Moderator Circulation mode of CANDU-6 reactor

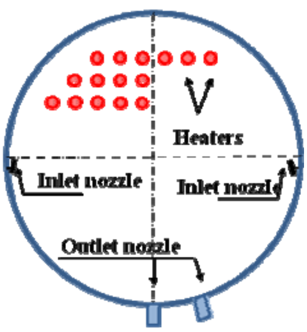
Tae Kwang Eom[a], Manwoong Kim[b], Jae Young Lee[a*]
[a]*School of Mechanical and Control Engineering, Han Dong Global University , Pohang, Korea*
[b]*Korea Institute of Nuclear Safety, Daejun, Korea*
* *Corresponding author: jylee7@handong.edu*

1. Introduction

In a CANDU reactor, heavy water moderator plays a role as the ultimate heat sink. When a loss of coolant accident occurs, a contact between PT (pressure tube) and CT (Calandria tube) threatens the fuel channel safety. The coolability of the moderator has an effect on the prevention of PT/CT contact. Therefore, thermal hydraulic analyses on moderator analyses are of importance. Series of studies have been made for enhancing the safety and performance of CANDU reactor. Among them, we developed the 1/8 scaled HU-KINS based on the scaling Laws for the experimental investigation of Thermal-Hydraulic Effect of CANDU-6 Moderator. For the CFD analysis, we investigated the performance of various turbulent models including k-ε. The major reference of the CFD justification is the experimental of HU-KINS.

2. Schematic diagrams and Computational Meshes of 1/8 scaled Moderator cooling system

Lee et al (2006) produced the scaling laws for circulation flow in calandira tank. In the scaling law, the linear scaling based on the power density and local dimensionless numbers for the buoyancy and inertial force balance were used. As shown in Fig.1, schematic diagram of the 1/8 scale calandria tank and geometry value and scale were depicted.

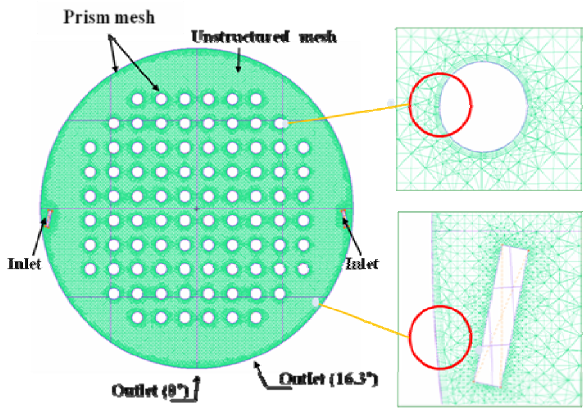


Geometry	HGU-KINS	
	Value	Scale
D_{cal} (m)	0.95	1/8
L (m)	0.0384	1/156.3
P (m)	0.072	1/4
D_{pipe} (m)	0.033	1/4
Heater	88	1/4.3
A_{in} (m^2)	0.006	1/767
A_{out} (m^2)	0.006	1/767

(a) (b)

Fig.1. Schematic diagrams of HU-KINS experiments and calculation domain (a) experimental facility (b) geometry value and scale.

In the present study, we utilized CFX, the three dimensional computational fluid dynamics codes, to find the flow structure in the calandria tank. We developed a mesh structure to realize the turbulent dissipation near the wall and nuclear fuel channels. As shown in Fig. 2, the unstructured meshes were mostly used for calculation and prism meshes were used for fuel channels and calandria vessel surface wall to make an accurate calculation the turbulence generation (k) and dissipation rate (ε).



(a) (b)

Fig. 2. Computational Meshes for calculating a HU-KINS reactor on central plane. (a) Inside calandria tank of HU-KINS (b) Fuel surface & the inner wall surface.

3. CFD analyses model

Table 1. Analyses model for computational calculation.

	Analysis Model
Mesh	Unstructured mesh :Mostly
	Prism mesh :Fuel channel wall Inlet nozzle reflector
	Total elements : 1,881,618 Total nodes : 448148 with over 2.5 quality
Fluid flow	Incompressible & single-phase flow
	The standard k-ε turbulence model
	Buoyancy effect : Boussinesq approximation
Calculation	Using a high resolution
	Residuals in the numerical solution of governing equation is less than 10^-3 but heat-transfer is nearly 10^-3
CFD code	CFX-11

In CFD analysis, the buoyancy and momentum forces should be modeled. We used k-ε turbulence model and the buoyancy force assumed Boussinesq approximation which changes density by temperature. Fluid flow is assumed by incompressible and single-phase flow. The convergence criteria for simulation was set to 10^-3 of RMS error.

4. Comparison between experimental and predicted flow pattern of the HU-KINS.

With the variation of inlet flow rate and 10kW power supply, we can produce the mixed flow regime and the buoyancy flow regime based on the Reynolds number and Archimedes number. In order to conduct an experimental visualization of the circulation mode of the moderator in the HU-KINS, we adopted chemical method. At first, we dissolved BTB into the moderator which changes into blue color when it mixes with NaOH solution.

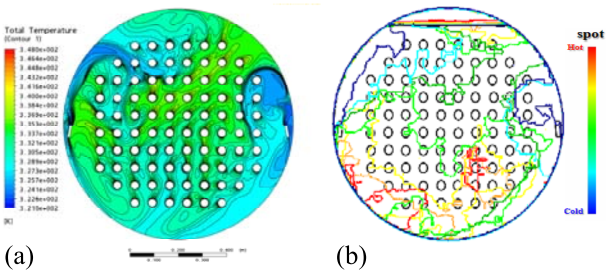


Fig. 3. Comparison of CFD and experiment on the flow of the mixed flow regime (a) CFD (b) Experiment

As shown in Fig. 3, we can see asymmetric circulations of flow in CFD and experimental data, and flow patterns are similar to each other. The mixed flow represents the circulation and buoyancy but the circulation hits the top of the tank so there is no stratified horizon in the tank. Therefore, there is a low risk of hot spot in the tank.

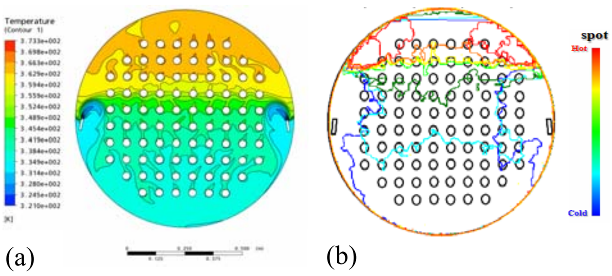


Fig. 4. Comparison of CFD and experiment on the flow of the buoyancy driven flow regime (a) CFD (b) Experiment

As shown in Fig. 4, we can see the stratified horizon of flow in CFD and experimental data. The flow pattern has a similar pattern except the stratified horizon area. The buoyancy-driven flow is depicted in Fig. 4. As noted, it takes a long time to cover up the whole tank with the chemical. The flow pattern shows that the injected jet flows down immediately near the nozzle and makes a circulation near the bottom of the tank. Then it finally makes the rising flow to the upper part. However, at a certain height, the jet cannot penetrate the stratified line due to the temperature difference. Due to the risk of hot spot, the CANDU needs to avoid the buoyancy driven circulation mode.

5. Conclusions

When we compare the experimental visualization results and the CFD result of k-ε turbulence model in the 1/8 Scaled HU-KINS of Calandria vessel of CANDU-6, we found that flow patterns are suitably matched with each other in the mixed and the buoyancy driven flow regime.The present study showes that the experimental works with HU-KINS can be used to evaluate the computational analysis and safety index for the moderator cooling system. We will also apply to other turbulence models to estimate validation of a CFD analysis model for predicting HU-KINS by comparing the experimental results with the CFD result. Once we have the reliability of the CFD code, we can easily extend the present work to the real case of CANDU-6. It may be expected to set up the safety regulation index, archimedes number for calandiria cooling system.

Acknowledgment

The present work is supported partially by Korea Institute of Nuclear Safety and BAERI of Ministry of Science and Technology in Korea.

REFERENCES

[1] Lee, J.Y., Kim, M.W., and Kim, N.S., 2006, "Design for the 1/8 Scaled HU-KINS Based on the Scaling Laws for the Experimental Investigation of Thermal-Hydraulic Effect of CANDU-6 Moderator", The Korean Society of Mechanical Engineers B, vol. 30, pp. 825~833

[2] Manwoong Kim, Seon-Oh Yu, Hho-Jung Kim, 2006, “Analyses on fluid flow and heat transfer inside Calandria vessel of CANDU-6 using CFD” ,Nuclear Engineering and Design 236 (2006) pp.1155–1164

[3] Jung, Y.S., Lee, J.Y. and Kim, M.W., 2004, “The Transition Criteria of Circulating Flow Pattern of Moderator in the Calandria Tank of CANDU Nulcear Power Plant,” N6P141, Proceedings of NUTHOS-6: 6th International Topical Meeting on Nucelar Reactor Thermal Hydraulics, Operation and Safety, October 4-8(2004)Nara, Japan.

[4] Yu, S.O and Kim, M.W., “Assessment of Moderator Integrity Using Realistic Model and Parametric Studies on Thermal-Hydraulic Characteristics in SPEL, the Korean Nuclear Society, Transactions of the Korean Nuclear Society Autumn Meeting (2002)

[5] Yoon, C., Rhee, B. W., and Min, B. J., "Validation of a CFD Analysis Model for Predicting CANDU-6 Moderator Temperature against SPEL Experiments," Proceedings of ICONE10, April, Virginia, (2002)