Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 29-30, 2014

Numerical Simulation for Flow Distribution in ACE7 Fuel Assemblies affected by a Spacer Grid Deformation

Jong-Pil Park [a*], Ji Hwan Jeong [b*]
[a] *School of Mechanical Engineering, Pusan National University, Busandaehak-ro, Geumjeong-gu, Busan, Korea*
(Now with the Korea Atomic Energy Institute, Daedeok-daero, Yuseong-gu, Daejeon, Korea)
[b] *School of Mechanical Engineering, Pusan National University, Busandaehak-ro, Geumjeong-gu, Busan, Korea*
[*] *Corresponding author: pjp3381@keari.re.kr*

1. Introduction

In the event of safety shutdown earthquake (SSE) with LOCA in the PWR, spacer grid will readily be deformed by reactor vibration. If flow area is reduced (flow blockage) by deformed spacer grid, coolant flow will not be sufficiently provided to cool down decay heat though blocked subchannel. In spite of various efforts to understand hydraulic phenomena in a rod bundle containing deformed rods due to swelling and/or ballooning of clad [1,2], the studies for flow blockage due to spacer grid deformation have been limited. In the present work, 3D CFD analysis for flow blockage was performed to evaluate coolant flow within ACE7 fuel assemblies (FAs) containing a FA affected by a spacer grid deformation. The real geometry except for inner grids was used in the simulation and the region including inner grid was replaced by porous media.

2. Methods and Results

The high computational power is required to solve full conservation equations using CFD codes with real geometry of FAs. Therefore, in this study, the region including inner grid of spacer grid was replaced by porous media. The unit subchannel analyses were conducted to calculate resistance coefficients of porous media.

2.1 Unit Subchannel Analysis

The coolant centered subchannel of Mid, IFM, and deformed Mid grid without mixing vane of ACE7 FA were selected as unit subchannel model. The shape of deformed Mid grid was determined by assumption that approximately 40% flow area reduction is occurred by spacer gird deformation. Fig. 1 shows the schematic of unit subchannel models, the entrance region was generated to inlet flow fully develop and the downstream region was generated to avoid flow recirculation at outlet. The average mesh size is about 0.25 mm and fine meshes are clustered densely around the wall. The starting point of the entrance region was selected as Dirichlet boundary and the outlet boundary was specified at the end of downstream region. The symmetry boundary condition is applied on four side plane of computational domain. In the present work, steady-state RANS equation was solved by FVM using a commercial CFD code, STAR-CCM, and the non-linear quadratic k-ε model was selected. The standard wall function was used to treat the flow within turbulent boundary layer.

Fig. 2 shows the pressure drop though the spacer grid of unit subchannel models against the inlet velocity. Based on this result, the resistance coefficients of porous media were obtained by least square method using polynomial equation (see Eq. 1). The calculated resistance coefficients are summarized in Table 1.

$$\frac{\Delta P}{L} = -\alpha V - \beta V^2 \qquad (1)$$

Table 1: Resistance coefficients (RC) of porous media

Grid type	Viscous RC (α)	Inertia RC (β)
Mid	1412	7614
IFM	1927	14495
Deformed Mid	2047	11041

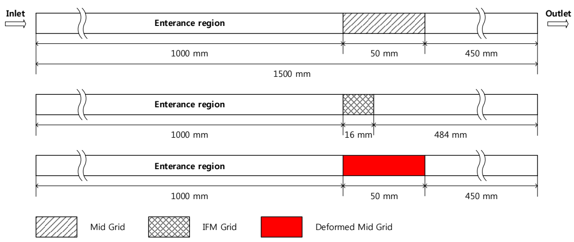

Fig. 1. The schematic for unit subchannel models

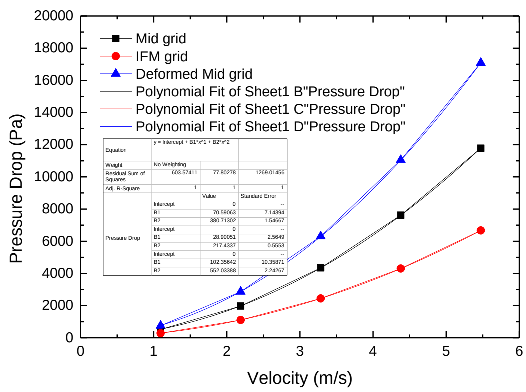

Fig. 2. Pressure drop across grid spacer under normal operating condition

2.2 Flow Blockage Analysis

Fig. 3 shows the schematic for computational domain of flow blockage analysis for ACE7 FAs. The domain consists of four ACE7 FAs with vertically 1 m. In this domain, one of ACE7 FAs includes a Mid grid affected by grid deformation. The region including inner grid of spacer grid was replaced by porous media and the real geometry of outer grid of spacer grid and mixing vanes is used in the simulation.

Using polyhedral type mesh with prism layer, about 25 million meshes were generated and the meshes in upstream and downstream region were generated using extrude mesh. The bottom of domain is selected as Dirichlet boundary and the outlet boundary was specified at the end of downstream region. The symmetry boundary condition is applied on four side plane of domain. The numerical method (turbulence model and boundary conditions, etc.) was kept the same as the previous unit subchannel analysis.

For flow blockage analysis under normal operating condition, the working fluid was selected as single-phase water at 1.551 MPa, 307.5 °C and Reynolds number for this condition is approximately 360,000 based on FSAR for Ulchin units 1 and 2.

Fig. 4 shows the velocity magnitude at horizontal plans in the mixing region. The coolant velocity in a FA containing deformed a spacer grid (FA1) is relatively lower then adjacent FAs (FA2, 3, 4). That means the coolant flow redistribution is occurred by deformed Mid grid. The coolant flow in a FA1 was restricted by deformed Mid, as high flow resistance, and, consequently, deformed Mid grid more coolant flow to be divert to the adjacent FAs.

Fig. 5 shows the normalized mass flux through FAs. The effect of flow blockage due to a spacer grid was seen approximately 60 subchannel hydraulic diameter upstream form the end of deformed Mid grid.

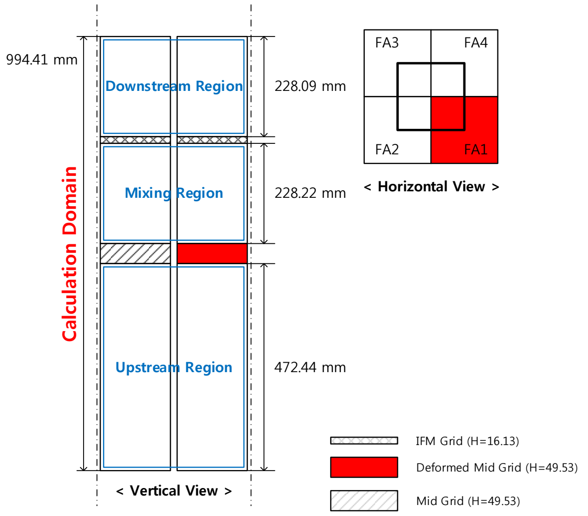

Fig. 3. The schematic for computational domain of flow blockage analysis of ACE7 fuel assemblies

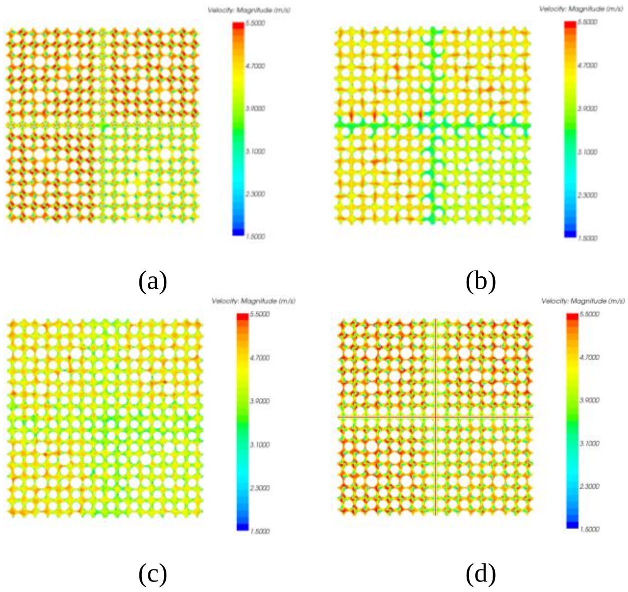

Fig. 4. Coolant velocity at horizontal plan 12 mm (a), 50 mm (b), 100 mm (b), 180 mm (b) apart from the end of Mid grids.

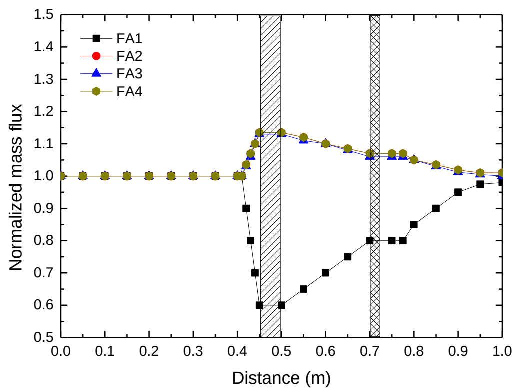

Fig. 5. Normalized mass flux though FAs affected by a spacer grid deformation under normal operating condition

3. Conclusions

In the present work, the numerical simulation was performed to predict coolant flow within ACE7 FAs affected by a Mid grid deformation. The 3D CFD result shows that approximately 60 subchannel hydraulic diameter is required to fully recover coolant flow under normal operating condition.

REFERENCES

[1] J. M. Creer, J. M. Bates, A. M. Sutey, Turbulent Flow in a Model Nuclear Fuel Rod Bundle Containing Partial Flow Blockage, Nuclear Engineering and Design, 52, p. 15, 1979.
[2] M. L. Ang, A. Aytekin, A. H. Fox, Analysis of Flow Distribution in a PWR Fuel Rod Bundle Model Containing a 90% Blockage, Nuclear Engineering and Design, 103, p. 165, 1987.