The Effect of the Holes Size Change of Lower-Support-Structure-Bottom Plate on the Reactor Core-Inlet Flow-Distribution

Gong Hee Lee [a]*, Young Seok Bang [b], Ae Ju Cheong [c]
[a]*Kori Residence Inspection Team, Korea Institute of Nuclear Safety, Daejon, 305-338*
[b]*Safety Evaluation Department, Korea Institute of Nuclear Safety, Daejon, 305-338*
[c]*Nuclear Safety Research Department, Korea Institute of Nuclear Safety, Daejon, 305-338*
**Corresponding author: ghlee@kins.re.kr*

1. Introduction

Complex thermal-hydraulic phenomena exist inside PWR because reactor interiors include a fuel assembly, control rod assembly, ICI (In-Core Instrumentation), and other internal structures. Because changes to reactor design may influence interior, thermal-hydraulic characteristics, licensing applicants commonly conduct a flow-distribution test and use test results (e.g., core-inlet flow-rate distribution) as the input data for a core thermal-margin analysis program.

Because the APR+ (Advanced Power Reactor Plus) had more fuel assemblies (241EA→257EA) and the design of some internal structures was changed (from those of APR1400), the core-inlet flow-rate distribution for a 1/5 scaled-down reactor model was measured and high flow-rates were found especially near the outer region of the reactor core [1,2]. Such a result may be undesirable in terms of both the mechanical integrity of fuel assembly and the core thermal-margin. To solve the above-mentioned problem, additional tests with a 50% blockage of the flow holes in the outer region of the Lower-Support-Structure-Bottom Plate (LSSBP) were conducted under the 4-pump balanced flow condition, and the measured data were compared with those of the original LSSBP [3].

In this study, to examine the effect of the holes size change (i.e. smaller diameter) in the outer region of the LSSBP, not a 50% blockage of the flow holes, on the reactor core-inlet flow-distribution, simulations were conducted with the commercial CFD (Computational Fluid Dynamics) software, ANSYS CFX R.14. The predicted results were compared with those of the original LSSBP.

2. Analysis model

2.1 APR$^+$ Flow Distribution Test Facility

APR+ Core Flow & Pressure Test Facility (ACOP), installed in the KAERI (Korea Atomic Energy Research Institute), is a 1/5 scaled-down model of APR+. It consists of a reactor vessel with two coolant loops (i.e., four cold legs and two hot legs). The scaling ratios applied to the test facility are summarized in Table I.

The internal structures of the reactor model (e.g., flow skirt and upper/lower core structures) had almost the same shapes as those in the original APR+, and satisfied geometrical similarity [1,2]. The core-inlet flow-rate distribution could be obtained by measuring the differential pressure and discharge coefficients at the venturi region of each core simulator. A total of 257 core simulators, which corresponded to the fuel assemblies, were installed in the reactor model. The upper head of the reactor, and some core-bypass flow-paths were neglected in the reactor model because these parts were expected to have little influence on the core-inlet flow-rate distribution. The criteria of the allowable data scattering for each core simulator inlet flow-rate distribution was ±1.5% [1].

Table I: Summary of scaling parameters [1].

Parameters	APR+	ACOP
Temperature, ℃	310	60
Pressure, MPa	15	0.2
Density, kg/m^3	705.8	983.2
Viscosity, Ns/ m^2	8.88×10^{-5}	4.66×10^{-4}
Length ratio	1	1/5
Area ratio	1	1/25
Volume ratio	1	1/125
Aspect ratio	1	1
Velocity ratio	1	1/2.17
Mass flow ratio	1	1/38.9
Core exit Re ratio	1	1/40.9
ΔP ratio	1	1/3.38

2.2 Test Conditions

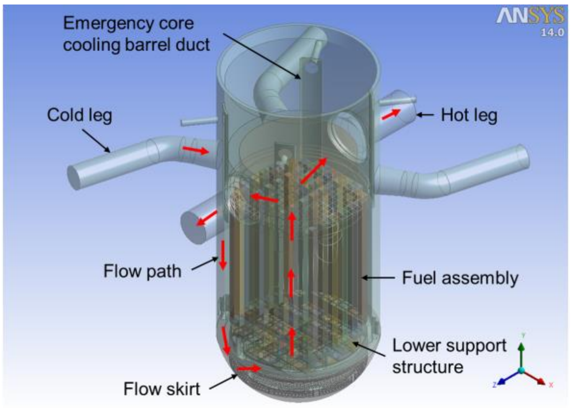
The test matrix consists of three flow conditions, i.e., the symmetric or asymmetric flow conditions for 4-pumps operation, and the flow condition for 3-pumps operation. In this study, CFD simulation was conducted under the symmetric flow condition for 4-pumps operation. Under this condition, the Reynolds number was about 8.6×10^5 in the downcomer.

2.3 Geometry Modeling

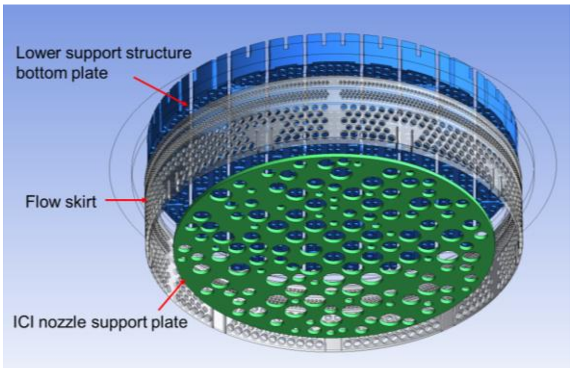
2.3.1. Porous medium assumption. APR+ reactor internals are complex structures which support fuel assemblies, control rods and measuring instruments.

The internal structures, especially those located in the upstream of the reactor core, may have a significant influence on the core-inlet flow-rate distribution; depending on both their shapes, and the relative distance between the internal structures and the core inlet [4]. Therefore an exact representation of these internal structures is needed for CFD simulation of the core-inlet flow-rate distribution. However, such an approach requires a great deal of computing resources to analyze the real-flow phenomena inside a reactor.

In this study, as shown in Fig. 1, among the reactor internal structures located upstream of the reactor core, the real geometries of a flow skirt, LSSBP and ICI nozzle support plate, were considered because these internal structures could significantly influence the flow-rate distribution at the core inlet.



(a) Full geometry



(b) Details of lower support structure

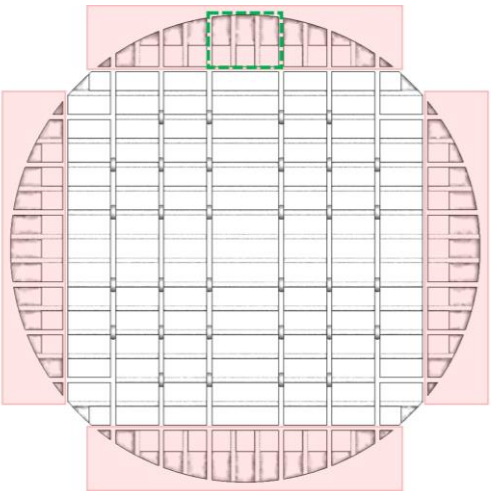
Fig. 1. The computational domain.

Meanwhile, to reduce total numbers of elements and thus minimize the required amount of computation, fuel assemblies and some internal structures (e.g., control-element guide tubes) were simply considered as each bulk volume (porous domain). Then, in order to reflect the velocity field and pressure drop occurring in the real-flow region; porosity and Isotropic Loss Models [5] were applied to the porous domain.

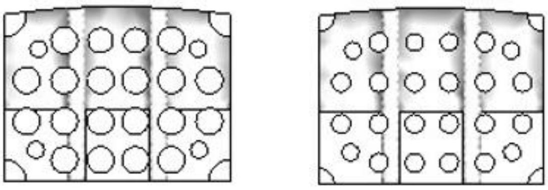
Porosity is the ratio of the volume of fluid region to total volume; including both fluid and solid regions. It has an effect on flow acceleration in the porous domain. In this study, the porosity was determined by considering the real geometry of the reactor internal structures. A momentum source was used to model the momentum loss in the porous domain; which corresponds to a pressure drop in real reactor vessel. Loss coefficients were adjusted to match the magnitude of the pressure drop found in the porous domain, with those of the measurement.

2.3.2. LSSBP holes pattern. With the aid of a flow skirt, the LSSBP plays a significant role in a uniform mass-flow rate distribution at core inlet. In general, holes size in the center region of the LSSBP are smaller than those in the outer region to prevent the flow from building up in the core center region.

In this study, in order to examine the appropriateness of the original LSSBP holes pattern and the effect of holes size change of LSSBP on the reactor core-inlet flow-distribution, the original LSSBP holes pattern was modified. As shown in Fig. 2, holes size in the outer region of the LSSBP, represented by four rectangular boxes in red, was reduced. The diameter ratio of a hole in the original LSSBP to that in the modified LSSBP was about 1.414, which corresponded to 50% reduction in the flow area per a hole. In case of several holes with small diameter, a hole size was not changed.



(a) Regions of the LSSBP holes pattern change



(b) Original

(c) Modified

Fig. 2. LSSBP holes pattern.

3. Numerical modeling

3.1 Numerical Method

The flow inside the scaled-down APR+ model was assumed to be steady, incompressible, isothermal and turbulent. Spatial discretization errors result from both the numerical order of accuracy of the discretization scheme, and from grid spacing. It is well known that second, or higher, order discretization schemes are potentially able to produce high-quality solutions. In addition, when either the flow is not aligned with the grid, or is complex, it is recommended that the first order discretization scheme not be used for the convection term, if possible. In this study, a high resolution scheme was used for both the convection-terms-of-momentum equations and -turbulence equations. The solution was considered 'converged' when the residuals of the variables were below 3×10^{-4}, and the variations of the target variables were small. Simulation was conducted with the commercial CFD software, ANSYS CFX R.14.

3.2 Turbulence Model

The k-ε model, which is one of the most prominent Reynolds Averaged Navier-Stokes (RANS)-based turbulence models, was used to simulate the turbulent flow inside the scaled-down APR+. The reason is that this model has proven to be numerically stable and has offered a good compromise in terms of accuracy and robustness. In a previous study [6], turbulence models available in ANSYS CFX R.13, for example k-ε model, Shear Stress Transport (SST) model, and SSG (Speziale, Sarkar and Gatski) Reynolds Stress model, were used to examine the turbulent flow inside the scaled-down APR+. Although the reactor internal-flow pattern differed locally; depending on the turbulence models used, the k-ε model showed the best agreement with the experimental data. More detailed descriptions of the k-ε model can be found in the ANSYS CFX-solver modeling guide [5].

3.3 Grid System

As shown in Fig. 3, a hybrid mesh, made up of tetrahedrons, pyramids and prisms, was generated to prevent the oversimplification of the geometry, and to have more efficient mesh distribution. Prism layers were used to get higher resolution in the near-wall region.

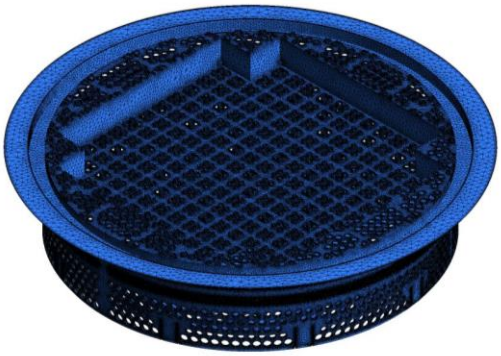


Fig. 3. Grid system – Lower support structure.

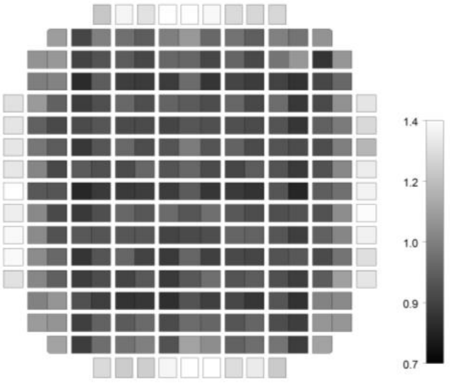
Table 2: Grid information for the modified LSSBP.

Domain	No. of elements	
	Type1	**Type2**
Downcomer	9.3×10^6	9.3×10^6
Lower support structure	7.0×10^6	1.4×10^7
Fuel assembly	7.8×10^6	7.8×10^6
Others	2.8×10^7	4.4×10^7
Total	5.2×10^7	7.5×10^7

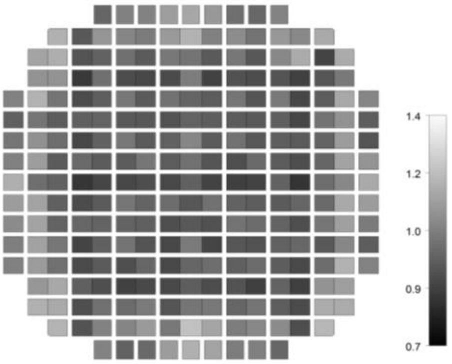
Detailed information for two grid types is shown in Table 2. Because the average difference of the normalized, mass-flow rate at the core-inlet plane between two grid types was about 0.4%, the predicted results with grid type1 was explained in this study.

3.4 Boundary Conditions

By referring to the test condition [1,2]; an inlet flow-rate of 135 kg/s was imposed at each cold leg. Turbulence intensity at the inlet was assumed to be 5 %. Light water at 60℃ was used as the working fluid. The 'average pressure over the whole outlet' option; with a relative pressure of 0 Pa, was used at each hot leg as an outlet-boundary condition. A no-slip condition was applied at the solid wall. To model the flow in the near-wall region, scalable wall functions were applied.



(a) Original LSSBP holes pattern



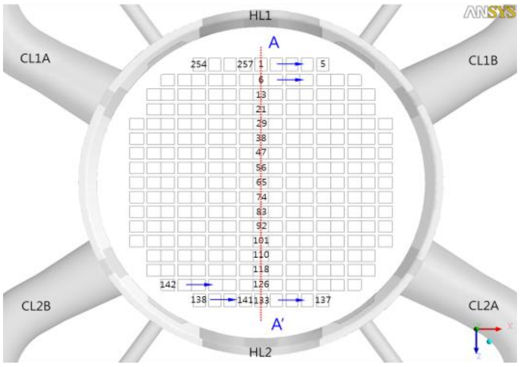
(b) Modified LSSBP holes pattern

Fig. 4. The normalized mass flow rate at core inlet plane.

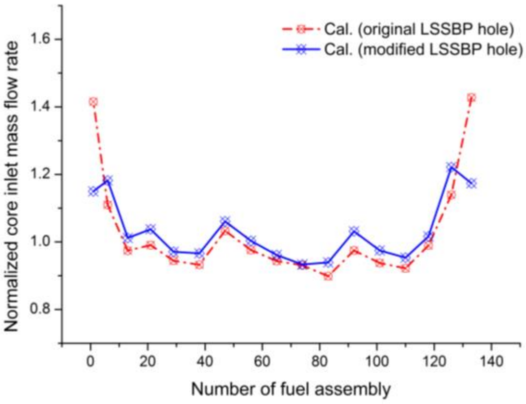
4. Results and Discussion

Fig. 4 shows the normalized, mass-flow rate at the core-inlet plane. The magnitude is the ratio of the mass-flow rate per each fuel assembly, to the average mass-flow rate at the core-inlet plane. The original and modified LSSBP holes pattern predicted core-inlet mass-flow rates in the range of 80~143%, and 84~122%, respectively. In the outer region of the reactor core (i.e. the region of the holes size change), core-inlet mass-flow rates were decreased, ranging from 15% to 28%.

Fig. 5 shows distribution of the normalized core-inlet mass-flow rate along the core centerline (A-A'). Except that a relatively high inlet-mass flow-rate in the outer region of the reactor core decreased, core inlet-mass flow-rate increased on the whole. This trend may be desirable in terms of improving both the mechanical integrity of fuel assembly in the outer region of the reactor core and the core thermal-margin.



(a) Numbering of fuel assemblies



(b) Normalized core inlet mass-flow rate

Fig. 5. Distribution of the normalized core inlet mass-flow rate along core centerlines (A-A′).

Fig. 6 shows the frequency distribution of the mass-flow rate at the core-inlet plane. The standard deviation (σ) of the mass-flow rate for the modified LSSBP hole pattern was smaller than that for the original LSSBP hole pattern. This means that the former predicted a more uniform distribution of the mass-flow rate at the core-inlet plane. In addition, the modified LSSBP hole pattern showed a relatively narrow minimum/maximum mass-flow rate distribution at the core-inlet plane.

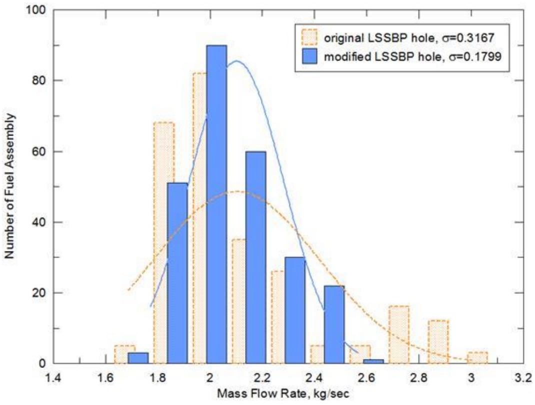


Fig. 6. Frequency distribution of the mass-flow rate at the core-inlet plane.

5. Conclusions

In this study, to examine the effect of the holes size change (smaller diameter) in the outer region of the LSSBP on the reactor core-inlet flow-distribution, simulations were conducted with the commercial CFD software, ANSYS CFX R.14. The predicted results were compared with those of the original LSSBP. Through these comparisons it was concluded that a more uniform distribution of the mass-flow rate at the core-inlet plane could be obtained by reducing the holes size in the outer region of the LSSBP. Therefore, from the nuclear regulatory perspective, this kind of the design change of the holes pattern in the outer region of LSSBP may be desirable in terms of improving both the mechanical integrity of fuel assembly and the core thermal margin.

ACKNOWLEDGEMENT

This work was supported by the Nuclear Safety Research Program through the Korea Radiation Safety Foundation (KORSAFe) and the Nuclear Safety and Security Commission (NSSC), Republic of Korea (Grant No. 1305002).

REFERENCES

[1] D. J. Euh, K. H. Kim, J. H. Youn, J. H. Bae, I. C. Chu, J. T. Kim, H. S. Kang, H. S. Choi, S. T. Lee, T. S. Kwon, A Flow and Pressure Distribution of APR+ Reactor under the 4-Pump Running Conditions with a Balanced Flow Rate, Nuclear Engineering and Technology, Vol.44, p.735, 2002.

[2] K. H. Kim, D. J. Euh, I. C. Chu, Y. J. Youn, H. S. Choi, T. S. Kwon, Experimental Study of the APR+ Reactor Core Flow and Pressure Distributions under 4-Pump Running Conditions, Nuclear Engineering and Design, Vol.265, p.957, 2013.

[3] K. H. Kim, D. J. Euh, H. S. Choi, T. S. Kwon, Effect of the Design Change of the LSSBP on Core Flow Distribution of APR+ Reactor, Transactions of the Korean Nuclear Society Autumn Meeting, Pyeongchang, Korea, October 30-31, 2014.

[4] B. J. Lee, H. C. Jang, J. S. Cheong, S. J. Baek, Y. S. Park, A Review on the Regionalization Methodology for Core Inlet Flow Distribution Map, J. Korean Nuclear Society, Vol.33, p. 441, 2001.

[5] ANSYS CFX-Solver Modeling Guide, ANSYS Inc., 2011.

[6] G. H. Lee, Y. S. Bang, S. W. Woo, A. J. Cheong, Sensitivity Study on Turbulence Models for the Prediction of the Reactor Internal Flow, ICONE22-31255, Proceedings of the 22nd International Conference on Nuclear Engineering, July 7-11, 2014, Prague, Czech Republic.