Numerical Simulation for a Core Simulator of ACOP

Jun Ho Bae[a*], Dong Jin Euh[a], Tae-Soon Kwon[a]
Korea Atomic Energy Research Institute, P.O.Box 105, Yuseong, Daejon, 305-600, Korea
[*]*Corresponding author: bjh@kaeri.re.kr*

Min Gu Kang[b], Do-Hyeong Kim[b]
ANFLUX Co., Ltd ,Sinseong-dong, Yuseong, Daejon, 305-600, Korea

1. Introduction

An experimental facility, called ACOP, is being constructed in order to evaluate the flow and pressure distribution in the APR+ reactor core. The ACOP facility has a 1/5 linear scale of the prototype. The design is based on the conservation of Euler number which is a ratio of pressure drop to dynamic pressure under a sufficient turbulent flow condition [1]. The each fuel channel in the reactor core, which consists of 257 HIPER fuel assemblies, will be modeled as a 1/5-scale core simulator, which preserves the hydraulic characteristics of the HIPER fuel assembly. The present study is to develop and verify the design parameters applied to the core flow simulator by using a commercial CFD code.

2. Design of Core Simulator

Figure 1 shows the schematic diagram of the core simulator for the fuel channel of APR+. The core simulator has a venturi tube at the front part to measure the flow rate through the channel, and several perforated plates to make the hydraulic resistance in the core simulator. The total axial pressure drop of the core simulator is adjusted by a hole diameter of perforated plate, together with the throat diameter of venturi tube. The each side of the core simulator has several cross flow holes in order to conserve the cross flow characteristics between adjacent core simulators.

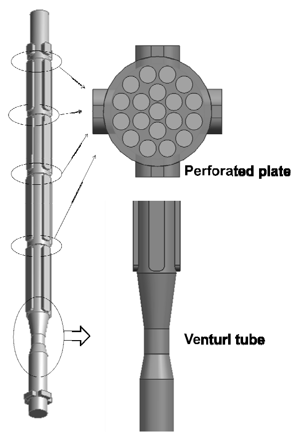

Fig. 1 Configuration of 1/5-scale core simulator

In order to preserve the flow characteristics, the reactor core of APR+ is linearly reduced with a scaling ratio of 1/5 and the velocity scale of 1/2 is applied. Table 1 shows a summary of the scaling parameters adapted in the ACOP facilities with respect to the APR+ reactor.

Table 1 Major scaling factor of ACOP

	APR+	ACOP	Remarks
Temperature, ℃	309.6	60	
Pressure, MPa	15.5	0.2	
Density, kg/m^3	705.8	983.2	1.393
Length, mm	4527.5	905.5	1/5
Flow Area, m^2	2.35e-2	9.41e-4	1/25
Velocity, m/s	4.936	2.463	1/2
Mass Flow, kg/s	81.78	2.278	1/50×1.393
Reynolds number	4.96e+5	1.79e+5	
Pressure Drop, KPa	150.75	52.5	1.393/4

3. Computational Procedure

A commercial CFD code of CFX version 12 has been used to evaluate the flow characteristics of the core simulator. The simulation type is steady state and the turbulent flow is simulated numerically using the k-ε model. The fluid flows into the core simulator with a uniform mass flow rate of 2.278 kg/s. At the downstream exit, the usual Neumann-conditions are applied for the fully-developed flow. The tetra-prism mesh was generated by using ICEM code as shown in Fig. 2. Approximately, 1.2 million nodes are used for the computation of a single core simulator.

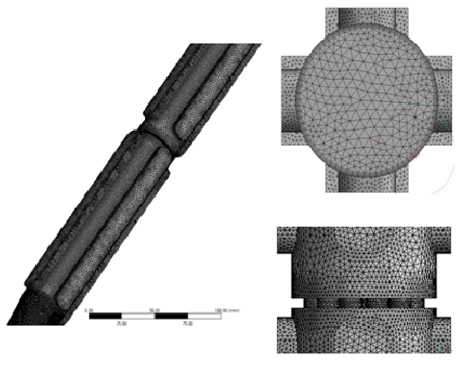

Fig. 2 Grid system of core simulator

3. Results

3.1 Computation for a single core simulator

The main purpose for the computation of single core simulator is to decide the hole diameter of perforated plate and the throat diameter of venturi tube, which makes the core simulator to have the target pressure drop. Through a series of computation for a simple tube channel with the venturi, the throat diameter of 19.2mm is revealed to satisfy the target pressure drop of 3.98kpa. In the same way with the venturi tube, the hole diameter of perforated plate are secured and it is decided to 5.35mm from the result of Table 2.

Table 2 Pressure drop of a 1/5-scale core simulator

Hole diameter (mm)	Pressure drop (kpa)	Target value (kpa)
5.3	55.4	52.5
5.35	52.6	
5.4	50.7	
5.5	45.9	

3.2 Computation for 3-core simulators

At the entrance region of reactor core, the different mass flow rate among fuel channels may induce the cross flow between fuel channels. In order to simulate the cross flow effect between fuel channels, the core simulator is designed to have the cross flow holes on each side of quadrant. The cross flow area of core simulator is designed to have a 1/25 scale ratio of the projected cross flow area of fuel channel. In the cross flow simulation, three parallel core simulators are connected by the cross flow holes and the boundary condition is same as that of single core simulator, except the mass flow inlet of each core simulator, which has the 90%, 100% and 110% of reference flow rate, respectively.

Figure 3 shows the velocity and pressure fields for 1/5-scale 3-core simulators. Due to the reduced flow area at the perforated plate, most of the pressure drop of core simulator is occurred when the flow passes through the perforated plate. In the downstream of perforated plate, the recirculation flow is generated at the side corner region by the abrupt expansion of flow area.

The cross flow characteristics of core simulator are observed along the axial location and compared with that of HIPER fuel assembly. Fig. 4 shows the variation of normalized cross flow rates for the HIPER fuel assemblies and 1/5-scale core simulators along the axial location of fuel channel. It is noted that the values of core simulator are displayed at the corresponding location on the HIPER fuel assembly.

It is revealed from the figure that the cross flow characteristics are similar for both cases of the HIPER fuel assemblies and 1/5-scale core simulators. The most of cross flow mixing are occurred within the 1/3 region from the entrance of fuel assembly. And the cross flow is not significant at the downstream of the simulator, where a sufficient uniformed flow condition is reached.

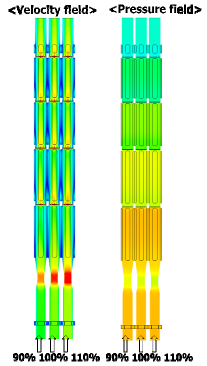

Fig. 3 Velocity and pressure fields for 3-core simulators

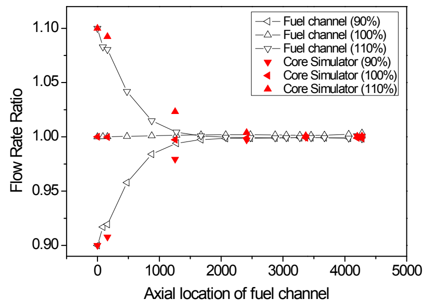

Fig.4 Comparison of cross flow ratio between the HIPER and 1/5-scale core simulator

4. Conclusion

From the numerical simulation for a single core simulator, the venturi diameter and the hole diameter of perforated plate was determined. The cross flow characteristics of 3-core simulators are revealed to be similar with those of the HIPER fuel assemblies. Most of cross flow is taken place at the lower part of the fuel assemblies and core simulators. Flow has a sufficient uniformed flow pattern at the downstream part.

Acknowledgement

This research has been performed as a part of the nuclear R&D program supported by the Ministry of Knowledge Economy of the Korean government.

REFERENCES

[1] D.J. Euh, Y.J. Youn, H. Bae, I.C. Chu, Y.J. Ko, T.S. Kwon, Design development of SCOP facilities for core flow and pressure distribution of SMART reactor, KNS autumn meeting, Korea, October, 2010.