The Preliminary Study for Numerical Computation of 37 Rod Bundle in CANDU Reactor

Yu Mi Jeon*, Jun Ho Bae and Joo Hwan Park
Korea Atomic Energy Research Institute, 150-1 Dukjin-Dong, Yuseong-Gu, Daejeon 305-353, Republic of Korea
**Corresponding author: Jeonym85@kaeri.re.kr*

1. Introduction

A typical CANDU 6 fuel bundle consists of 37 fuel rods supported by two endplates and separated by spacer pads at various locations. In addition, the bearing pads are brazed to each outer fuel rod with the aim of reducing the contact area between the fuel bundle and the pressure tube. Although the recent progress of CFD methods has provided opportunities for computing the thermal-hydraulic phenomena inside of a fuel channel, it is yet impossible to reflect the detailed shape of rod bundle on the numerical computation due to a lot of computing mesh and memory capacity. Hence, the previous studies [1, 2] conducted a numerical computation for smooth channels without considering spacers, bearing pads. But, it is well known that these components are an important factor to predict the pressure drop and heat transfer rate in a channel. In this study, the new computational method is proposed to solve the complex geometry such as a fuel rod bundle. In front of applying the method to the problem of 37 rod bundle, the validity and the accuracy of the method are tested by applying the method to the simple geometry. Based on the present result, the calculation for the fully shaped 37-rod bundle is scheduled for the future works.

2. Methods and Results

2.1. Flow configurations

In order to reflect the geometric characteristics of a 37-rod bundle, the channel flow in a long cylindrical tube with a heated rod has been considered. The typical geometry of the simplified model is shown in Fig.1. The fluid flows into the channel with a uniform velocity of 0.1 m/s and the temperature of 25℃. At the downstream exit, the usual Neumann-conditions are applied for fully-developed flow and temperature fields. The heat source of solid domain has the value of 100,000 W/m. The usual Boussinesq-fluid assumption is invoked, and the physical properties of the fluid and the heated rod are taken to be constant. The computation was conducted by using commercial CFD code, CFX-ver 12.0 [3]. The hexagonal mesh was generated by using ICEM code [4] as shown in Fig. 1. Totally 512,000 nodes are used for each case.

The convergence of the solution was declared when the maximum relative change between two consecutive iteration levels fell below 10^{-4} for velocity and temperature.

2.2. Computational method

In a channel flow, the thermal condition of upstream flow has an effect on the characteristics of downstream flow, but the condition of downstream flow does not affect to the characteristics of upstream flow. That is, the thermal-hydraulic characteristics of a certain position in a channel are only determined from the condition of upstream flow. This is the main concept of unwind scheme for treating convective term. This concept is applicable to the high speed flow in a smooth channel. However, the accuracy is deteriorated for the case of a low Reynolds number flow, and it is more difficult to apply to the problem with secondary flow by the existence of blockage in a channel. In a CANDU rod bundle, there are a number of blockages in a channel such as spacers, bearing pads. However, the size of blockage is very small compared to the channel length, which result for the most regions in a channel to have the uniform flow pattern. This geometric characteristics, together with the high speed flow in a channel, make it possible to apply the unwind scheme concept to solve the complicated channel flow such as CANDU fuel channel.

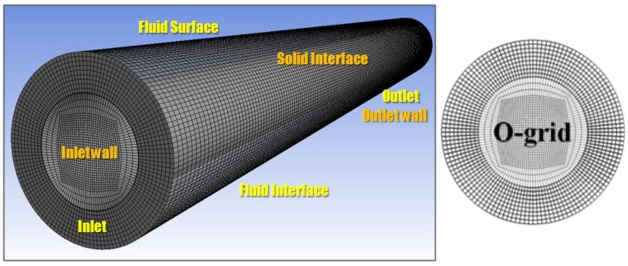

Fig. 1 Grid systems for singular rod

In order to apply the unwind scheme concept to the actual problem, the computational domain of a channel is divided into several parts, as shown in Fig.2. In computation, the steady solution of a split channel 1 is first secured with the given inlet condition. Then the thermal-hydraulic data (velocity, pressure, temperature) at the downstream exit of split channel 1 is exported to a file and these data are used as the inlet condition for obtaining the solution of a split channel 2. The same way is applied to solve a split channel 3. Note that the Neumann condition was applied to the downstream exit of each split channel. Figure 2 shows the graphical explanation for the new-proposed method (split channel method) based on the upwind scheme concept and the generalized method (single channel method).

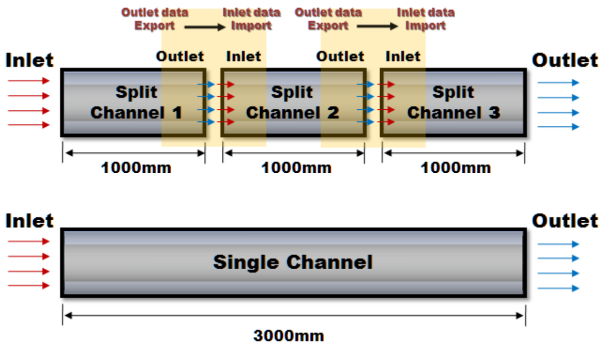

Fig. 2 Graphical explanation for split channel method

2.3. The calculation results

In order to establish the validity for the split channel method, it is necessary to compare the computational results by the split channel and single channel methods. Figure 3 shows the temperature distribution at the downstream exit, solved by the split channel and single channel method. The outlet temperature is shown to have a similar distribution for two methods.

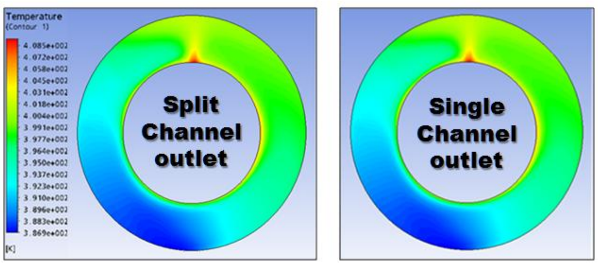

Fig. 3 Comparison of outlet temperature distribution

Figure 4 shows the relative difference of temperatures at the exit plane of each split channel for two methods. Although it has a slight difference for the maximum and minimum temperature, the average temperatures of an exit plane have the similar values. The average temperature is shown to have the relative difference lower than 1.0% for all the measured points along the channel length. The plausible explanation for the deviation of local temperatures is that the upwind scheme concept is applicable to the convective heat transfer in the fluid region of a channel, while it gives rise to a numerical error in computing the conduction heat transfer in the solid region of a rod.

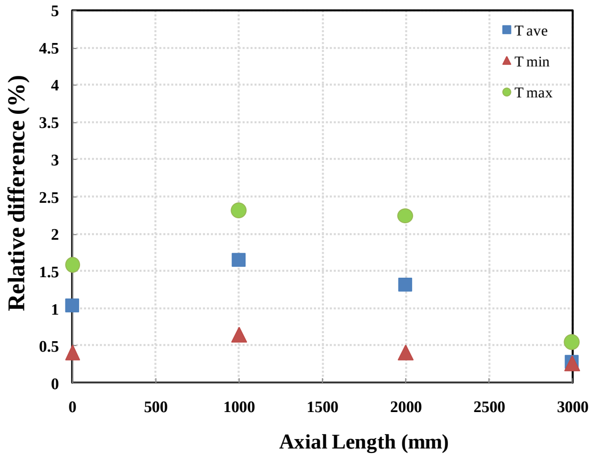

Fig. 4 Comparison for relative difference of temperature

3. Conclusion

The split channel method has been proposed with the aim of computing the fully shaped CANDU fuel channel with detailed components. The validity was tested by applying the method to the single channel problem. The average temperature has the similar value for the considered two methods, while the local temperature shows a slight difference by the effect of conduction heat transfer in the solid region of a rod.

ACKNOWLEDGMENT

This work was supported by Nuclear Research & Development Program of Korea Science and Engineering Foundation (KOSEF) grant funded by the Korean government (MEST). (grant code : M20702040005-08M0204-00510)

REFERENCE

[1] H. T. Kim, B. W. Rhee, J. H. Park., CFX simulation of high temperature thermal-chemical, Annals of Nuclear Energy Nuclear Energy, p677-689, 2008
[2] M. W. Kim, Y. S. Choi., CFD Prediction of Two-phase Fluid Flow Characteristics Inside a Fuel Channel of a CANDU-6 Reactor, Korean Nuclear Society Autumn Meeting, 2005
[3] ANSYS, Inc., ANSYS CFX-Solver, Release 10.0: Theory, Canonsburg, 2005a
[4] ANSYS, Inc., ANSYS ICEM CFD, Release 10.0: Tutorial Manual, Canonsburg, 2005b