Comparative Study for Modeling Reactor Internal Geometry in CFD Simulation of PHWR Internal Flow

Gong Hee Lee [a]*, Sweng Woong Woo [a], Ae Ju Cheong [b]
[a]*Safety Analysis & Evaluation Department, Korea Institute of Nuclear Safety, Daejon, 305-338*
[b]*Safety Issue Research Department, Korea Institute of Nuclear Safety, Daejon, 305-338*
**Corresponding author: ghlee@kins.re.kr*

1. Introduction

Estimating the local subcooling of the moderator in a CANDU calandria under transient conditions is one of the major concerns in the CANDU safety analysis. Therefore extensive CFD analyses have been performed for predicting the moderator temperature in a CANDU calandria or its similar shape. However most of previous studies used a porous medium assumption instead of considering the real geometry of calandria tube [1,2,3].

A porous medium assumption has some possible weaknesses; (1) The increased production of turbulence due to vortex shedding in the wake of the individual tubes is not considered in the turbulence model. (2) It is difficult to identify the true effects of the outer ring of calandria tubes on the generation of the highly non-uniform flows in the reflector region. (3) It is not clear how well the pressure loss models quantitatively represent the three-dimensional effects of the turbulent flows through the calandria tubes.

The main objective of the present study is to compare the results predicted by using either the real geometry of tubes or porous medium assumption and to assess the prediction performance of both methods.

2. Analysis Model

2.1 Test Facility and Test Conditions

A schematic diagram of the test rig is shown in Fig. 1. The vessel had a transparent polycarbonate cylindrical shell with a diameter of 2m and a length of 0.2m.

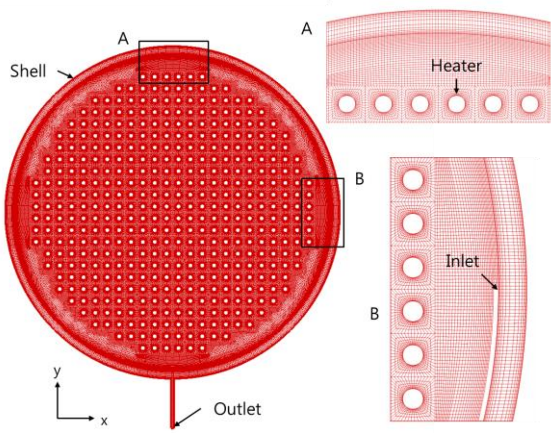

Fig. 1. Computational domain and grid system

A total of 440 electrical tube heaters (0.033m in diameter and 0.2m in length) were installed on a 0.071m square pitch pattern. Light water entered the vessel through two symmetrically placed inlet nozzles, which shot vertically directed jet-type flow at the horizontal centerline. One outlet nozzle whose thickness is same as that of the vessel was installed at the bottom of the vessel.

Tests were conducted both in isothermal condition and nominal (100 kW heat addition) condition. Fluid velocity and temperature were measured with a laser Doppler anemometer (LDA) and thermocouples.

2.2 Porous Medium Model

An approach considering the real geometry of calandria tubes requires much more computation resource to analyze the real flow phenomena inside a calandria. Porous medium assumption may be one of candidates to solve this problem. In this assumption, porosity has an effect on flow acceleration in the porous domain and its magnitude is generally determined by considering the real geometry of calandria tubes. Empirical correlation for the pressure loss coefficient, which should be independent of the calandria tube configurations, is used to model the momentum loss in the porous domain which corresponds to pressure drop in real calandria vessel. More detailed descriptions of porous medium model can be found in the reference [3].

3. Numerical Modeling

3.1 Numerical Method

The flow was assumed to be steady, incompressible and turbulent. The first-order accurate upwind differencing for the convection terms of each governing equation was used to enhance the convergence. The second-order accuracy was maintained for the viscous terms. The pressure-velocity coupling was handled by the SIMPLE algorithm. The solution was considered to be converged when the residuals of variables were below 10^{-5} (10^{-7} for energy equation) and the variations of the target variables were small.

In case of nominal condition, the effect of buoyancy force was modeled with the Boussinesq approximation. A constant volumetric expansion coefficient of 4.6×10^{-4} was used. The generation of turbulent kinetic energy due to buoyancy was considered in k and ε equation.

Simulation was conducted with the commercial CFD software, FLUENT R.14 [4].

3.2 Turbulence Model

Standard k-ε model, which is one of Reynolds-averaged Navier-Stokes (RANS)-based two equation turbulence models, was used. It is well known that this model has been widely used in the various industrial applications and has a superior convergence in comparison with other turbulence models.

3.3 Grid System and Boundary Conditions

As shown in Fig. 1, the hexahedral grid system was generated for the computational domain that had the same dimension as the test facility. Detailed information for the grid system was shown in Table 1.

Constant flow rate of total 2.4kg/s (nozzle width of 8mm) was imposed at normal to inlet boundary. Inlet fluid temperature was 55.5℃. Turbulence intensity at inlet was assumed to be 5.0%. Average pressure over whole outlet option with the relative pressure of 0 Pa was used as an outlet boundary condition. No-slip condition was applied on the solid wall. Uniform heat flux (W/m^2) which corresponded to 100kW DC power, was imposed on the heater wall and adiabatic condition on the other walls. In order to model the near-wall region, wall function approach was used.

Table 1. Summary of grid system

Items	Grid A	Grid B
Total number	1,152,195	4,270,780
Min. orthogonal quality	0.56	0.507
Max. aspect ratio	111.2	227.5
Max. y+	177/274	117
Note	*Isothermal & Nominal*	*Isothermal*

4. Results and Discussion

4.1 Isothermal Condition

Fig. 2 shows the vertical (y-direction) velocity profile along the vertical centerline (x=0). The prediction with real geometry model showed the better agreement with measurement. Porous medium model underpredicted the magnitude of the vertical velocity. Meanwhile, relatively dense grid (grid B) showed a little bit the improved prediction accuracy.

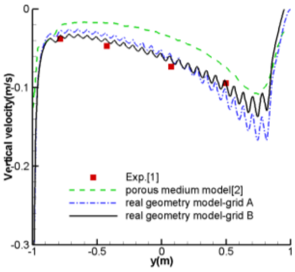

Fig. 2. Vertical velocity profile at the vertical centerline (x=0)

Fig. 3 shows the tangential velocity profile near the circumferential wall along a radial line which is at 30 degree and 60 degree from horizontal centerline. The prediction with real geometry model showed the better agreement with measurement. The effect of total number of grids was not nearly significant.

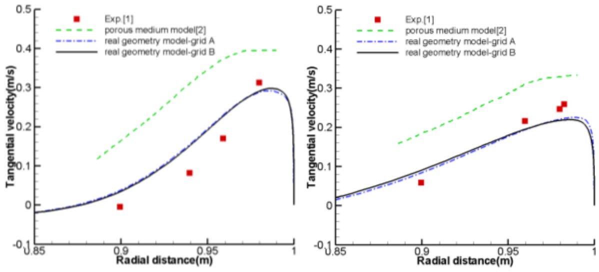

Fig. 3. Tangential velocity profile at 30 degree (left) and 60 degree (right) from the horizontal centerline in the counter-clockwise direction

4.2 Nominal Condition

Overall continuity and energy balance were satisfied with the difference in 0.001% of the total flow rate and 0.06% of total heat flux. The predicted results showed the constant pattern at the convergence criteria.

Fig. 4 shows the temperature profile at the vertical centerline (x=0) and the horizontal line (y=0.57m). At the vertical centerline, the prediction with real geometry model showed the better agreement with measurement in the range of y > 0.0m.

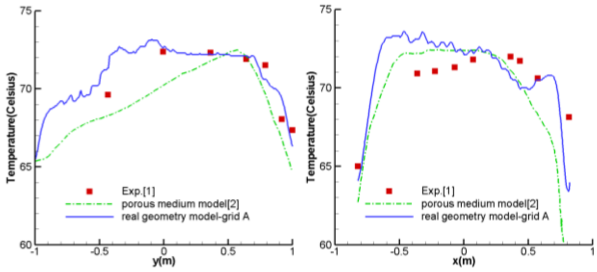

Fig. 4. Temperature profile at the vertical centerline (x=0; left) and the horizontal line (y=0.57m; right)

Acknowledgments

This study was conducted under the financial support of the Nuclear Safety and Security Commission of Korea [project title: Development of Regulatory Evaluation Technologies for Thermal-hydraulic Safety].

REFERENCES

[1] R. G. Huget et al., Status of Physical and Numerical Modeling of CANDU Moderator Circulation, Proc. 10th Annual Conference of the Canadian Nuclear Society, 1989.
[2] R. G. Huget et al., Experimental and Numerical Modelling of Combined Forced and Free Convection in a Complex Geometry with Internal Heat Generation, Proc. 9th Int. Heat Transfer Conference, Jerusalem, Israel, 1990.
[3] C. Yoon, H. H. Park, Development of a CFD Model for the CANDU-6 Moderator Analysis using a Coupled Solver, Annals of Nuclear Energy, Vol.35, p.1041, 2008.
[4] ANSYS FLUENT, Release 14.0, ANSYS Inc.