Transactions of the Korean Nuclear Society Autumn Meeting
Yeosu, Korea, October 25-26, 2018

CFD Calculation for Investigating Geometrical Distortion of Reactor Flow Distribution Test Facility for PGSFR

Woo Shik Kim*, Dong-Jin Euh
Korea Atomic Energy Research Institute, 111, Daedeok-Daero 989Beon-Gil, Yuseong-Gu, Daejeon, Republic of Korea
**Corresponding author: wooshik@kaeri.re.kr*

1. Introduction

A prototype Generation IV Sodium-cooled Fast Reactor (PGSFR) is being developed by KAERI in Korea. This is a pool-type reactor in which the major components are installed inside the reactor vessel. It has four IHXs (Intermediate heat exchangers), four DHXs (Decay heat exchangers), and two PHTS pumps (Primary heat transport system pumps) inside the reactor vessel. In such a pool type reactor, the flow distribution and pressure drop strongly depend on the arrangement and geometrical configuration of the components inside the reactor vessel. [1, 2] It is important to evaluate hydrodynamic characteristics for the optimum thermohydraulic design of the reactor. In the absence of a mathematical model for the flow distribution in such a complex geometry, experimental techniques are an alternative way to reveal the flow behavior. [3]

For this concern, the experimental facility using a one-fifth down-scaled model with water flow at room temperature was constructed to observe the flow characteristics. [4, 5] The purpose of the hydrodynamic test with the down-scaled model are 1) to evaluate the degree of non-uniformity and asymmetry of the flow in the reactor vessel, 2) to estimate the flow rate distribution of the fuel assemblies in the core, and 3) to evaluate the flow resistance across complex internal structures.

There are 313 fuel and non-fuel assemblies in the reactor core of PGSFR, which constitute 12 groups according to the functional classification. Among them, each fuel assembly from group 1 to 9 (112 fuel assemblies) has 217 fuel rods inside a hexagonal duct housing. Most coolant flow from the inlet plenum passes through these fuel assemblies of groups 1 through 9. Thus the flow path through the fuel assemblies belonging to groups 1 through 9 are simulated in the present test facility. The complex flow path of the fuel assembly was simulated by a single flow path. The fuel assembly simulators consist of a receptacle, variable-resistance rotating orifice spool, venturi tube, and connection pipes.

To setup the fuel assembly simulators in the reactor vessel, three pressure impulse line per fuel assembly simulator should be drawn out and connected to the pressure transmitters without any significant interference of the reactor flow. To minimize the perturbation due to instrumentation, a total of 336 pressure impulse lines were guided inside the CRDM guide tubes and drawn out from the top of the test section as shown in Fig. 1.

To investigate the effect of the pressure impulse line on the reactor flow, a CFD calculation had been performed prior to construction of the test facility.

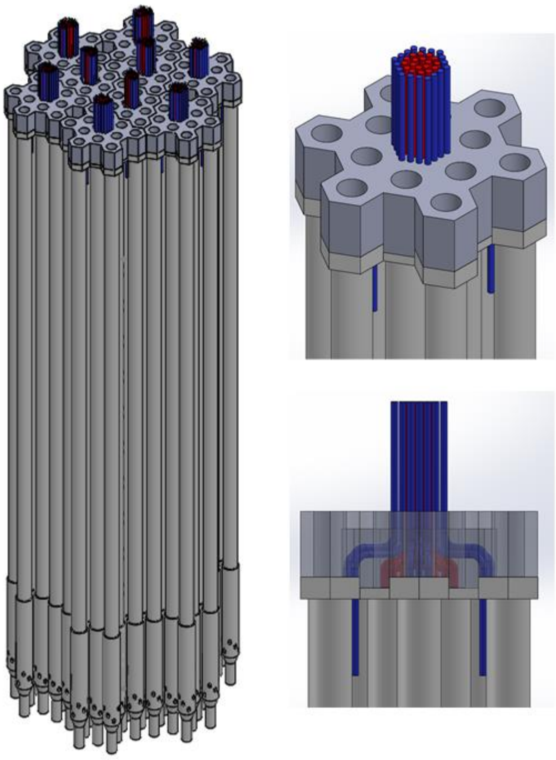


Fig. 1. Drawn out configuration of pressure impulse lines

2. Calculation Domain

Fig. 2 shows the computational domain of the present CFD calculation. In the present CFD analysis, each flow path of 112 fuel assemblies was postulated as a porous medium for the computational efficiency, and the flow resistance through the porous region was estimated through a separate CFD calculation. The considered region of the analysis includes all internals of a reactor vessel such as the IHXs, UIS and pump inventories, as well as the core. The total number of grid cells for the analysis was about 30 million. Fig. 3 shows the grid deployments of the whole domain. To investigate the effect of the pressure impulse lines, two different calculation domains were employed. For the first case (Case 1), the calculation domain without pressure impulse line, which corresponds to the prototype reactor, was used as reference case. In the second case (Case 2), each group of pressure impulse lines was treated as a single rod, which corresponds the test model. Since the dimension of the pressure impulse line tube bundle is similar to the diameter of the CRDM guide tube, the CRDM guide tubes are extended to downward direction to reach the upper surface of the core exit in the second calculation case.

The boundary conditions used in this analysis are summarized in Table 1.

Table 1 Boundary conditions

Boundary Type	Name	Value
Mass flow Inlet	InletUp	23.235 (kg/s)
	InletDown	22.235 (kg/s)
Pressure Outlet	OutletUp	0 (Pa)
	OutletDown	0 (Pa)
Top Boundary	Slip Wall	

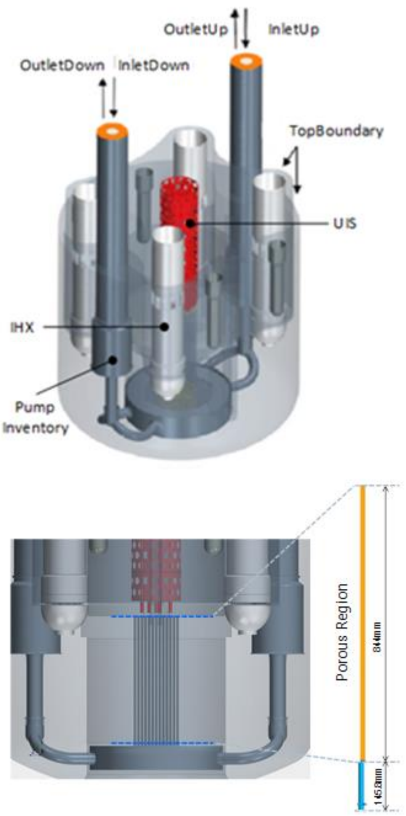


Fig. 2. Calculation Domain

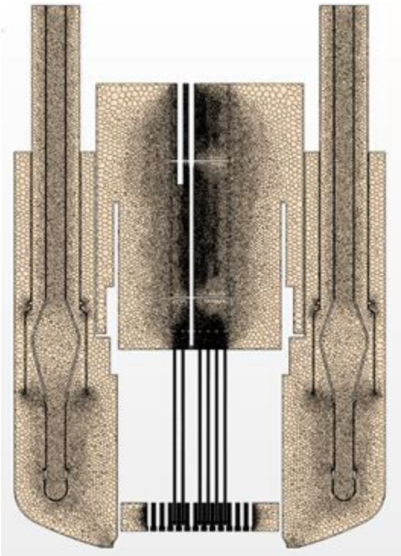
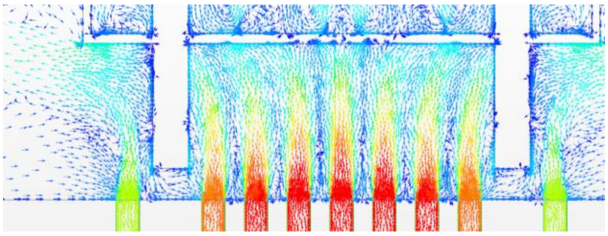


Fig. 3. Grid Generation

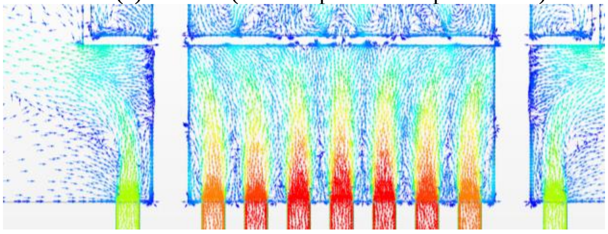
3. Results

Fig. 4 shows the velocity vector at the core exit region. There exist gap flow between CRDM guide tube and the core exit in Case 1, but the magnitude of the velocity through the gap is relatively low compared to velocity for overall domain. As shown in Fig. 5, overall velocity magnitude distribution showed no significant difference between two cases.

Fig. 6 and 7 describe the group averaged mass flow rate pressure drop through the fuel assembly simulators, respectively. From these results, it can be verified that the existence of the pressure impulse line has little effect on the flow and pressure distribution of the core.

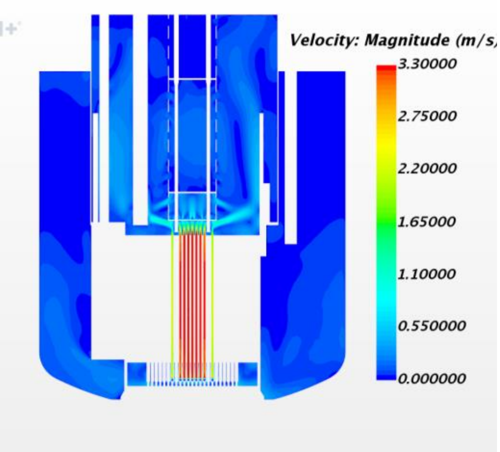


(a) Case 1 (without pressure impulse lines)

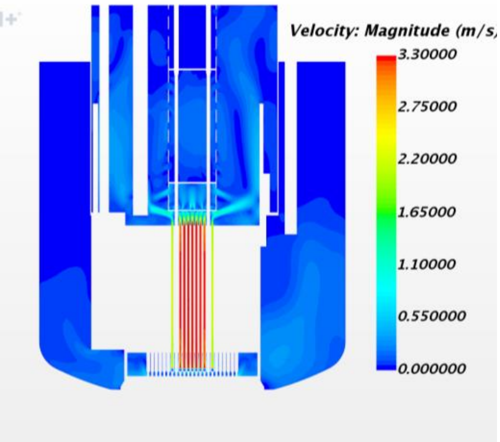


(b) Case 2 (with pressure impulse lines)

Fig. 4. Velocity Vector at Core Exit



(a) Case 1 (without pressure impulse lines)



(b) Case 2 (with pressure impulse lines)
Fig. 5. Velocity distribution

3. Conclusions

CFD calculations of the reactor flow distribution test section, which has been constructed for the simulation of the flow behavior of the PGSFR, were conducted to investigate the effect of geometrical distortion of the test section. To simulate the pressure impulse lines from the fuel assembly simulators, which are inevitably located between the core exit and CRDM guide tube, a single rod geometry was employed to the calculation domain and the result of the CFD analysis was compared to the reference case.

Even there exist gap flow between the core exit and CRDM guide tubes for the reference case, the velocity magnitude of the gap flow was relatively small. Furthermore the global velocity distribution, mass flow rate through the fuel assembly simulator, and pressure drop through the fuel assembly simulator were reveal to be little affected by the existence of the pressure impulse lines.

ACKNOWLEDGEMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIP; No. 2012M2A8A202568, 2012M2A8A2025635).

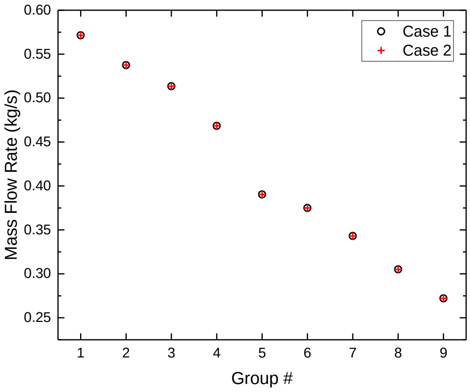


Fig. 6. Group averaged mass flow rate through fuel assembly simulator

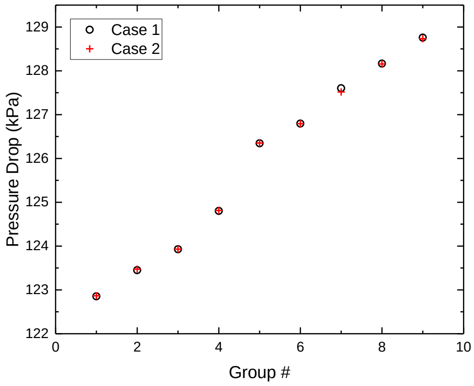


Fig. 7. Group averaged pressure drop through fuel assembly simulator

REFERENCES

[1] J. Yoo, J. Chang, J. Y. Lim, J. S. Cheon, T. H. Lee, S. K. Kim, K. L. Lee, H. K. Joo, Overall System Description and Safety Characteristics of Prototype Gen IV Sodium Cooled Fast Reactor in Korea, Nuclear Engineering and Technology, Vol. 48, pp. 1059-1070, 2016.

[2] J.W. Han, S.K. Choi, S.O. Kim, E.K. Kim, D. Kim, J. Hong, H.Y. Ye, S. Yeom, S. Ryu, J. Yoon, S.R. Choi, J.S. Park, T.H. Lee, Overview of NSSS Fluid System Design of PGSFR, Transactions of the Korean Nuclear Society Spring Meeting, Jeju, May 12-13, 2016.

[3] G. Hetsroni, "Use of Hydraulic Models in Nuclear Reactor Design," Nuclear Science and Engineering, 28, pp. 1-11, 1967.

[4] W. S. Kim, S. K. Chang, Y. J. Yoon, J. K. Park, H. S. Choi, J. W. Kim, S. H. Hong, H. S. Seol and D. J. Euh, Construction Report of SFR Reactor Flow Distribution Test Facility, KAERI/TR-6950/2017, KAERI, 2018.

[5] W. S. Kim, S. K. Chang, Y. J. Yoon, J. K. Park, H. S. Choi, J. W. Kim, S. H. Hong, H. S. Seol and D. J. Euh, Report of SFR Reactor Flow Distribution Test, KAERI/TR-6952/2017, KAERI, 2018.