CFD Analysis for Verification of SFR Reactor Flow Distribution Test Facility

Woo Shik Kim*, Seok-Kyu Chang and Dong-Jin Euh
Korea Atomic Energy Research Institute (KAERI), 111 Daedeok-daero 989beon-gil, Yuseong-gu, Daejeon, Korea
**Corresponding author: wooshik@kaeri.re.kr*

1. Introduction

PGSFR is a pool type reactor in which the major components are installed inside the reactor vessel. PGSFR has four IHXs, four DHXs, and two PHTS pumps inside the reactor vessel [1]. The flow distribution will show multi-dimensional phenomena which depend on the geometrical configurations of the components. The evaluation of the flow distribution in the reactor and the pressure drop across each major component is required for licensing of a new reactor. The reactor flow distribution has high phenomenological importance in safety analysis, but the knowledge level such as experimental database is relatively low. Therefore the evaluation and validation of the design and safety performance have utmost importance.

In 'Thermal-Fluid Validation Test of Prototype Gen-IV Sodium Cooled Fast Reactor' project, test facility for investigating the flow characteristics inside the reactor vessel is being constructed. In the present test facility, reactor vessel and the main in-vessel components are linearly reduced at a scaling ration of 1/5 and water is used as the working fluid. In the reactor vessel of the test facility, the main components such as core, UIS, PHTS pump and IHX are installed. The exteriors of the main components are conserved following the scaling ratio of 1/5, but the internal flow paths inside the fuel assemblies and IHXs are uniquely designed for precise measurement of flow rate and conserving the pressure drop characteristics. Pressure and flow rate at the main measuring points could be obtained by the test. However overall pressure distribution and velocity distribution in the reactor vessel are not easily quantified, so the separate CFD analysis is needed. In the present analysis, adequate CAD and grid for the flow simulation inside the reactor vessel of the test facility were modelled, and the pressure distribution and velocity distribution inside the reactor vessel were calculated.

2. Computational Domain and CFD Code

Figure 1 shows the computational domain for the flow simulation of the flow distribution test facility. In the test facility, the coolant is supplied from two external pumps to two inlet pipes of the test section and flowed out through two outlet pipes. In the present CFD calculation, mass flow inlet boundary condition was applied at the inlet pipes from each pump, and pressure outlet boundary condition was set at two outlet pipes.

There are 451 fuel assemblies in a reactor core of the PGSFR, which are divided into 12 groups according to the functional classification. Among them, only 112 fuel assemblies belong to the group 1 through 9 are designed for the test facility, which are named fuel assembly simulators, because most of mass flow rate is concentrated on them. The coolant flow through the other fuel assemblies is neglected in the test, so that the flow paths of them are simply blocked. The flow path of the fuel assembly simulator for the test is composed of venturi tube and orifice holes. For the computational efficiency, in the present CFD calculation, this complex flow path is simplified as the porous region with having appropriate flow resistance whose value is estimated by the separate calculation [2].

The free surfaces of the liquid at the hot pool inside the redan and cold pool outside the redan were treated as the slip boundary conditions. The level of the free surfaces are determined from the similarity analysis.

In the present CFD calculation, STAR-CCM+[3], a commercial CFD code, was used. Total 30,996,670 cells were generated for the calculation.

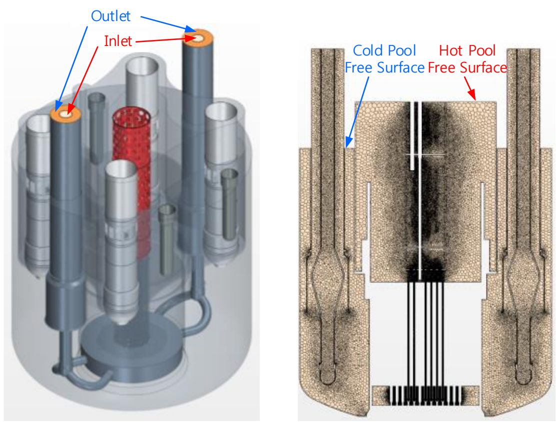


Fig. 1 Computational Domain and Grid

3. Calculation Results

Figure 2 shows the pressure distribution along the main flow path from the inlet to the outlet in the vessel. Total pressure drop of the test facility was calculated as 142 kPa, which gives useful information for the test loop design especially for the estimation of the pump capacity. The pressure drop from the inlet plenum (point 6) to the center of the core outlet (point 7) was revealed to be 118 kPa, which is less than target value (125.6 kPa) from the similarity analysis. The difference between the target and the calculated values is originated from the spatial pressure distribution at the outlet of the core, which will be discussed next paragraph. The pressure drop between the hot pool and cold pool (point 12 – 13) was calculated as 3.66 kPa, which is almost identical to the target value (3.71 kPa).

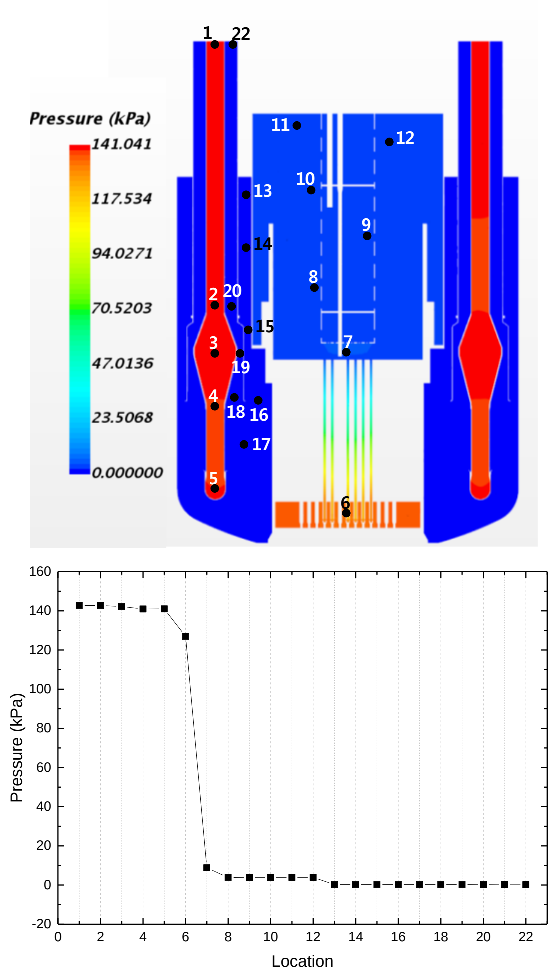
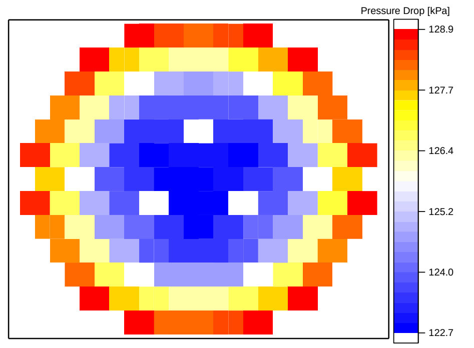


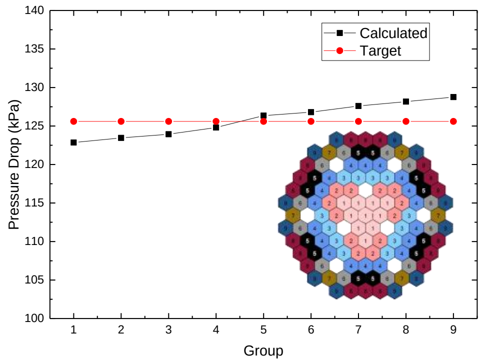
Figure 2 Pressure Distribution inside Reactor Vessel

The calculated pressure drop across each fuel assembly simulator is depicted as shown in Fig. 3(a), and the averaged values by the group are compared with the target value as shown in Fig. 3(b). The calculated pressure drop is lower than the target value at the center region while it becomes gradually higher as approaching to the outer region and finally exceeded the target value at the group 5. This phenomenon may be caused from some reasons such as geometrical configuration of the fuel assembly simulators and the flow conditions at the upstream (inlet plenum) and the downstream (UIS).

Figure 4 shows the mass flow rate distribution of the fuel assembly simulators. The flow rate across the fuel assembly simulators in the same group showed no significant difference. The calculated flow rate was well distributed in accordance with the degree of the target flow rate as shown in Fig. 4(a), however overall flow rate tends to exceed the target value as shown in Fig 4(b). As mentioned section 2, the flow paths through the fuel assemblies for the group 10 to 12 are not simulated

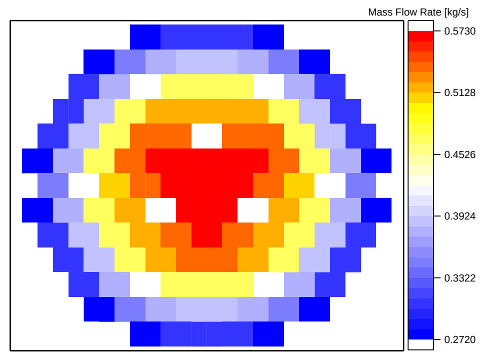


(a) Each Fuel Assembly Simulator

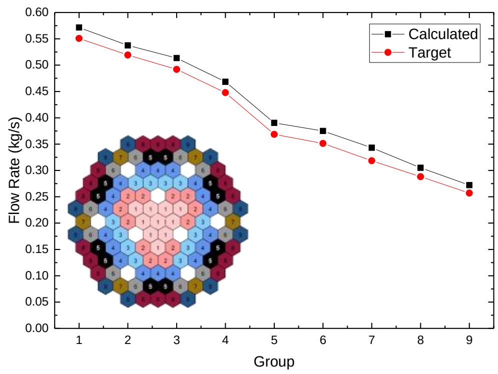


(b) Averaged by Group

Figure 3 Pressure Drop across Fuel Assembly Simulators



(a) Each Fuel Assembly Simulator



(b) Averaged by Group

Figure 4 Mass Flow Rate across Fuel Assembly Simulators

in the present calculation in the same manner as the test facility. The mass flow rate introduced into the inlet pipes includes that for the non-fuel assembly simulators and the leakage flow, whose portion is about 5% of the total mass flow rate. This amount of the mass flow rate is naturally added to the fuel assembly simulators, which resulted in the overall increase in the mass flow rate in the present calculation.

4 Conclusion

In the present study, the CFD analysis for the SFR reactor flow distribution test facility have been performed. From this calculation, overall pressure drop as well as the pressure distribution in the test facility could be estimated. The mass flow rate and pressure drop across each fuel assembly simulators have been fully obtained and compared with the target values. The experimental data base is planned to be obtained in near future, and the results will be compared with the present calculation results.

ACKNOWLEDGMENTS

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

REFERENCES

[1] Description of PGSFR System, SFR-000-SP-403-001, Rev.01, KAERI, 2016.
[2] S.K. Chang, W.S. Kim, I.C. Chu and E.J. Euh, CFD Analysis for the Verification of Test Vessel for the Flow Simulation of PGSFR, Proceedings of the 11th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, Operation and Safety (NUTHOS-11), N11P0396, Oct.9-13, 2016, Gyeongju, Korea.
[3] Star-CCM+ Documentation, Ver. 10.04, CD-adapco, 2015.