Transactions of the Korean Nuclear Society Autumn Meeting
Jeju, Korea, October 21-22, 2010

Performance Evaluation of Core Simulator for SMART Reactor Core Flow and Pressure Distribution by Using STAR-CD

Yung Joo Ko [a*], Dong Jin Euh [a], Tae Soon Kwon [a]
[a] *Korea Atomic Energy Research Institute, P.O.Box 105, Yuseong, Daejeon, 305-600, KOREA*
[*] *Corresponding author: yungjoo@kaeri.re.kr*

Dong Ho Suh [b], Yeong Ryeol Baek [b]
[b] *# 915 JEI Platz Building, 459-11 Gasan-Dong, Geumcheon-Gu, Seoul 153-803*

1. Introduction

Recently, a conceptual design of the SMART reactor has been developed at KAERI. In order to verify the performance of the SMART design with respect to flow and pressure distribution, a core simulator is being designed with 1/5 reduced scale. The rod bundle type of the SMART fuel assembly is modeled by a simple geometry of structure with a preservation of hydraulic resistance. The core simulator includes the design for the measurement of inlet flow and outlet pressure. This paper describes the design feature and performance evaluation of the core simulator. The evaluation was performed with a proven CFD software, STAR-CCM+ (Ver. 5.02).

Table 1 Major scaling factor

Parameter	Scale	Ideal Ratio	Applied Ratio	Ratio
Length	l_R	1/5		Comment
Height	l_R	1/5		
Volumetric Flow	$V_R A_R$	1/25	1/25	At Core Boundary
Hydraulic Resistance	R_R	1	1	
Flow Area 1	$(l_R)^2$	1/25	1/25	At Core Boundary
Flow Area 2	$(l_R)^2$	1/25	1/25 x 72%	Inside Core
Velocity 1	V_R	1	1	At Core Boundary
Velocity 2	V_R	1	1 x 139%	Inside Core
Pressure Drop	$\rho_R (V_R)^2$	1.4	1.4	

2. Design of Core Simulator

In general, linearly reduced scaling ratio was adapted to preserve the hydraulic characteristics. The scaled ratio was decided to be 1/5 according to the scales of the SCOP facilities, a separate effect test for SMART hydraulic characteristics. The core simulator preserves a hydraulic resistance of the SMART fuel assembly. However, due to the consideration of the measurement inside core region, the core flow area was reduced further compared with the ideal value. By consideration of the flow area scaling distortion, the cross flow area was initially designed to be reduced with same ratio of axial flow area distortion. The major scaling factor is summarized in Table 1.

Fig. 1 shows a shape of the core simulator at the 1st step. Inlet flow rate and exit pressure will be measured at all the 57 core simulators representing SMART core. To measure the inlet flow rate, a venturi type of design was selected as shown in figure 1.The total axial pressure drop of the core simulator is adjusted by three orifices installed at the downstream part during a calibration process. The orifice type was selected as perforated plate having same size holes. The each side of the core simulator has several cross flow holes simulating cross flow between adjacent fuel assemblies.

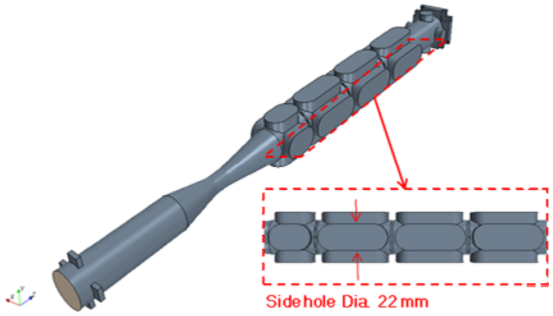


Fig. 1 A shape of 1/5-scale core simulator

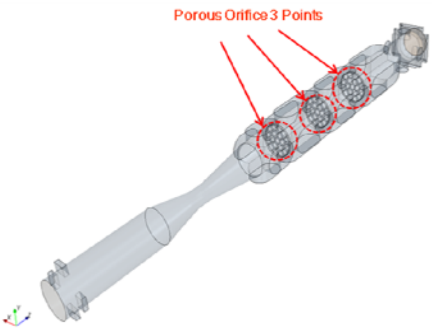


Fig. 2 Three orifices at the 3points

3. CFD Analysis

3.1 Pressure Drop at a single simulator

3.1.1 Mesh Generation and Analysis Method

Only the fluid volume of a core simulator is considered for current CFD analysis with polyhedral mesh, as shown in figure 3. The total pressure drop between the core inlet and outlet is controlled by using hole size of orifice, which was selected from the current simulation.

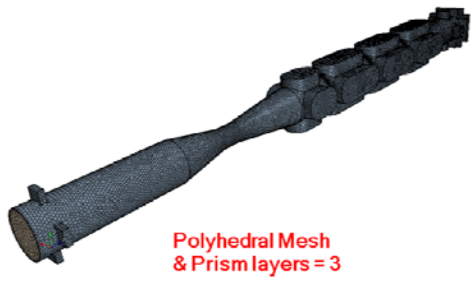


Fig. 3 CFD analysis with polyhedral mesh

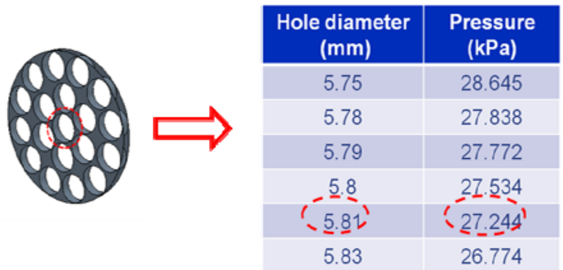
Table 2 shows the simulation conditions of CFD analysis. Uniform exit pressure conditions and inlet velocities were set as the boundary condition.

Table 2 The simulation conditions of CFD analysis.

Turbulence Model	Realizable K-ε two-layer model Two-layer all y^+ wall treatment
Inlet mass flow rate	2.048 [kg/s]
Material Property	ρ : 983.2 [kg/s] μ : 4.67e-4 [Pa·s]
Water Temperature	60 [℃]

3.1.2 Result

The results of the parametric studies for the hole diameter of porous orifice are shown in figure 4. The best fitted hole diameter was found to be about 5.81mm matching a desired pressure drop.



Hole diameter (mm)	Pressure (kPa)
5.75	28.645
5.78	27.838
5.79	27.772
5.8	27.534
5.81	27.244
5.83	26.774

Fig. 4 The results of the CFD analysis

3.2 Side Holes Sensitivity Tests

For the simulation of a cross flow between fuel assemblies, the cross flow holes were designed on both the sides of the core simulator. The side hole size of the simulator was selected as based on an equivalent diameter for the projection flow area of the SMART core in lateral direction. Three core simulators were configured as a parallel channel as shown figure 5. The cross flow characteristics were observed along the axial location. Periodic boundary condition was applied on the lateral boundary of each simulator. Figure 6 shows a new design of the cross flow holes.

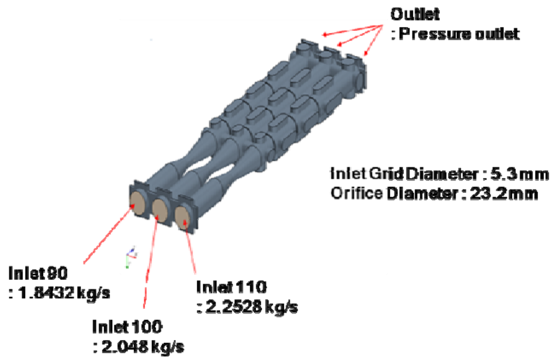


Fig. 5 A definition of the problem for side holes sensitivity tests

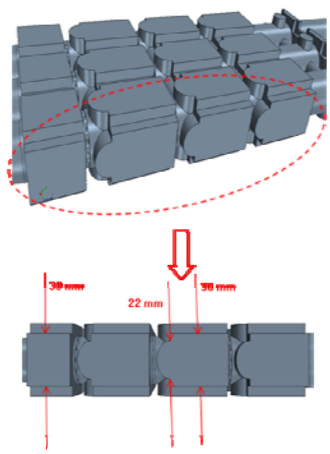


Fig. 6 A new design of the cross flow holes.

4. Conclusions

A design of the core simulator representing the hydraulic characteristics of SMART fuel assembly was developed in this paper. A CFD analysis was performed to evaluate the design characteristics of the core simulator for SMART. From a single simulator analysis, a desired the hole diameter of orifice representing the pressure drop of the SMART was determined. A new design of cross flow hole was proposed based on the observed characteristics of the core simulator.

REFERENCES

[1] T.S. Kwon and B.D. Chung, CFD Simulation of Bubble Mixing in a Square Duct, Transactions of the Korean Nuclear Society Autumn Meeting, 2007

[2] Y.J. Ko, H.Bae, D.J. Euh and T.S. Kwon, A CFD Simulation for a Design of Core Flow Simulator for SMART, Transactions of the Korean Nuclear Society Spring Meeting, 2010

[3] H. Bae, D.J. Euh, B.S. Shin, Y.J. Ko and T.S. Kwon, Design Concept of a 1/5-Scale Core Simulator, Transactions of the Korean Nuclear Society Spring Meeting, 2010