Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 12-13, 2016

A Study on the Turbulence Models for Full-Scale Analysis of Reactor Vessel Internals

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

Reactor vessel internals (RVIs) support reactor core and control element assembly. Also, it reduces vibration that may occur in the fuel assemblies and control rod assemblies due to the flow of coolant as well as protect reactor core and fuel assemblies from the external lead. Since there are complex thermal-hydraulic phenomena inside a reactor, understanding of the flow characteristics in the RVIs is an important factor when designing nuclear power plants (NPPs).

Many studies have been conducted to analyze the flow characteristics of the RVIs in the steady states [1]. However, there are a few study on flow analysis and the suitability of the turbulence model in transient conditions. The rapid flow transients may cause water hammering [2], so that the characteristics of reactor coolant flow have great effect on the safety of NPPs.

In this paper, a numerical study was performed to analyze coolant flow distribution in RVIs of optimized power reactor (OPR-1000). An analysis method was established to evaluate operating transient conditions. Simulation was carried out by using the commercial computational fluid dynamics software, ANSYS CFX V.16.2 [3].

2. Computational Model

In general for steady state flow analyses, shear stress transport (SST) turbulence model is more appropriate than others [4]. However, it is necessary to examine which turbulence model is proper for analysis of transient flow. Thereby, two representative turbulence models were briefly examined.

2.1 SST Turbulence Model

SST 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. Governing equation of SST model is below (1)

$$\frac{\partial(\rho k)}{\partial t} + \nabla\cdot(\rho U k) = \left[\left(\mu + \frac{\mu_t}{\sigma_{\omega 3}}\right)\nabla\omega\right] - \beta_3\rho\omega^2 + (1-F)2\rho\frac{1}{\sigma_{\omega 2}\omega}\nabla k\nabla\omega + \alpha_3\frac{\omega}{k}p_k \quad (1)$$

where, ρ is the fluid density, U is the velocity vector, μ is the viscosity, μ_t is the turbulent viscosity that is linked to the turbulence energy (k) and frequency (ω) as $\mu_t = \rho k/\omega$, σ_k, σ_ε and σ_ω are the turbulent Prandtl numbers, P_k is the turbulence production caused by the viscous forces and F is the blending function. In addition, α, β and so on are constants used in the analysis.

2.2 DES Turbulence Model

Detached eddy simulation (DES) model has been used as a three-dimensional unsteady numerical solution. Its functions are combination of large-eddy simulation as a sub-grid scale model in regions where the grid density is fine enough and Reynolds-averaged model in regions where it is not [5]. Particularly, in DES model the nearest-wall-distance n that governs the eddy viscosity in the original Spalart-Allmaras one-equation model is replaced in the DES by a new length scale $\tilde{n}$ defined as

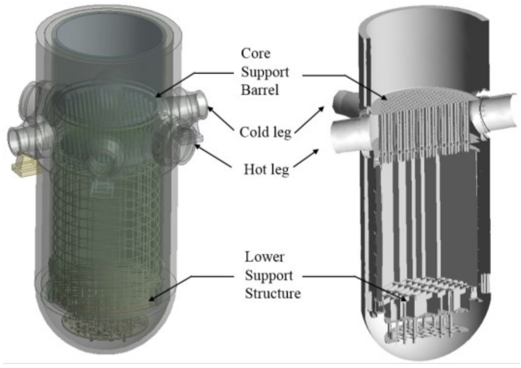
$$\tilde{n} = \min\ [n, 0.65\Delta_{max}], \quad \Delta_{max} = \max\ [\Delta x, \Delta y, \Delta z]$$

where $\Delta x, \Delta y$ and Δz denote the size of the grid spacing in each direction. Recently, many studies reported that DES model was better agreement with the experimental result [6].

3. CFD Analysis

3.1 Analysis Model and Methods

Fig. 1 depicts a schematic of OPR-1000. From the analysis of existing studies and expert opinion, the cooling water moving to the upper head of reactor pressure vessel (RPV) through upper guide support (UGS) was less than 0.1% of total quantity in its flow. So, in the present study, upper part of RPV was not modeled. Total number of elements and the y^+ value were 3.4×10^7 and 0.102 for RVIs. The flow inside the reactor internal was assumed to be incompressible, isothermal and turbulent.

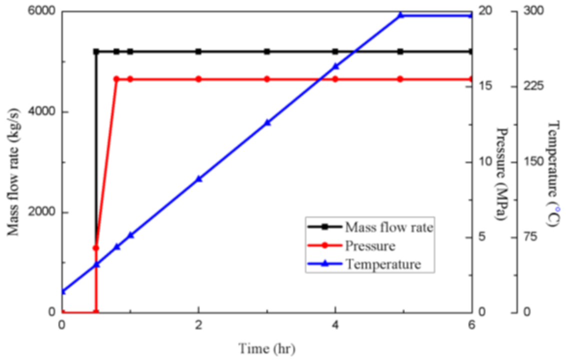


(a) Real arrangement (b) CFD Model

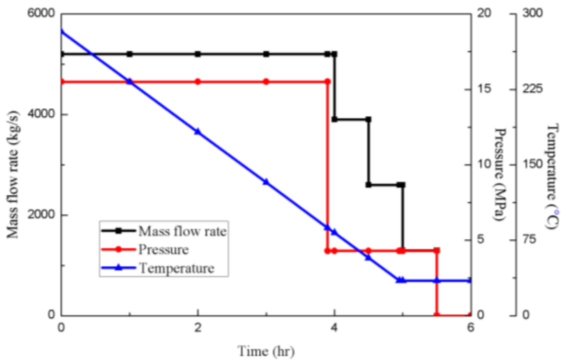
Fig. 1. Schematic of OPR-1000

3.2 Analysis Conditions

Fig. 2 represents typical thermal-hydraulic data of heat-up and cool-down conditions, which were applied to the cold leg depicted in Fig. 1.



(a) Heat-up condition

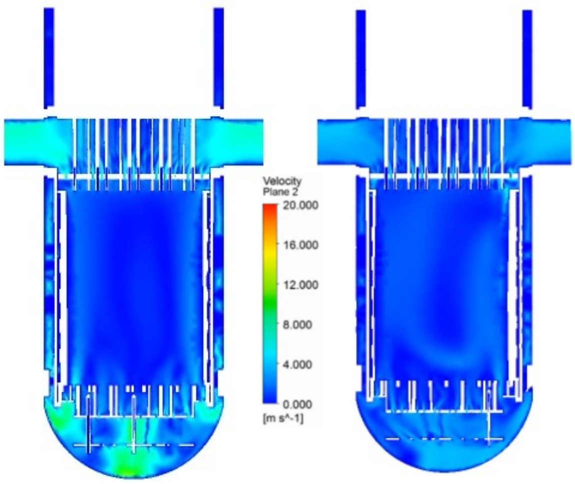


(b) Cool-down condition

Fig. 2. CFD analysis conditions

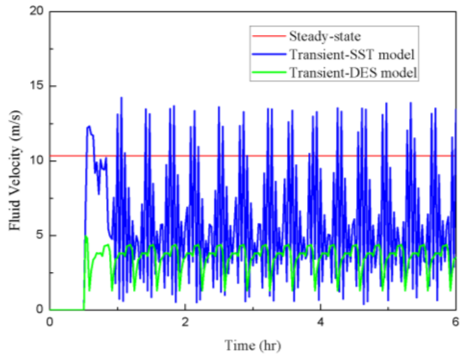
3.3 Analysis Result

Fig. 3 shows typical fluid velocity distributions at outlet plane obtained from DES model for each heat-up and cool-down condition. Also, Fig. 4 compares fluid velocities calculated from both SST model and DES model at the center of LSS.

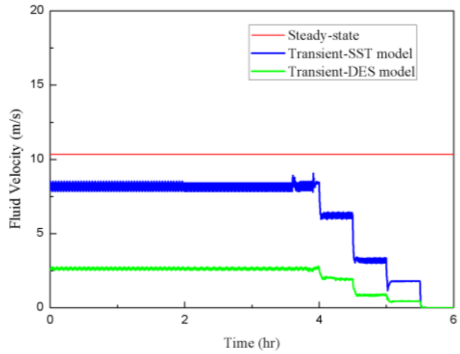


(a) Heat-up at 1.2hr (b) Cool-down at 4.8hr

Fig. 3. Velocity distribution at outlet plane



(a) Heat-up condition



(b) Cool-down condition

Fig. 4. Comparison of fluid velocities at the center of LSS

4. Conclusions

This study was to examine two representative turbulence models for full-scale analysis of RVIs. Thereby, the following key findings were observed.

(1) Fluid velocity and pressure distributions obtained from DES model were lower than those obtained from SST model.

(2) Temperature profiles calculated from both turbulence models were the same.

REFERENCES

[1] 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.

[2] Q. Zuo, S. Qiu, W. Lu, W. Tian, G. Su and Z. Xiao, Water hammer characteristics of integral pressurized water reactor primary loop, Nuclear Engineering and Design, Vol.261, p.165-173, 2013.

[3] ANSYS CFX, Version 16.2, ANSYS Inc., 2015.

[4] 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.

[5] A. Travin, M. Shur, M. Strelets and P. Spalart, Detached-eddy simulations past a circular cylinder, Flow Turbulence and combustion, Vol.63, p.293-313, 2000.

[6] P. Ding, M. Chen, W. Li, Y. Liu and B. Wang, CFD simulations in the nuclear containment using the DES turbulence models, Nuclear Engineering and Design, Vol.287, p.1-10, 2015.