CFD Analysis on KAERI SFR 37-Pin Wire-wrapped Fuel Bundle Experiment

Min Seop Song[a], Jae Ho Jeong[b], Eung Soo Kim[a*]
[a]*Dep. of nuclear engineering, Seoul National Univ., 1 Gwanak-ro, Gwanak-gu, Seoul 08826*
[b]*Korea Atomic Energy Research Institutde, 111 Daedoek-daero 989beon-gil, Yuseong-gu, Daejeon 34057*
[*]*Corresponding author: kes7741@snu.ac.kr*

1. Introduction

The Wire-wrapped fuel bundle is a fuel assembly design used in Sodium Fast Reactor. Due to the high heat transfer rate of sodium, compact triangular geometry required for fast neutron reactors is possible. The wire wound on the fuel rod prevents the impact between the rods and converts the axial momentum of the fluid into the transverse momentum in the direction in which the wire is wound [1]. Therefore, the swirl flow is generated in the same direction as the winding direction of the wire. This makes the coolant to flatten temperature distribution inside fuel bundles. At the same time, the wire induces vortex generation by flow separation at the downstream of wire. These result in complex cross-flow [2]. This lead to researchers to investigate flow distribution inside of the wire-wrapped fuel bundle.

An experiment had been conducted to measure the overall pressure drop and subchannel flow distribution for a 37-pin wire-wrapped fuel bundle to obtain reference data for validation of subchannel codes in KAERI [3]. In this experiment, the pressure drop was measured in three channels and flow rate at each subchannel was measured using iso-kinetic sampling probe. The CFD simulation was also performed to predict flow behavior inside of fuel bundle. As a result, the experimental data showed some difference with CFD results calculated from commercial program, STAR-CCM, for pressure drop and flow distribution. This difference is presumably due to inadequate turbulence model and imperfect configuration of mesh.

The methodology of mesh construction of Jeong et al [1, 4] has been successfully applied to the analysis of flow in various size of wire-wrapped fuel bundles. The heart of this methodology is to divide fluid region into inner fluid region and outer fluid region. Inner fluid region includes fluid region around rod and rest of fluid region is belongs to outer region. The efficiency and accuracy of this methodology have been proven by comparing result of various experimental cases [1, 4]. Therefore, it is needed to apply this methodology to this analysis case. In addition, it is necessary to confirm whether the measured flow distribution is valid through CFD analysis and previous researches.

In this study, through CFD study with grid structure made from methodology of Jeong and more various turbulence models, validation study and analysis on the flow distribution for 37-pin test section have been conducted. The pressure drop and flow distribution data from experiment and CFD was compared for the same condition. In addition, the validity of both result were investigated in terms of flow pattern.

2. Numerical Analysis Methods

2.1 Geometry of 37-Pin Test Section

The 37-pin experiment was made by scaling down the design parameters of the PGSFR [3]. The major geometry parameters of the 37-pin experiment are shown in Table 1.

Table I: Major Geometrical Parameters of 37-Pin Test Section [3]

Parameters	Test Section
Rod Diameter (mm)	8
Rod Length (mm)	1500
Wire Diameter (mm)	1
Lead Length (mm)	221.5
Rod Pitch (mm)	9.05

The height of the test section of the 37-Pin experiment is 1500mm, which is sufficient to develop the flow as the wire rotates more than 3 times. [5] The bottom of actual test section was fixed to the grid, but the upper part was not fixed. This could change the flow area in each subchannel at the outlet [3]. Figure 1 represents the cross plane of test section at the outlet with number of rods and subchannel.

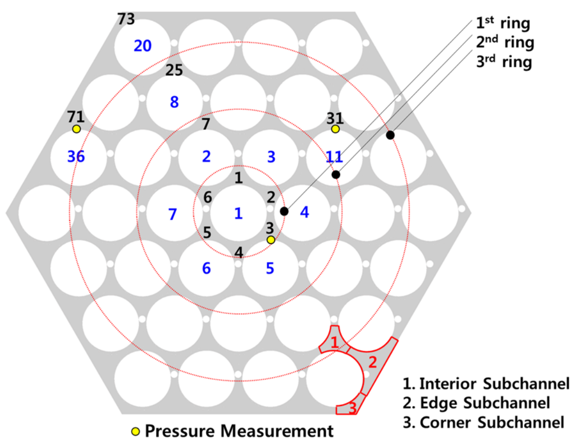


Fig. 1. Cross-section of 37-Pin test section at the outlet.

2.2 Analysis Conditions

The CFD analysis was performed under the same flow conditions as the experiment. In the experiment, water heated to 60 °C was used to increase Reynolds number. Density and dynamic viscosity of heated water is 983.4 kg/m^3 and 4.67×10^{-4} Ns/m^2. Experiments were conducted at Reynolds numbers of 7420, 18550, 26712, 37100 and 42665 based on the hydraulic diameter and bulk velocity. The inlet flow rate is 5.49kg/s when the Reynolds number is 37100.

The SST, LRRSM, EARSM-*k-e*, *k-w* and RNG-*k-e* models were used as the turbulence models. [5] Chang et al reported CFD result is calculated with cubic-k-e turbulence model. Therefore, the turbulence model based on the *k-w* model and the Reynolds stress model is added considering the influence of the low Reynolds region and the swirl motion of flow.

The tool used for calculation was ANSYS CFX 17.0 [6]. High-resolution option for advection scheme and turbulence numeric scheme were applied. The solver iteration was performed 500 times and the residuals were converged under 1e-5 for momentum and mass equations.

2.3 Mesh Structure of 37-Pin Test Section

The mesh structure used in this study already showed high accuracy of pressure drop calculation in the analysis using the SST k-w model in comparison with various experimental cases. [1, 4] This grid resolves a narrow area between the wire and the rod with a structured grid, which allows the computation of local flow field. The number of grid is about 7 million for the whole test section and average y+ is under 2. Figure 2. Shows the part of grid of 37-Pin test section. The yellow circles represents interface for General Grid Interface (GGI), where grid of two side meet.

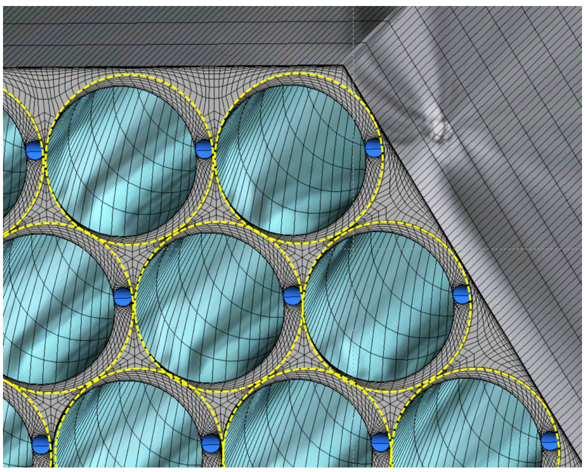


Fig. 2. Image of gird structure of 37-Pin Test Section.

As in the measurement of the actual experiment, the pressure was probed at 553.78, 1218.30mm from the inlet at the wall of 5, 11, 36 Pin as shown in Fig 1. In addition, flow area at the outlet was divided by subchannel to extract the mass flow rate of each subchannel.

3. Results and Discussion

3.1 Validation Study – Friction Factor

The validity of the CFD analysis was firstly investigated by comparing the friction factor calculated from the pressure drop. The difference of experimental friction factor between measuring points was under 0.016% based on SC#31 case. The comparison result were drawn in Figure 3. The *k-w* model shows closer results than the *k-e* model. With increase of Reynold number, the difference is even reduced. These results are consistent with the result of the different test section using this grid. [1,4] It is suspected that result from *k-e* models are less accurate due to wall function, which affect the calculation of skin friction. The *k-w* based model that resolved the wall well predict the experimental results, among which the SST model best matched.

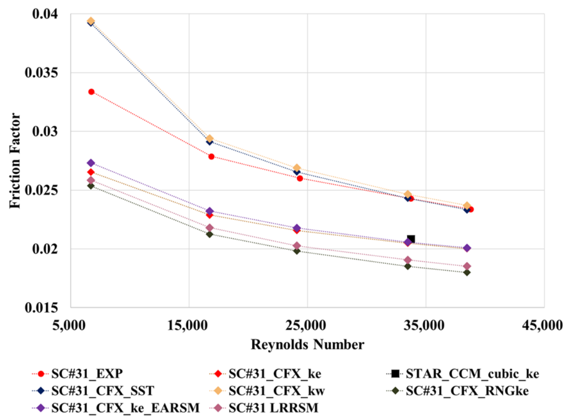


Fig. 3. Friction factor comparison between experiment and CFD with different turbulence models. [3]

3.2 Validation Study – Flow Rate in subchannel

In order to analyze the flow distribution, comparison was made for the flow rate data obtained from the iso-kinetic sampling probe and the data extracted from the CFD results from the SST, *k-e* and *k-w* models as in Figure 4. This is meaningful as it is able to compare the flow rate at the level of subchannel. As the experimental data are not measured in all interior subchannel, it is impossible to grasp trends for whole location. The data obtained through CFD shows that the flow rate changes sinusoidally with increasing subchannel number for each ring marked in Fig 1. In addition, in the interior subchannel area, the magnitude of flow rate of the experiment and analysis results varied, but in the edge and corner areas, the experimental data were clearly smaller than the CFD analysis data. This may be because the top of the pin is not fixed during measurement and the position of the pin is changed so that the subchannel in the outer region is formed smaller than the ideal case [3]. Therefore, the validation of the pressure drop, which is a global parameter, has been performed, but the validation at the subchannel level has not been achieved.

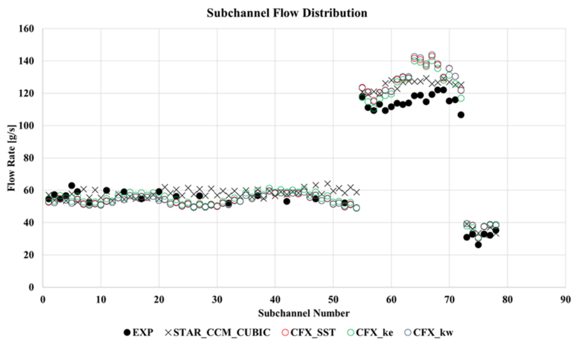


Fig. 4. Flow rate at each subchannel from experiment and CFD result. [3]

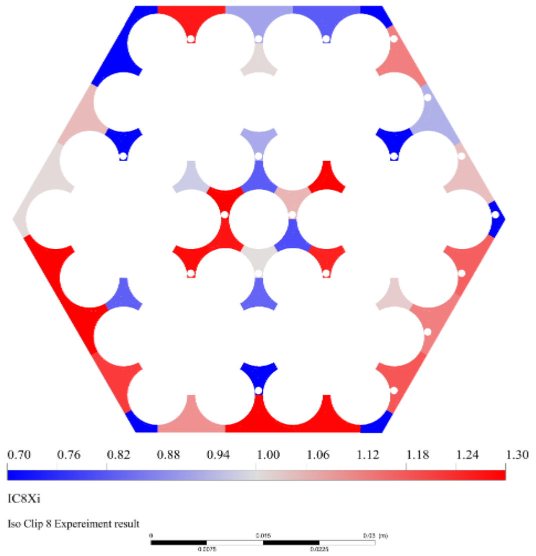
3.3 Investigation of Outlet Flow Distribution

The flow split factor is the ratio of the mass flow rate measured through the iso-kinetic sampling probe to the total flow rate obtained by dividing the cross-sectional area of each subchannel. This allows confirming the relative magnitude of the flow rate in each subchannel based on the average flow rate and the tendency of the flow distribution by location.

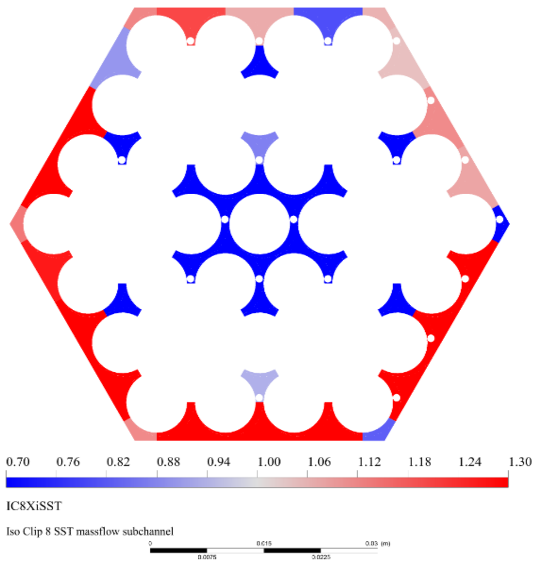
$$X_i = \frac{V_i}{V_T} = \frac{\dot{m}_i/(\rho A_i)}{\dot{m}_T/(\rho A_T)} \quad (1)$$

Figure 5 shows the distribution of flow split factor obtained from the experiment and CFD calculation. The flow split factor calculated from the experimental results showed a relatively irregular distribution when compared with the result from CFD. In addition, there is a clear distinction between the interior subchannel and the outer subchannel in (b). This characteristic shown in (b) is due to the dominant cross-flow that depends on the relative position of the wire in the outer region (Edge + Corner Subchannel). As can be seen in Figure 6, this periodic strong axial flow is most strongly induced when the wire in the edge region passes generating a vortex (Red circles). Since the vortex generated by the wire entering the inner subchannel from the edge subchannel rotate in a direction that suppresses the dominant cross-flow, [5] the linearity of the axial momentum increases in this region. Both of (a) and (b) in Fig. 5 reveal the periodic flow pattern in the outer region, but in (a) the flow at the outer region is weaker than the CFD result. This means that flow rushes inner region. This tendency is presumably because only the lower region is fixed, so that the outer region is narrowed and the region of the inner region is widened at the upper end.

The above analysis gives an answer to why the results for the pressure drop are consistent and the results for the flow distribution are not. The main factor of the pressure drop inside the wire-wrapped fuel bundle is the pressure



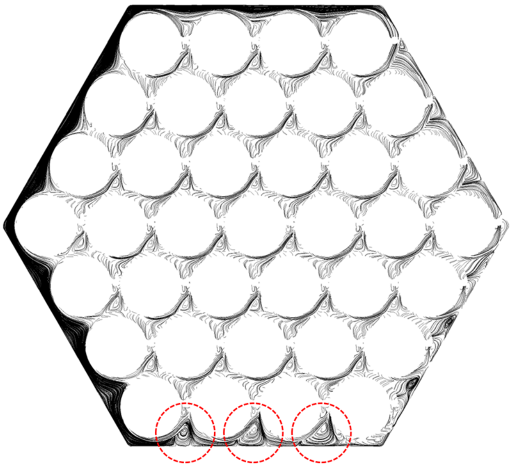
(a) Experiment Result [3]



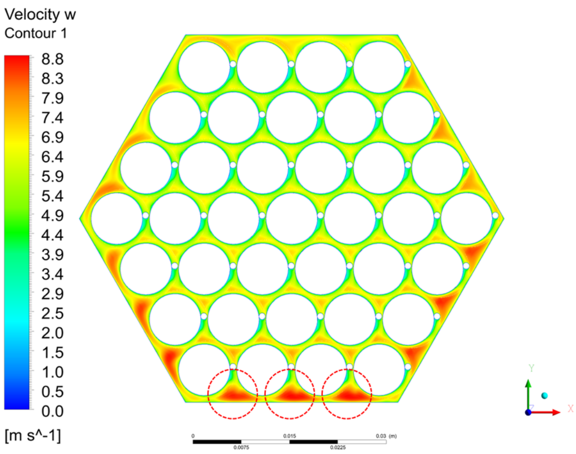
(b) CFD Result

Fig. 5. Distribution of Flow Split Factor at the Outlet.

-drop due to skin friction which is related to the contact area of flow and velocity gradient. The arrangement of the wire-wrapped fuel bundle is different from the ideal one, but the internal flow area is almost same with the ideal case. The comparison results on the pressure drop are therefore in good agreement, and the distribution of the flow rate associated with the arrangement of bundle is different. Since the flow distribution obtained through the analysis is a regular pattern identified through other analysis cases [1, 4, 5], the flow pattern of the test section is likely to be deformed in the experiment. Therefore, the 37-pin experimental data provides the possibility of validation of pressure drop analysis, but the flow distribution by subchannel region may not be suitable due to deformation of passage area in fuel bundle [3].



(a) Streamline at the Outlet (CFD Result)



(b) Axial Velocity at the Outlet (CFD Result)

Fig. 6. Streamline and Axial Velocity Contour at the Outlet.

3. Conclusions

The experimental data including pressure drop and distribution of flow rate obtained from 37-pin test section was compared with CFD simulation for validation. The grid structure made by structured grid with GGI option showed good agreement with friction factor for the flow region of high Reynolds number and with SST *k-w* turbulence model. For the flow distribution at the outlet, the experimental data showed more flatten flow distribution than CFD result. Considering flow behavior inside wire-wrapped fuel bundle, it is suspected that the unfixed pins lead to the broadening of the inner subchannel area at the outlet. In conclusion, the simulation result was validated for overall flow characteristic pressure drop. However, the comparison of local flow distribution was inadequate.

ACKNOWLEDGEMENT

This research was supported by National R&D Program through the National Research Foundation of Korea(NRF) funded by the Ministry of Science, ICT & Future Planning(No. 2016M2B2A9911846) and National Nuclear R&D Program through the National Research Foundation of Korea (NRF) funded by MSIP; Ministry of Science ICT & Future Planning (No. NRF-2013M2B2B1075735).

REFERENCES

[1] J. H. Jeong, J. Yoo, K.L. Lee & K.S. Ha, Three-dimensional flow phenomena in a wire-wrapped 37-pin fuel bundle for SFR. Nuclear Engineering and Technology, 47(5), 523–533, 2015.
[2] R. Ranjan, C. Pantano and P. Fischer, Direct simulation of turbulent heat transfer in swept flow over a wire in a channel. International Journal of Heat and Mass Transfer, 54(21–22), 4636–4654, 2011
[3] S. K. Chang, D. J. Euh, H. S. Choi, H. Kim, S. R. Choi and H. Y. Lee, Flow Distribution and Pressure Loss in Subchannels of a Wire-Wrapped 37-pin Rod Bundle for a Sodium-Cooled Fast Reactor, Nuclear Engineering and Technology, Vol.48, p. 376, 2016
[4] J. H. Jeong, M. S. Song and K. L. Lee, RANS based CFD methodology for a real scale 217-pin wire-wrapped fuel assembly of KAERI PGSFR. Nuclear Engineering and Design, 313, p.470–485, 2017
[5] R. Gajapathy, K. Velusamy, P. Selvaraj, P. Chellapandi, and S. C. Chetal, A comparative CFD investigation of helical wire-wrapped 7, 19 and 37 fuel pin bundles and its extendibility to 217 pin bundle. Nuclear Engineering and Design, Vol.239(11), p.2279–2292. 2009
[6] ANSYS® Academic Research, Release 17.0, Help System, CFX Solver Manual, ANSYS, Inc.