Preliminary Simulation Results for 6x12 Rod Bundle Test Using OpenFOAM

Byong Guk Jeon[1*], Gun Hong Kim[2], Seok Kim[1], Hyeokjun Byeon[1], Sang-Ki Moon[1]

1 Korea Atomic Energy Research Institute, Daedeok-Daero 989-111, Yuseong-Gu, Daejeon, Republic of Korea
2 OpenCAE, Republic of Korea
[*]Corresponding author: bgjeon@kaeri.re.kr

1. Introduction

In the core of a pressurized water reactor, coolant flows upward through sub-channels removing heat from fuel rod-bundle assemblies. Under accident conditions, e.g. loss of coolant accident, fuels can be partially exposed and three dimensional cross-flow mixing can take place among assemblies with different burn-up, or decay heat levels. To explore single-phase turbulent-mixing phenomena across sub-channels, a 4 x 6 rod bundle test, PRIUS-I, was conducted and currently a 6 x 12 rod bundle test, PRIUS-II, is being conducted [1].

Until now, many sub-channel codes were developed and utilized. These codes use porous media approach, which has the advantage of fast calculation with constituting correlations [2-3]. On the other hand, a CFD code can simulate flow in detail along complex geometry at the expense of high calculation cost. In this research, we used OpenFOAM, a free CFD software, for preliminary simulation of PRIUS-II [4].

2. Method

2.1 PRIUS-II facility

Figure 1 shows the schematic diagram of the test facility, PRIUS-II. At two bottom inlets, fluid is injected at different flow rates into separate inlet channels. After the injected flow is developed through the inlet channels, the flow enters the test section comprised of a rod bundle and is mixed. The rod bundle has a 6 x 12 array of rods with a diameter of 10 mm and a pitch of 13.35 mm. A PIV-matching index of refraction (MIR) technique is used for full velocity visualization of subassemblies.

2.2 Calculation setup

The computational domain encompasses a part of inlet region (0.5 m) and the test section (1.5 m) (red box in Fig. 1). Three types of grids were generated using the SALOME software. The characteristics of the three types are given in Table I and Fig. 2. OpenFOAM v2006(the latest version) was used and parallel computing was made with 30 cores. The simpleFoam solver, which handles incompressible flow, was used and k-ω SST turbulence model was chosen. Lastly, steady calculation was made.

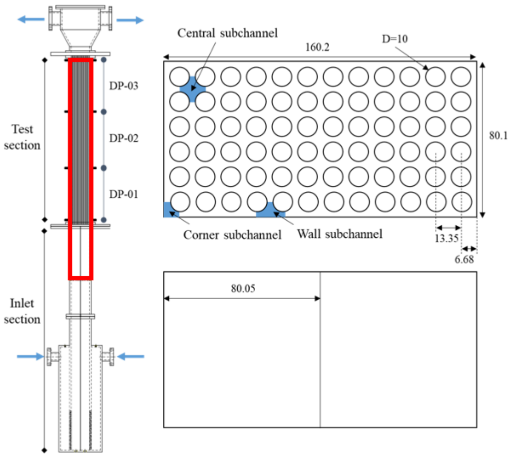

Fig. 1 Schematic diagram of PRIUS-II

Table I: Grid Characteristics

	Grid1	Grid2	Grid3
Cell number	7.1 millions	12.1 millions	23.4 millions

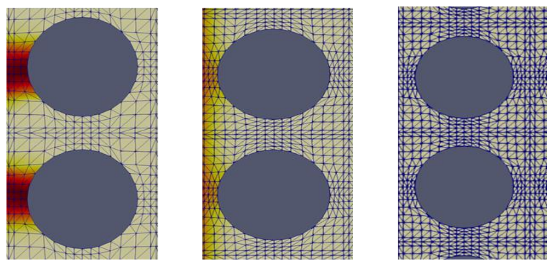

Fig. 2 Grids near PRIUS-II rod bundles

3. Results and discussions

A grid sensitivity study was made and compared with two preliminary experiment results conducted with water. The experiment condition is given in Table II and calculated DP results are given in Fig. 3 (DP locations are given in Fig. 1).

Table II: Preliminary experiment condition

	Case 1	Case 2
Mass flowrate (left/right)	2.164 kg/s/ 1.165 kg/s	2.164 kg/s/ 2.164 kg/s/

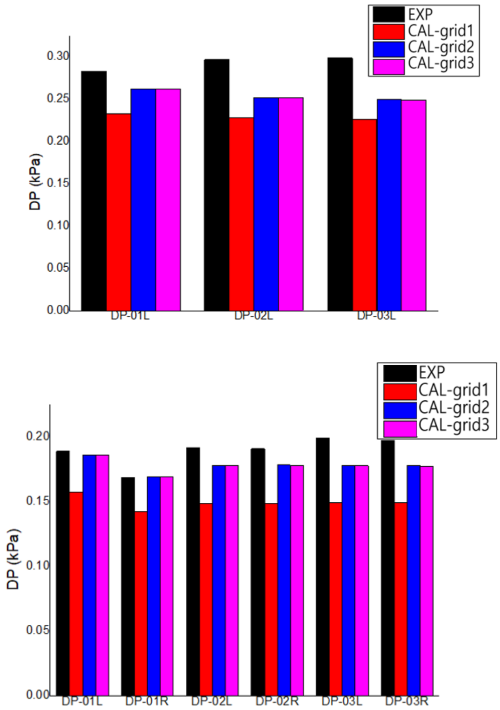

Fig. 3. Grid sensitivity results for water test

From the grid sensitivity results, we can see that grid 2 gave the converged results and the calculation predicted reasonably against the experimental data.

A preliminary calculation result for a main test, which uses water-NaI solution, is given in Fig. 4. Mixing of water was simulated along the sub-channels.

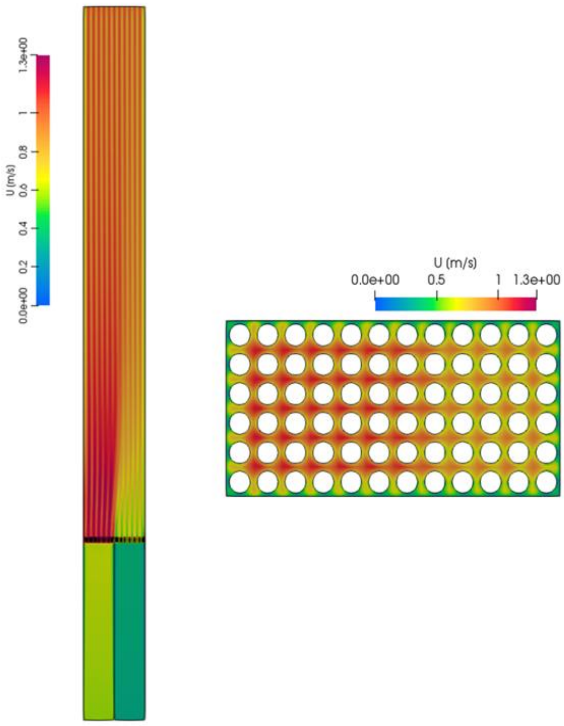

Fig. 4. Preliminary velocity calculation results for a main test (flow rates: 2.164 kg/s, 1.165 kg/s) (left: side view, right: top view)

4. Conclusion

PRIUS-II experiments are being conducted for examining cross-flow mixing models. In parallel to the experiment, a CFD code calculation using openFOAM was prepared to evaluate turbulence models and to provide unmeasured local values.

A preliminary calculation was made using an incompressible solver for steady flow. The adequacy of the generated grid was confirmed through a grid sensitivity study and the preliminary experiment results. A sample calculation was made for a main test as well. In the future, different turbulence models will be validated against the main experiment results.

ACKNOWLEDGEMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2017M2A8A4015026).

REFERENCES

[1] S. Kim, et. al., Experimental visualization of flow structure inside sub-channels of a 4 x 6 rod-bundle, Annals of Nuclear Energy, 140, 107097, 2020
[2] J.J. Jeong, et al., Development and preliminary assessment of a threedimensional thermal hydraulic code, CUPID. Nuclear Engineering Technology, 42 (3), 279–296. 2010
[3] D. H. Hwang, Development of Validation System for Sub-channel Analysis Codes under Steady-State PWR conditions, KAERI-TR-4143, 2010
[4] OpenCFD Ltd, OpenFOAM User Guide v2006, 2020