CFD Analysis of Reactor Vessel Internals with Porous Media

Tae-Jin Kim [a], Sang-Yun Je [a] and Yoon-Suk Chang [a*]
[a]*Dept. of Nuclear Engineering, Kyung Hee University, 1732 Deokyoungdae-ro, Yongin, Kyunggi, 446-701, Korea*
[*]*Corresponding author: yschang@khu.ac.kr*

1. Introduction

Flow distribution at the Reactor Vessel Internals (RVIs) is one of the most important factor in nuclear reactor design. Since it perform important safe-related functions such as supporting the control rod and fuel assembly as well as providing the coolant passage of the reactor core. Therefore, simulations of the flow distribution are essential for guarantee structural integrity of RVIs.

Although the capability of computer hardware technology have been rapidly increased, there are still limitation of Computational Fluid Dynamics (CFD) simulations to analyzing the internal flow distribution at the RVIs. For this matter, porous media was adopted at the fuel assemblies to predicting the reactor internal flow at the normal operating condition of Nuclear Power Plants (NPPs) in detail.

In this paper, CFD analyses have been conducted to investigating the complex thermal - hydraulic characteristics with porous media in the RVIs of 1000MW NPPs. Simulations were carried out by using the commercial multi-purpose CFD software, ANSYS CFX V.17 [1].

2. CFD Analysis

CFD analyses were conducted to investigate modeling effect of the fuel assemblies as porous media in the RVIs.

2.1 Analysis Model

Fig. 1 represents a schematic of RVIs in typical pressurized water reactors. While reactor internals are complex structures, generally, they can be classified into two major parts such as the Core Support Barrel (CSB) assembly and the Upper Guide Structure (UGS) assembly.

From the analysis of existing studies and expert opinion [2], the cooling water moving to the upper head of reactor pressure vessel (RPV) through the UGS was less than 0.1% of total quantity in its flow. So, in the present study, upper part of RPV was not modeled due to the requirement of computational resource to analyze the flow phenomenon in the RVIs model.

As shown in Fig. 1, a hybrid mesh was generated with composition of tetrahedron and hexagon. The number of cells in the RVIs were about 1.48×10^{7} with considering porous media at the fuel assemblies and 2.94×10^{6} without considering fuel assemblies at the RVIs.

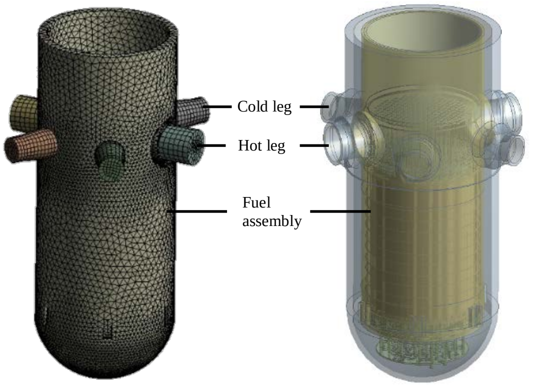


Fig. 1. Schematic of RVIs

2.2 Analysis Condition

The flow inside the reactor internal was assumed to be incompressible, isothermal and turbulent. Physical properties of the coolant are as follows: pressure is 15.5 MPa, temperature is 569 K and density is 750 kg/m^3.

As a boundary condition, non-slip condition was applied on the solid wall. Turbulence intensity at cold leg was assumed to be 5%. Relative pressure was 0 Pa at the whole model. Mass flow rate of normal operation condition was considered at the cold leg of the RPV.

2.3 Turbulence Model

For flow analyses of RVIs at the steady state, Shear Stress Transport (SST) turbulence model is more appropriated than others [3]. SST model, which is one of Reynolds-Averaged Navier Stokes (RANS) based turbulence model, has an advantage in combination of the k-ε model and k-ω model. SST model uses k-ω model at wall and k-ε model at freestream. In this study, SST model was used to simulate the turbulent flow inside the RVIs.

2.4 Porous Media

In this study, fuel assemblies were considered as porous media. In order to reflect the fluid velocity and pressure drop at the real geometry, porosity and isotropic loss model were applied to the porous region.

Porosity is the ratio of the volume which are available for flow and total volume which contains both flow region and solid structure region. The porosity were determined by considering the real geometry of the reactor internal structures and their magnitudes [4]. The porosity value of the fuel assemblies was 0.54, as defined by the following equation:

$$\gamma = 1 - \frac{A_s}{A_c} \tag{1}$$

where γ is porosity of the fuel assemblies, A_c is cross-sectional area of the core and A_s in cross-sectional area of the fuel assemblies.

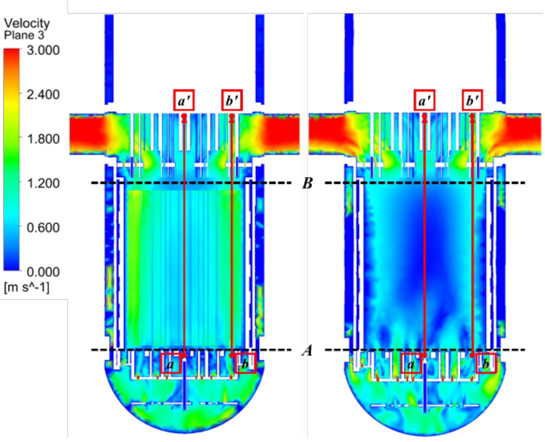
The porous model can be calculated by the velocities based on the volumetric flow rate in a porous region. A momentum source term S_i^M is added to the Navier - Stokes equations to model porous media, as shown in Eq. (2), and then the flow resistance caused by the porous material in fluid domain cab be simulated [5]:

$$\frac{\delta(\rho U_i)}{\delta t} + \frac{\delta(U_j U_i)}{\delta x_j} = -\frac{\delta \rho}{\delta x_i} + \rho g_i + \frac{\delta \tau_{ij}}{\delta x_j} + S_i^M \tag{2}$$

where $U_{i,j}$ is the velocity vector, ρ is the fluid density, τ_{ij} is the stress tensor and S_i^M is a momentum source which includes a contribution $-R \cdot U$ where R represents a resistance to flow in the porous medium.

3. Analysis Results

Fig. 2 shows typical fluid velocity distributions at hot leg plane obtained from each conditions. Also, Fig. 3 compares the velocity distributions marked as '*a-a*, *b-b*' in Fig. 2. Fig. 4 represents fluid velocity distributions at the cross-section of RVIs marked as '*A* and *B*' in Fig. 2.



(a) With fuel assemblies (b) W/o fuel assemblies

Fig. 2. Velocity distributions at hot leg plane

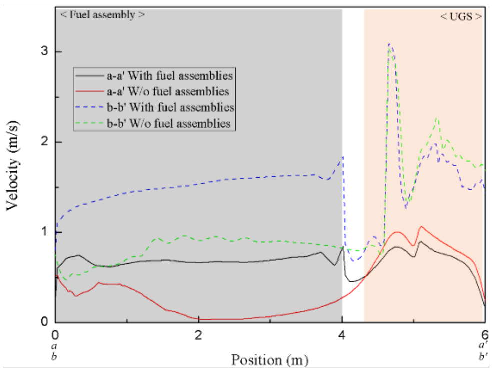
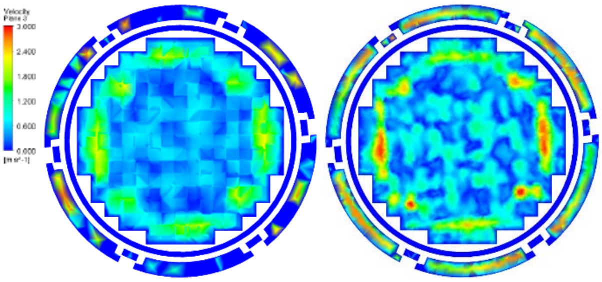
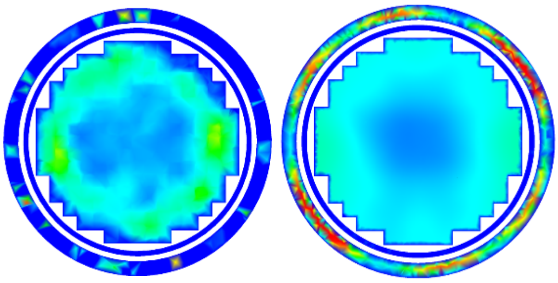


Fig. 3. The velocity distributions at *a-a*' and *b-b*'



(a) With fuel assemblies at *A* (b) W/o fuel assemblies at *A*



(c) With fuel assemblies at *B* (d) W/o fuel assemblies at *B*

Fig. 4. Velocity distributions at two cross-sections

4. Conclusion

In this research, mainly, CFD analyses of RVIs were carried out to investigate the flow characteristics. In particular, modeling effect of fuel assemblies as porous media was examined and the following key findings were observed.

(1) Fluid velocity distribution obtained from RVIs with fuel assemblies seemed reasonable than those obtained from RVIs without fuel assemblies.
(2) Fluid velocity was 4 times higher at the center of CSB (a-a') and 2 times higher near the hot leg nozzle (b-b') when fuel assemblies were considered. In the UGS, however, differences were less than 10% at each cases.

REFERENCES

[1] ANSYS CFX, Version 17.0, ANSYS Inc., 2016.
[2] D.H. Kim, Y.S. Chang and M.J. Jhung, Numerical study on fluid flow by hydrodynamic loads in reactor internals, Structural Engineering and Mechanics, Vol.51, p.1005-1016, 2014.
[3] U. Bieder and E. Graffard, Qualification of the CFD code Trio_U for full scale reactor applications, Nuclear Engineering and Design, Vol.238, p.671-679, 2008.
[4] G.H. Lee, Y.S. Bang, S.W. Woo and A.J. Cheong, Comparative study on the effect of reactor internal structure geometry modeling methods on the prediction accuracy for PWR internal flow distribution, Annals of Nuclear Energy, Vol.70, p.208-215, 2014.
[5] J.P. Cheng, L.M. Yan and F.C. Li, CFD simulation of a four-loop PWR at asymmetric operation conditions, Nuclear Engineering and Design, Vol.300, p.591-600, 2016.