# Cross Flow Characteristics of the Core Simulator in SMART Reactor Flow Distribution Test Facility

J. Yoon [a*], Y.I. Kim [a], Y.J. Chung [a], W.J. Lee [a]
[a]*Korea Atomic Energy Research Institute, SMART development Dept., 1045 Daedeok-daero, Daejeon*

## 1. Introduction

To identify the flow characteristic of SMART reactor, flow distribution model test and numerical simulation has being performed in KAERI. Fuel assemblies are simulated by using simulators because of the complexity. The geometries of the core in SMART reactor and simulator are different, but its similarity is maintained such as the ratio of pressure drop in vertical and cross direction. There are cross flow holes in each core simulator to reproduce the cross flow of SMART fuel assemblies. So it is necessary to know the flow characteristics of cross flow.

In this paper, numerical analysis is performed to confirm the cross flow characteristics with the variation of inlet flows and cross flow areas.

## 2. Methods and Results

### *2.1 Geometry and Condition*

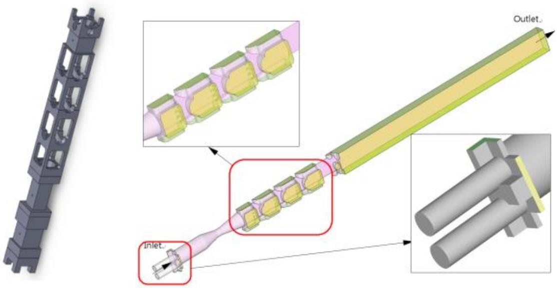



Fig. 1. Geometry of Cross Flow Holes in Core Simulator

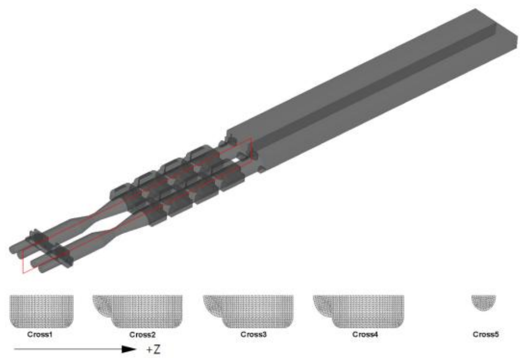



Fig. 2. Geometry of Core Simulators for CFD Analysis

The geometry of core simulator is shown in Fig. 1. The simulator is consisted of a venturi for flow measurement and three perforated plates for axial loss coefficient. Also there are several cross flow holes on the side of simulator. The geometry for numerical analysis is shown in Fig. 2. Two simulators are connected with cross flow holes in parallel and each simulator has its own inlet to simulate the condition with different mass flow rates. Then flow distribution in cross flow holes, pressure drop in whole simulator and influence of cross flow area change are investigated.

### *2.2 Methodology*

#### *2.2.1. Numerical Simulation*

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad (1)$$

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial \mathrm{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_j}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \qquad (2)$$

$$\frac{\partial}{\partial x_i}(\rho u_i k) = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) + G_k + G_b - \rho\varepsilon \qquad (3)$$

$$\frac{\partial}{\partial x_i}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_i}\left(\frac{m_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k + (1 - C_{3\varepsilon})G_b\right) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad (4)$$

Numerical analysis is performed by FLUENT 12.0[1]. Ignoring gravity, three-dimensional, 1/2 axisymmetric, steady-state flow, and constant properties such as density and viscosity are assumed. The continuity, momentum equation, and standard $k-\varepsilon$ model which is one of the turbulence models used in this study are shown in Eq. (1) ~ (4).

Mesh sensitivity and turbulence model test are performed with single core simulator. Standard, Realizable and RNG $k-\varepsilon$ and SST $k-\omega$ model are used in turbulence model test, and the standard wall function is applied for $k-\varepsilon$ series, and the low-Re correction option is not used for SST. Also a straight pipe(20D) is added in a downstream of simulator to remove the influence of downstream.

### *2.3 Result*

Mesh sensitivity test results using realizable $k-\varepsilon$ are shown in Table. 1. In all cases, the deviation of the differential pressure compared with a maximum value is less than 1.5%.

Turbulence model test results are shown in Table. 2 and Fig. 4. The deviation between experimental and numerical result shows the smallest value in Realizable

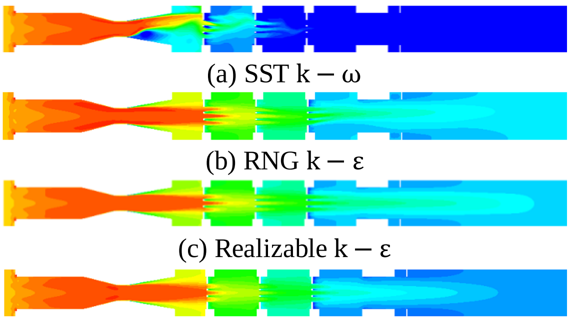
$k-\varepsilon$. In RNG $k-\varepsilon$ and SST $k-\omega$, the flow pattern is unstable because the flow shows a transient behavior. So these two models are not appropriate in the cases.

Table. 1. Mesh Sensitivity Test Result

| Case | Mesh | ΔP(Pa) | Deviation (vs Case A05) |
|---|---|---|---|
| A01 | 1,397,043 | 31,506 | 1.31% |
| A02 | 2,334,704 | 32,006 | 0.26% |
| A03 | 2,826,408 | 32,288 | 1.14% |
| A04 | 4,011,570 | 31,709 | 0.67% |
| A05 | 5,084,766 | 31,923 | - |

Table. 2. Turbulence Model Test Result

| Case | Turb. Model | ΔP(Pa) | Deviation (vs Exp.) | Remark |
|---|---|---|---|---|
| B21 | Real | 32,006 | 8.68% | |
| B22 | RNG | 35,191 | 0.4% | Unstable |
| B23 | STD | 38,427 | -9.63% | |
| B24 | SST | 61,586 | 75.71% | Unstable |



(a) SST $k-\omega$
(b) RNG $k-\varepsilon$
(c) Realizable $k-\varepsilon$
(d) Standard $k-\varepsilon$

Fig. 3. Pressure Distribution for Turbulence Model

With the inlet mass flow deviation of 0, 5, 10, 15, and 20% and the cross flow area variation of 100, 90, 70 and 50%, cross flow and pressure drop variations are calculated. In the case of 10% flow imbalance, the mass flow rate normalized with an average axial flow rate is shown in Fig. 4. In the figure, X-axis represents the dimensionless height which is normalized with the height where the cross flow is occurring. As the normalized height is bigger, the average axial flow rate become constant.

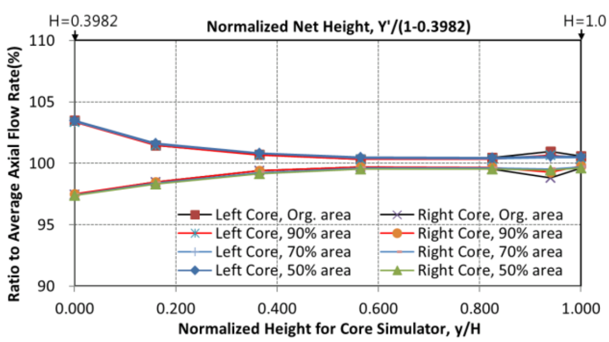



Fig. 4. Cross Flow Distribution for 2 Parallel Core Simulators

Also as shown in Fig. 5, even if the cross flow area is reduced by up to 50%, cross flow differences are a maximum of 0.3% or less. This result means that even though the cross flow area is reduced, cross flow itself is not affected as the cross flow area is larger enough.

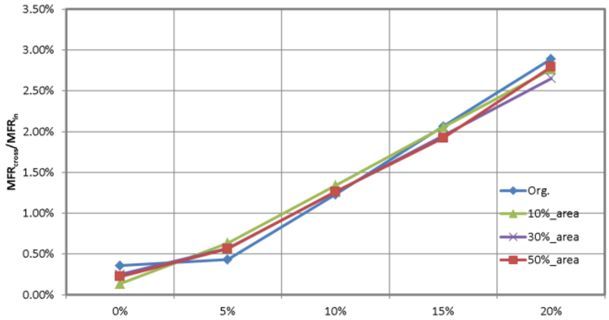



Fig. 5. Mass Flow Rate for 2 Parallel Core Simulator

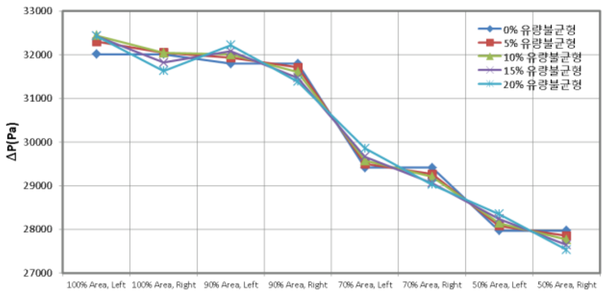



Fig. 6. Pressure Difference for 2 Parallel Core Simulator

The pressure drop in each simulator is shown in Fig. 6. As the cross flow area is reduced, the pressure drop in whole core simulator is reduced. With the reduction of cross flow area, sudden expansion effect around perforated plates is decreased. So even though total inlet mass flow is same, if the cross flow area is reduced, the pressure drop is reduced.

## 3. Conclusions

Numerical analysis is performed to investigate the cross flow characteristics of core simulator. In case the flow imbalance is constant, the cross flow area doesn't significantly affect the cross flow. Also as the flow imbalance is increased, the cross flow is increased.

## Acknowledgement

This study has been performed under a contract with the Korean Ministry of Educational Science and Technology.

## REFERENCES

[1] FLUENT 12.0 Theory Guide, ANSYS Inc., New Hampshire, U.S.A., 2009.
[2] I.E. Idelchik, Handbook of Hydraulic Resistance, 3rd Edition, Begell House, pp. 239-329, 2005.
[3] S. Gencay, A. Tapucu, Cross-flow Between Identical Subchannels Caused by a Severe Blockage, Nuclear Engineering and Design, Vol. 127, pp. 33-45, 1991.
[4] M. Osakabe, H. Futamata, Effect of Inserted Rod and Cross Flow on Top Flooding of Pipe, International Journal of Multiphase Flow, Vol. 22, No. 5, pp. 883-891, 1996..