Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 26-27, 2023

Development of Automated System for Numerical Investigation of SFR Fuel Assembly

Yohan Jung*, Jongtae Kim, Jonggan Hong
Korea Atomic Energy Research Institute, 111, Daedeok-daero 989 Beon-Gil, Yuseong-gu, Daejeon, Korea
**Corresponding author: yhjung81@kaeri.re.kr*

1. Introduction

SFR (Sodium-cooled Fast Reactor) is one of a next-generation nuclear power reactor that can effectively process spent nuclear fuel. Since the SFR uses liquid sodium as a coolant and sodium has a high boiling temperature, high operating pressure is not required, and hence, the design pressure for components inside the reactor is nearly atmospheric. In addition, since sodium has high thermal conductivity, the SFR can effectively remove the heat generated in the reactor.

The fuel assembly of the SFR generally consists of several fuel pins, wires and a hexagonal duct. The fuel pins are arranged in a triangular pitch and wound with helical wire spacers. Because these wire spacers improve the mixing of the coolant, they have a great influence on the flow and heat transfer characteristics. It is important to predict and analyze the flow and temperature distributions for fuel safety.

Various fuel assembly analysis codes have been developed to predict the flow and temperature distribution in the subchannels [1-2] and many experimental studies on fuel assemblies have been conducted for the core thermal-hydraulic design of the SFR [3-4]. Recently, Many CFD (Computational Fluid Dynamics) studies have been performed due to the development of computational resources [5-6]. Since detailed flow phenomena that are difficult to investigate in experiments can be examined, numerical studies using CFD are being conducted more and more. CFD simulation largely consists of pre-processing, simulation, and post-processing, and it is necessary to automate the entire complex process of CFD.

In this study, development of automated system for numerical investigation of SFR fuel assembly is presented. Numerical methods and validation results for wire-wrapped 37-pin fuel assembly are addressed.

2. Benchmark descriptions

The experimental benchmark is the Toshiba 37-pin SFR fuel assembly [7]. The objectives of the experiment are to measure the buoyancy effect on pin bundle heat transfer and to verify the validity of the analysis code (COBRA-IV-I) under low flow rates typical of natural circulation conditions. A series of experiments were conducted with the sodium heat transfer test loop at Toshiba Nuclear Engineering Laboratory. The loop simulates a heat transfer system for a typical LMFBR (Liquid Metal Fast Breeder Reactor) plant and a test assembly is installed in the primary loop. A layout view of the test assembly is shown in Fig. 1.

The test assembly contains 37 simulator pins within a hexagonal can, which has 50.4 mm flat to flat dimensions. The 6.5 mm diameter simulator pins are wound with wire-spacers 1.32 mm in diameter and the wire-wrap pitch is 307 mm. The triangular pitch between pins is 7.87 mm. Each simulator pin contains an electrical heated element to generate heat and numerous thermocouples were embedded in the simulator rods to measure heat transfer characteristics with the pin bundle. The total length of the fuel assembly is 3043 mm, and the heated length is 930 mm. The detailed geometry parameters are listed in Table 1.

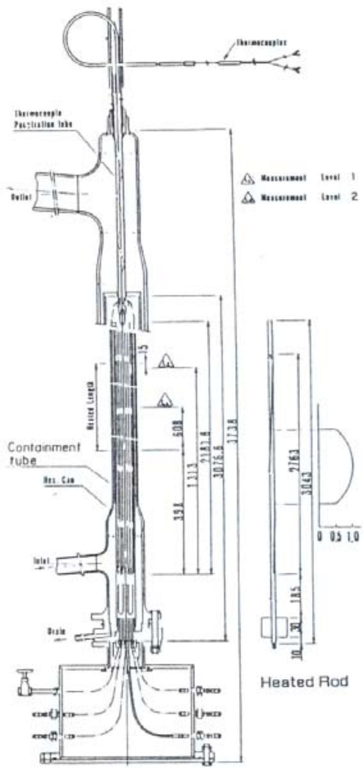

Fig. 1 Test section geometry for the Toshiba 37-pin bundle

Table 1. Design parameters of the Toshiba 37-pin fuel assembly

Parameters	Value (mm)
Pin number	37
Pin diameter	6.5
Pin pitch	7.87
Wire diameter	1.32
Wire lead length	307
Duct flat to flat distance	50.4
Total length	3043
Heated length	930

3. Automated system for numerical simulation

A program that can automate pre-processing, simulation and post-processing of CFD was developed using open source software as shown in Fig. 2.

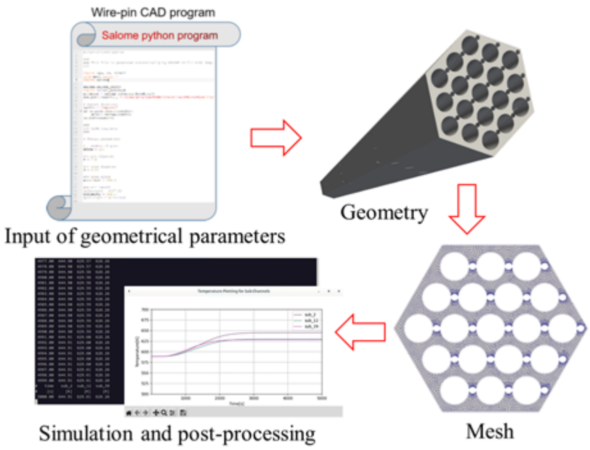

Fig. 2 Process of automated system for numerical simulation

The CFD simulations were performed using OpenFOAM code [8]. The geometry of the 37-pin fuel assembly for CFD simulations was modeled using SALOME [9] as shown in Fig. 3. The geometry can be generated automatically through input in python code. Meshes were generated according to the programmable mesh generation strategy for wire-pin bundles [10]. In the programmable mesh generation strategy, directional hybrid mesh is adopted. Unstructured mesh is used in spanwise or radial direction of the bundle, while structured mesh is used in axial or streamwise direction by extruding the spanwise mesh.

The computational meshes were created using OpenFOAM mesh generator snappyHexMesh. As shown in Fig. 4, meshes were densely created around wires that were relatively small in size compared to duct and pins, and the total number of computational mesh cells was about 17,600,000.

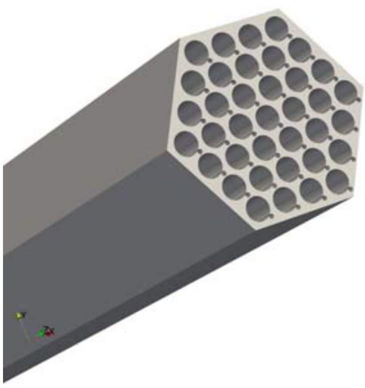

Fig. 3 The computational geometry for the Toshiba 37-pin bundle

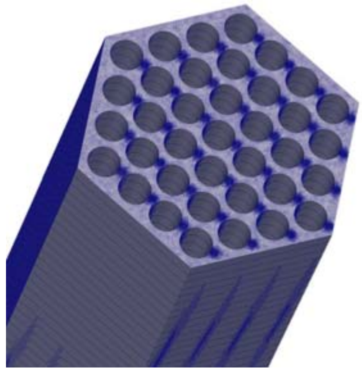

Fig. 4 Meshes for the Toshiba 37-pin bundle

The three dimensional, steady state, compressible RANS (Reynolds Averaged Navier-Stokes) based simulations were performed. The mass flow rate was applied to the inlet and the pressure was given at the outlet, and no-slip condition was applied on the walls. The working fluid is sodium and the properties of sodium such as density, viscosity, specific heat, and conductivity according to temperature were applied. Heat transfer simulation was performed by giving the same heat flux to all pins and the heat flux value was set the same as in the experiment. The boundary conditions are shown in Table 2. In this study, the SST (Shear Stress Transport) model was used as a turbulence model. Constant turbulent Prandtl number was used to predict the thermal eddy diffusivity and turbulent Prandtl number can be specified by the user.

Table 2. Boundary conditions of the Toshiba 37-pin fuel assembly

Parameters	Value
Inlet temperature (°C)	211.3
Inlet density (kg/m^3)	900.2
Inlet viscosity (Pa·s)	0.000436
Inlet mass flow rate (kg/s)	1.34
Outlet pressure (Pa)	0
Power (W)	53,580

4. Results

In order to investigate the heat transfer characteristics in the subchannels, the temperature distribution at the heated outlet is shown in Fig. 5. As shown in this figure, the temperature is high in the inner subchannels and low in the edge and corner subchannels. For comparison with the experiment, the temperature values in the subchannel numbers shown in Fig. 5 were calculated and shown in Table 3. In the same way as the experimental results, the temperatures at the outlet was normalized to the temperature difference between the outlet and inlet. The CFD results show good agreement with the experiment with a maximum error of about 7%.

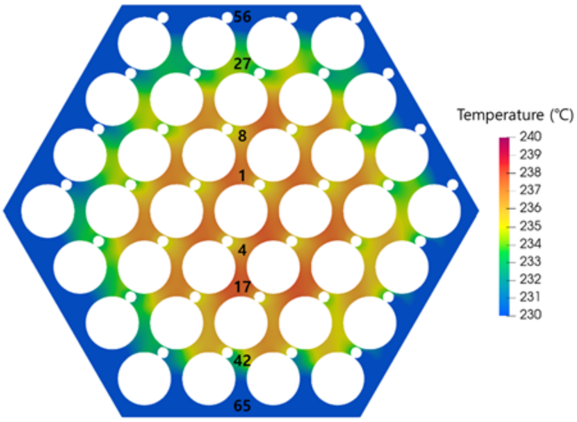

Fig. 5 Temperature distribution at the outlet

Table 3. Comparison of normalized outlet temperatures in the subchannels

Subchannel number	Experiment	CFD	Error (%)
65	0.796	0.812	2.0
42	0.974	1.044	7.2
17	1.203	1.204	0.1
4	1.210	1.145	-5.4
1	1.206	1.178	-2.3
8	1.211	1.129	-6.8
27	1.052	1.023	-2.7
56	0.768	0.790	2.8

5. Conclusions

In this study, numerical investigations of heat transfer characteristics in wire-wrapped 37-pin fuel assembly are presented. A program that can automate pre-processing, simulation and post-processing of CFD was developed using open source software. The CFD simulations were performed using OpenFOAM code for the Toshiba 37-pin SFR fuel assembly. The temperature distribution at the heated outlet was investigated to analyze heat transfer characteristics and to validate the numerical simulations.

ACKNOWLEDGEMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant and National Research Council of Science & Technology (NST) grant funded by the Korean government (MSIT) [grant numbers 2021M2E2A2081061, CAP-20033-100].

REFERENCES

[1] Khan, E.U., Rohsenow, W.M., Sonin, A.A., Todreas, N.E., A porous body model for predicting temperature distribution in wire-wrapped rod assemblies, Nucl. Eng. Des. 35, 1–12, 1975.
[2] Yang, W.S., An LMR core thermal-hydraulics code based on the ENERGY model, J. Korean Nucl. Soc. 29, 406–416, 1997.
[3] Cheng, S.K., Constitutive correlations for wire-wrapped subchannel analysis under forced and mixed convection conditions, Ph.D. Thesis. MIT, 1984.
[4] Chang, S.-K., Euh, D.-J., Kim, S., Choi, H.S., Kim, H., Ko, Y.J., Choi, S.R., Lee, H.-Y., Experimental study of the flow characteristics in an SFR type 61-pin rod bundle using iso-kinetic sampling method, Ann. Nucl. Energy 106, 160–169, 2017.
[5] Gajapathy, R., Velusamy, K., CFD investigations of helical wire-wrap fuel pin bundle and its comparison with straight wire bundle, Prog. Nucl. Energy 89, 57–68, 2016.
[6] Jeong, J.-H., Song, M.-S., Lee, K.-L., RANS based CFD methodology for a real scale 217-pin wire-wrapped fuel assembly of KAERI PGSFR, Nucl. Eng. Des. 313, 470–485, 2017.
[7] Namekawa, F., Ito, A., Mawatari, K., Buoyance Effects on Wire-Wrapped Rod Bundle Heat Transfer in an LMFBR Fuel Assembly, AIChE symposium series, 80, 128-133, 1984.
[8] OpenFOAM, https://www.openfoam.com/, 2023.
[9] Salome, https://www.salome-platform.org/, 2023.
[10] J. Kim, Y. Jung, D. Kim, J. Hong, Programmable Mesh Generation Strategy for Flow Analysis of Wire-pin Bundles, Transactions of the Korean Nuclear Society Autumn Meeting, 20-21 Oct., 2022, Changwon, Korea.