Performance Assessment of Turbulence Models for the Prediction of the Reactor Internal Flow in the Scale-down APR+

Gong Hee Lee [a*], Young Seok Bang [a], Sweng Woong Woo [a], Do Hyeong Kim [b], Min Ku Kang [b]
[a]*Safety Analysis & Evaluation Department, Korea Institute of Nuclear Safety, Daejon, 305-338*
[b]*ANFLUX Inc., Seoul*
[*]*Corresponding author: ghlee@kins.re.kr*

1. Introduction

Complex thermal-hydraulic characteristics exist inside reactor because the reactor internals consist of fuel assembly, control rod assembly, and the internal structures. Either flow distribution test for the scale-down reactor model [1] or computational fluid dynamics (CFD) simulation [2] have been conducted to understand these complex thermal-hydraulic features inside reactor.

The types of errors in CFD simulation can be divided into the two main categories: numerical errors and model errors. Turbulence model is one of the important sources for model errors.

In this study, in order to assess the prediction performance of Reynolds-averaged Navier-Stokes (RANS)-based two equations turbulence models for the analysis of flow distribution inside a 1/5 scale-down APR+, the simulation was conducted with the commercial CFD software, ANSYS CFX V.14 [3].

2. Analysis Model

2.1 Test Facility and Test Conditions

Test facility is a 1/5 scale-down model of APR+ and internal structures of the reactor model, for examples flow skirt, core upper and lower structures, had almost the same shape and satisfied the geometrical similarity [1]. A number of the differential pressure transmitters were used to measure the core inlet flow rate and core outlet pressure distribution [1].

The test matrix consists of the symmetric/asymmetric flow conditions for four-pump operation and the flow conditions for three-pump operation. In this study, CFD simulation with the symmetric flow conditions for four-pump operation was conducted.

2.2 Geometry Modeling

The internal structures, especially located in the upstream of reactor core, may have a significant influence on the core inlet flow rate distribution according to the shapes and relative distance between the internal structures. Therefore an exact representation of these internal structures is needed for the reactor internal flow simulation. However, such an approach requires much more computation resource to analyze the real flow phenomena inside reactor model.

In this study, the real geometry of flow skirt and beams in the lower support structure were considered. On the other hand, fuel assembly and some internal structures, for examples instrument nozzle support, fuel alignment plate, and upper plenum were considered as each simple bulky volume (porous domain) due to the limitation of computation resource. Then, in order to reflect the velocity field and pressure drop occurring in the original flow region, porosity and isotropic loss model [4] were applied to porous domain.

3. Numerical Modeling

3.1 Numerical Method

The flow inside a 1/5 scale-down APR+ model was assumed to be steady, incompressible and isothermal. High resolution scheme was used for the convection terms of momentum equations and 1st order upwind scheme was used for the convection terms of turbulence equations. The solution was considered to be converged when the residuals of variables were below 6×10^{-4} and the variations of the target variables were small.

3.2 Turbulence Model

Among Reynolds-averaged Navier-Stokes (RANS)-based turbulence models, both standard k-ε model and Shear Stress Transport (SST) model were used to simulate the turbulent flow inside a 1/5 scale-down APR+ model. More detailed descriptions of SST turbulence model can be found in the ANSYS CFX user's guide [4].

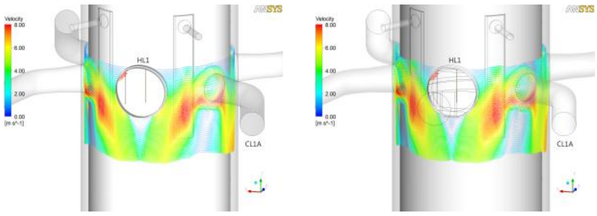
3.3 Grid System and Boundary Conditions

The grid system was generated for the computational domain that had same size as the test facility. The grid type was a hybrid mesh, made up of tetrahedron and prism.

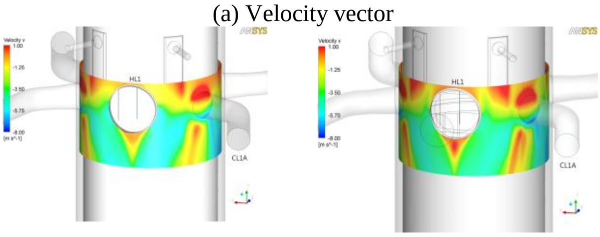
Inlet flow rate of 135 kg/s was imposed at each cold leg. Turbulence intensity at inlet was assumed to be 5 %. Light water of 60℃ was used as the working fluid. Average pressure over whole outlet option with the relative pressure of 0 Pa was used at each hot leg as an outlet boundary condition. No-slip condition was applied on the solid wall.

4. Results

Fig. 1 shows the velocity vector and velocity (y-direction) contour near the cold leg entrance region. Flow entering from the cold leg accelerated when it passed over the emergency core cooling barrel duct. The secondary flow was formed near the lower part of the hot leg. Although there was a local small difference in the velocity magnitude, the flow pattern predicted by both turbulence models was similar.



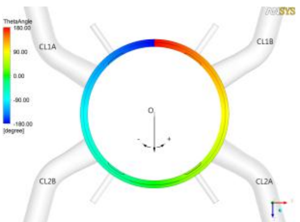
(a) Velocity vector



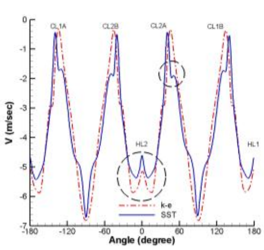
(b) Velocity contour

Fig. 1. Velocity vector and velocity (y-direction) contour near the clod leg entrance region (left side; k-ε model, right side; SST model)

Fig. 2 shows the circumferential distribution of velocity (y-direction) in the downcomer (-0.6m from the center of cold leg). Standard k-ε model predicted a little large local velocity in comparison with SST model under the hot leg. While there was some variation of velocity profile in the SST model between the cold legs, it was not seen in the standard k-ε model.



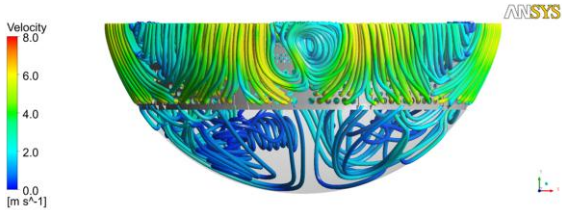
(a) Definition of angle



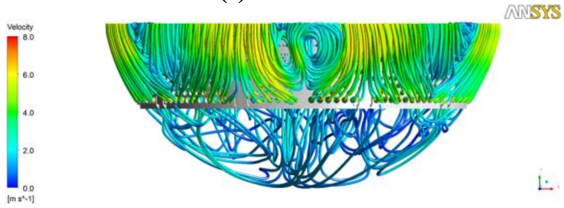
(b) Velocity distribution

Fig. 2. Circumferential distribution of velocity (y-direction) in the downcomer (-0.6m from the center of cold leg)

Fig. 3 shows the streamlines near the reactor lower plenum. Flow passing through the flow skirt mixed in the reactor lower plenum and the flow velocities were relatively low in this region. The secondary flow in the flow skirt region was formed due to the snubber lug and its shape was similar in both turbulence models.



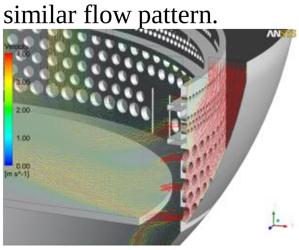
(a) k-ε model



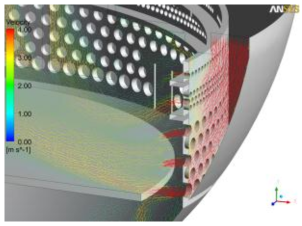
(b) SST model

Fig. 3. Streamlines at lower plenum region

Fig. 4 shows velocity vector near the flow skirt. Flow passing through the holes in the upper rows mixed with the flow near the outer boundary of the lower plenum and moved upward because it didn't have the sufficient momentum to go to the core center. On the other hand, flow through the holes in the lower rows, having relatively large momentum, moved toward the core inner region. Both turbulence models predicted the similar flow pattern.



(a) k-ε model



(b) SST model

Fig. 4. Velocity vector near the flow skirt

5. Conclusions

In this study, in order to assess the prediction performance of turbulence models for the analysis of flow distribution inside a 1/5 scale-down APR+, the simulation was conducted with the commercial CFD software, ANSYS CFX V.14. Both standard k-ε model and SST model predicted the similar flow pattern inside reactor. Therefore it was concluded that the prediction performance of both turbulence models was nearly same.

REFERENCES

[1] D. J. Euh et al., A Flow and Pressure Distribution of APR+ Reactor under the 4-Pump Running Conditions with a Balanced Flow Rate, Nuclear Engineering and Technology, Vol.44, p.735, 2012.

[2] U. Rohde et al., Fluid Mixing and Flow Distribution in a Primary Circuit of a Nuclear Pressurized Water Reactor - Validation of CFD Codes, Nuclear Engineering and Design, Vol.237, p.1639, 2007.

[3] ANSYS CFX, Version 14.0, ANSYS Inc.

[4] ANSYS CFX-Solver Modeling Guide, ANSYS Inc. (2011)