Comparison of RANS-Based and LES Turbulence Models for Core Flow Analysis in a 1/5-Scale APR1000 Model

Uiju Jeong*, Gilim Kim, Juyeon Lee, Nuri Oh, Sin Jung Chang
[a]*SMR Development Lab., KHNP Central Research Institute, 70, Yuseong-daero 1312beon-gil, Daejeon, Korea*
**Corresponding author: ujjeong88@khnp.co.kr*

***Keywords : APR1000, CFD, RANS-Based, LES, Core inlet flow distribution**

1. Introduction

The APR1000 reactor, which combines proven technologies from APR1400 and OPR1000 with advancements from APR and EU-APR, has been selected for the Czech Republics's nuclear new-build project. KHNP has been officially selected as the preferred bidder for the project.

The present study describes the CFD model and its results, focusing on identifying areas for improvement in the current CFD model through comparative analysis with experiment. Extensive CFD simulations have been conducted to evaluate core flow distribution using a APR1000 reactor 1/5 scale model. In addition to the Reynolds-Averaged Navier-Stokes (RANS)-based models, we have implented a Large Eddy Simulation (LES)-based CFD model to enhance the prediction of complex turbulent flow behaviors within the reactor vessel, especially lower plenum. LES, by directly resolving large-scale turublent eddies while modeling only the smaller scales, offers improved accuracy in simulating flow mixing phenomena, which are critical for predicting the core inlet flow distribution. The application of LES is particularly beneficial for resolving large-scale vortexs and recirculation zones in regions such as the downcomer, lower plenum, and core inlet, where traditional RANS models often fall short.

The primary objective of this study is to compare the results obtained from LES-based CFD simulations with those from RANS-based simulations and experimental data, highlighting the advantages of LES in predicting core inlet flow uniformity. We hope to share the accuracy level of the core flow distribution simulation using a commercial CFD analysis tool.

2. Method and Results

The CFD model has been developed for a 1/5 scale model of the APR1000 using a commercial CFD software ANSYS CFX. The CFD results are compared and analyzed against the experimental data [1].

2.1. Geometry model

Figure 1 shows a geometry of the 1/5 scale model, mainly consists of six parts such as cold legs, downcomer, lower plenum, core, upper plenum, and hot legs. The lower plenum contains complex structures such as flow skirt and lower support structures, which facilitate flow mixing while also making it difficult to accurately predict the flow. The core consists of 177 core simulators, each of which simulates a single actual fuel assembly, and is specifically designed to have similarity in terms of inlet and outlet pressure drop and crossflow mixing characteristics [1].

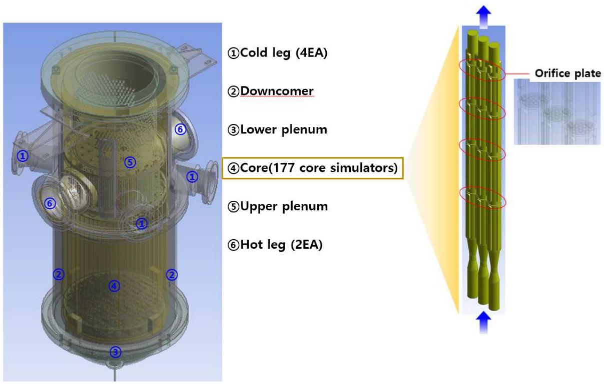


Fig. 1. Geometry of a 1/5 scale model of the APR1000

2.2.Grid model

A grid structure for the geometry model has been made by using ANSYS Workbench mesh program, shown in Fig. 2.

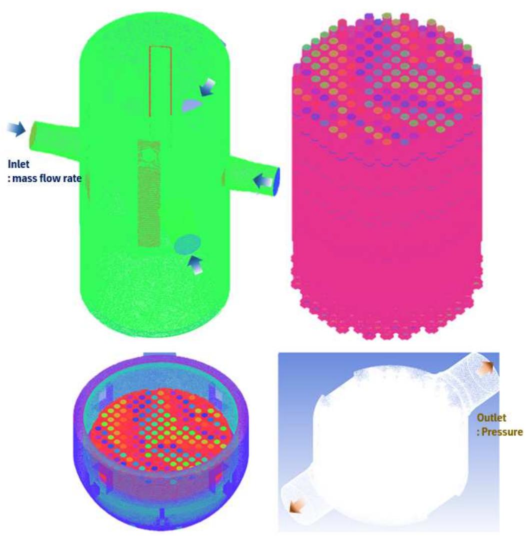


Fig. 2. Grid model of a 1/5 scale model of the APR1000[2]

In a core simulator, four thin perforated plates are installed to simulate differential pressure characteristics, shown in Fig.1. By setting these perforated plates as porous medium domains, the number of mesh cells could be significantly reduced.

2.3.Solver setting

2.3.1. RANS-based Turbulence model

A sensitivity analysis of turbulence models was conducted by applying three different RANS-based turbulence models (standard k-ε, shear stress transport, k-ε EARSM model). Based on this sensitivity analysis, the standard k-ε turbulence model is selected.

In the steady state simulation, the inlet boundary was set as the mass flow rate with a specific value at 60ºC determined by the methdology of scaling analysis, while the outlet pressure was set to 0 Pa. All wall boundaries were defined as non-slip, and the scalable wall function was employed to model the flow behavior near the walls. The calculation was considered to have converged when the RMS residual dropped below 10^{-3}, and also both the core inlet flow rates and differential pressure at the core simulators stably converged to specific values.

2.3.2. Large Eddy Simulation

The LES approach is implemented as follows. Large-scale eddies, which exhibit strong anisotropic characteristics such as vortex stretching, are directly resolved due to the difficulty in mathematical modeling.

Although LES provides high accuracy, accurate boundary layer modeling requires the first grid point near the wall to be located in the laminar sublayer. While a wall function can be applied in the log-law layer, LES still requires significantly finer grids compared to RANS, which leads to a substantial increase in computational cost when solving the entire domain.

There are three primary models in LES: the WALE (Wall-Adapting Local Eddy-Viscosity) model, the Smagorinsky model, and the Dynamic Smagorinsky-Lilly model. Among these, the WALE model is generally preferred over Smagorinsky-based LES models for better near-wall accuracy and improved numerical stability.

The central difference scheme (CDS) is used for the treatment of diffusion as it is preferred for the following reasons.

- Second-order accurate : the high resolution need for turbulence simulations
- Minimal artificial viscosity : no excessive numerical viscosity
- Compatibility with LES modelinlg : well-balanced velocity field, which CDS naturally provides due to its symmetric formulation

To mitigate numerical instability, however, Bounded CDS is used for the present study. CDS is inherently non-dissipative, causing small errors to amplify. To retain the benefits of CDS while preventing numerical instability, Bounded CDS, combining Upwind Scheme with CDS as follows, is applied in this study.

$$\varphi = \alpha\varphi_{Upwind} + (1-\alpha)\varphi_{CDS} \qquad (1)$$

where $0 \le \alpha < 1$

A constant value of 0.2 is set to α for prevention of numerical instability. The automatic wall function was employed to model the flow behavior near the walls. The total simulation time was set to 2 seconds, with a time step of 0.00005 seconds. The simulation results were obtained by applying a time-averaging process over a duration of 0.3 seconds.

2.4.Results

2.4.1. Turbulence model sensitivity

For the turbulence model sensitivity evaluation, the differences between the calculation and experimental values of the inlet flow rate of the core simulator were statistically analyzed. The standard deviations of the flow rate deviations are presented in Table I. Among RANS-based models, standard k-ε model, which provided results most similar to the experimental data, was selected as the optimal turbulence model. However, when incorporating the LES model, the simulation results exhibit the highest agreement with the experimental data. As shown in Figure 3, the pressure profiles from the cold leg to the hot leg exhibit similar trends in both the simulation and experimental results. However a slight discrepancy is observed in the pressure drop from the cold leg to the core inlet. This diffeence is primarily attributed to the symmetric veolcity profile imposed at the cold leg inlet in the simulation, whereas in the experiment, an asymmetric velocity profile is induced by the upsteram bend section of the cold leg.

It is noteworthy that the simulation results show a higher pressure drop compared to the experimental results as the fluid enters the core simulator from the lower plenum. Although the cause for this has not yet been clearly identified, it is important to forcus on the phenomenon of 'flow mixing' to explain this. What we can clearly observed from Fig. 4 is that the flow distribution at the core inlet is much more uniform in the experiment compared to the RANS-based simulation results. In other words, less flow mixing occurs in the simulation than in the experiment, leading to a more severe velocity gradient in the simulation. Considering that a larger velocity gradient tends to result in a greater pressure drop, the higher pressure drop observed in the simulation can be explained by the reduced flow mixing, compared to the experiment. The LES model, which simulates flow mixing more realistically by modeling large eddies, shows a pressure drop more closely matching the experimental results compared to that of

RANS-based simulation results. This support the above explanation.

Table I. Summary of deviation analysis

Category	Standard deviation
Exp. - SST	5.69%[2]
Exp. - Std. k-ε	4.89%[2]
Exp. - EARSM	5.51%[2]
Exp. - LES	4.5%

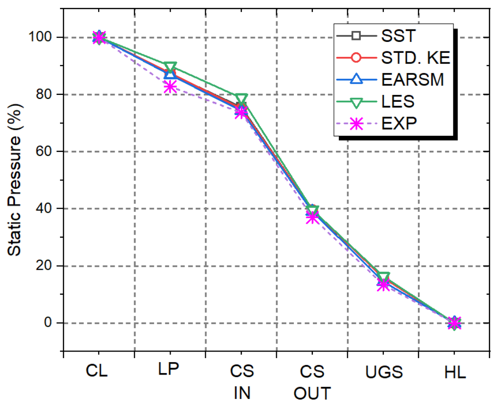
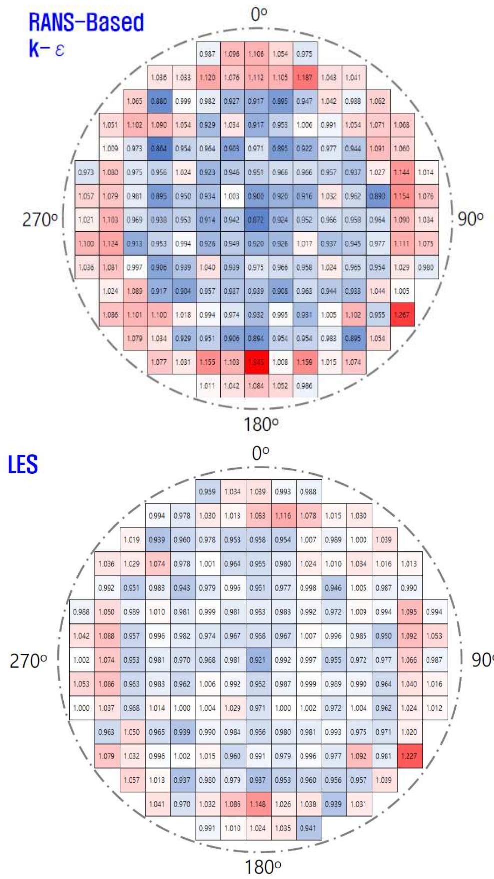


Fig. 3. Static pressure profiles along the reactor flow path

2.4.2. CFD results and its validation



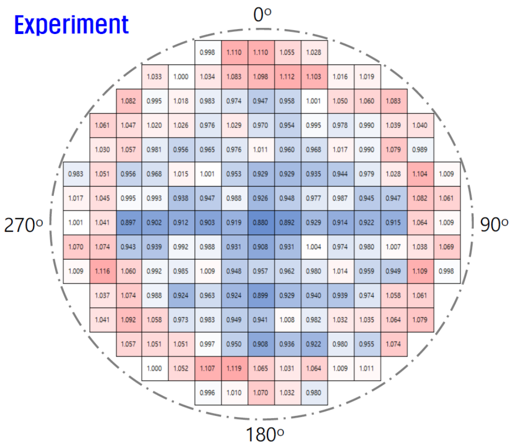


Fig. 4. Core inlet flow distribution from CFD and Exp.

Both the CFD simulation and experiments showed similar results, with higher flow rates forming in the outer region and relatively lower flow rates forming in the central region. Howerver, in the flow distribution results from the simulations, it was observed that some specific core simulator exhibited more than a 15% flow deviation compared to adjacent core simulator, which is somewhat difficult to consider as a realistic deviation. Compared to the RANS-based turbulence model results, the LES model significantly reduced the regions where large flow deviations occur.

The numerous complex and even asymmetrically arranged structures in the lower plenum make it difficult to accurately predict turbulent flow through a CFD simulation. It is presumed that the significant flow rate deviations among the adjacent core simulators were caused by the limited simulation performance of turbulent behavior, which in turn restricted the flow mixing phenomena. The first basis for this assumption is the fact that the pressure drop occurring as the flow enters the core from the lower plenum shows the greatest discrepancy between the experiment and the analysis, as observed in Figure 3.

The second basis is that the locations where significant flow rate deviations between core simulators occur are mostly in the outer regions of the core. It can be suggested that the flow path from the downcomer to the core outer region is the shortest, which could result in the least flow mixing. In the experiment, rapid flow homogenization occurs as the flow passes through the flow skirt and lower support plate. However, in the CFD simulation, it is observed that when the flow path is shorter, sufficient flow mixing does not occur, resulting in less flow homogenization compared to the experiment, especially in the case of RANS-based turbulence model.

The results of the statistical comparison between the simulation and experimental data regarding the core inlet flow distribution are presented in Table II. Although it was found that the simulation accuracy for the flow distribution in the core outer region is low, as show in Fig. 4, the overall flow differences between the experiment and CFD were within 10% with a 95% confidence interval.

Table II. Summary of deviation analysis

Category	Standard deviation
Std. k-ε	7.65%[2]
LES	4.22%
Exp.	5.59%
Exp. - Std. k-ε	4.89%[2]
Exp. - LES	4.50%

3. Conclusison

In this study, the core inlet flow distribution in a 1/5 scale model of the APR1000 was calculated using the ANSYS CFX software, and it was confirmed that the error compared to the experimental values was within 10%. It is important to note that the accuracy of the RANS-based simulation results for the inlet flow distribution in the core outer region was somewhat low. This results is presumed to be due to the limitations in simulating large eddies. This is because large eddies, which occur when the fluid passes through obstacles or small holes, facilitate rapid flow mixing.

Improving the simulation performance of flow mixing phenomena in the domain between the downcomer and the core outer region is considered a key point for enhancing the predictive accuracy of the core inlet flow distribution. Accordingly, the LES model, capable of resolving large eddies, was applied for the simulation. Notably, the computational time was approximately 30 times longer than that of the RANS-based turbulence model.

The LES model, which simulates flow mixing more realistically by modeling large eddies, shows a core inlet flow distribution more closely matching the experimental results compared to that of RANS-based simulation results

REFERENCES

[1] K. Kim, W.-S. Kim, H.-S. Choi, H. Seol, B.-J. Lim and D.-J. Euh, An Experimental Evaluation of the APR1000 Core Flow Distribution Using a 1/5 Scale Model, Energies, Vol.17, p. 2714, 2024.

[2] U. Jeong, Y. Choi, K. Kim and D.-J. Euh, CFD simulation for predicting core inlet flow distribution in a 1/5 scale model of the APR1000, Transactions of the Korean Nuclear Society Autumn Meeting, Korean Nuclear Society, 2024

Acknowledgements

This work was supported by the Innovative Small Modular Reactor Development Agency grant funded by the Korea Government(MOTIE) (No. RS-2024-00404240).