Transactions of the **14th International Conference on Structural Mechanics in Reactor Technology** (SMiRT 14), Lyon, France, August 17-22, 1997

C03/3

Flow characteristics for spacer grid with mixing vane of PWR fuel assembly

Lim H.T., Jeon K.L., Choi J.H., Park B.S.
Korea Nuclear Fuel Co., Korea

Abstract

Some Computational Fluid Dynamics(CFD) analyses were recently performed to simulate flow behavior around spacer grid of PWR fuel assemblies. A part of KOFA fuel is modeled to see the effects of mixing vane of spacer grids on flows. A typical cell of spacer grid is simulated using the commercial CFD code. Lastly, to check the validity of CFD analysis the calculation results are compared with experimental results that were performed in the downstream of the spacer grid.

1. INTRODUCTION

The KOFA(Korea Optimized Fuel Assembly) is the PWR fuel used in Korean nuclear power plant in the past. That fuel is the first fuel designed in Korea. In the design stage of the fuel, the flow characteristics are checked by only the experimental approach. Currently, with the help of CFD technique, the design verification is been performing. This paper presents a part of this work.

The evaluation of the thermal-mixing performance of the spacer grid is very important in the PWR fuel assembly design. The use of CFD code is very helpful in improving the design approval of the PWR fuel assembly, especially for the fuel grid spacer. From the view point of spacer grid design, there are two main objects ; one is the pressure drop and the other is the

DNB performance and usually they have been evaluated by flow test and DNB test. Nowadays, Computational Fluid Dynamics(CFD) has made it possible to calculate the flow characteristics for spacer grid with mixing vane of PWR fuel assembly.[1,2]

The finite volume method with the k- ε turbulence model is employed in the analysis. Also, a flow test was conducted to verify the analytical method, using a part of the actual spacer grid that is composed by typical cells. The pressure distribution around the spacer grid was measured, and the analytical results were compared with the test results.

2. Analysis and Experiment for Spacer Grid

Governing Equations

The Reynolds Averaged Navier-Stokes equations with the k- ε turbulence model were utilized in the analysis. These equations that include the conservation of mass, momentum and energy are expressed as:

$$\frac{\partial}{\partial t}(\rho_l)+\nabla\cdot(\rho_l\overline{u}_l)=0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\rho_l)+\nabla\cdot(\rho_l\overline{u}_l\otimes\overline{u}_l-\mu_l(\nabla\overline{u}_l+(\nabla\overline{u}_l)^T))=-\nabla p \qquad (2)$$

$$\frac{D}{Dt}(\rho_l h_l)-\nabla\cdot(\lambda_l\nabla h_l)=q \qquad (3)$$

where, ρ, $\overline{u}$, p and h are the density, velocity, pressure and enthalpy of the water respectively. The heat flux into the water is given by q. In the PWR, the water flow is highly turbulent. It is not possible to resolve all scales in time and space by the numerical simulation. The common approach in engineering analyses is to resolve the mean flow field, and to model the fluctuating flow. A widely used model is the *k*- ε turbulence model. Here, *k* stands for turbulence kinetic energy, and ε stands for the dissipation rate. The effective viscosity in the governing equations has been redefined as,

$\mu_{1,t} = C_\mu \ k^2 / \varepsilon$

$\mu_{1,\lambda}$ is laminar viscosity

C_μ is a model constant

The partial different equations are solved by the CFX code, which uses a finite volume approach.[3]

Modeling of Spacer Grids with Mixing Vanes

A single subchannel of one grid span was modeled in the CFX code. The split mixing vane generates a swirling flow pattern around the fuel rod. In Figure 2 two vanes are placed on top of the spacer and bent in opposite directories to promote a swirling flow pattern in the subchannel. Figure 1 also shows a larger array of the spacers. In the axial geometry of the split vane model, the simulating starts 45 mm upstream of the leading edge of the spacer and the total length of the models 645 mm. To model the diversion of flow between subchannels and minimize the size of the computational model, a special treatment of the boundary conditions was developed making use of flow symmetry. This reduces the computational model to a function of a subchannel using traditional periodic boundary conditions. The strip and the mixing vanes of spacer grid were modeled with the thin surfaces option in CFX. This way of modeling thin details makes the grid generation much easier and it allows a more coarse grid since no cells must be as thin as the thin detail. The computational grid for the split vane model was 82x12x12 cells. The grid of computational cells in the cross section should have enough detail to resolve the flow field, therefore a detailed computational grid was needed. The inlet page profiles used in the simulations were taken from a model without spacer grid. The inlet boundary condition was a top hat profile with mean inlet velocity 5m/sec. Side boundary conditions were symmetric at all four sides and a usual outlet boundary was used. A pressure drop boundary was also used at the outlet with constant pressure in the cross section.

Evaluation of Calculation with Experiments

The experiment was performed by KAERI(Korea Atomic Energy Research Institute).[4] The test loop consisted of a test-section storage tank and pumps. The velocity of flow through the test sections was controlled by adjusting the pump rotation speed. Loop temperature during measurement was maintained by controlling a bypass flow through the heat exchanger. The test assembly used in this test was a 5x5 square rod array bundle at KOFA, which consisted of typical cells only. The cross section of square housing of 68mm in size of 25 rods of 9.5mm in diameter is shown with measuring locations and a coordinate system in Figure 2. Working fluid is water and flows upward. The flow rate was 20kg/sec at 25 ° C resulting is an average flow velocity 5m/sec. They used the Laser Doppler Velocimeter(LDV) to measure velocities. The measurements were made at various axial elevations downstream of several grid designs with mixing vanes. Pressure drop measurements were also performed. The LDV measurements were taken in two paths traversing the full width of the test section in increments of 1.26 mm at the center line between rods.

3. Results and Discussion

In Figure 5, the calculated axial velocity was compared to the experimental data. As seen in the figure, the data shows qualitatively good agreement. Close to the spacer and the mixing vane, the measured axial velocities in Figure 5 show the big difference between maximum and minimum values, due to the thickness of the spacer strip and the rod support of which features were not modeled. Further downstream, the differences between maximum and minimum velocites decrease. This trend is common in calculation as well as experiments.

The predicted pressure drop was also compared for experiment data. The measured value was about 1.45 mm and the calculated value was 1.8 mm. The difference is small. Figure 4 shows the axial turbulent intensity obtained from the calculation and experiments. In this analysis, the k-ε turbulence model was used. Hence, the turbulent intensity was reduced from the turbulent kinetic energy with the assumption of isotropic nature of turbulence. As can be seen, the difference is some large. It is believed that the isotropic assumption may not be applied because of anisotropy of flow around mixing vane and the measuring characteristics of LDV. LDV measuring volume is so large that the length is about 3 mm which is almost quarter of channel size. But, Further and further, the turbulent intensity distribution becomes more uniform. This can be explained by energy transfer of turbulence. That is, the flow changes into uniform or nearly isotropic state.

4. Conclusion

To improve design in the dynamic flow field, the CFD method to evaluate flow characteristic for PWR fuel assembly has been developed. The relatively good comparison between CFX predictions and LDV measurements indicate that CFD methods can be helpful in optimizing spacer grid design relative thermal performance and in performing more detailed subchannel analysis downstream of spacer grids. This method has already been brought into practical use for design evaluation of the grid spacer grid for fuel assembly.

To verify the calculation, more detail experiments are needed in the future.

References

1. Imaizumi, M., et al., "Development of CFD Method to Evaluate 3-D Flow Characteristics for PWR Fuel Assembly," Trans. of 13th Int. Conf. on Structural Mechanics in Reactor Technology, pp. 3-14, Porto Alegre, Brazil, Aug.13-18,1995.
2. Z. Karoutas et al., "3-D Flow Analyses for Design of Nuclear Fuel Spacer," NURETH-7, NUREG/CP-0142, pp.3153-3174, 1995.
3. CFX 4.1 User Guide, AEA Industrial Technology, Harwell Laboratory, Oxfordshire, UK.
4. S. K. Yang and M. K. Chung, "Spacer Grid Effects on Turbulent Flow in Rod Bundles," J. of the Korean Nuclear Society, Vol.28, pp. 56-71, Feb. 1996.

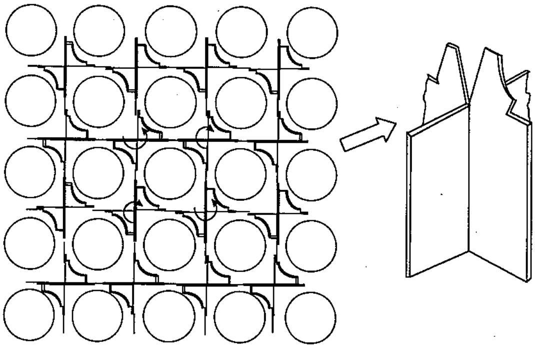

Fig. 1. Typical Cell of Spacer Grid with Split Mixing Vane

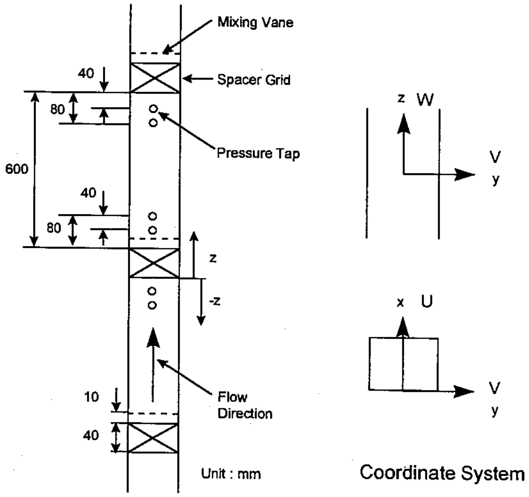

Fig. 2. Coordinate system

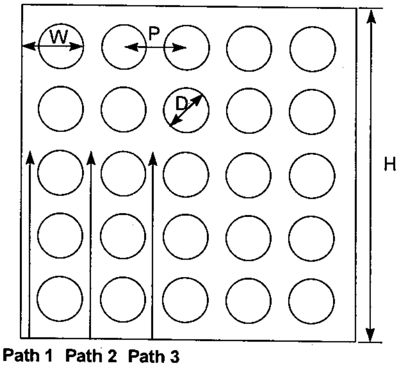

Fig. 3. Measuring Position

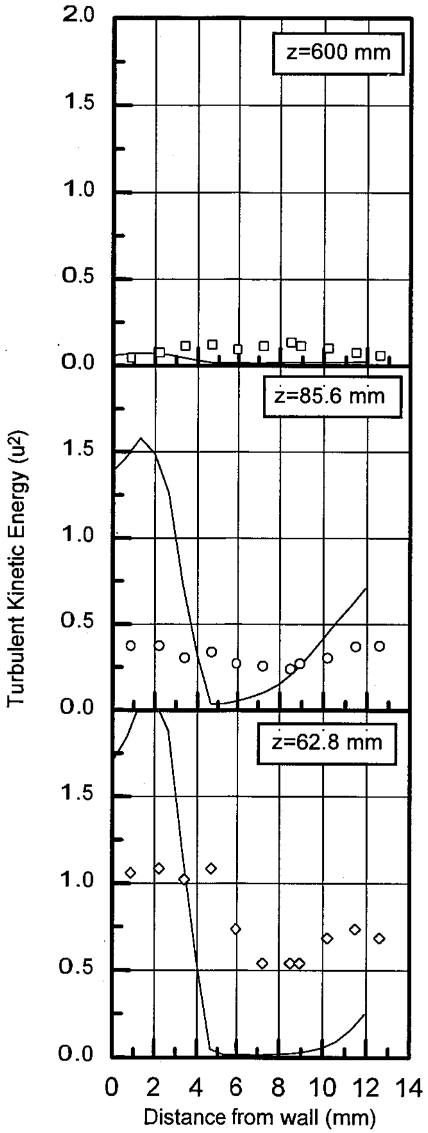

Fig.4 Distribution of Turbulent Kinetic Energy

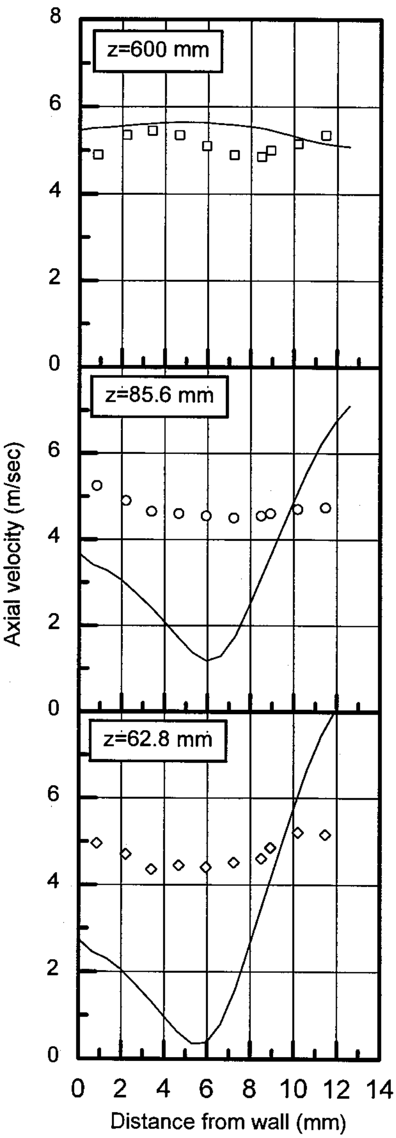

Fig.5 Distribution of Axial Velocity