CFD ANALYSIS OF FLOWING FIELD IN 5×5 ROD BUNDLE WITH MULTI-GRIDS

J.B. Zhao, X.Y. Zhang[1], L. Qiao
School of Electric power, South China University of Technology
Wushan Tianhe, Guangzhou 510640,China
zhaojingbo0119@126.com; zxy1119@scut.edu.cn; 858806770@qq.com

ABSTRACT

To study the characteristics of flowing field in fuel bundle, the 5×5 rod bundle with one spacer grid and two mixing grids has been taken for investigation. Geometry model of the mixing vane grids is simplified to be made of mixing vane, steel convex, spring and stripe. The 3-D flow field and heat transfer have been simulated with the CFD code. Much effort has been made to analyze the effect of different grid type and geometry on flowing field in the bundle channels. Also the turbulent mixing performances of different type of grids have been investigated. What's more, the effect of mixing vane on the flowing field, with change of its deflection angle and spread length, have been investigated. The research shows that the spacer grid in the upstream has stronger effect on the flowing field than spacer grid in the downstream. Both the mixing vane grids and support grids have positive effect on heat transfer enhancement. Geometry of the vanes, the deflection angle and length of mixing vanes, has significant effect on cross flow between the flowing channels, increase the deflection angle and length of mixing vanes will make the lateral mixing stronger, but enhance the pressure drop at the same time.

KEYWORDS
5×5 rod bundle; multi-grids; CFD analysis; flow field; heat transfer

1. INTRODUCTION

Fuel assembly is the most important structure in the nuclear reactor core, the mixing vane grids on the assembly not only acts to support the fuel bundle, but also strengthen cross flow between flowing channels, that is helpful for cooling of hot rod. The widely used grids in the PWR fuel assembly have two types, support grids and mixing vane grids. The support grid is shorter and doesn’t has mixing vane while the mixing vane grids is longer and be fixed with groups of mixing vanes at the top. The mixing vane at top of the mixing vane grid has effect of enhancing cross flow and heat transfer. Concerning safety design of the fuel assembly, research on the cross flow mixing and heat transfer characteristics of rod bundles has attracted focus of study in nuclear researches for past decades to improve the safety and thermal-hydraulic performance of fuel assemblies.

[1]Corresponding author: X.Y. Zhang

Much research has been made on the flowing characteristics in downstream of a single spacer grid with CFD analysis. C.C.Liu, et.al [1] had studied the effect of turbulence model on convective heat transfer coefficient in downstream of the 5×5 spacer grids mixing vane, and proposed that the SST k-ω model has been the best turbulence model in CFD simulation of fuel assembly. M.Holloway[2] has studied the influence of spacer grid on convective heat transfer in downstream area of the grid, a general formula for the convection heat transfer coefficient in downstream of the spacer grid has been proposed. The CFX code has been used to study the axial flowing velocity and pressure drop of a 5×5 spacer grid in research of M.A.Navarro, et.al [3]. K.Ikeda, et.al [4] has studied the cross flow and axial flow in downstream of a 5×5 spacer grid in high temperature and high pressure condition, the mixing vane and steel convex has been considered in geometry of that work, and the location of DNB has been studied. E.D Elvis, et.al [5] has studied simulation of flowing velocity in downstream of the spacer grid using a CFD code, and compared the simulation results with experiment. S.K.Chang, et.al [6] has applied a CFD code to study the flowing mixing characteristics for different geometry types of mixing vanes. Beside study on flowing in downstream of grids, some researchers also concern about flowing field in upstream of a spacer grid. X.W.Yu, et.al [7] has calculated the axial flow velocity and the cross flow velocity in upstream and downstream of a 2×2 rod bundles spacer grid with CFX program. Also C.W.Hong, et.al [8] and T.R.Feng et.al [9] have used the CFX code to calculate flowing velocity in upstream and downstream flow field of a 5×5 rod bundle's spacer grid. Some other works have put attention on comprehensive effect of geometry structure of spacer grid such as G.Ye, et.al [10], who has calculated the flowing field and heat transfer of a full lenghth 2×2 rod bundles with FLUENT code, where the geometry is composed with seven spacer grids. A.Gandhir et.al [11] and M.E.Conner, et.al [12] have also considered a comprehensive geometry of rod bundle in CFD study of flowing field of a 5×5 rod bundle, where mixing vane, steel convex, spring and stripe have been included in the mixing vane spacer grids.

Considering the complex geometry of grids, most published work on CFD study of rod bundle only considered mixing vanes in grid geometry, and usually only studied effect of one single grid. The concerned topic in CFD study is mainly about the turbulent model and mixing characteristics of the spacer grids. The more precise geometry is necessary for accurate simulation of flowing field in a fuel assembly, as different geometry parts has different effect on the flowing field, and a quantitative coefficient for mixing is important for evaluating lateral mixing of fuel assembly. On the other side, the actual fuel assembly contains multi grids of different type.

For CFD simulation of flowing filed of fuel assembly with a more precise geometry model, this paper will study flowing field of a 5×5 fuel assembly including two mixing grids and one spacer grid. Geometry of the mixing grid includes mixing vane, steel convex, spring and stripe, that of the spacer grid is similar except not includes mixing vane. A mixed mesh model has been established and the CFD simulation has been carried out to study the flowing and heat transfer characteristics of the 5×5 fuel assembly. A cold test case of flowing in the 5×5 fuel assembly by N. Cinosi, et.al [13] has firstly been taken for verification, and the computed lateral velocity is compared with the test results. The second computation considers a heated condition of the same 5×5 fuel assembly. The flowing field and heat transfer characteristics of the fuel assemblies with whole geometry parts have been investigated, the turbulent mixing characteristics and pressure drop of two types of grids: spacer grid and mixing grid, were studied in detail. The mixing coefficient will be presented as a quantitative evaluation of cross mixing.

2. GEOMETRIC MODEL AND MESH

The actual structure of mixing grid is very complex, which is made up with mixing vane, steel convex, spring and stripe, as shown in figure 1. The stripe is the skeleton frame of the mixing vane on the fuel assembly. The mixing vanes are fixed on top of the grid, which have the function of lateral mixing to enhance cross flowing and heat transfer between neighboring channels. There are two vanes set in opposite direction on the middle stripe, and one single vane located on the edge stripe. Steel convexes are set on top and bottom of the grid, with an arc shape. There are eight convexes in every flowing channel. Springs are fixed in interior side of a stripe along the length. Both steel convexes and springs act to fix bundles in the fuel assembly. Geometry of the spacer grid is a little different with the mixing grid. It's commonly located at the front and back ends of fuel assembly. Geometry of the spacer grid is made of stripe, steel convex and spring, without mixing vane.

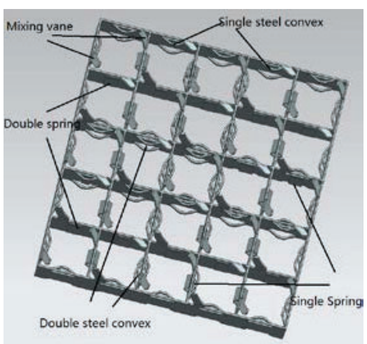


Fig.1 Sketch of mixing vane fuel

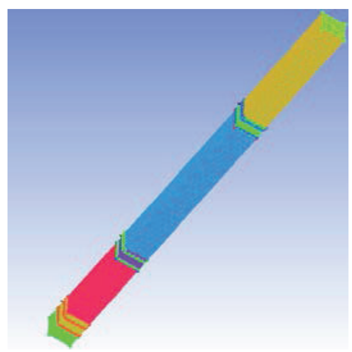


Fig.2 Sketch and mesh of 5×5 fuel assembly

Two types of grids, spacer grid and mixing grid are fixed on the 5×5 fuel assembly. The first one is spacer grid which is located at the bottom, and the other two are mixing grids. Distance from the spacer grid to the first mixing grid is 260mm, that is 520mm between the two mixing grids. To investigate the flowing characteristics in upstream and downstream of the three grids, computation domain is taken from front of the first grid to downstream of the third grid. The peripheral size of computation domain is 65 mm×65 mm×1200 mm. The geometry model and meshes of the 5×5 rod bundle assembly is shown in figure 2.

Based on geometry complexity of the 5×5 rod bundle assembly, a compound type of mesh has been established in the flowing field. The tetrahedral mesh is used for mixing vane grids, while the hexahedral mesh is used for spacer grid and all other area. The generated mesh in fluid domain of the 5×5 rod bundle assembly is shown in figure 2.

To study the size sensitivity of meshes, four mesh sizes have been used for CFD solution, from 0.5mm to

4mm. The resulted mesh number and maximum difference of axial velocity and lateral velocity comparing with results of 0.5mm mesh size have been shown in Table.1. Comprehensively considering the result difference and solution time, the mesh size was set to 1mm for CFD solution.

Table.1 Sensitivity analysis of mesh size

maximum mesh size (mm)	**mesh number**	**Maximum difference of axial velocity($u/U_{0.5}$)**	**Maximum difference of lateral velocity($w/W_{0.5}$)**	**Solution time (min)**
0.5	11283877	0	0	350
1	8086472	3.18%	2.55%	264
2	7208496	4.94%	4.83%	236
4	6702348	7.15%	6.92%	178

3 . CALCULATING MODEL SELECTION

A suitable calculating model is also important for CFD solution to give a precise result. Calculation in this work is steady state. In our solution, the whole flowing zone is regarded as a fluid domain, the working medium is water. Hydraulic diameter of each flowing channel is 9.76mm. Flowing velocity at entrance of fuel assembly is 1~1.5m/s. Entrance coolant temperature is 35℃ in cold test case, but is 300℃ in heated fuel rod case. And Reynolds number at entrance is 80100. A non slippery boundary is used for velocity on the bounding face of flowing domain and surface of fuel rods. At outlet of fuel assembly, a pressure balance requirement is assumed. As for the turbulence model, we take the SST k-ω model as this model could better simulate separation of vapor from the liquid coolant in condition of high rod heat flux. A high resolution scheme has been used for advection term. The RMS model is adopted to evaluate the residual error, and the error limit is set to be 10^{-5}.

4. VERICICATION CALCULATION IN TEST CASES

The first simulation case is the test case in non heated, medium pressure condition of N.Cinosi, et.al research [13]. Comparing the CFD simulated cross velocity with experiment results to validate the adaptability and precision of geometry and calculating model. The entrance coolant velocity of the computational domain is 1.5m/s, entrance coolant temperature is 35℃. Cross flowing velocity on cross-section $0.5D_H$ downstream of the spacer grid has been measured with LDV in work of N.Cinosi, et.al. The flowing data are taken from gap y1 and y2. Gap y1 is between the central and medium row's fuel rods, and gap y2 is between the outmost and medium row fuel rod. The axial and lateral position is shown in figure 3.

CFD simulation with same entrance data and boundary condition has been made in this work. The computed vector chart of cross velocity is shown in Fig. 4. Vortex pair of velocity vector can be found close to the mixing vane. The computed cross velocity on gap y1 and y2 has been compared with test data of N.Cinosi, et.al. in Fig.5, one can see well agreement between the two groups of results.

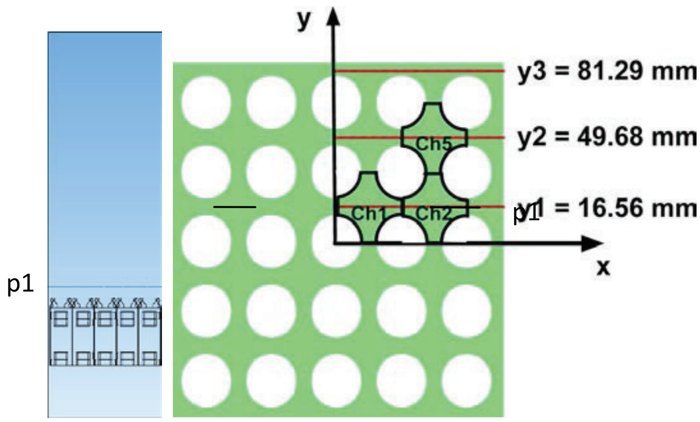


Fig.3 Location of study sections

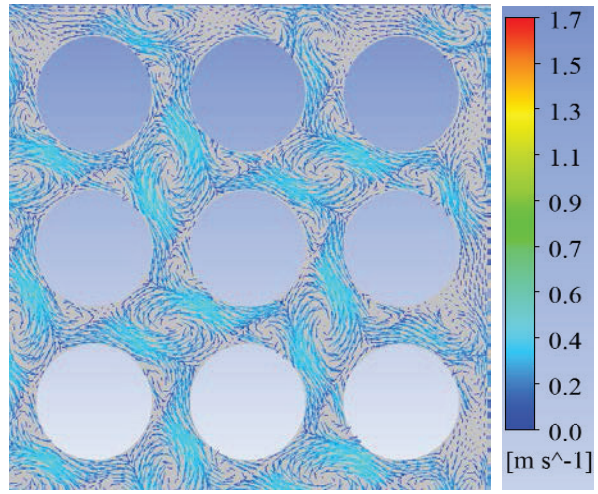


Fig.4 Velocity vector on axial section p1

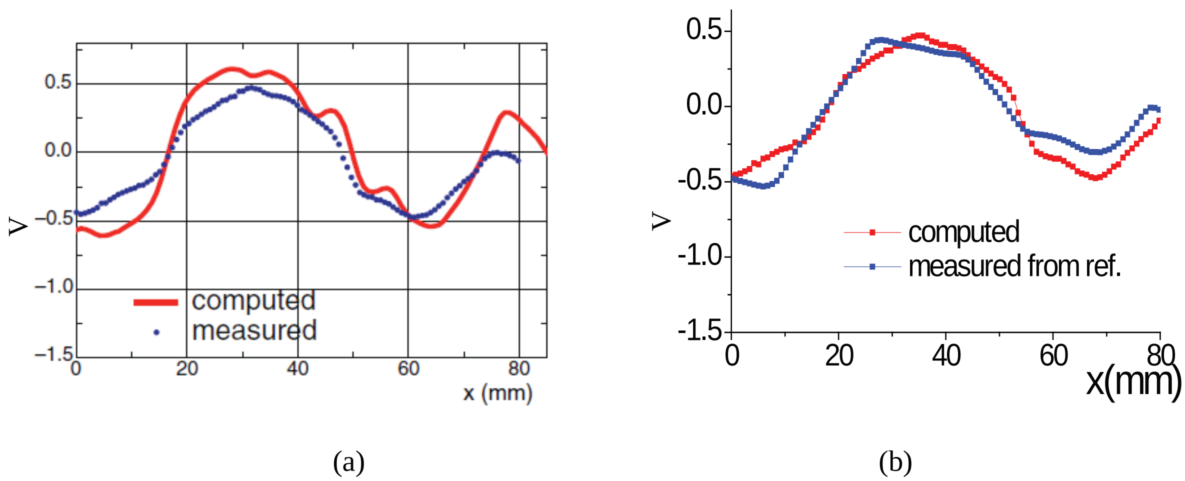


Fig.5 Comparing computed lateral velocity with tested results. (a) line y1; (b) line y2

5. CALCULATION OF FLOWING FIELD OF 5×5 FUEL ASSEMBLY WITH THREE GRIDS IN HEAT CONDITION

5.1 Comprehensive simulation result of flowing field

In order to obtain the comprehensive flowing characteristics of the 5×5 fuel assembly with three grids in working condition of PWR, a heated case has been taken for simulation. Pressure for that case is 15.5MPa. Non-uniform radial heat power is considered as shown in Fig.6, where the central rods have a peak heat flux that is 1.0MW/m^2, the intermediate rods have a peak power that is 0.8 MW/m^2, the surrounding rods have a peak power that is 0.7 MW/m^2. Along the length, heat power of each fuel rod takes a cosine distribution and the half length has the peak heat power. Coolant at the entrance has velocity being 1m/s, and temperature being 300°C. The three dimensional flowing field and heat transfer in the 5×5 fuel assembly has been simulated in CFX. Here results in four types of channels have been taken for discussion. The four channels are chosen from their locations in the fuel assembly as shown in Fig.6, named as center channel, middle channel, side channel and corner channel.

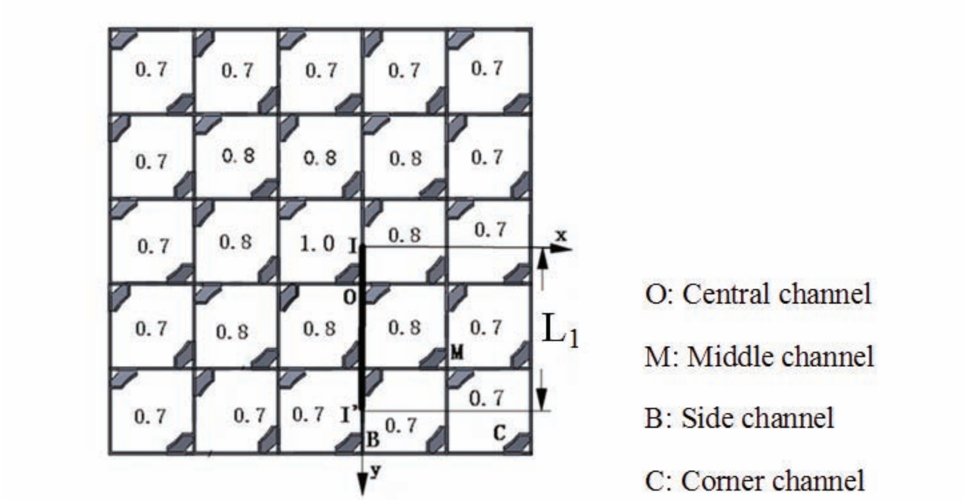
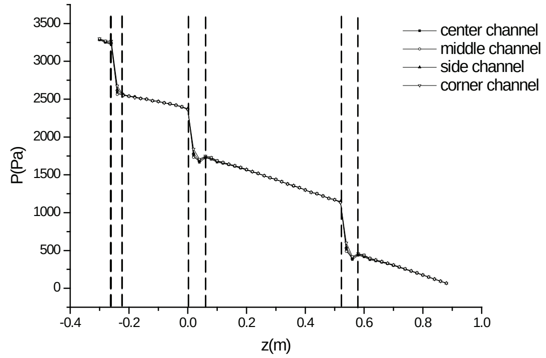


Fig.6 Radial heat power of the 5×5 fuel assembly

Fig.7 shows the pressure drop along the fuel length in the four channels, where position of the three grids have been marked out with three dotted lines in the figure. We can find that total pressure drop of the four channels are basically same. There is an obvious pressure drop at positions of mixing vane grid and spacer grid, while pressure drop in other range of fuel rod is very small. There is another distinct characteristic in the figure, which is the small pressure rise after the mixing vane grids. The reason is the increase of flowing area and loss of kinetic energy after the mixing vane grids.

Fig.8 shows the fluid temperature in the four channels. As shown in the figure, fluid temperature tends to rise along length of the channel with continuous heating from the fuel rod. Comparing the temperature increment of the four channels, temperature of the center channel is the largest due to the higher power. The middle channel and side channel have lower temperature, and the corner channel has the lowest channel. The maximum temperature difference between the four channels is about 20K, appears at the third grids. Another characteristic of fluid temperature along the length is the slight decrease at the mixing vane grid, with a reverses after the grid. The reason is decrease of heat transfer coefficient by blocking of flowing by the grids and mixing vane, causes decreasing of fluid temperature at the mixing grid. While increase of heat transfer comes after the mixing grids, makes the fluid temperature increasing at that position.



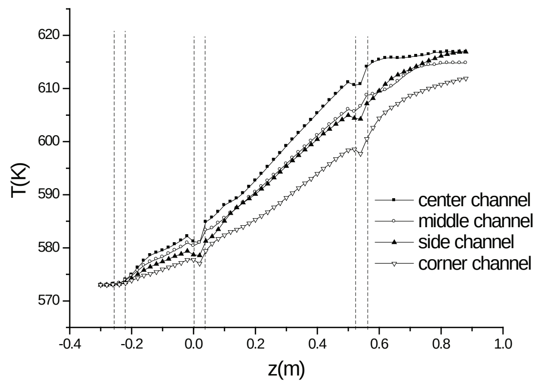
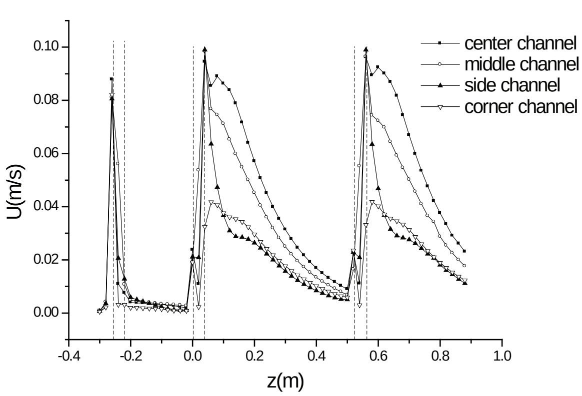


Fig.7 Pressure drop in four channels. **Fig.8 Fluid temperature in four channels.**

Fig.9 shows the cross velocity in the four channels along fuel length. One can find that cross flow is much stronger in downstream of mixing vane grids, which will decrease gradually with increase distance to the grid. Comparing cross flow of spacer grid and mixing grids, cross velocity of the spacer grid is smaller. Cross flow disappears just after the spacer grid, while a remarkable cross flow still exist after a longer distance after the mixing grid. Comparing cross flow of the four channels, it can be find that cross velocity in the center and middle channel is larger, while that of side and corner channel is smaller.

Fig.10 shows axial flowing velocity in the four channels along length of the 5×5 fuel assembly. One can find that the axial flowing velocities in the four channels rise up quickly before the grid and decrease soon after the grid, with blockage of flowing area by the grid. Variation of flowing axial velocity near the mixing grids is larger than spacer grid. Comparing axial velocity in the four channels, a similar rule with cross velocity can be found. Axial velocity in center channel is the largest, the second comes that in middle channel and side channel, that in the corner channels is the smallest.



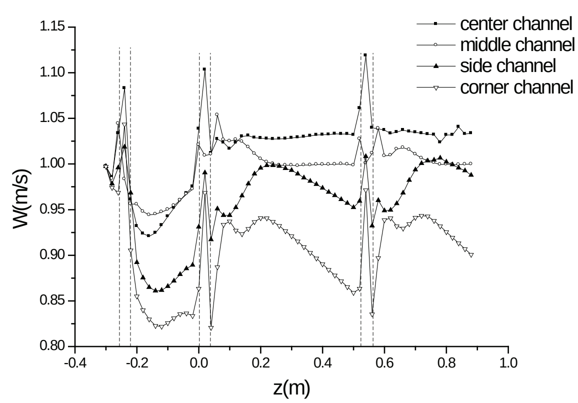


Fig.9 Cross velocity in the four channels **Fig.10 Axial velocity in the four channels**

Graph of streamline on cross sections at exit of mixing grid and spacer grid around center rod is shown in Fig.11 and Fig.12. One can see that stream lines at exit section of the two grids are quite different. Stream lines on exit section of the mixing grid form large ellipse vortex in each flowing channels, caused by lateral flowing of the mixing grid, the length of the vortex is along the stretching direction of the vanes.

Those of the spacer grid are separated by the grid form and develops from the grid to the rod surface.

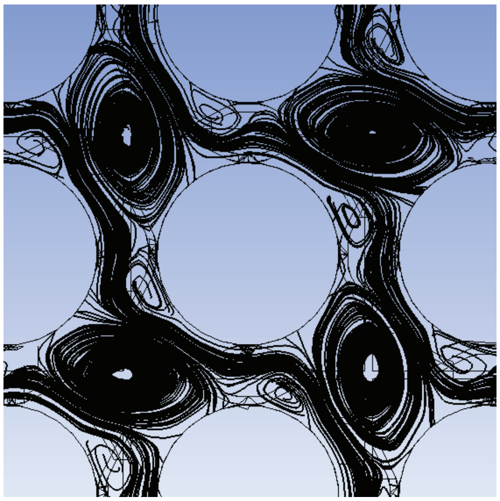


Fig.11 Streamline of mixing vane grid exit.

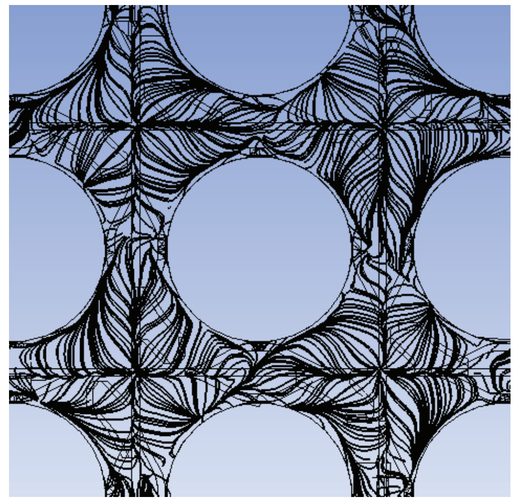
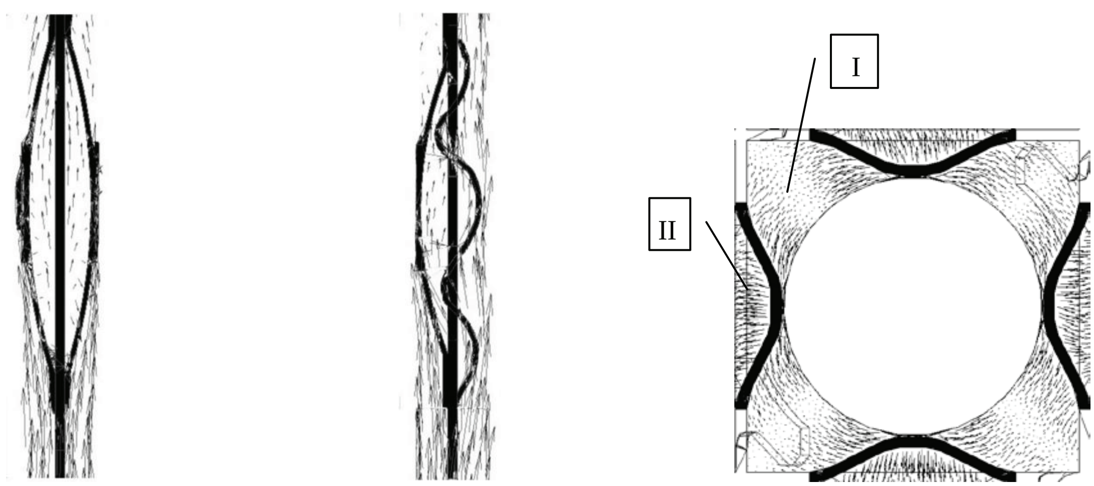


Fig.12 Streamline of support grid exit.

In order to study effects of spring and steel convex on flowing field of mixing grid and spacer grid in the 5×5 fuel assembly, Fig.13 shows vector graph of the flowing velocity on lengthwise section near the double spring, single spring. One can find that both the two types of spring only have a slight effect on the flowing velocity, make the velocity vector turning direction slightly from straight upward to the spring surface. As for effect of the steel convex, vector graph of velocity on cross section in the flowing channel with four steel convexes is also shown. One can see that the steel convex has a noticeable effect on cross flowing velocity, which makes cross flow lies in the tangential direction of the fuel rod in area I of flowing channel, while going from the steel convex to the stripe in area II.



double spring single spring steel convex

Fig.13 Velocity vector of three structures of spacer grid and mixing grid.

5.2 Contour lines of cross and axial velocity at exit section of mixing grids

To fully investigate the turbulence characteristics of mixing grids, the contour lines of cross and axial velocity on 1/4 exit section have been presented in Fig.14 and Fig.15 respectively. One can see the symmetrical characteristics of cross velocity nearby the dual set of mixing vane in part I and II in Fig.14. Area with high cross velocity is in orthogonal direction of stretching wise of the mixing vanes. Because the dual set of mixing vanes forms a positive pressure zone along the stretching direction of vanes on the grid exit section, while a negative pressure zone forms in the orthogonal direction. The pressure difference makes cross flowing from the positive pressure zone to the negative pressure zone. Also, an obvious cross flow can also be found near the single mixing vane, as shown in part III of Fig.14. But magnitude of cross flowing velocity near single mixing vane is much smaller than that of dual mixing vanes. Distribution of axial flowing velocity near the mixing vanes is closely related to lateral velocity, which is apparently smaller in area with high cross flow, and vice versa.

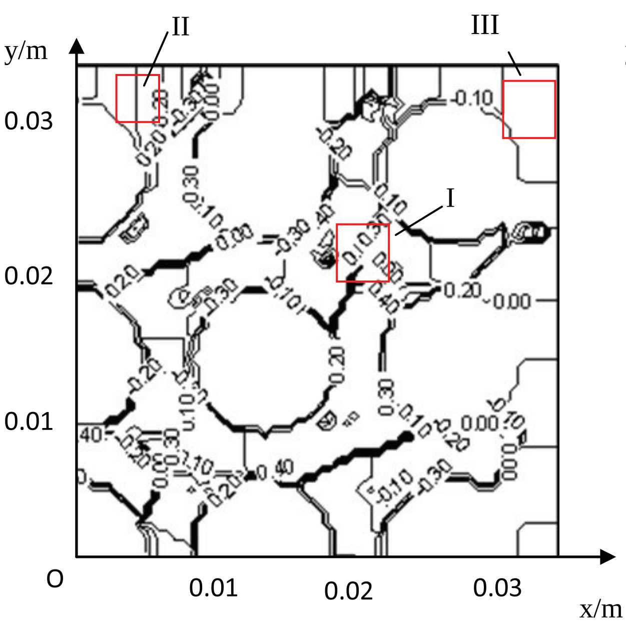


Fig.14 Cross flow velocity on exit of mixing grid

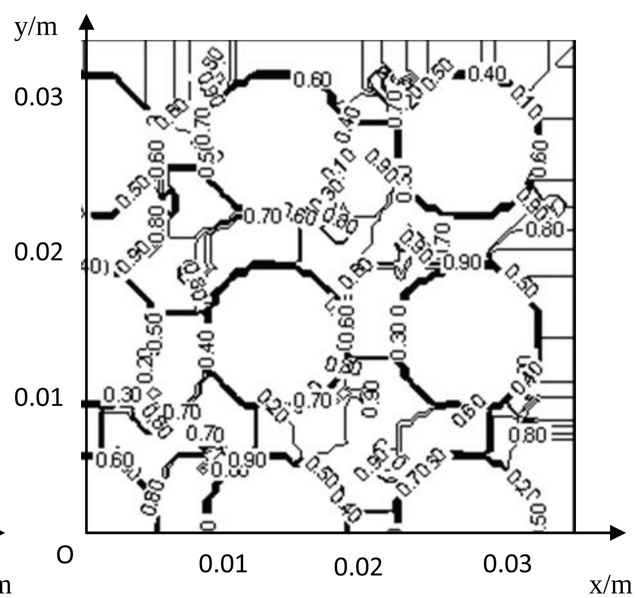


Fig.15 Axial flow velocity on exit of mixing grid

5.3 Influence of mixing vane's geometry on mixing effect and pressure drop

In order to investigate effect of mixing vane geometry on lateral flowing and pressure drop of the fuel assembly, our work also calculated the mixing coefficient and pressure drop along length of the fuel assembly with four different deflection angles and vane lengths. The four deflection angles are φ =25°, 31°, 37° and 43°, and the five lengths of mixing vane are $L=0.8L_0$, $0.9L_0$, L_0, $1.1L_0$ and $1.2L_0$, where φ =25° and L_0 is the common designed geometry parameter of mixing vanes in PWR fuel assembly. Calculation of mixing coefficient β is based on the Ref.14, which is ratio of the area averaged lateral velocity in each cross section to the entrance velocity,

$$\beta = \frac{|\bar{u}|}{W} = C \times 0.2 \times \mathrm{Re}^{-0.125}$$

Fig.16 shows the mixing coefficient along length of the fuel assembly with the four deflection angles of mixing vane. The results show great change of mixing coefficient near mixing grids, which increase steeply from the entrance of grids, arrives at a peak on the exit and drops down sharply after exit of the grids. The first mixing grid has a higher mixing coefficient compared with the second mixing grid in the downstream. Comparing mixing coefficient in downstream of mixing grids for the four deflection angles, one can find that mixing grids with the higher deflection angle has higher mixing coefficient. While the spacer grid located in z=-0.1m doesn't cause obvious lateral mixing flow. Figure 17 shows the total pressure drop along whole length of fuel assembly with the four deflection angles for mixing vanes. One can find that the pressure drop along the whole fuel length increases slightly with increasing deflection angles, the increase amplitude of pressure drop is about 10% when deflection angle changes from 25° to 43°.

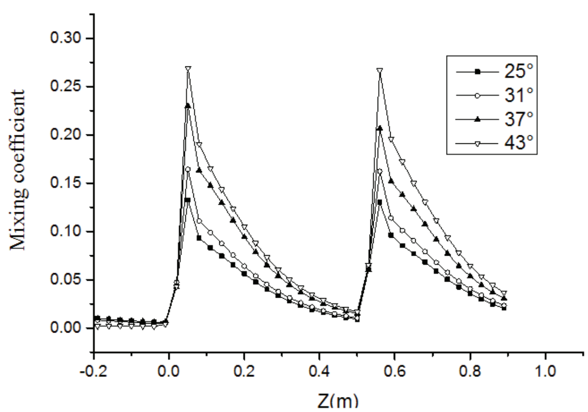


Fig.16 Mixing coefficient for mixing vanes with different deflection angles mixing

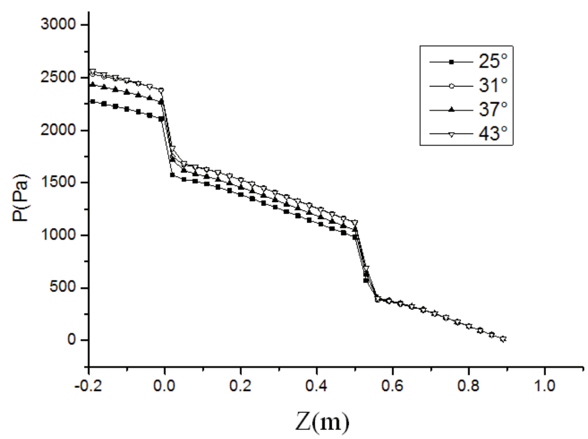


Fig.17 Pressure drop of the fuel assembly for vanes with different deflection angles

Fig.18 shows the mixing coefficient along length of the fuel assembly with the five lengths of mixing vane, that is L=0.8L_0, 0.9L_0, L_0, 1.1L_0 and 1.2L_0. It's shown that the mixing coefficient in downstream of grids goes up with increasing mixing vane length, but change of the peak value is very small. Fig.19 shows the total pressure drop along the fuel assembly with the five different lengths of mixing vane. One can find that when the length of mixing vanes is 0.8L_0, 0.9L_0, L_0 , pressure drop of the assembly is basically same, when L= 1.1L_0 and 1.2L_0, the pressure drops rise a little, the maximum difference is only 3.6% for L= 1.1L_0.

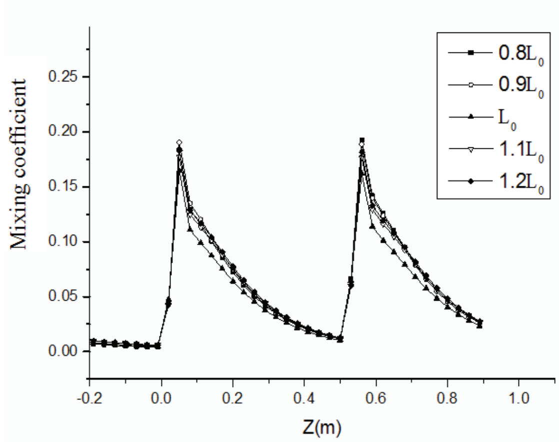


Fig.18 Mixing coefficient for mixing vanes with different vane lengths

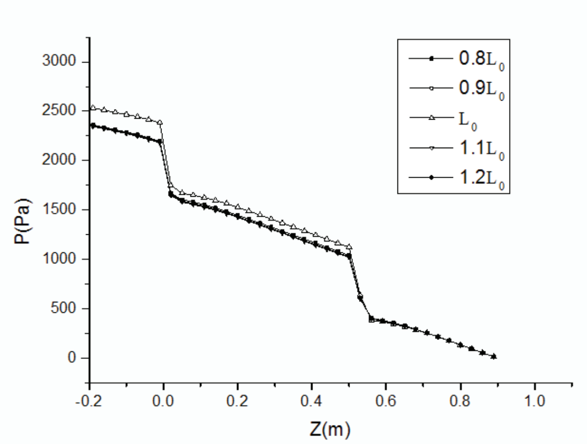


Fig.19 Pressure drop for mixing vanes with different vane lengths

6 CONCLUSION

Concerning the flow and the turbulent mixing characteristics of fuel assembly with multi-grids, a complete geometric model of a 5×5 fuel assembly and the compound mesh model has been established. A CFD code has been used to calculate the three-dimensional flowing field in bundle channels and mixing characteristics under non-heated and heated conditions. The following finding can be given from this study. Both the mixing grid and spacer grid are important structure acts to enhance cross mixing and heat transfer of fuel assembly, the effect of mixing vane is much stronger, that is most remarkable in downstream of the grid. The cross flowing develops in the stretching direction of mixing vanes at the mixing grid exit section. Increasing the deflection vane angle will enhance the mixing coefficient of mixing grid, but will also lead to a rise in pressure drop of the whole fuel assembly. Increasing the vane length when $L>L_0$ will also strengthen mixing coefficients of the grid, but variation of the vane length has little influence on pressure drop of fuel assembly if $L<L_0$.

ACKNOWLEDGMENTS

Acknowledge sponsorship from the National Natural Science Foundation of China (51376065, 51176052), Guangdong Key Scientific Project (2013B010405004), Guangdong Province Key Laboratory of Efficient and Clean Energy Utilization (2013A061401005).

REFERENCES

1. C.C. Liu , Y.M. Ferng, C.K. Shih. CFD evaluation of turbulence models for flow simulation of the fuel rod bundle with a spacer assembly [J], Applied Thermal Engineering, 40:389-396(2012).
2. M.Holloway, Single-phase convective heat transfer in rod bundles[J].Nuclear Engineering and Design 238: 848–858(2008).

3. Moysés A. Navarro , André A.C. Santos. Evaluation of a numeric procedure for flow simulation of a 5×5 PWR rod bundle with a mixing vane spacer [J]. Progress in Nuclear Energy, 53(8): 1190-1196 (2011).
4. Kazuo Ikeda， Yasushi Makino, Masaya Hoshi. Single-phase CFD applicability for estimating fluid hot-spot locations in a 5×5 fuel rod bundle [J]. Nuclear Engineering and Design, 236(11)：1149-1154(2006).
5. E.D Elvis,Y.A.H, Experimental benchmark data for PWR rod bundle with spacer-grids[J].Nuclear Engineering and Design 25(3):396–405 (2012).
6. Seok Kyu Changa, , Sang Ki Moona, Won Pil Baeka, Young Don Choi b, Phenomenological investigations on the turbulent flow structures in a rod bundle array with mixing devices[J]. Nuclear Engineering and Design 23(8): 600–609(2008).
7. XIONG Wan-yu, CHENBing-de, XIAOZe-jun. 3-D Flow Field of Rod Bundles With Spacer Grids [J]. Atomic Energy Science and Technology, 39(4):326-329(2005).
8. CHEN Wei-hong, ZHANG Hong, ZHU Li, XIONG Wan-yu. Research on Application of CFD Method in Thermal-Hydraulic Performance Analysis of Rod Bundle Grid [J]. Nuclear Power Engineering, 30(5): 34-38(2009).
9. TIAN Rui-feng, MAO Xiao-hui, WANG Xiao-jun. Study on 3-D Flow Field in 5×5 Rod Bundles with Spacer Grids [J]. Nuclear Power Engineering, 29(5): 48-51(2008).
10. GAO Ye， LI Xiaochang. Numerical simulation of single-phase flow in a PWR four-subchannel model with overall length [J]. Journal of Harbin Engineering University, 34(3):1-6(2013).
11. A. Gandhir, Y. Hassan， RANS modeling for flow in nuclear fuel bundle in pressurized water reactors[J]， Nuclear Engineering and Design 24(1): 4404–4408(2011).
12. Michael E. Conner, Emilio Baglietto, Abdelaziz M. Elmahdi. CFD methodology and validation for single-phase flow in PWR fuel assemblies [J]. Nuclear Engineering and Design, 24(9): 2088-2095(2011).
13. N. Cinosi, S.P. Walker et al. CFD simulation of turbulent flow in a rod bundle with spacer grids(MATIS-H) using STAR-CCM+[J]. Nuclear Engineering and Design 27(9): 37–49 (2014)
14. X. Cheng, B. Kuang, Y.H. Yang. Numerical analysis of heat transfer in supercritical water cooled flow channels[J]. Nuclear Engineering and Design 23(7) : 240–252(2007).