preliminary step in developing evaluation methods for natural circulation in steam generators in nuclear reactors. A steam generator tube simplification modeling technique was developed for use in nuclear power, and benchmarking tests were conducted against existing experimental studies.

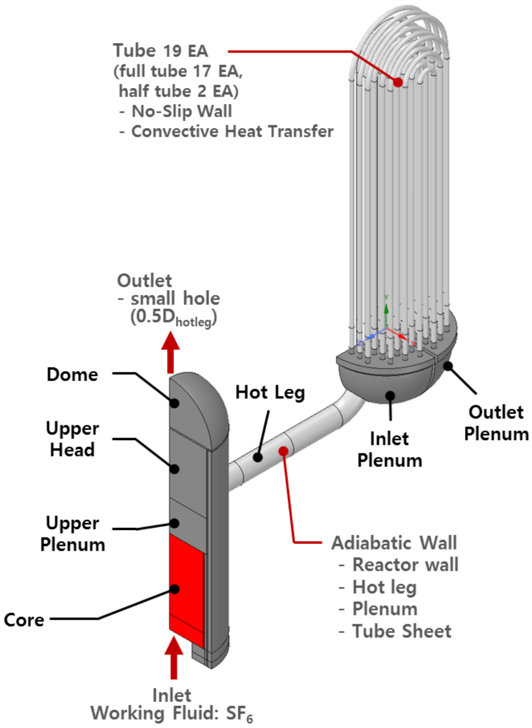


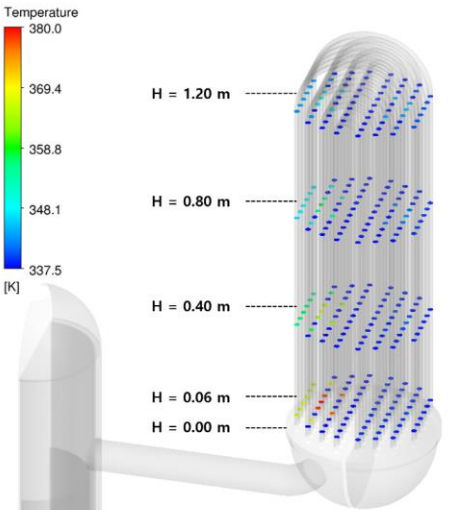
Fig. 1. Analysis domain and boundary conditions for numerical analysis.

2. Numerical Analysis Method

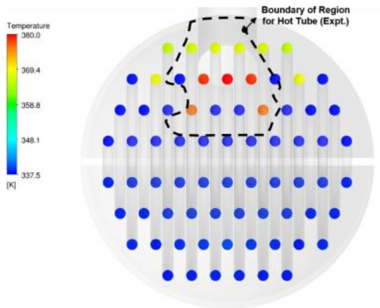
Three-dimensional steady state CFD(Computational Fluid Dynamics) analysis were conducted using ANSYS Fluent(Ver.18) based on the finite volume method. A continuity equation, an energy equation, and a momentum equation were used to analyze the thermal flow in RCS. The three-dimensional compressible flow, turbulent flow, and conjugate heat transfer in the steam generator tube were considered to analyze the natural circulation in the RCS(Reactor Coolant System). Turbulence models were applied to consider the boundary layer properties affected by turbulent flow and turbulent process viscosity. The RSM turbulence model significantly consistent with the experimental results was selected for the CFD model [2].

The CFD analysis methodology used in this study was validated in comparison to previous 1/7th scale experimental studies [3]. CFD analysis requires extensive computing resources for simulating tens of thousands of steam generator tubes to assess the natural circulation in actual nuclear power plants. Thus, numerous steam generator tubes were modeled in this study as a single tube, and the fluid within each tube maintained the same thermal characteristics. In this study, six tubes were modeled as a single equivalent tube (Fig. 1). The simplified tube region was modeled as a porous medium to mimic the pressure drop generated in the original tube. Furthermore, the effective thermal conductivity (k_{eff}) of the working fluid in the steam generator tube was applied to generate similar thermal characteristics concerning the original tube.

Fig. 1 shows the boundary conditions of the CFD analysis in this study. SF_6 was used as the working fluid, with an operating pressure of 300 psia. The flow inlet condition was applied to the lower part of the reactor core, and the temperature and flow rate of SF_6 passing the reactor inlet were adjusted to ensure that they correspond to the experimental parameters (447.5 K and 0.23 kg s^{-1}, respectively). Convective heat transfer condition was used to simulate cooling in the hot fluids through heat exchange as they moved through the steam generator tube and reached the secondary system. The temperature conditions of the secondary system (64.7 °C/337.85 K) in an existing experiment [3] were used, and a convection heat transfer coefficient of 250 W m^{-2} K^{-1} was defined, similar to previous study [4]. The insulation condition was applied to all areas except the tube bundle to prevent heat transfer. In the analysis, the physical properties of SF_6 were treated as a function



(a) Model 2 (ISO view)



(b) Model 2 (H=0.06 m)

Fig. 2. Distribution of temperature on the vertical plane

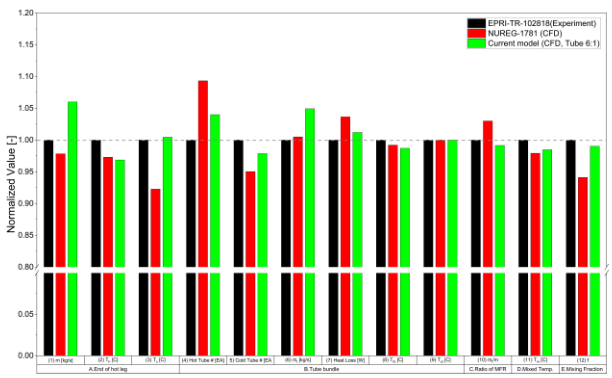


Fig. 3. Comparison of the evaluation factors between previous studies and CFD analysis

of temperature [4]. The simplified tube model for the steam generator was mesh constructed using the same rules as the previous study[2]. A total of 2,805,000 meshes were used in this model.

3. Results and Discussion

The temperature distribution in the horizontal section of the steam generator tube is shown in Fig 2. The high-temperature SF_6 flowed from the inlet plenum to the outlet plenum and cooled in the steam generator tube. CFD analysis showed that the distribution of the high-temperature tube was similar to the experimental result (Fig. 2(b)). A total of 78 and 138 high- and low-temperature tubes, respectively, were evaluated in this study.

A comparison of the CFD results for the evaluation factors with previous studies is presented in Fig. 3. The experimental and CFD results differed by approximately 0.85% and 1.52% for the flow ratio and mixed temperature, respectively. On comparing the experimental and CFD analysis results for the main comparison factors, it was determined that the difference was -7.7~9.3% with the previous study and -3.14~6.01% with the present study. The results of this study were similar to experimental results, which considered all steam generator tubes. Thus, it can be concluded that the CFD modeling methods of this study could be exploited to simulate the RCS in nuclear power plants.

3. Conclusions

As a preliminary step toward developing a technique for evaluating the natural circulation in RCS of nuclear power plants, the CFD model was developed in this study. In addition, this study developed a simplified tube model with a simplification ratio of 6:1. The natural circulation in the reactor cooling system was evaluated, and benchmarking tests were conducted against existing 1/7th scale experimental data. The benchmarking test results indicate that the CFD analysis produced similar results as the experiments.

The thermal flow characteristics of the steam generator can be compromised if an excessive number of tubes are simplified into a single equivalent tube. Consequently, to develop simplified steam generator tube models in subsequent studies, sensitivity analysis should be conducted for the simplified steam generator tube ratio. The results of this study will also serve as a foundation for future CFD analysis of natural circulation in RCS of nuclear power plants.

ACKNOWLEDGEMENT

This work was supported by the Nuclear Safety Research Program through the Korea Foundation of Nuclear Safety, using financial resources by the Nuclear Safety and Security Commission, Republic of Korea (No. 1805001)

REFERENCES

[1] S. Sancaktar et al., Consequential SGTR Analysis for Westinghouse and Combustion Engineering Plants with Thermally Treated Alloy 600 and 690 Steam Generator, Technical Report, NUREG-2195, U.S.NRC., 2018.
[2] Kukhee Lim et al., CFD modeling for validation of the 1/7th scale steam generator inlet plenum mixing experiment, Transactions of the Korean Nuclear Society Spring Meeting, Jeju, Korea, 2020.
[3] W. A. Stewart et al., Natural Circulation Experiments for PWR High-Pressure Accidents (Report no. TR-102815), EPRI, 1993.
[4] C. F. Boyd et al., CFD Analysis of 1/7th Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident, NUREG-1781, Technical Report, U.S.NRC, 2003.