RESEARCH ON FLOW CHARACTERISTICS FOR THE OPTIMAL DESIGN OF EXPERIMENT FOR THE TYPICAL PRESSURED WATER REACTOR

G.L. Chen, Z.J. Zhang, Z.F. Tian*
Harbin Engineering University
National Defense Key Subject Laboratory for Nuclear Safety and Simulation Technology,
Harbin Engineering University, Harbin 150001, China
chenguangliang.heu@gmail.com; zhangzhijian@hrbeu.edu.cn; tian_zf@hrbeu.edu.cn

X.M. Dong
Harbin Engineering University
National Defense Key Subject Laboratory for Nuclear Safety and Simulation Technology,
Harbin Engineering University, Harbin 150001, China
dongxiaomeng@hrbeu.edu.cn

ABSTRACT

Experiment can be used for the research on the flow physical process of coolant in the typical pressured water reactor which is with complex structures, such as spacer grid, mixing vane and narrow channels. Usually, the data obtained from the experiments can also be used as the CFD benchmark. However, this kind of approach could not be avoided to use some simplification and approximation: only several rods with special structures are used; solid walls are imported; gaps between the walls and the rod bundle are created. The influence of these operations has not been considered and discussed clearly, though the influence certainly exists. To obtain the quantitative and qualitative understanding, this research has done some work to study the impact created by the region size, region boundary and region shape in the experiment. The regularities found in this work can be used as the guidance on the optimal design of experiments so that the data is more near the real physical process in a reactor.

OPTIMAL DESIGN, EXPERIMENT, PRESSURED WATER REACTOR

1. INTRODUCTION

Nowadays, the diminishment of the safety margin and the improvement of the safety assessment of nuclear power plant are needed greatly. One reason is that many nuclear reactors are at the last stage of their lifetime. Another reason is that both the nuclear resource and the public acceptability are limited. So the power uprate, lifetime extension, higher fuel burnup and high-fidelity predictive capability for component performance have been the focuses of research team and program, like CASL. However, all of the objectives delay on the deep insight of the physical process of the coolant in the reactor.

The experiment is one of the most credible approaches to study the physical process in reactor. Recently, many experiments have been built for the benchmark building and simulation improvement, such as the work done by the Westinghouse and Texas A&M for CASL[1,2], and the work done by OECD/NEA[3].

However, the complex structures of a reactor increase the difficult to carry out the experiments. The subchannel, spacer grid and mixing vane are the important regions in a fuel assembly of a typical pressured water reactor. The subchannel where the coolant exists in is narrow; the spacer grid which supports the fuel rods through the springs and dimples is complex; the mixing vane what disturbs the flow and optimize the heat transfer is also with complex structure. Furthermore, the number of these structures in a reactor is all very large. The large number, complex and narrow structure feather will create great difficulties on the measurement in an experiment. So a local small region with several fuel rods, a special region shape and boundary is always been designed as the research object in an experiment. To get a better understand of the physical process, some important issues have to be considered well.

Firstly, the research objects with various kinds of region sizes have been used in previous experiments. For example, 5×5 rod bundle is used for the benchmark research on the flow field of coolant[1,2], 2×2 rod bundle is used for the research on CHF to consider the influence created by the angles and positions of mixing vane[4], 2×2 rod bundle is also used for the research on the distribution and turbulent of coolant[5], 2x3 rod bundle is used for the research on CHF to consider the influence created by the pressure and mass flux of coolant[6], 4×4 rod bundle is used in research[7], 10x10 rod bundle is used for the research on the two phase flow[8]. However there are seldom discussions on the accuracy of the experimental results. It's necessary to investigate whether, why and how much the region size will create effect to the flow and heat transfer process of the coolant. The conclusions based on the investigation will provide the guidance for the design of the region size used in the experiment.

Another matter needs to be considered well is that the import of the solid walls surrounding the fluid domain of coolant, which will cause a certain effect to the fluid domain in an experiment. However, the flow house including solid walls could not be avoided to be used in each experiment. It is necessary to be clear how large and the reasons of the differences exist between the measurement and the real physical process in a reactor, so that we can avoid or modify the error in the experiment.

What's more, gaps between the solid walls and the spacer grid or rod bundle will also be generated in the experiment. These gaps are not with the same magnitude due to the location error or the design of bypass channel. For example, the experimental object[2] is with a bypass channel, but other objects in this research team are with equal gaps of 0.5mm surrounding the bundle[1]. The region shape of the fluid domain is usually different in different research objects. So another focus is to consider whether the region shape with different fluid domain of gaps will create influence on the real physical process of the coolant in the rod bundle of a reactor. If the influence is obvious, it is also needed to be studied to give the guidance on the design of the establishment of an experiment.

2. OBJECTIVES AND APPROACHES

2.1. Research objectives

According to the previous description, the objective of this paper is to obtain the guidance for the optimal design of the experiment, and the guidance is based on the detailed information of the physical flow process of the coolant in pressured water reactor with typical spacer grid and mixing vane. The analysis on the region boundary, region size and region shape can increase the understanding of the physical process occurred in the experiment. To realize the objective, this research needs to analyze the impact on the flow characteristics from these three aspects.

2.2. Research approaches

In this research, the analysis of the flow characteristic in the experiment is done by the CFD simulation. However, the experimental conditions were applied in this approach, so that it can reflect the influence created by the experimental conditions.

To get a better understanding, the flow physical process has to be reflected. In this research, some characteristic parameters are chosen and designed. They are the turbulent intensity, the secondary intensity and the pressure drop. The turbulent intensity and secondary intensity are used for the indication of the lateral and turbulent mixing state respectively. The pressure drop is used for the indication of the flow resistance. The definitions of these parameters and the reasons of the choice are introduced as follow.

(1) Secondary flow intensity

The expressions of the secondary flow intensity can be written as Eq.1. I_{sec} stands for the secondary flow intensity. V_{xi} and V_{yi} stand for the lateral velocities of local position at the x and y directions respectively. $\bar{V}_z$ stands for the mean axial velocity. A_i stands for the area of cross section in each local cell, and A is the sum of the areas of cross section.

$$I_{\text{sec}} = \frac{1}{A}\sum_i \frac{A_i\sqrt{V_{xi}^2 + V_{yj}^2}}{\bar{V}_z} \tag{1}$$

(2) Turbulent intensity

The expressions of the turbulent intensity (TI) can be written as Eq.2. I_{tub} stands for the turbulent intensity. V_x', V_y', V_z' stand for the pulsation velocity at x, y, z directions. $\bar{V}$ is the mean velocity.

$$I_{tub} = \frac{\sqrt{\frac{1}{3}\left(\overline{V_x'^2} + \overline{V_y'^2} + \overline{V_z'^2}\right)}}{\bar{V}} \tag{2}$$

Based on the definition of TI, we can see that TI is just for the physical process in one single cell. Only the distribution of TI in a surface has a practical value for the flow physical research. However, the comparison of the distribution is inconvenient. The distributions may be the same in one's eyes, however there may be still some little differences. The mathematic expression for one parameter may be better when we want to find the differences. So this research designed an expression for TI. The expression can be written as Eq.3. I_{tub}' stands for the area weighted average value of turbulent intensity of a cross section (TICS). The subscript i is the mark number of each mesh cell. A_{i} and A have the same meaning as the explanation in the expression of SI.

$$I'_{tub} = \frac{1}{A}\sum_i \frac{A_i\sqrt{\frac{1}{3}\left(\overline{V'^2_{xi}} + \overline{V'^2_{yi}} + \overline{V'^2_{zi}}\right)}}{\overline{V_i}} \tag{3}$$

(3) Pressure drop

The large number of complex structures and narrow channels also create obvious pressure drop. The more pressure drop means the more energy the primary pumps must provide to drive the coolant, which will affect the economy of the nuclear power plant. So the pressure drop is another important parameter chosen for the research on the flow physical characteristic.

The pressure drop is made up by friction resistance loss and local resistance loss. However, these two kinds of resistances are not distinguished obviously in the flow process of coolant in reactor. The local resistance is mainly created in the grid region (spacer grid and mixing vane), however the lateral flow between adjacent subchannels also create it. Meanwhile, the friction resistance exists in all length of the flow channel. So the pressure drop is analyzed not based on separated principles.

The computation of the total pressure drop can be written as Eq.4. The area weighted average value of pressure is used in the equation. The P in the equation stands for the total pressure which is the sum of the static pressure and the dynamic pressure. The subscripts 1 and 2 stand for the mark of the cross section chosen for the comparison. The subscripts i and j stand for the mark of the mesh cell in the selected section. A_{sum} stand for the sum of the areas of cross section.

$$\Delta P = \frac{\sum_i A_i P_{i\_1}}{A_{\text{sum1}}} - \frac{\sum_j A_j P_{\text{j\_2}}}{A_{\text{sum2}}} \tag{4}$$

3. RESEARCH ON FLOW CHARACTERISTICS

3.1. Analysis on region boundary

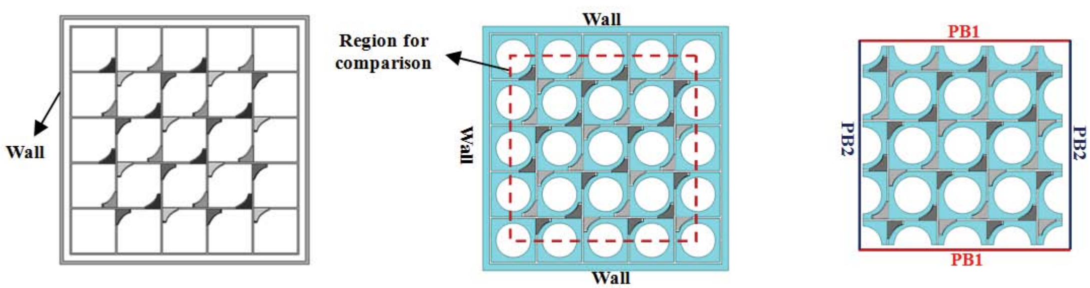
The influence of region boundary could not be avoided in experiments. This kind of influence is created by solid walls surrounding fuel rods and coolant as shown in Fig. 1 (a). The spacer grid and coolant have to be limited in a transparent house environment so that the measurements can be done by PIV technology as the introduction done by [1]. Meanwhile, the region size in an experiment could not be designed too large, or the difficult on measuring will increase greatly. Generally, the rod bundle in an experiment is just with several rods (such as 5×5, 3x3). Both the experimental objects are 5×5 rod bundle with solid walls in the establishment of the CFD benchmark for CASL and OECD/NEA, so this research choose a 5×5 rod bundle with solid walls as one of the research objects.

For an experimental research, the coolant region is designed with solid wall. However, the real physical condition of a typical pressured water reactor is that there are hundreds of fuel rods in one assemble, and hundreds of assembles in one reactor. Most important is that there is no solid wall around local fuel rods. So the periodic boundary will be most suitable for the local research object in the CFD simulation, if it is just for the flow characteristic analysis. However, this boundary is also just a nearly real boundary for the local flow process in a reactor. Usually, the results of an experiment are used as the benchmark for the CFD simulation to find better physical models and simulation methods & schemes. However there are some questions:

- Are these two kinds of research approaches equal?
- Which approach is better for the prediction of the real flow process?
- Is it suitable if we choose the experimental results as the benchmark for the CFD simulation?

The following part will give the analysis on these questions.

This research chose a 5×5 rod bundle as the research object based on Karoutas's experiment as shown in Fig. 1 (a), and the detail geometrical parameters of this object can be found in the work done by [9]. The coolant region for the CFD simulation is surrounded by the solid walls to model the experimental condition as shown in Fig. 1 (b). Another research object is shown in Fig. 1 (c). It is the inner region of the object of Karoutas's experiment. The CFD simulation will be done with the periodic boundary. The former one is for cross section with 5×5 rods and 4 solid walls. The other one is for the cross section with 4×4 subchannels and periodic boundary. The comparison could not be implemented, because the cross sections are different. So the inner region of 4×4 subchannels was separated by the red lines from the simulation region as shown in Fig. 1 (b).



(a) Experimental object **(b) Coolant region with EB** **(c) Coolant region with PB**

Fig. 1. Resear Objects with Different Boundaries

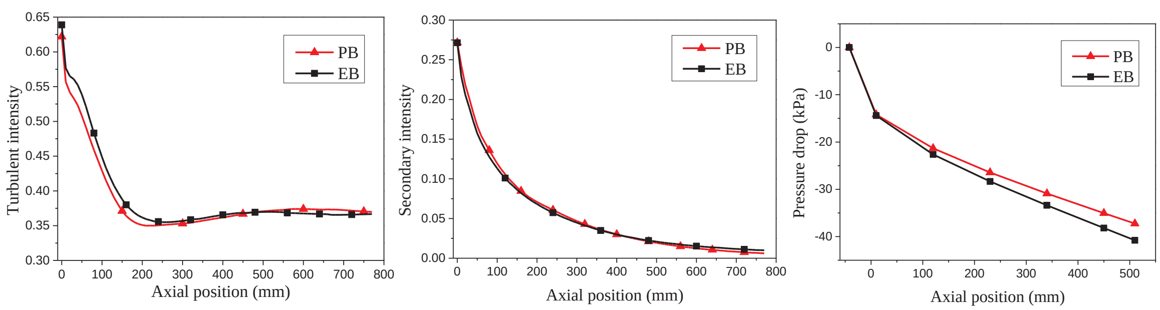
The flow mixing status and the flow resistance of the simulations with experimental and periodic boundaries can be seen from Fig. 2. The difference of SI is not obvious, but it also can be seen. The differences of TICS can be seen more obviously. The profiles of pressure drop for the two boundaries coincide well at the location where the spacer grid exists (-50~0 mm), however they will be different after the location with mixing vane (0~510 mm).

The regularities and the differences are mainly decided by the structures of the flow channel. The spacer grid and mixing vane will create great impact on flow resistance. The swirl and turbulent flow status was created and increased by the mixing vane.

At the location (-50~0 mm), the grid and mixing vane exist. The SI and TICS increase more at the simulation with each boundary. The TICS and the SI reach at the highest value when the coolant leaves the location with mixing vane (0mm). Furthermore, the influence of the surrounding walls is limited at this location, because the obstruction of the spacer grid between the walls and the inner coolant, thus the values of SI, TICS and pressure drop are nearly the same at the simulations with EB and PB.

After the location (0 mm), the disturbance of the mixing vane will disappear when the coolant flow away from the mixing vane. So TICS and SI decrease immediately when there is no obstruction (0~200 mm). At the location (200~510 mm), the TICS will keep in a value range. However, the SI will continue to decrease with slower speed. The friction resistance of the surrounding walls will be the reason for the flow characteristics in this location. The boundaries of EB and PB create different TICS, which can be seen in Fig. 2 (a). On the contrary, the results of SI with EB and PB coincide well as seen in Fig. 2 (b). The reason is that SI is affected mainly by the lateral force. However the force created by surrounding walls is mainly on the vertical direction, thus the difference is little for SI. What's more, TICS can be affected by the force in any direction, thus the difference exists in TICS. The difference of pressure drop exists in the simulations with boundaries of EB and PB. The difference is more obvious and it will increase if the coolant flows on. That means the solid walls in experiment will result in misunderstanding if the distance exceeds a certain length at the direction of the flow. Compared with the use of EB boundary, the simulation with PB may be more suitable to study the flow resistance, because the influence of the solid walls is avoided. Meanwhile, the experimental data may be closer to the real value if the axial length of the flow channel was limited at a certain length in an experiment.

It can be known from the analysis above: 1) the experimental approach may be not the best research approach, because the wall boundary around the rod bundle could make obvious different effect to the flow characteristics compared with the nearly real boundary; 2) the wall boundary creates obvious influence on the resistance and TICS; 3) the wall boundary makes less effect on SI.



(a) TICS profile **(b) SI profile** **(c) Pressure profile**

Fig. 2. Analysis on the Flow Characteristics with Different Region Boundaries

3.2. Analysis on region size

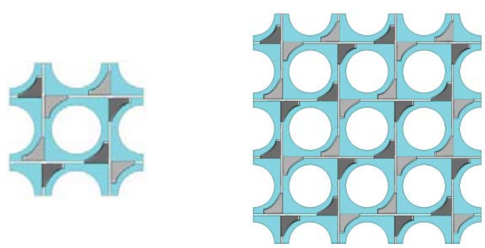
As the analysis above, the rod bundle could not be designed with many fuel rods in an experiment. Usually 5×5 or 3×3 rod bundle are chosen as the research object. The principle of the choice and design hasn't been introduced and analyzed. This research has paid some effort to analyze the influence of the region size of rod bundle in experiments.

The simulation objects with 4×4 and 2×2 subchannels were chosen for the research as shown in Fig. 3. The first objective in this part is to find whether the region size will create influence on the flow characteristics. These two regions were both with the periodic boundary.

The results of the simulations using different region sizes coincide well at different axial positions as shown in Fig. 4. The various trends of each main parameter for 2×2 and 4×4 subchannels are nearly the same. However the differences still exist though they are very little. The difference value of 2×2 and 4×4 subchannels for SI and TICS can be seen in Fig.5. The source of the difference may come from the numerical error or the influence created by the region size. Two feather stages can be found in Fig.5. They are one horizontal fluctuated stage (L_1 in Fig.5) and one inclined fluctuated stage (L_2 in Fig.5). However, there should be only one feather stage if the differences of SI and TICS are mainly created by the numerical error. So the differences are most probably created by the region size. That means different region size may create different physical mechanism.

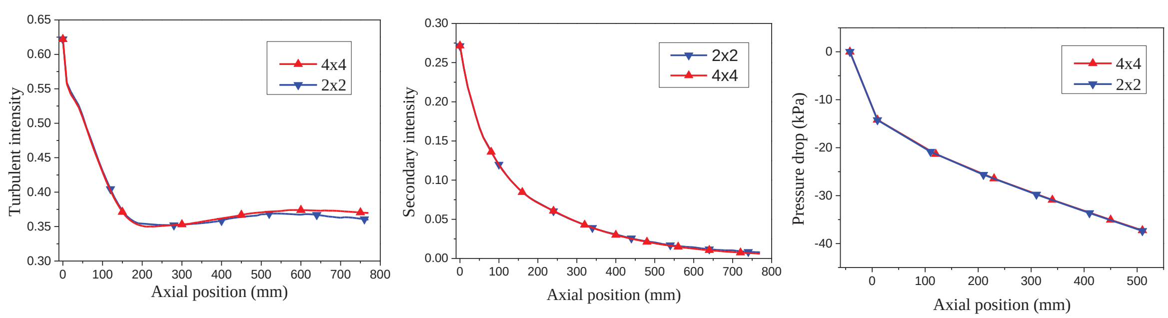
The coolant will have the lateral velocity when it flows through the mixing vane. The physical mechanism will be nearly the same at the simulations with 2×2 and 4×4 subchannels in a certain axial distance (L_1, the first flow stage), because both regions can reflect the lateral scope of lateral velocity's influence. Then the region with 4×4 subchannels is more suitable for the simulation at the next flow part (L_2, the second flow stage), because the lateral scope of lateral velocity's influence has already exceeded the area of 2×2 subchannels that the real physical mechanism cannot be reflected well.

What's more, the simulations with 2×2 and 4×4 subchannels are performed just for one span of spacer grid and coolant subchannel. The difference between the two sizes for SI or TICS will be enlarged after the coolant flows through a whole length of an assemble which is made up with several spans of spacer grid, thus the experiments based on a region with large lateral scale is very necessary, especially for the research with long axial length.



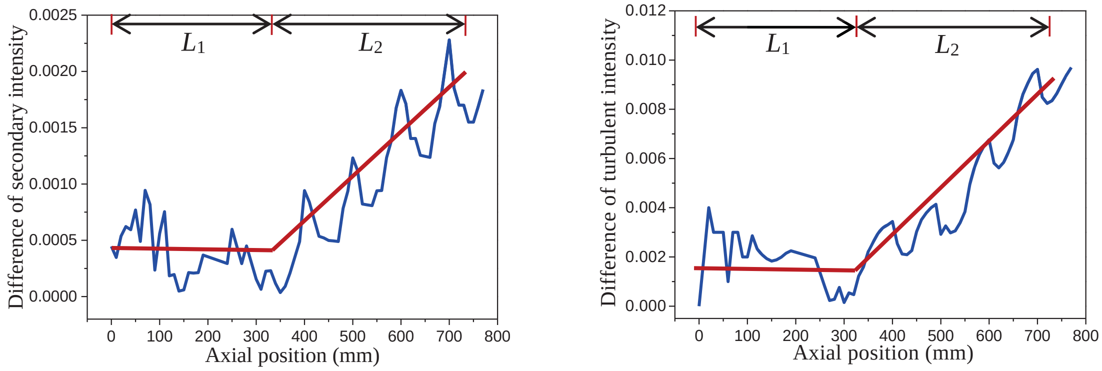
(a) 2×2 **(b) 4×4**

Fig. 3 Research Regions with Different Region Sizes



(a) TICS profile **(b) SI profile** **(c) Pressure profile**

Fig. 4. Comparison for Different Characteristic Parameters

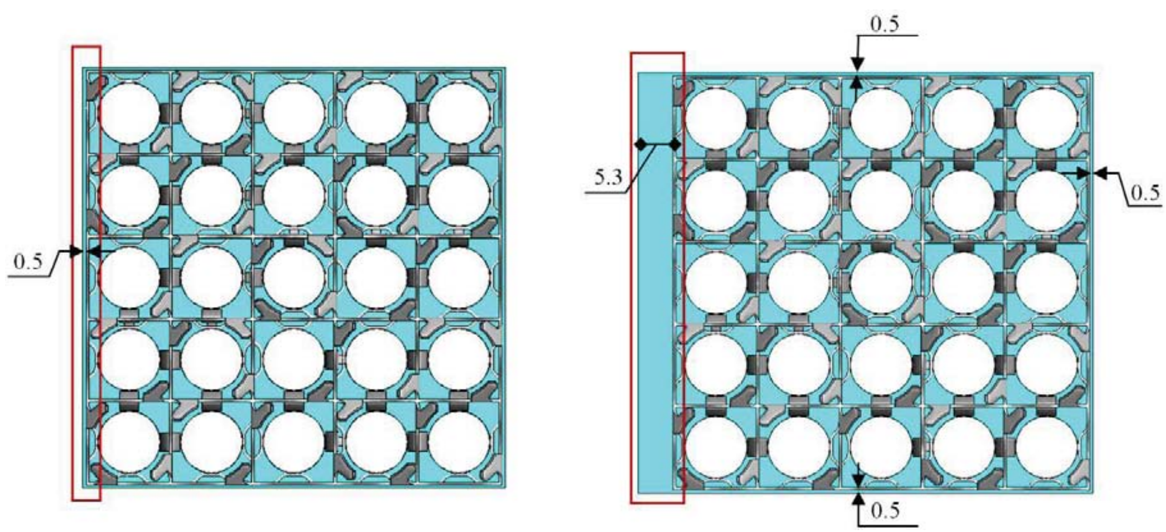


(a) Difference of SI **(b) Difference of TICS**

Fig.5. Difference of SI and TICS Between 2×2 and 4×4 Rod Bundle

3.3. Analysis on region shape

The structures of spacer grid and mixing vane are always treated carefully in experiments, because they will bring influence into the flow and heat transfer process. In addition, the region shape is another important condition which will also create impact on the flow characteristics of coolant. However, this condition gets not enough attention. The structures of spacer grid and mixing vane can be designed similar to the real structure in an experiment. However, the creation of differences could not be avoided for the region shape. For example, the rod bundle and spacer grid have to be settled in a transparent house in an experiment. The gap will be generated as shown in Fig.6 (a)[1]. Sometimes, bypass channel is used in the experiment as shown in Fig.6 (b)[2]. So the region shapes of the coolant domain are different in some experiments. The following part will discuss whether this condition is able to create influence and whether the influence can be ignored.

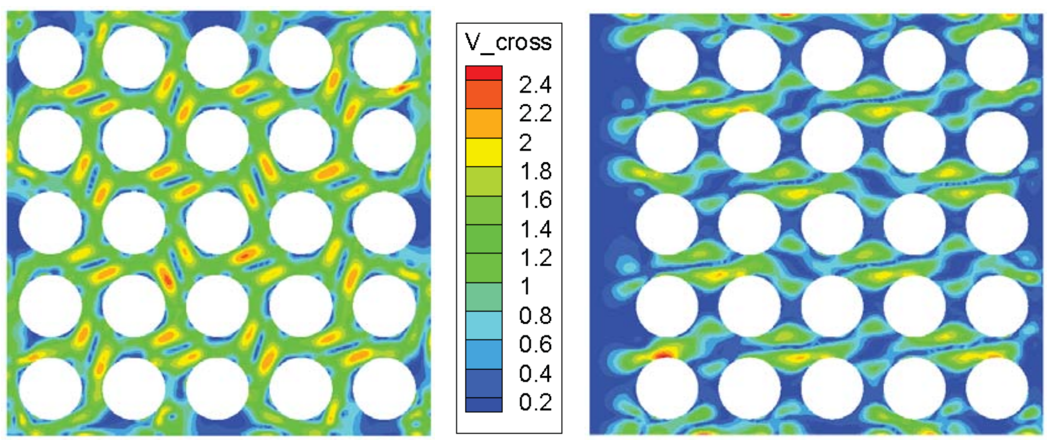


(a) Region with equal gap **(b) Region with bypass channel**

Fig.6 Region Shape of Different Research Objects

The results of the simulations with these two kinds of region shapes are listed in Fig. 7, and we can name them as R1 and R2 respectively. As the comparison, the result of the simulation for 4×4 subchannels with PB is listed in Fig. 8, and we name it as R3. R3 is not affected by the surrounding walls, and it can be taken as the nearly real flow field. The gaps between the house walls and the spacer grid are little and equal for the object in Fig. 7 (a). We can see the similar distribution of the flow field exists in R3 and in the same region scope of R1. The different of R1 and R3 is mainly created by the different structure of mixing vane, which can be seen in Fig.6 and Fig. 8. However, the distribution of the flow field in R2 is obviously different from R1 and R3. It can be easily understood that the bypass region is with less flow resistance compared with the grid region. Then less pressure will be created in bypass region, so the flow field is different from R1 and R3.

For the similar reason, the influence of the pressure will also be imported if the gaps are not little enough or they are not equal. That means the region shape of the coolant is also needed to be designed carefully in experiments. The gaps between the walls and spacer grid must be little enough and equal enough. Otherwise the misunderstanding of the flow characteristics will be created.



(a) Region with equal gap **(b) Region with bypass channel**

Fig. 7 Comparison of the Results for Different Region Shapes

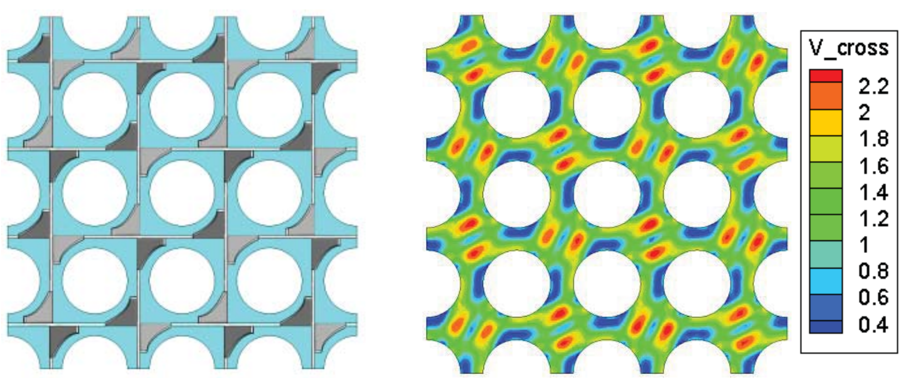


Fig. 8. The Structure and the Result for the Comparison

4. CONCLUSIONS

The influence created by region boundary, region size and region shape of the research object has been considered in this paper. Some regularities have been found, and they can be used as the guidance on the design of experiment for the better understanding of the real physical process in a reactor.

Firstly, from the research on resign boundary, we can know: 1) SI can be utilized with high confidence in the experiment. SI is mainly corresponding with the mixing vane, so the design and optimization of the mixing vane can be considered well in the experiment. 2) Pressure drop obtained in an experiment needs to be modified before it is used for the reflection of the resistance characteristic in a real reactor. Meanwhile the error will be more as the increasement of the length of the flow channel, so the length has to be limited for the research on resistance. Secondly, from the research on region size, we can know: 1) Difference exists between different region sizes. The difference will continue to increase as the coolant flows on. 2) The region size should be designed carefully based on the structure feature of mixing vane and the distance between the two spans of spacer grids. 3) Large region size is necessary if the condition permits, because it is better for the reflection of the real flow characteristics. Thirdly, from the research on region shape, we can know: 1) the basic flow distributions are obviously different for different region shapes of the coolant domain. 2) The design of the distances between the rod bundle and the corresponding wall of the flow house should be little and equal, and the location error of the rod bundle also should be kept little enough.

ACKNOWLEDGEMENTS

This paper is funded by the International Exchange Program of Harbin Engineering University for Innovation-oriented Talents Cultivation. Special thanks are to Prof. Zhang Z.J. and Prof. Tian Z.F. for their continuous encouragement and guidance. The authors also appreciate deeply the discussion and suggestions from Li L., Li X.C., Dong X.M., Xu J.Y. of Harbin Engineering University. The authors are grateful to all of them.

REFERENCES

1. M.E. Conner, Y.A. Hassan, E.E. Dominguez-Ontiveros, "Hydraulic benchmark data for PWR mixing vane grid," *Nucl. Eng. and Des.*, **264**, pp. 97–102 (2013).
2. E.E. Dominguez-Ontiveros, Y.A. Hassana, M.E. Conner, Z. Karoutas, "Experimental benchmark data for PWR rod bundle with spacer-grids," *Nucl. Eng. and Des.*, **253**, pp. 396–405 (2012).
3. C.H. SONG, H.C. KIM, "OECD/NEA Experts Activities on the Application of CFD Codes to Nuclear Reactor Safety Problems: Issues and Future Direction," *Transactions of the Korean Nuclear Society Autumn Meeting Gyeongju*, Korea, Oct. pp. 29-30 (2009).
4. B.S. Shin, S.H. Chang, "Experimental study on the effect of angles and positions of mixing vanes on CHF in a 2×2 rod bundle with working fluid R-134a," *Nucl. Eng. and Des.*, **235**, pp. 1749–1759 (2005).
5. S. Hosokawa, T. Yamamoto, J. Okajima, A. Tomiyama, "Measurements of turbulent flows in a 2×2 rod bundle," *Nucl. Eng. and Des.*, **249**, pp. 2– 13 (2012).
6. B.S. Shin, S.H. Chang, "CHF experiment and CFD analysis in a 2×3 rod bundle with mixing vane," *Nucl. Eng. and Des*., **239**, pp. 899–912 (2009).
7. J. Chen, B.D. Chen, H. Zhang, "CFD Method Research on Characteristic Cells in Rod Bundle Fuel Assembly," *Nucl. Pow. Eng.*, (China), **32**(3) pp. 68-72 (2011).
8. T. Arai, M. Furuya, T. Kanai, K. Shirakawa, "Development of a subchannel void sensor and two-phase flow measurement in 10×10 rod bundle," *Int. Jour. of Multiph. Flow*, **47**, 183–192 (2012).
9. M.A. Navarroa et al, "Evaluation of a numeric procedure for flow simulation of a 5×5 PWR rod bundle with a mixing vane spacer," *Prog. in Nucl. Ener.*, **53**, 1190 (2011).