Thermal-hydraulic analysis of a heavy-water reactor moderator tank using the CUPID Code

Su Ryong Choi[a], Hyoung Tae Kim[b], Han Young Yoon[b], Jae Jun Jeong[a]*
[a]School of Mechanical Engineering, Pusan National University, Busan 609-735, Korea
[b]Korea Atomic Energy Research Institute, 1045 Daedeok-daero, Yuseong-gu, Daejeon, Korea, 305-353
*Corresponding author: jjjeong@pusan.ac.kr

1. Introduction

When the core cooling system fails to remove the decay heat from the fuel channels during a loss of coolant accident (LOCA), the pressure tube (PT) could strain to contact its surrounding Calandria tube (CT), which leads to sustained CTs dry out, finally resulting in damages to nuclear fuel. This situation can occur when the degree of the subcooling of the moderator inside the Calandria vessel is insufficient. In this regard, to estimate the local subcooling of the moderator inside the Calandria vessel is very important. However, the local temperature is measured at the inlet and outlet of the vessel only. Therefore, we need to accurately predict the local temperature inside the Calandria vessel [1, 2].

Numerous experimental studies have been carried out for flow distribution inside the Calandria vessel. For example, the Stern laboratory established an experimental facility, which is scaled down one quarter of a CANDU Calandria vessel. Extensive experimental database had been produced to enhance the understanding of the complex thermal-hydraulic flow in the moderator tank and these were used for the validation of computer codes.

In this study, the thermal-hydraulic analysis of the real-scale heavy-water reactor moderator is carried out using the CUPID code. The applicability of the CUPID code to the analysis of the flow in the Calandria vessel has been assessed in the previous studies [3, 4].

2. Mathematical Model of the CUPID Code

The CUPID code has been developed for a high-resolution component-scale thermal-hydraulic analysis of two-phase flows in nuclear reactor components. The code adopts a two-fluid, three-field model to analyze two-phase flows. The three fields represent continuous liquid, droplets, and vapor, respectively. The governing equations of the CUPID code have been established to be applicable to both open media and porous media

A special pressure drop model for a porous media zone is needed to accurately simulate the flow behavior there. The hydraulic resistance consists of two factors; cross flow and axial flow. In the tube bundle region, the frictional pressure drop for the cross flow is represented as follows [5];

$$PLC = \frac{\Delta P}{N_f \cdot \rho \cdot v_{fs}^2/2} = 4.54 \cdot Re^{-0.172}, \qquad (1)$$

where $v_{fs} = \varepsilon v = \varepsilon\sqrt{\sum u}$, and PLC, P, N_f, v_{fs} are the pressure loss coefficient, pressure, number of tubes in the direction of flow, and free stream velocity, respectively. For an axial flow, the hydraulic resistance could be expressed by the conventional correlations for the pressure drop in a cylindrical pipe;

$$\left.\frac{\Delta P}{\Delta L}\right)_z = \frac{\Delta P}{\Delta z} = \frac{f\rho u_z^{\,2}}{2D_e}, \qquad (2)$$

where $f = 0.316Re^{-0.25}$. This model was implemented in the CUPID code [5].

3. Modeling of the Calandria Tank of a CANDU Reactor

3.1 Description of the Calandria vessel

The Calandria vessel of the CANDU reactor is a cylindrical tank with an axial length of 6 m and a diameter of 7.6 m as shown in Fig. 1. Inside the Calandria shell, there is a coaxial cylindrical tank with a diameter of 6.8 m in the core region. At the middle of left and right sidewall, the four inlet nozzles with an inclination of 14 degrees towards the center of the vessel, pointing to the top, are located symmetrically. At the bottom of right side in the Calandria vessel, two outlets are installed [3].

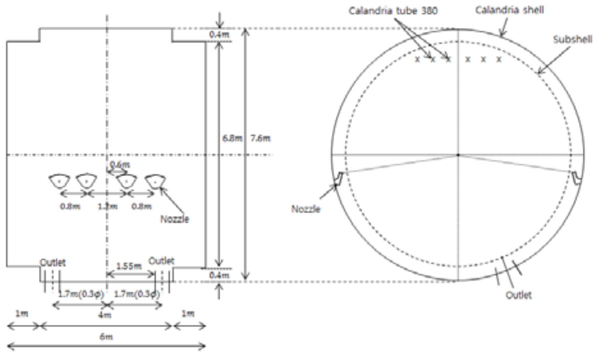

Fig. 1. CANDU moderator system

3.2 Mesh generation

Since the Calandria vessel in the core region is composed of a matrix of 380 CTs, it is not effective to realistically simulate all subchannel regions in between the tubes. Therefore, a porous media approach is adopted for the tube bundle region. And an open media approach is used for the outer region of the bundle [2].

The three dimensional grid is made by the CUPID_POP code, which can generate polyhedral mesh. Polyhedral grids were used for the core region in the center of the tank, which is modeled as a porous media. And the fluid region, regarded as an open media region, is made up of bent structured grids.

In this mesh generation, it was very important to decide the proper size of bent structured mesh for the fluid zone. Because the mesh size in the fluid zone has a considerable effect on the moderator flow near the inlet nozzles. In this study, the mesh which has a radial length of 0.181m, a circumferential length of 0.04m, and an axial length of about 0.1m is used for considering computer resource. Fig. 3 shows the three-dimensional grids of the Calandria vessel .

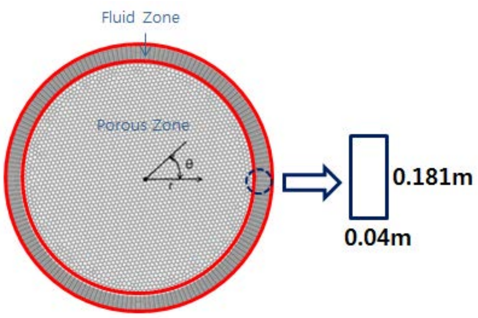

Fig. 2. A bent structured grid in the fluid zone

Fig. 3. Isometric view for the grids of the Calandria vessel

3.3 Local power distribution in the Calandria tank

For the 2,064 MW_{th} nuclear reactor, the total heat load in the moderator is about 103MW, which is divided into 96.7 MW in the core region and 6.3 MW in the reflector region [1].

We intend to simulate the actual flow phenomenon of the moderator system. Therefore, we suppose that the local power distribution is similar to the real power distribution in the core region. Fig. 4 shows the power distribution in the radial direction. Regardless of the azimuthal angle, the thermal power is equally distributed. The red line in Fig. 4 is the 5-th order polynomial fit based on the actual data in the power distribution. Fig. 5 presents the relative power distribution in the axial direction. The red line is the step function based on the actual data.

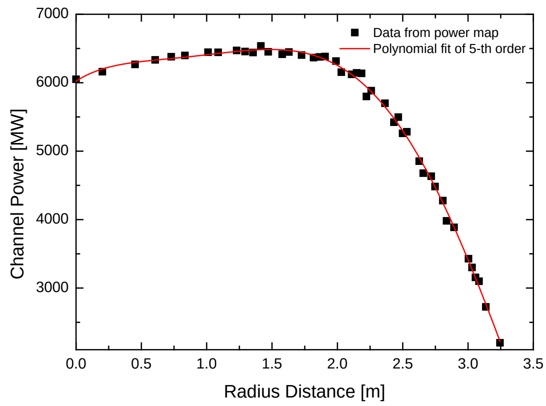

Fig. 4. Power distribution in the radial direction

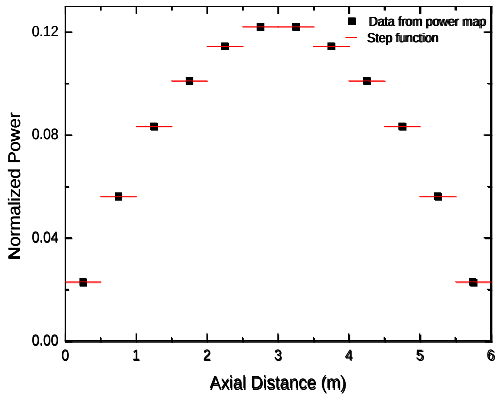

Fig. 5. Power distribution in the axial direction

4. Results of Simulations in the Calandria tank of the CANDU Reactor

4.1 Simulation with the basic case input

The mass flow rate in to the inlet nozzles is 1019kg/s (v_{in}=2.123 m/s), where the cross-sectional area of each inlet nozzle is 0.4864m^2, and the inlet temperature is 320K. The simulation results with the basic case input showed a buoyant-dominant flow pattern; a thermal stratification was predicted, which is caused by lacking the momentum of the inlet nozzles as depicted in Fig. 5. This leads to a boiling at the top of the tank as shown in Fig. 7. It is clear that this result is not realistic.

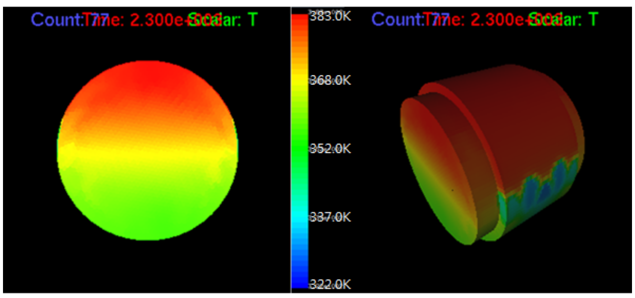

Fig. 6. Temperature distribution; 103% full power, 2.123m/s

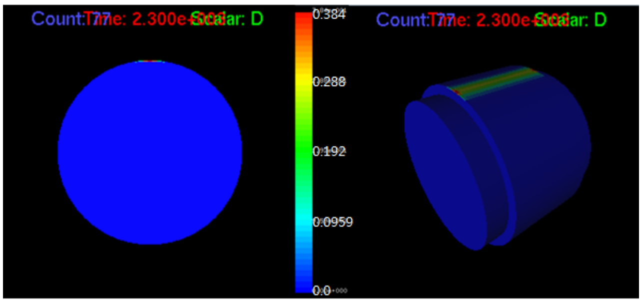

Fig. 7. Void fraction distribution; 103% full power, 2.123m/s

4.2 Simulation with reduced inlet nozzle area

In the previous calculation, the moderator flow did not reach to the top of the Calandria tank due to the lack of the momentum. To improve the unphysical simulation results, the cross-sectional area at each inlet nozzle is reduced to the half ($0.2432m^2$) of the former case, while maintaining the mass flow rate. Other parameters remain unchanged.

Fig. 8 and Fig. 9 shows the steady-state temperature distribution and velocity fields at z=1.2m and 3.0m, respectively. The flow pattern is very similar to other calculation [3]. In the beginning of the calculation, the moderator flow reaches to the top of the vessel from the slanting inlet nozzles. At about 400 s, the flow pattern becomes asymmetrically leant to the left side because of the grid structure or initial instability. After 1800 s, the results show a mixed flow regime in a quasi-steady state. The local maximum temperature is 363.0K in the upper right region.

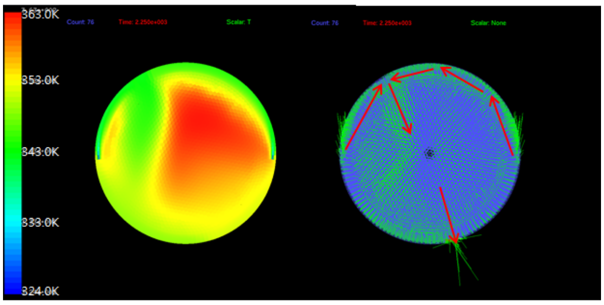

Fig. 8. The temperature distribution and vector of liquid for steady-state; 103% full power, 4.245m/s (z=1.2m)

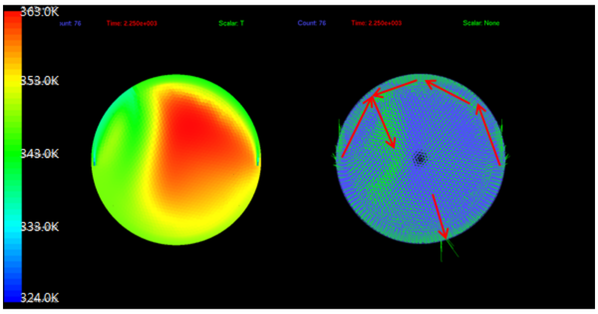

Fig. 9. The temperature distribution and vector of liquid for steady-state; 103% full power, 4.245m/s (z=3.0m)

It is very clear from the two calculations that the modeling of the flow from the inlet nozzles is very important for the accurate prediction of the flow in the Calindria vessel.

5. Conclusions

In this study, a preliminary analysis is performed for the CANDU moderator tank. The calculation results using the basic case input showed a unrealistic, thermal stratification in the upper region, which was caused by the lack of the momentum of the cooling water from the inlet nozzle. To increase the flow momentum from the inlet nozzle, the cross-section area of each inlet nozzle was reduced by half and, then, the calculation showed very realistic results. It is clear that the modeling of the inlet nozzle affects the calculation result significantly. Further studies are needed for a realistic and efficient simulation of the flow in the Calandria tank.

Acknowledgment

This work was supported by the Nuclear Research and Development Program of National Research Foundation of Korea (NRF) grant funded by the Korean government (MEST) and the Nuclear Safety Research Center Program of the KORSAFe grant funded by Nuclear Safety and Security Commission (NSSC) of the Korean government (Grant code 1305011).

References

[1] C. Yoon, B.W. Rhee, H.T. Kim, J.H. Park, and B,J Min, “Moderator Analysis of Wolsong Units 2/3/4 for the 35% Reactor Inlet Header Break with a Loss of Emergency Core Cooling Injection”, Journal of Nuclear Science and Technology, Vol.43, No.5, p.505–513, 2006.

[2] S.G. Park, J.J. Jeong, J.R. Lee, and H.T. Kim, “Preliminary Study of CANDU-6 Moderator System with the CUPID Code”, Transactions of the Korean Nuclear Society Spring Meeting Jeju, Korea, May 16-18, 2012.

[3] C. Yoon, J. H. Park, “Development of a CFD Model for the CANDU-6 Moderator Analysis using a Coupled Solver”, Annals of Nuclear Energy 35, 1041-1049, 2008.

[4] R.G. Huget, J.K. Szymanski, and W.I. Midvidy, “Status of Physical and Numerical Modelling of CANDU Moderator Circulation”, Proceedings of the 10th Annual Conference of the Canadian Nuclear Society, Ottawa, 1989.

[5] G.I. Hadaller, R.A. Fortman, J. Szymanski, W.I. Midvidy, and D.J. Train, “Frictional Pressure Drop for Staggered and In Line Tube Bank with Large Pitch to Diameter Ratio”, Proceedings of 17th CNS Conference, Federiction, New Brunswich, Canada, June 9-12, 1996.