Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 24-25, 2013

Recommendations for CFD Simulation of Reactor Internal Flow

Gong Hee Lee [a]*, Young Seok Bang [a], Sweng Woong Woo [a], Ae Ju Cheong [b]
[a]*Safety Analysis & Evaluation Department, Korea Institute of Nuclear Safety, Daejon, 305-338*
[b]*Safety Issue Research Department, Korea Institute of Nuclear Safety, Daejon, 305-338*
**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, In-Core Instrumentation (ICI), and the internal structures. Either flow distribution tests for the scale-down reactor model or Computational Fluid Dynamics (CFD) simulations have been conducted to understand these complex thermal hydraulic characteristics inside reactor.

Core inlet flow rate and core outlet pressure distribution measured in the flow distribution test were used as input data for the core thermal margin code. In addition, the inlet nozzle-to-outlet nozzle pressure losses inside reactor model were used for the verification of the magnitudes calculated by pressure loss methods.

Although the competitiveness of CFD is continuously growing due to the rapid developments in computer hardware technology, computer capacity is still a limiting factor for CFD calculations to produce completely accurate results in the prediction of reactor internal flow. Therefore simplified geometries and turbulence models have to be used, and the computer capacity puts restrictions on the resolution in space and time. This leads to modeling errors and numerical errors that give more or less inaccurate results.

In this paper a summary of the recommendations drawn from the analysis of the scale-down APR+ (Advanced Power Reactor Plus) internal flow [1,2,3,4] are explained in the regulatory viewpoint.

2. Recommendations

2.1 Geometry Modeling of Reactor Internals

Reactor internals are complex structures which support the fuel assemblies, control rods and measuring instruments. The internal structures, especially located in the upstream of reactor core, may have a significant influence on the core inlet flow rate distribution depending on both their shapes and the relative distance between the internal structures and the core inlet. Therefore if the sufficient computation resource is available an exact representation of these internal structures, for examples lower support structure bottom plate, ICI nozzle support plate, and flow skirt, is needed for the accurate reactor internal flow simulation [1,2].

2.2 Porous Medium Assumption

An approach considering the real geometry of reactor internals requires much more computation resource to analyze the real flow phenomena inside reactor. Porous medium assumption may be one of candidates to solve this problem. In this assumption, some internal structures were considered as each simple bulky volume (porous domain). Then, in order to reflect the velocity field and pressure drop occurring in the original flow region, porosity and isotropic loss model were applied to porous domain [1,2].

Porosity has an effect on flow acceleration in the porous domain and its magnitude is generally determined by considering the real geometry of the reactor internal structures. A momentum source was used to model the momentum loss in the porous domain which corresponds to pressure drop in real reactor vessel. However, because pressure drop in reactor vessel is locally different due to the complex geometry of reactor internals, isotropic loss model may have a limited applicability and give an inaccurate result. Therefore licensing applicant should be very careful to use isotropic loss model.

2.3 Turbulence Model

Because the reactor internal flow analysis require a huge number of grids it may be practically impossible to use the advanced turbulence model such as Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS). Therefore Reynolds-averaged Navier-Stokes (RANS)-based two equation turbulence models can be used as the potential candidates.

It is well known that standard k-ε model has been widely used in the various industrial applications and has a superior convergence in comparison with other turbulence models. Shear Stress Transport (SST) model has the possibility of giving the superior performance in the reactor internal flow where flow impingement and reattachment, swirling and re-circulation flow, flow with strong buoyancy effects and high streamline curvature can exist.

In the previous work [3], sensitivity study was conducted to select the turbulence models suitable for the analysis of the turbulent flow inside the scale-down APR+ model. It was confirmed that among the two-equation turbulence models in CFX R.14, standard k-ε model showed that its prediction performance was equivalent to SST model. On the other hand, Renormalization Group (RNG) k-ε model didn't show reasonably the complex mixed flow pattern in the lower plenum.

2.4 CFD Software

Although recently licensing applications supported by using the commercial CFD software are increasing, there is no commercial CFD software which obtains a licensing from the domestic regulatory body until now. In addition, there is no guideline for the comprehensive evaluation of CFD software. Therefore, from a regulatory perspective it is necessary to perform the systematic assessment and prepare the guideline for the prediction performance of the commercial CFD software.

According to the previous work [4], although there was a limitation in estimating the prediction performance of the commercial CFD software due to the limited number of the measured data, CFX R.14 showed the more reasonable predicted results in comparison with FLUENT R.14 when considering the range of mass flow rate at core inlet plane and flow mixing pattern in the lower plenum.

Because of the difference of discretization methodology, FLUENT R.14 (node based discretization scheme) required more the computational memory than CFX R.14 (cell based discretization scheme) for the same grid system. Therefore the CFD software suitable to the available computational resource should be selected for the massive parallel computation.

2.5 Grid Sensitivity

The sufficient number of grid should be used such that an adequate resolution can be obtained. A good grid quality is essential for obtaining an excellent CFD analysis result. Therefore, the grid quality before performing a numerical analysis of reactor internal flow should be assessed. To resolve boundary layers in the reliable level, non-orthogonal (e.g. unstructured tetrahedral) grid should not be used in boundary layers. A hybrid grid, made up of tetrahedron and prism, may be a candidate for improving grid resolution in the near wall region. Because it may be difficult to obtain the grid independence result for reactor internal flow analysis, the conservative analysis result obtained from the sensitivity study for several grid systems using the same grid topology should be used for the licensing application.

2.6 Discretization Accuracy

Spatial discretization errors result from both the numerical order of accuracy of the discretization scheme and the grid spacing. When the flow is not aligned with either triangular or tetrahedral grids, first-order convective discretization scheme increases the discretization error (numerical diffusion) and second-order convective discretization scheme may give more accurate results. According to the previous work [5], if higher order discretization scheme was used for the convection terms of momentum equations, the accuracy order of the convection terms of turbulence equations had little influence on the final result. In addition, higher order convective discretization did not always guarantee the more accurate solution. Therefore the conservative analysis result obtained from the sensitivity study for the discretization accuracy should be used for the licensing application.

3. Conclusions

In this paper a summary of the recommendations for CFD simulation of reactor internal flow were explained in the regulatory viewpoint. Among them, an exact representation of the internal structures, especially located in the upstream of reactor core, may be the most important item for the accurate reactor internal flow simulation. To enhance the completeness of this study an additional CFD simulation to consider real geometry of reactor internal structures is on-going and the simulation results will be explained in the separate papers.

Acknowledgments

This study was conducted under the financial support of the Nuclear Safety and Security Commission of Korea [project title: Development of Regulatory Evaluation Technologies for Thermal-hydraulic Safety]. The authors gratefully thank Dr. Kim and Mr. Yim in the Central Research Institute of KHNP (Korea Hydro & Nuclear Power) for providing the research materials.

REFERENCES

[1] G. H. Lee, Y. S. Bang, S. W. Woo, A. J. Cheong, D. H. Kim, M. K. Kang, A Numerical Study for the Effect of Flow Skirt Geometry on Reactor Internal Flow, Annals of Nuclear Energy, Vol.62, p.452, 2013.

[2] G. H. Lee, Y. S. Bang, S. W. Woo, D. H. Kim, M. K. Kang, CFD Simulation of Reactor Internal Flow in the Scaled APR+, Journal of Energy and Power Engineering, Vol.7, p.1533, 2013.

[3] G. H. Lee, Y. S. Bang, S. W. Woo, D. H. Kim, M. K. Kang, Performance Assessment of Turbulence Models for the Prediction of the Reactor Internal Flow in the Scale-down APR+, Transactions of the Korean Nuclear Society Spring Meeting, 2013, Gwangju, Korea.

[4] G. H. Lee, Y. S. Bang, S. W. Woo, D. H. Kim, M. K. Kang, Comparative Study of the Commercial CFD Software Performance for the Prediction of the Reactor Internal Flow, The Korean Society of Mechanical Engineers Spring Meeting, 2013, Jeju, Korea.

[5] G. H. Lee, Y. S. Bang, S. W. Woo, Numerical Analysis of Turbulent Flow around a Tube Bundle by Applying CFD Best Practice Guideline, The Korean Society of Mechanical Engineers Autumn Meeting, 2012, Changwon, Korea.