THE EFFECT OF DEFLECTION ANGLE OF MIXING VANE ON THE FLOW CHARACTERISTICS IN ROD BUNDLE BASED ON CFD METHODS

Li Zhongchun[1,2], Du Sijian[1,2], Li Songwei[1,2], Ding Shuhua[1,2], Zhang Hong[1,2], Zhang Yu[1,2]

[1] *Thermal Hydraulic and Safety Analysis, Nuclear Power Institute of China, Chengdu, China*
[2] *Thermal Hydraulic and Safety Analysis, Science and Technology on Reactor System Design Technology Laboratory, Chengdu, China*

EXTENDED ABSTRACT

I. BACKGROUND AND MOTIVATION

The critical heat flux (CHF) is one of the design limit for thermal hydraulic design and one of the key issues in nuclear reactor safety. Many CHF enhancement methods of fuel assembly in pressurized water reactor (PWR) have been proposed and spacer grid with mixing vane is the most widely used. Mixing vane generated strong swirl flow and enhanced the turbulence intensity level and the channel mixing. These effects increased CHF significantly. Available experiments have been shown that the deflection angle is one of most important geometrical parameters which significantly changed the flow, heat transfer and hence CHF enhancement ratio (Shin et al.,2005; Cosby, 2007; Shin et al., 2009). The results showed that the deflection angle around 30^{o} showed the best heat CHF performance. The mechanism for the CHF enhancement is a combination of multi-effects, such as formation of swirl flow, increasing enthalpy mixing and bubble breakup and migration characteristics. However, there is still no consensus on the deflection angle effect on CHF enhancement mechanism.

To obtain clear understanding on the deflection angle effects on flow characteristics, adiabatic CFD models on simple 3*3 fuel assembly channel with mixing vane with different deflection angle, which are 20^{o}, 30^{o}, and 40^{o}, respectively, had been developed. The CFD models included adiabatic single phase flow and two phase flow. The detailed flow characteristics for deflection angle effect were obtained and compared.

II. CFD MODEL AND SETUP

3×3 rod bundle geometry with mixing vane of different deflection angle is adopted to the calculation domain, as shown in Figure 1. The mixing vane has the same shape, and only the deflection angle is different. The calculation domain is rectangle with size of 40 mm. The rod diameter is 9.5mm and rod to rod distance is 12.6mm. Typical spacer grid is assigned in the middle of the flow channel. The simulation domain height is 500mm and the distance between spacer grid and outlet is 350mm. The mesh in the spacer grid is generated by tetrahedrons with prism mesh. The region in the downstream and upstream of spacer grid is generated using the hexagonal sweep methodology.

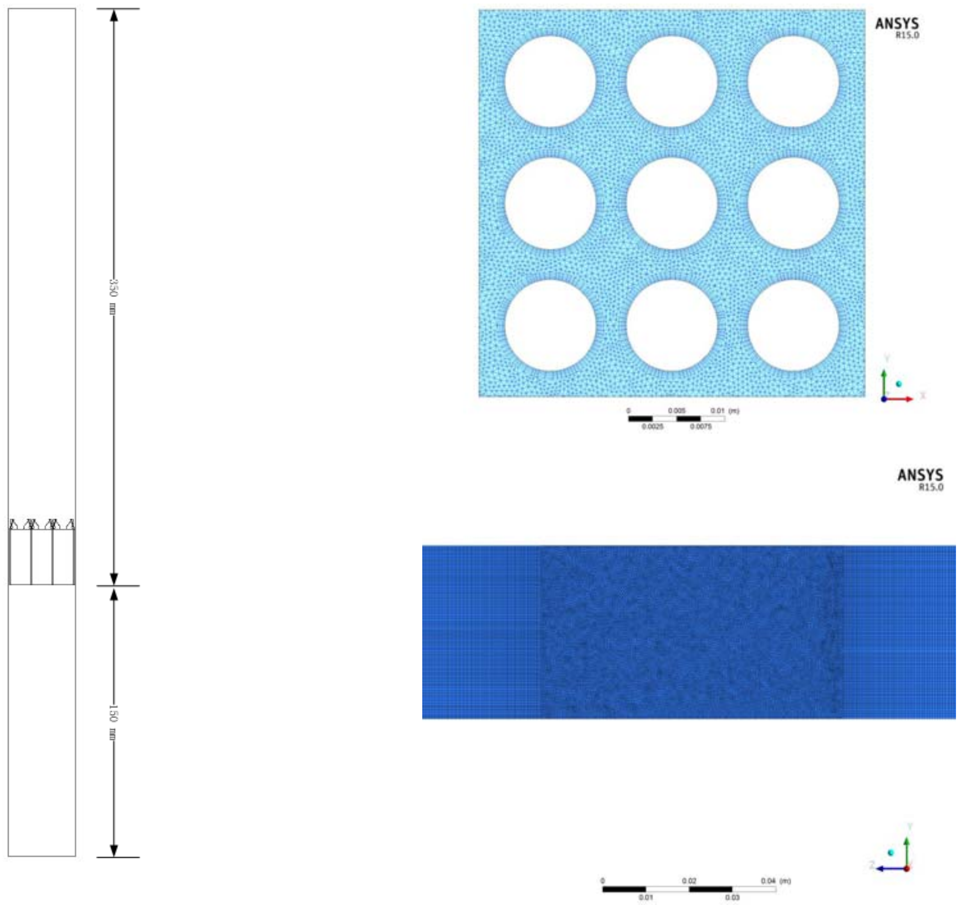


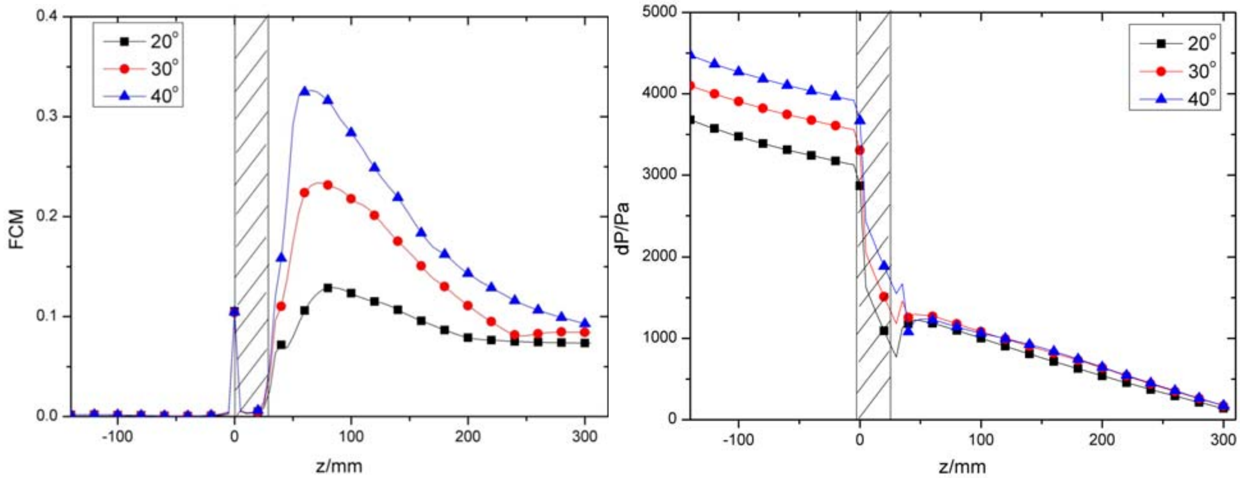
Figure 1. The calculation region and mesh

The adiabatic CFD models are developed in the present simulation with single phase and two phase flow. The working fluid is water at saturation condition under 15.5MPa for single phase flow. For two phase flow simulation, steam at saturation condition under 15.5MPa is added as a dispersed phase. The standard k-e model is used in the single phase flow. In the two phase flow simulation, the standard k-e model and dispersed phase zero equation was used as turbulent models about liquid and gas phases, respectively. The homogenous MUSIG model with 12 groups of bubble is considered for the interfacial area. Various interfacial momentum transfer terms, including drag and non-drag forces were determined. No mass and energy transfer is considered in the simulation. The inlet is set as uniform flow velocity for water (and) steam. The outlet is set as pressure outlet. The channel wall and spacer grid is set as wall condition and no-slip for continuous phase and free slip for dispersed phase. The calculation is steady state and two-order numerical scheme in space. The residual RMS value set as 10^{-4}. The correlations and coefficient in CFD models is determined based on model sensitivity and calibration study.

III. RESULTS AND DISCUSSIONS

The flow characteristic is obtained from the single phase and two phase CFD simulation results and compared on the deflection effect. After the flow passed the spacer grid, the flow parameters first rise up to a maximum value and then decayed along the flow direction. The parameters' longitude distribution is compared. The parameters include pressure drop, cross flow, swirl flow, turbulence level, flow distribution parameters and factors based on the parameters. The results revealed that the deflection angle has great effect on the parameter distribution.

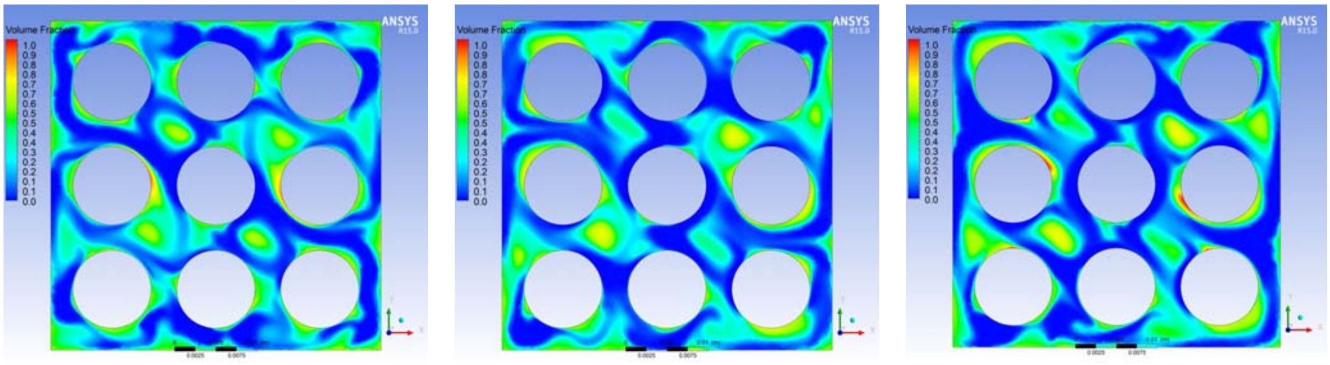
The pressure drop and cross flow magnitude increased with deflection angle for both single phase and two phase flow. The reason is mainly caused by the increase of flow blockage ratio. This is also the same for the swirl flow factor which is indicated by SM factor and FSW factor, and turbulence level, as shown in Figure 2.



(a) FCM (b) Pressure Drop

Figure 2. The calculation results for different deflection angle

The void fraction distributions with the different deflection angle were shown in Figure 3. It was confirmed that the mixing cane with deflection of 30 ^{o} resulted better distribution pattern, where the center void fraction is higher and the void fraction around the rod bundle is not so high at same height. It is also confirmed with two phase flow streamline that the streamline for gas phase is trended to the center of the channel. These effects both demonstrated that it would obtain better CHF performance, which is consisted with the experiment result.



(a) (b) (c)

Figure 3. The calculation results for different deflection angle at z = 300mm

IV. CONCLUSIONS

The present study focused on the deflection angle effect on the flow characteristics based on adiabatic single phase and two phase flow in 3×3 rod bundle. The results indicated that deflection angle has great effect on the flow parameters. The pressure drop, cross flow, swirl flow and turbulence level are increased with deflection angle. The void fraction distribution in the deflection angle of 30 ^{o} hold better distribution pattern which would be good for CHF enhancement. However, quantitative analysis and real case simulation is strongly needed to get more information on the geometry effect on the CHF performance.

REFERENCE

[1] Shin, B.S., et al., 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. Des. 235, 1749–1759(2005).
[2] Cosby M C. Effects of support grid vane angles on single phase heat transfer within rod bundle subchannels [J]. Dissertations & Theses - Gradworks(2007).
[3] Shin, B.S., et al., CHF experiment and CFD analysis in a 2×3 rod bundle with mixing vane, Nucl. Eng. Des. 239, 899–912(2009).