Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 18-19, 2017

Numerical Study of Thermal Hydraulic behavior of Moisture Separator of Steam Generator by CUPID Code

Jae Ryong Lee[a*], Yun Je Cho[a], Yong shin Lee[b] and Han Young Yoon[a]
[a]*Korea Atomic Energy Research Institute,989-111 Daeduk-daero, Daejeon, Korea, 305-353*
[b]*KEPCO Nuclear Fuel,989-242 Daeduk-daero, Daejeon, Korea, 305-353*

[*]*Corresponding author: jrlee@kaeri.re.kr*

1. Introduction

Moisture separator in a pressurized water reactor (PWR) steam generator is utilized to separate steam and liquid and provoke that the dry steam is supplied to the turbine system. It is important to obtain the steam as dry as possible during normal operation and examine how to work in transient scenario. Thus, there are numerous effort to investigate thermal hydraulics of the moisture separator. Due to the complexity of the geometry, experimental researches have been conducted for couple of decades. By using the appropriate physical models from the experiments, it has been attempted to apply to best-estimated thermal hydraulics codes [1]. Nowadays, A Rapid development of computing resources makes it possible to perform a simulation for multi-dimensional thermal hydraulics. Visualization of the two-phase flow behavior inside of the moisture separator is able to provide insights about complicated two-phase swirl flow and corresponding carry-over performance which is defined as a ratio of mass flow throughout the separator to the inlet mass flow.

In this study, the three-dimensional thermal-hydraulic behavior inside the moisture separator is numerically investigated by the CUPID code. At first, real geometry is treated with find grid, and then, the carry-over performance is investigated according the various inlet mixture velocity.

2. Numerical Methodology

2.1 Governing equation

The CUPID code [3] adopts the two-fluid model for two-phase flows. In the two-fluid model, the mass, energy, and momentum equations for liquid and vapor phases are established separately, and then, they are linked by the interfacial mass, momentum, and energy transfer models. For a mathematical closure, the constitutive relations for the interfacial momentum transfer, the interfacial heat transfer and the wall heat partitioning are necessary. Standard k-ε turbulence model is implemented to handle complicated turbulent flow behavior.

2.2 Modeling moisture separator

In this study, the CE (Combustion Engineering) typed moisture separator is taken into account. CE-typed separator is consisted of stationary swirl vane at bottom to generate upward swirl flow and perforated cylinders as shown in Fig 1. At the top, it has a screen assembly of wire-mesh and perforated plates which also plays a role to remove small amount of moisture. The 3D CAD in Fig 1 is generated by SolidWorks®. Since it heavily costs to consider the circular shape of the perforated holes of the cylinder for real geometry scale, the circular shape is changed to square cylinder to conserve an equal hydraulic diameter in order to easily generate structured computational mesh near the square hole. Open-source preprocessor, SALOME [4], is used to generate computational mesh. 1.4 millions of both structured and unstructured meshes are employed. In this study, the screen assembly at top is modeled with an assumption that most of moisture is already separated by vertical perforated hole.

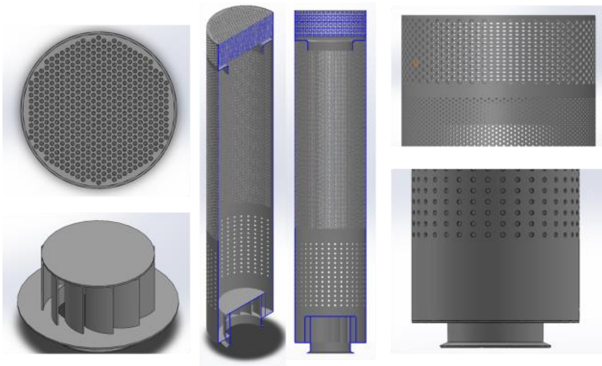

Fig 1. Schematics and computational mesh

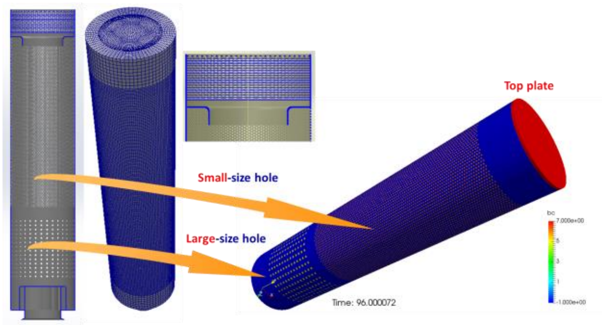

Fig 2. Boundary condition assignment

2.3 Feasibility study for Outlet and thin-wall boundary condition

The separator has countless numbers of perforated hole which should be assigned as a pressure outlet boundary condition. In addition, top perforated screen also needs to be as pressure outlet. In CUPID code, the pressure boundaries are assigned according to the hole size for vertical perforated hole and additional pressure boundary for top perforated screen as shown in Fig 2.

Since the swirl vane at bottom is no need to be modeled with finite thickness, a thin-wall boundary condition is assigned in this study. Fig 3 shows a feasibility study for the thin-wall boundary condition in CUPID code. The vane region is only taken into account for feasibility study. As shown in Fig3, the CUPID code can handle thin-boundary condition properly.

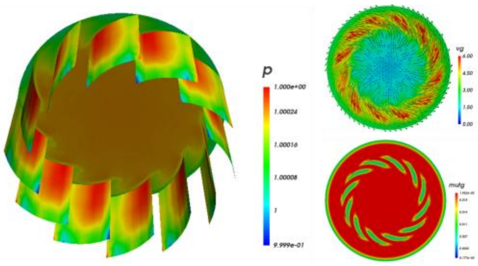

Fig 3. Boundary condition assignment

3. Results and Discussion

3.1 Simulation results

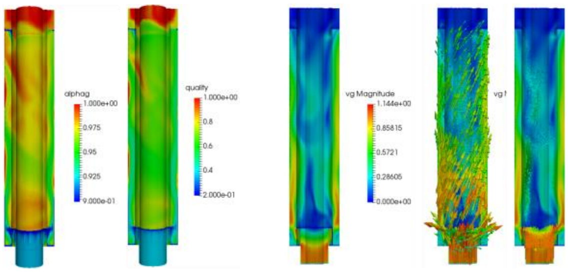

Fig 4. Flow visualization for $v_{g,in}$ = 1.0 *m/s*

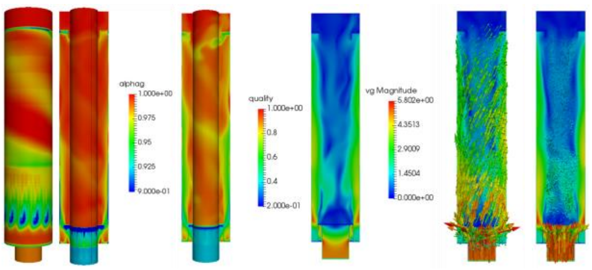

Fig 5. Flow visualization for $v_{g,in}$ = 5.0 *m/s*

In this study, the two-phase flow behavior inside separator is calculated according to the different inlet velocity; 1.0 m/s and 5.0 m/s of gas velocity is assigned. The boundary condition about void fraction and quality level at inlet during normal operation are set to be constant; 0.92 for void fraction and 25% quality, respectively. Design specification at the exit is supposed to be less than 99.75%. The result from the CUPID simulation shows reasonable range.

The liquid-gas mixture travels through a short vertical section and is redirected by the top plate to the swirl vane. The vanes impart a centrifugal force to the mixture. In a vortex motion, liquid flows up to inner wall and the gas tends to rise in the center of the separator. Most portion of the liquid exists through the vertical perforated wall, and the gas flows through the top exit plate.

Flow field for different inlet velocity is shown in Fig 4 and 5. Since larger inlet momentum of the mixture can converted higher level of swirl strength, liquid fraction of the mixture for high velocity tends to separate the liquid from the mixture easily. For relatively lower velocity inlet condition (1 m/s), more liquid fraction is captured in the middle of the separator.

4. Conclusions

The three-dimensional thermal-hydraulic behavior inside the moisture separator is numerically investigated by the CUPID code. Real geometry is taken into account and modified mesh system is assigned to reduce the cost. The flow fields highly depend on some parameters such as inserted mass flow rate. Thus, these parameters are examined in this study.

For the future work, simplified geometry by adopting porous media approach is going to be calculated in order to propose new physical model.

Acknowledgement

This work was performed as a collaborative research project of project C17001 (Computational science and engineering-based National R&D projects supporting efficiency) and supported by the KISTI, the National Research Foundation of Korea (NRF) and the Korea Foundation of Nuclear Safety (KoFONS) grant funded by the Korean government and supported by (MSIP & NSSC) (Nuclear Research and Development Program: 2017M2A8A4015005, Nuclear Safety Research Center Program: 1305011).

REFERENCES

[1] C.Y. Paik et al., “Steam separator modeling for various nuclear reactor transients”, Nuclear Engineering and Design, 108, pp.83-97 (1988).
[2] B.E. Lee et al., “Moisture removal performance of the steam generator applied to Korean standard nuclear power plants”, Proceedings of KSME, Nov.2004
[3] H.Y. Yoon et al., “Recent Improvements in the CUPID Code for a Multi-Dimensional Two-Phase Flow Analysis of Nuclear Reactor Components, Nuclear Engineering and Technology, 46, pp.655-666 (2014)
[4] http:// www.salome-platform.org.