Transactions of the Korean Nuclear Society Autumn Meeting
Busan, Korea, October 27-28, 2005

CFD Simulations on Flow Fields of the 16x16 ACE7™ Rod Bundle Spacer Grid

SeungGeun Yang, HyunMi Hong, KangHoon Kim, HongJu Kim, EungJun Park, SunTack Hwang, Young-Koo Kim
KNFC Fuel Technology Center 493 Deokjin-Dong Yuseong-Gu Daejeon, Korea 305-353
sgyang@knfc.co.kr

1. Introduction

Several CFD simulations have been performed to characterize the hydrodynamic flow fields produced by the rod bundle spacer grid with the mixing vanes. Based on the previous studies [1,2], the scope of the present work is extended to the full size 16x16 ACE7™ rod bundle spacer grid. The objective of this study is to evaluate the lateral flow characteristics and the heat transfer performance of the full size 16x16 ACE7™ rod bundle spacer grid.

2. CFD Model Descriptions

2.1 Computational Domain and Mesh

One of the leading commercial CFD code, FLUENT, is chosen to simulate the flow fields induced by the rod bundle spacer grid. The configuration of geometry of the 16x16 ACE7™ rod bundle spacer grid is given in Figure 1.

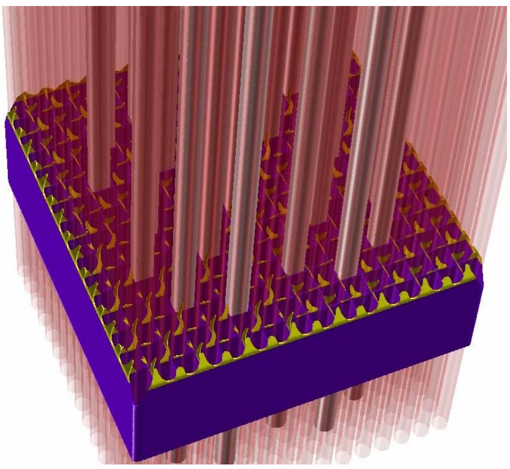

Figure 1. Schematics of the 16x16 ACE7™ Spacer Grid

The computational domain includes 2.25 inches height of a spacer grid with the mixing vanes. The diameters of the fuel rod and guide tube are 0.360 inches and 0.471 inches, respectively. For the consistency to the previous studies [1,2], the lengths of inlet and outlet portion are chosen as –5.25 inches and +9.375 inches including a 2.25 inches height strap of spacer grid. Due to the periodic array of the fuel rod, the guide tube, and the mixing vanes, the half model of the spacer grid as the computational domain is enough to simulate the full size spacer grid.

The minimum mesh size is set to a half of the strap thickness. The meshes are grouped as two categories. The tetrahedral meshes were used in the mixing vane region due to the very complex geometry. Others such as the upstream and downstream regions were filled with the hexahedral meshes. Specially, the multiple thin layers were specified in the near rod wall regions to count for the near wall effect. Following this treatment, the target y+ values could be kept between 50 and 150 in the near rod wall regions. The number of total meshes in the half size (16x8 array) spacer grid model is around 7 millions.

2.2 Analytical Conditions

The constant water properties at 610 °F (594.26 K) and 2250 psia were used in the computation. The constant heat flux (946372.2 W/m^2) condition was set at the entire surface of all fuel rods. The SIMPLE algorithm and the RNG k-ε turbulence model were used to analyze the three-dimensional flow fields. The uniform flow velocity at the inlet and the constant pressure at the outlet were the boundary conditions set with following values:

- Axial velocity at inlet : 5.1816 m/sec
- Lateral velocity at inlet : 0 m/sec
- Pressure (reference) at outlet : 0 pascal

3. Simulation Results and Discussions

The FLUENT code version 6.1.22 was used to solve the computational domain. At least 1,200 iterations were performed to converge the all residuals. As shown in Figure 2, the computational domain is allocated to the several specific regions to evaluate the local lateral flow characteristics and the heat transfer performance around the fuel rods and guide tubes.

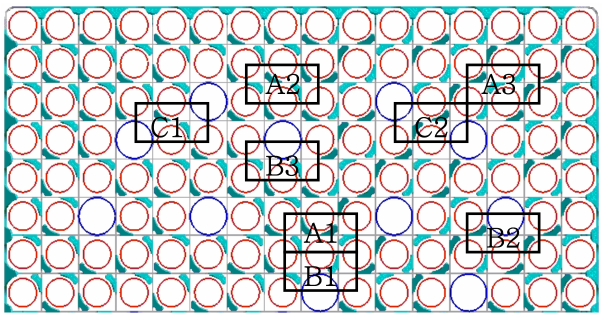

Figure 2. Schematics of the Local Computational Regions

The average and maximum lateral flow velocity magnitudes of the CHF test (5x5 array) spacer grid [2] and the half size (16x8 array) spacer grid of 16x16 ACE7^{TM} fuel is given in Figures 3 and 4, respectively. Figures 3 and 4 show that the lateral flow velocity magnitudes of the typical channels are larger than that of the thimble channels in all kinds of subchannels. The magnitudes of the lateral flow velocities from the tip of mixing vane (~1.0 inch) to the near next IFM grid (~8.0 inches) position are continuously decreased in every case. Also the overall trends are similar to each other. Though the magnitude of the lateral flow velocity between the CHF test array model and the half size array model shows some remarkable deviations, they are gradually reduced along the entire downstream region.

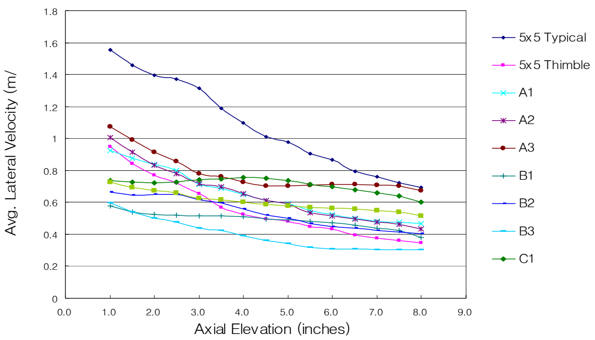

Figure 3. The Avg. Lateral Velocity of the 16x16 ACE7^{TM} Spacer Grid around the Top of Mixing Vane

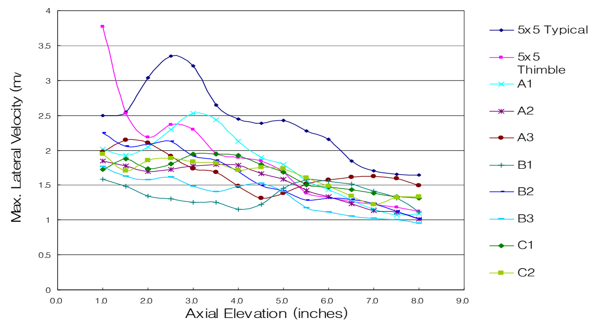

Figure 4. The Max. Lateral Velocity of the 16x16 ACE7^{TM} Spacer Grid around the Top of Mixing Vane

Figure 5 shows the surface-averaged heat transfer coefficients along the axial direction for several typical and thimble channels. The overall trend of the surface-averaged heat transfer coefficient is very similar to that of the overall average and maximum lateral flow velocities. This means that the intensity of the swirl and crossflow in the subchannels affects directly to the heat transfer coefficients on the fuel rod surfaces. Figure 5 shows that the surface-averaged heat transfer coefficients of each channel vary in accordance with the mixing vane pattern, flow area, heat rod, and so on. Also the surface-averaged heat transfer coefficients of the CHF test array model and the half size array model show the similar trends with the lateral flow velocity.

Basically, the geometric difference between the CHF test array model and the half size array model makes the local pressure drop difference. It also affects the local flow conditions and the flow velocity magnitudes. Therefore, the scalar quantities should be scaled to consider the geometric effects. As other effects, the mixing vane pattern, the heater rod configuration, the analytic boundary conditions also should be considered to reduce the model deviations.

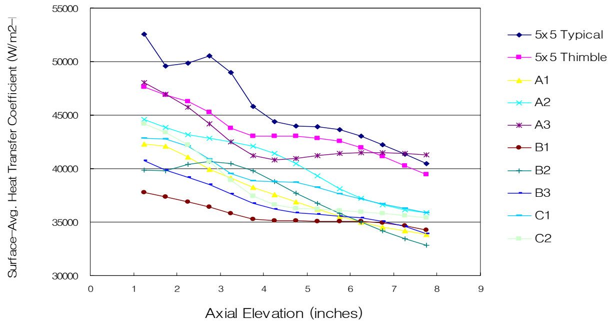

Figure 5. The Surface-Avg. Heat Transfer Coefficient on the Fuel Rod Surfaces

4. Conclusions

Based on the results of this study, it was concluded that the characteristics of the lateral flow velocity and the surface heat transfer coefficient for the CHF test array model and the half size array model show the overall similarity in the entire region of downstream in spite of the geometric effects. Therefore, the configuration of the CHF test spacer grid can represent properly that of the actual 16x16 ACE7^{TM} rod bundle spacer grid. To absorb the quantitative deviations, this analytical model must be examined more carefully through revising geometry and boundary conditions.

RFERENCES

[1] K.H. Kim, et. al., “Effects of Mixing Vane Design on Hydraulic Flow Fields in Subchannel,” Proceeding KNS Autumn Meeting, Yongpyong Korea, October 2004.

[2] S.G. Yang, et. al., “Heat Transfer Performance Analysis of a 5x5 Rod Bundle Spacer Grid by CFD Simulations,” Proceeding of ICAPP’05, Seoul Korea, May 2005.

Acknowledgement :

This study is supported by MOST under “Technology Development to Resolve Axial Offset Anomaly (AOA)” R&D program. Authors would like to express deep appreciation to the MOST personnel and program staff.