A CFD Simulation for a Design of Core Flow Simulator for SMART

Yung Joo Ko[a*], Hwang Bae[a], DongJin 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*

1. Introduction

The conceptual design of the SMART reactor has been developed at KAERI since 1997, for the generation of electric power and also for seawater desalination. In order to verify the performance of the SMART design in respect to flow and pressure distribution, an experimental test facility named SCOP has been developed. The core flow distribution will be measured by using a simulator preserving the flow characteristics of the prototype, which will be utilized for the evaluation of the core thermal margin. The present study is to develop and verify design parameters applied to the core flow simulator by using proven CFD software. A CFX version 11 has been used to evaluate the flow characteristics of the core simulator using a 1/5 length scale of SMART core assembly. To conserve flow similarity, the Euler number which is expressed as the ratio of a differential pressure to a dynamic pressure should be similar under a sufficient turbulent flow condition.

2. Design Concept of Core Simulator

Fig. 1 shows a schematic of the design of the core simulator for the fuel assembly of SMART. Inlet flow rate and exit pressure will be measured at all the 57 core simulators representing SMART core. To measure the axial flow rate of each fuel assembly, a venturi flow meter was installed at the lower part of the core simulator. The total axial pressure drop of the core simulator is adjusted by an orifice during a calibration process. The each side of the core simulator has several cross flow holes simulating cross flow between adjacent fuel assemblies.

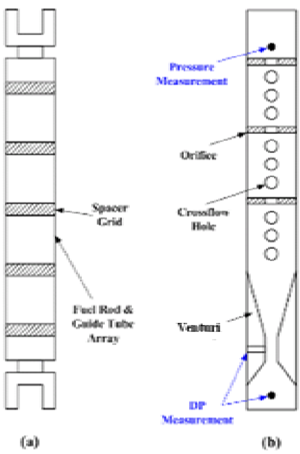


Fig. 1. SMART-330 fuel assembly and its simulator

Fig. 2. 1/5-scale core simulator

Fig.2 shows the 1/5-scale core simulator. The major scaling factor is summarized in Table 1.

Table 1 Major scaling factor

	Scaling Law		Comment
Parameter	Scale	Ratio	
Length	l_R	1/5	
Height	l_R	1/5	
Flow Area	$(l_R)^2$	1/25	At Core Boundary
Velocity	V_R	1	At Core Boundary
Volumetric Flow	$V_R A_R$	1/25	
Pressure Drop	$\rho_R (V_R)^2$	1.35	

3. CFD Analysis

3.1 Single Channel Test

3.1.1 Mesh Generation and Analysis Method

Only the fluid volume of a core simulator is considered for current CFD analysis with tetra mesh, as shown in Fig. 3. The total pressure drop between the core inlet and outlet is controlled by using orifice size, which was selected from the current simulation. From a 20mm sized orifice, the analysis was started to a desirable size inducing the target pressure drop in the core.

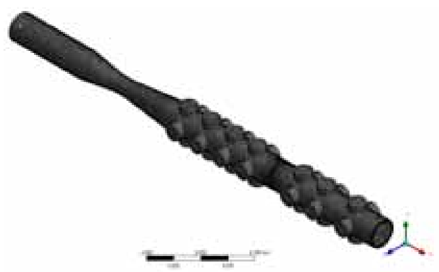


Fig. 3 A flow model of core simulator

Fig. 4 shows a definition of the current problem. Uniform exit pressure conditions and inlet velocities were set as the boundary condition. At the boundary of the side hole, a no slip condition was applied. The number of meshes was about 1,000,000. A sensitivity analysis for the mesh size was also performed.

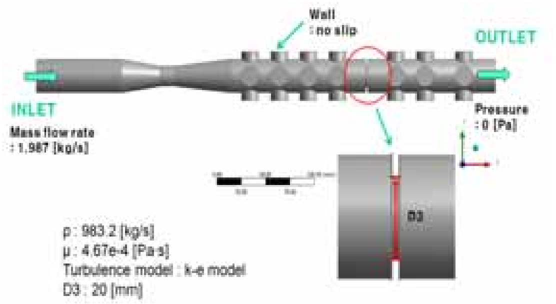
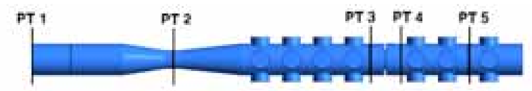


Fig. 4. A definition of the problem for a single test

3.1.2 Result

The results of the parametric studies for the orifice diameter are shown in Fig. 5. The best fitted orifice diameter was found to be about 18.5mm matching a desired pressure drop.



계산 순서	D3(mm)	PT1-PT5	PT1-PT2	PT3-PT4	P1	P2	P3	P4	P5
1	20	27570	62940	29080	27130	-35240	17370	-10540	-419.5
2	15	97873	62945	110675	96119	33747	83858	-30535	-1598
3	19	34310	62932	38704	33680	-28696	23844	-13422	-593
4	18.5	38700	62946	44634	37965	-24411	28005	-15071	-688

Fig. 5. The results of the CFD analysis

3.2 2X2 Channel Test

For the simulation of a cross flow effect between fuel assemblies, the cross flow holes were designed on both the sides of the core simulator. The hole size of the simulator was selected as an equivalent diameter for the projection flow area of the SMART core in lateral direction. Four core simulators were configured as a parallel channel as shown in top view of Fig. 6. The cross flow characteristics were observed along the axial location. Periodic boundary condition was applied on the lateral boundary of each simulator.

The possible asymmetry of the axial flow may induce a cross flow. Most of the cross flow mixing would be occurring at the inlet region of the core simulator. Meanwhile, the cross flow is not significant at the downstream of the simulator, where a sufficient developed flow condition is reached. In this simulation, the axial velocity difference was tested at about 20% in axial direction. Fig. 7 shows calculated results at the interfaces between each core simulator along the flow proceeds.

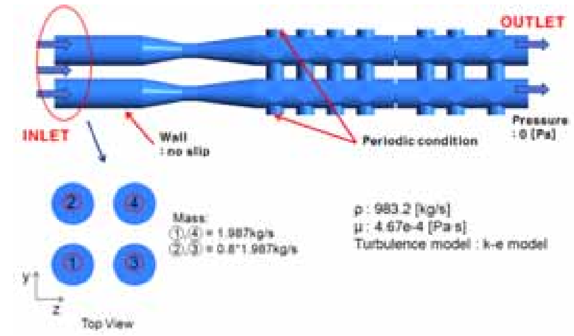
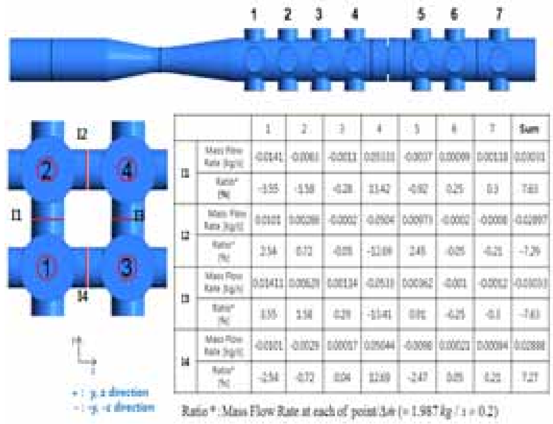


Fig. 6. A definition of the problem for 2X2 channel test



		1	2	3	4	5	6	7	Sum
I1	Mass Flow Rate [kg/s]	-0.0141	-0.0061	-0.0011	0.05111	-0.0037	0.00009	0.00118	0.03031
	Ratio* [%]	-3.55	-1.58	-0.28	13.42	-0.92	0.25	0.3	7.63
I2	Mass Flow Rate [kg/s]	0.0101	0.00286	-0.0002	-0.0504	0.00971	-0.0002	-0.0008	-0.02897
	Ratio* [%]	2.54	0.72	-0.05	-12.69	2.45	-0.05	-0.21	-7.29
I3	Mass Flow Rate [kg/s]	0.01411	0.00629	0.00114	-0.0533	0.00362	-0.001	-0.0012	-0.03031
	Ratio* [%]	3.55	1.58	0.29	-13.41	0.91	-0.25	-0.3	-7.63
I4	Mass Flow Rate [kg/s]	-0.0101	-0.0029	0.00017	0.05044	-0.0098	0.00021	0.00084	0.02888
	Ratio* [%]	-2.54	-0.72	0.04	12.69	-2.47	0.05	0.21	7.27

Ratio* : Mass Flow Rate at each of point/$\Delta\dot{m}$ (= 1.987 kg / s × 0.2)

Fig. 7 Cross flow mixing at interfaces

4. Conclusions

A CFD analysis was performed to evaluate the design characteristics of the core simulator for SMART. From a single channel analysis, a desired orifice size representing the pressure drop of the SMART was determined. The 2X2 channel test shows cross flow mixing behaviors along the channel. As a result, the 20% difference of inlet flows between channels was suppressed as 7.5% at the flow outlet region. An effort for the similarity for the cross flow is planned as a further study.

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