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

Experimental Program to Investigate Flow Characteristics inside Subchannels of 8×24 Rod Bundles

Seok Kim [a*], Hae-Seob Choi[a], Byong Gook Jeon [a], Sung-Uk Ryu, Young-Jung Youn[a], Sang-Ki Moon[a]
[a]*Thermal Hydraulics and Severe Accident Safety Research Division, Korea Atomic Energy Research Institute*
111 Daedeok-daero989beongil, Yuseong-gu, Daejeon, 34057, Republic of Korea
[*]*Corresponding author: seokim@kaeri.re.kr*

1. Introduction

The fuel element geometry frequently used in nuclear reactors is the rod bundle. The coolant is flowing axially through the subchannels formed between the rods. The mixing of cooling fluid in a rod bundle reduces the temperature differences in the coolant and along the perimeter of the rods. Flow inside rod bundles has a similarity with flow in porous media. To ensure thermal performance of a nuclear reactor, detailed information of the heat transfer and turbulent mixing flow phenomena taking place within the subchannels is required. The subchannel analysis is one of the key thermal–hydraulic calculations in the safety analysis of the nuclear reactor core [5]. At present, subchannel computer codes are employed to simulate fuel elements of nuclear reactor cores and predict the performance of cores under normal operating and hypothetical accident conditions. The ability of these subchannels codes to predict both the flow and enthalpy distribution in fuel assemblies is very important in the design of nuclear reactors. Recently, in accordance with the modern trend of the safety analysis for the nuclear reactor, a new component scale analysis code, named CUPID, has been developed in KAERI. The CUPID code is based on a two-fluid and three-field model, and both the open and porous media approaches are incorporated. In a view point of porous media approach, the momentum and energy equation are rearranged by using time- and volume-averaging method (double decomposition technique). To evaluate the performance of macroscopic transport equation in CUPID code, many experimental data should be utilized for developing new model for subchannel analysis. From this motivation, a test facility, called PRIUS (in-PWR Rod-bundle Investigation of Undeveloped mixing flow across Sub-channel), was newly designed and constructed to generate an experimental database for a multi-dimensional flow distribution in a rod-bundle geometry. The configuration for the rod bundle simulated in this study consists of an 8 × 24 array of parallel rods. In the present work, PIV-MIR (Matched Index of Refraction) technique is used to measure the velocity field of multi-dimensional flow driven by various combinations with inlet flow condition and flow area change. The measured velocity field data were produced for the nuclear safety analysis code verification, but were compared with the CFD analysis results on the turbulent dispersion and diffusion term of cross flow inside subchannels.

2. Test Facility

The test section of PRIUS shown in Figure. 1 will have rectangular geometry with a dimension of 327 mm × 113 mm × 1.5 m which is made of acryl of 15 mm thickness. The test section and the rod are all made of acryl for visualization. The typical configuration for the rod bundle will consist of 8 × 24 unheated rods representing three 1/4 scaled assemblies (Fig. 2). The rod will be approximately the same size as the APR1400 PLUS7 fuel assembly (16 × 16, D=9.5 mm, P=12.85 mm, D_h=12.64 mm). The 10 mm diameter is selected for rods considering commercial products. In order to focus on the flow phenomenon inside the sub-channels, the pitch is set to be 13.35 mm to be equal to the hydraulic diameter of the prototype.

Single phase liquid flow will be investigated. Pressure is close to atmospheric at top of the test section. The temperature of the fluid will be maintained at 30 °C.

A separate feeding of each assembly may create differences of velocity and tracer concentration between the left, center and right assemblies. The maximum water flowrate is 48 m^3/h. The Re number is up to 50,000, which includes the Re number of the SB/IB-LOCA and Low flow SLB accidents. The lower part of the main test section consists of three entrance chambers. Each inlet chamber is connected to an individual piping line. We will use a perforated plate and a honeycomb to create a flat velocity profile and measure the velocity field using the PIV technique at the inlet region of the main test section. The measured velocity field can be used as an inlet condition for code analysis.

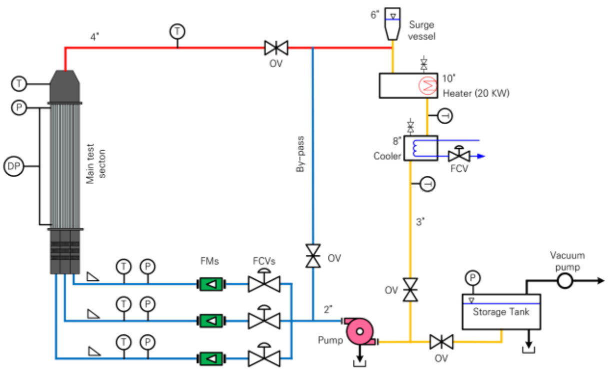


Fig. 1. A schematic diagram of PRIUS test facility.

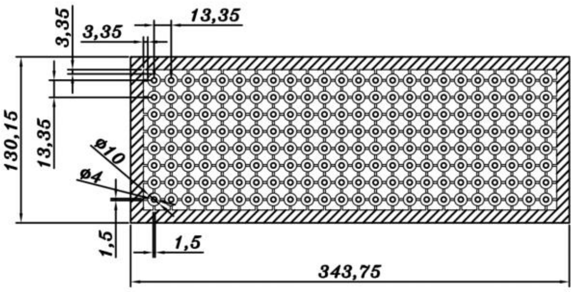


Fig. 2. The PRIUS test section (8 × 24 rod bundles).

3. Instrumentation

PIV-MIR technique will be adopted to measure profiles of axial and radial components [$V_z(x,z)$, $V_x(x,z)$] by visualizing the entire cross section of the sub-channels including the back side of the rods as well as the gap between the rods. (Fig. 3) Velocity vector field results will be used to calculate average velocity field, turbulence intensity, turbulent kinetic energy, turbulent dissipation rate, and so on using the ensemble averaging. 16 M pixels camera will be used as a recording device in the PRIUS program. The window offset and recursive scheme will be used to calculate the velocity field and enhance the sub-pixel accuracy [2,3]. The velocity fields for interrogation window size of 64 × 64 pixel2 calculated with 50% overlap is used for the final interrogation window size of 32 × 32 pixel2. This results in an effective spatial resolution of 16 × 16 pixel2. After calibration of the images, a resolution of 21 × 21 μm^2/pixel will be achieved. This will correspond to an effective spatial resolution of 0.34 × 0.34 mm2 for the final velocity vector field. This high spatial resolution will be helpful in evaluating the turbulent diffusion and dispersion terms. The MIR (matched index of refraction) technique [1] adopted to match the refractive index of the fluid and the acrylic rods and the test windows utilizes a fluid mixed with NaI (sodium iodide) in demineralized water. The viscosity of the mixed fluid is low enough to enable Reynolds-number identity with a feasible mass flow. The density and viscosity information of the mixed fluid will be provided.

LIF (laser induced fluorescence) technique is an optical measurement technique used to measure instant whole-field concentration or temperature field in flows. Flow information on a passive scalar or the velocity field is desirable in many fluid flow investigations. However, there are many limitations to acquire the temperature field information in complex rod bundle geometry. In general, heat-mass transfer analogy is applicable for predicting the mass transfer rate in turbulent flows, where the concentration field is correlated to the temperature field. The temperature gradient is directly responsible for establishing a concentration gradient. It will be able to measure passive scalar radial concentration profiles c(x,z) at various elevations. The tracing liquid (eg. Rhodamine B) is injected using a rod consisting of rod bundle, inside which a tracing tube is connected exquisitely. Several rods will be selected for the injection at various elevations. The number and position of the rods injecting the tracer solution will be studied.

The static pressure P and pressure differences ΔP will be measured at several positions on the wall of the test section.

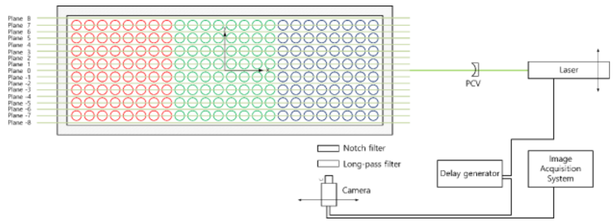


Fig. 3. Measurement of velocity and scalar concentration using PIV-MIR and LIF.

4. Test Matrix

Equal/unequal inlet velocity conditions will be simulated for turbulent diffusion term or turbulent frictional term evaluation to create the cross flow between the assemblies. (Fig. 4) The Reynolds numbers may be covered a various range: 0-50,000. Not only the bare rods but also the spacer grid imbedded bundles can be considered.

Table 1. Velocity distribution in cross-flows.

$\frac{V_{left}}{V_{center}} \mid \frac{V_{right}}{V_{center}}$	Re_{center}			
	3000	5000	10000	30000
0.6 | 0.4	⊕	⊕	⊕	⊕
0.8 | 0.2	⊕	⊕	⊕	⊕
1.0 | 1.0	⊕	⊕	⊕	⊕
1.2 | 0.8	⊕	⊕	⊕	⊕
1.4 | 0.6	⊕	⊕	⊕	⊕
1.6 | 0.4	⊕	⊕	⊕	⊕
1.8 | 0.2	⊕	⊕	⊕	⊕

To maximize the accuracy of the physical modeling, we will set the simplest experimental conditions and clarify the boundary conditions. The inlet velocity distribution for each assembly may be provided as inlet conditions. In order to enhance the reliability of the experimental data, a measurement uncertainty evaluation and CFD analysis will be carried out from the preliminary experimental stage.

Tests on the characteristics of a passive scalar mixing at equal/unequal velocity conditions will be performed with a concentration field measurement using the LIF technique.

Since NaI solution is used as a fluid, it is impossible to visualize by the discoloration of the fluid when the concentration of the fluorescent dye is injected to each assembly channel, resulting in a cost problem. Therefore, in the PRIUS test, as shown in Figure 5, the fluorescent dye injection rods are set in consideration of the characteristics of the sub-channel.

The flow configurations of Figure 6 are used with unequal inlet velocity and concentration differences. The cross flows in bare rod bundle and with spacer grids will be investigated separately.

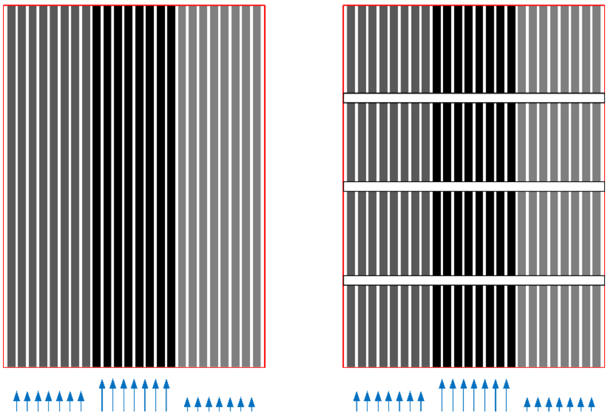


Fig. 4. Tests with unequal velocity.

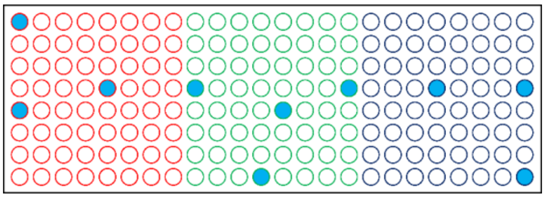


Fig. 5. Fluorescent dye injection location.

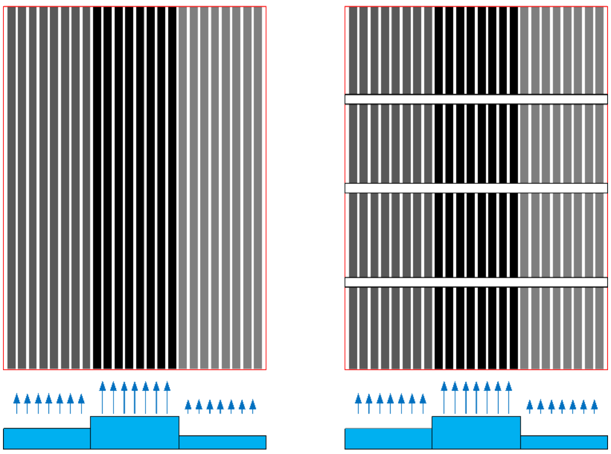
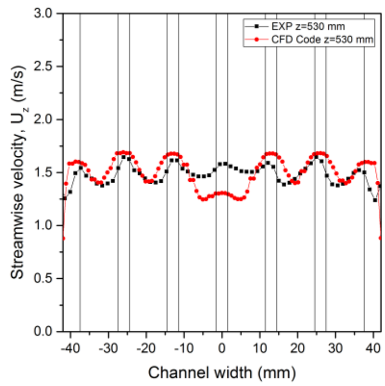


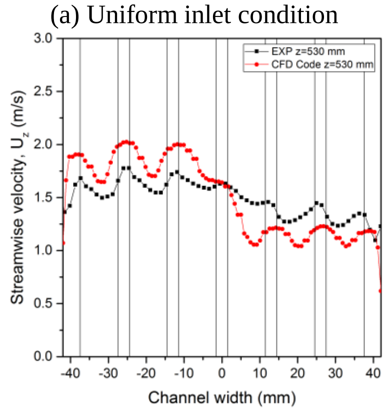
Fig. 6. Test conditions for measuring radial mixing of a passive scalar with cross flow

4. Results and Discussions

Preliminary experimental study has been done to validate the PIV-MIR technique using 4 × 6 rod array [4]. The experimental data of the Re=20,000 condition was utilized to validate the prediction capability of the CFD code (STAR-CCM+ 11.02). At the inlet regions, mass flow boundary conditions were imposed with measured flow rates. At the downstream of the rod bundle region, the pressure outlet boundary was imposed. The representative results were given in Figure 7 and 8. Figure 7 shows the streamwise velocity profile along the center plane at 530 mm downstream of the rod bottom. Figure 7 (left) is the uniform inlet condition, which has the symmetric velocity profile. Apart from the center region (i.e. width < 5 mm), the overall trend was reproduced. Figure 7 (right) is the asymmetric inlet condition, which has the higher velocity at the left region. In the CFD code calculation, flow was not mixed efficiently compared to the experimental results. Figure 8 shows the turbulence intensity at the location same with Figure 7. For both uniform and asymmetric inlet conditions, the turbulence intensity was under-predicted. This lower turbulence intensity in calculation can lead to less mixing in the asymmetric inlet condition. By examining different turbulence models and modifying the constituting parameters, the underlying physics will be explored further.

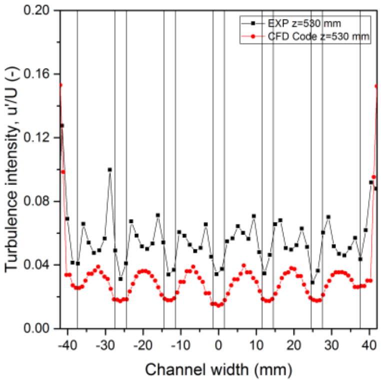


(a) Uniform inlet condition

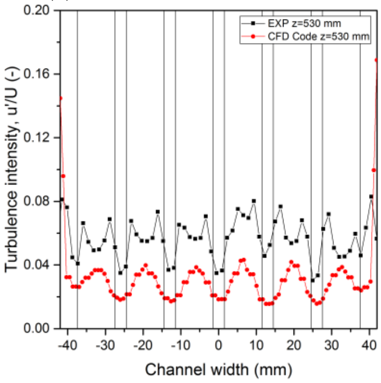


(b) Asymmetric inlet condition

Fig. 7. Streamwise mean velocity profile compared with CFD analysis results at the center plane.



(a) Uniform inlet condition



(b) Asymmetric inlet condition

Fig. 8. Turbulence intensity profile compared with CFD analysis results at the center plane.

5. Conclusions

To evaluate the performance of macroscopic transport equation in CUPID code, many experimental data should be utilized for developing new model for sub-channel analysis. From the motivation, a test facility, called PRIUS (in-PWR Rod-bundle Investigation of Undeveloped mixing flow across Sub-channel), is being newly designed and constructed to generate an experimental database for a multi-dimensional flow distribution in a rod-bundle geometry. It can also be useful for CFD in open medium validation.

ACKNOWLEDGMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2012M2A8A4004176 and 2017M2A8A4015026).

REFERENCES

[1] K. Bai, J. Katz, On the Refractive Index of Sodium Iodide solutions for Index Matching in PIV, Exp. in Fluids, Vol.55, p.1704, 2014.
[2] K. Rehme, Turbulent momentum transport in rod bundles, Nuclear Engineering and Design, Vol.62, p.137, 1980.
[3] K. Rehme, The structure of turbulence in rod bundles and the implications on natural mixing between the subchannels, International Journal of Heat and Mass Transfer, Vol.35, p.567, 1992.
[4] S. Kim, H.-S. Choi. B.-G. Jeon, D.-J. Euh, S.-K. Moon, Experimental Investigation on Flow Visualization inside Subchannels of PWR Rod Bundle, 19th International Symposium on the Application of Laser and Imaging Techniques to Fluid Mechanics, Lisbon, Portugal, July 16 – 19, 2018.
[5] D. Bestion, M. Valette, P. Fillion, P. Gaillard, 3D Core thermalhydraulic phenomena in PWR SBLOCAs and IBLOCAs, NURETH-17, Sept 2017, Xian, China