Assessment and Improvement of Macroscopic Turbulence Model Using 6×12 Rod Bundle Test

Seok Kim [a*], Jee Min Yoo [a], Hae-Seob Choi [a], Young-Jung Youn [a], Sang-Ki Moon [a]
[a] *Korea Atomic Energy Research Institute, 111 Daedeokdaero989beongil, Yuseong-Gu, Daejeon, Republic of Korea*
[*] *Corresponding author: seokim@kaeri.re.kr*

1. Introduction

The rod bundle is widely used geometry in heat exchangers, steam generators, and fuel assembly in a pressurized water reactor. In the rod bundle, fluid passes through subchannel defined by space surrounded by rods. Unlike flow in the simple channel flow, transverse flow between adjacent subchannel exists as well as axial flow. This large-scale motion results in momentum exchange and turbulent mixing which has a large effect on rod temperature. Therefore, better understanding on the subchannel flow is important to the design and optimization of fuel assemblies and studies on nuclear safety. Base on the previous study (Bestion et al., 2017), precise analysis on flow characteristics in rod bundle was justified. In the small break loss of coolant accident (SB-LOCA) and the intermediate break LOCA (IB-LOCA), because of different neutronic power in rods, thermal-hydraulic parameters show radial difference. It generates cross flow inside the rod bundle named as 'chimney effect' or 'divergent' depending on surrounding conditions. It results in difference in peak cladding temperature of rods. Therefore, development of turbulent model for these phenomena is crucial for nuclear safety. A macroscopic analysis of the PRIUS-II experiment was conducted using the CUPID code (Jeong et al., 2010; Yoon et al., 2014), and, through this study, the performance of the existing source term models of the macroscopic turbulence equation was evaluated. Based on the evaluation results, an improved source term model was proposed.

2. Macroscopic Turbulence Model

The macroscopic model provides the averaged flow characteristics by spatially and temporally integrating with respect to the microscopic point of view that analyzes the flow in detail. The macroscopic turbulence model has additional source terms and resulting from averaging the microscopic equations. Several source term models have been developed for various geometry conditions. The rod bundle geometry, which is the subject of this study, has a longitudinal flow condition in which the flow direction and the structure are parallel. Chandesris et al. (2006) and Nakayama and Kuwahara (2008) developed source term models for this geometry condition. The Improvements to the source term model were proposed to properly predict the TKE in the region with the velocity gradient. The basic concept of improvement is that the velocity gradient will affect the generation or dissipation of turbulence. Therefore, the improved model includes the axial-velocity gradient term. First, the modeling for S_ε is to add a velocity gradient term to the c_p of k_∞. Second, a term containing a velocity gradient is added to S_k. The descriptions of the two models are as follows.

Based on the assumption that the velocity gradient affects the turbulence dissipation, a new type of c_p was proposed. The new cp has the following form:

$$c_p = 0.01 + c_1 \mathrm{Re}_l^{c_2} \left(\frac{\partial u_z}{\partial x} \right)^2 \tag{1}$$

Equation (1) gives 0.01 under the condition where the velocity gradient is zero, and increases the c_p value in the region where the velocity gradient exists. The turbulent dissipation decreases, and the TKE increases accordingly. The c_1 and c_2, were obtained through trial and error based on the PRIUS-II experimental data, and were 0.6 and -1.0, respectively.

Just as eddy is caused by the velocity difference near the wall, eddy can be additionally generated by the axial-velocity gradient. Based on this assumption, a term including the velocity gradient was added to the source term of the TKE transport equation.

$$S_k = \varepsilon_\infty + S_a \tag{2}$$

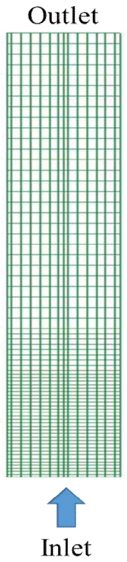
$$S_a = c_k \frac{\mu}{\rho} \left(\frac{\partial u_z}{\partial x} \right)^2 \tag{3}$$

The coefficient c_k was defined as follows based on the PRIUS-II experimental data:

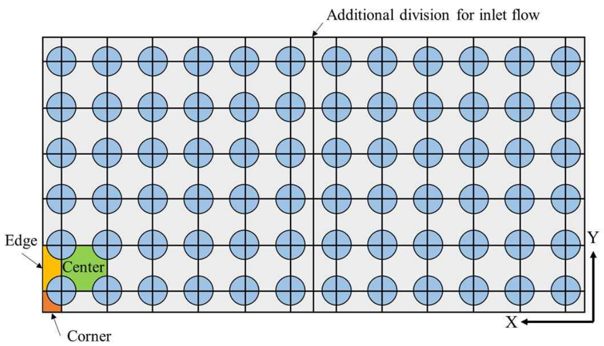
$$c_k = 0.225 \mathrm{Re}_l \tag{4}$$

3. CUPID Code Analysis

The PRIUS-II test facility was assessed with a component scale thermal hydraulic analysis code, CUPID, to investigate the macroscopic turbulence model for rod bundle geometry. The mesh for the CUPID calculation is shown in Figure 1. As shown in Figure 1(a) showing the axial grid, the grid was densely formed in the inlet region where fluid mixing occurs actively, and the grid density was set low in the upper part where there was relatively little mixing. The cross-sectional grid was divided into a corner cell, an edge cell, and a center (subchannel) cell as shown in Figure 1(b). Because there is a partition wall to separate the flow under the entrance of the experimental facility (the region under the rod bundle), the grid located at the center of the X-axis was further divided. In the CUPID analysis, the location where the inlet velocity was measured was set as the starting point of the calculation, and accordingly, the inlet region where rod bundles did not exist was included in the analysis.



(a) Axial mesh

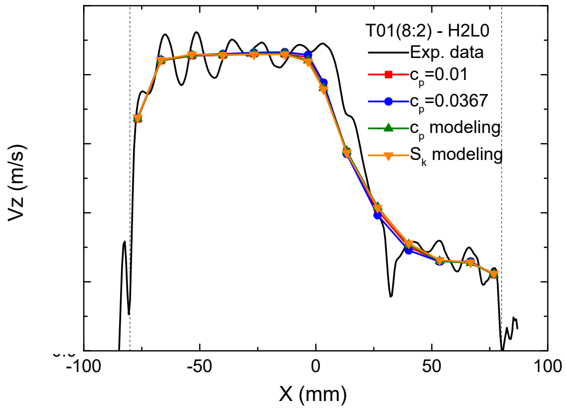


(b) Cross-sectional mesh

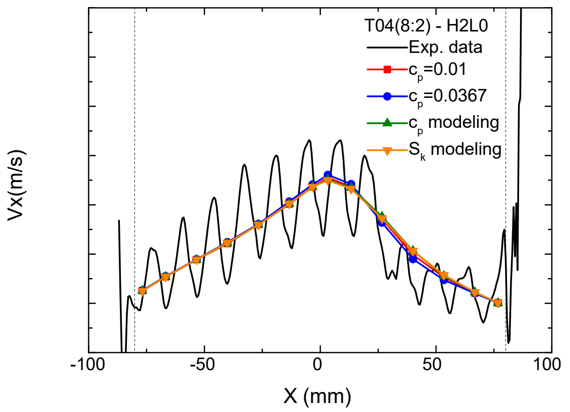
Fig. 1. Axial and cross-sectional mesh in the CUPID calculation

For the model assessment, PRIUS-II experimental data were used. The experiment was performed under the condition of Reynolds number of 6000, 9000, and 12000, and two asymmetric inlet conditions (8:2, 6.5:3.5) and one symmetric inlet condition (5:5) were conducted for each Reynolds number condition.

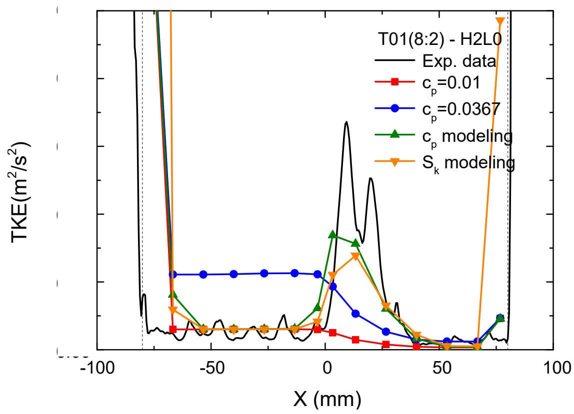
The calculation results applying the new source term model are shown in Figure 1 to Figure 4. In these figures, the red line is the result of c_p = 0.01, the blue line is the original Chandesris model (c_p = 0.0367), the green line is the c_p modeling, and the orange line is the S_k modeling. Figure 1 shows the calculation result at the H1L0 position of the T01 test (Re 12000, 8:2). In the graph comparing the axial and lateral velocities in Figure 1(a) and (b), it was identified that there is no change in the velocity prediction even if the new model is applied. And, looking at the results for the TKE and turbulence intensity in Figure 1(c) and (d), it is confirmed that the new model predicts the trend in the region with a velocity gradient well. In the region where the velocity gradient is zero, the results of the new model and the original model were almost the same, and the new source term was operated only in the region where the velocity gradient is not zero. The analysis results for different Reynolds number conditions are given in Figures 2 and Figure 3.



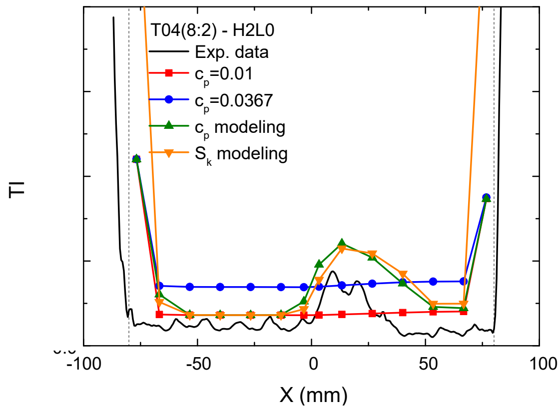
(a) Axial-velocity distribution



(b) Transverse-velocity distribution

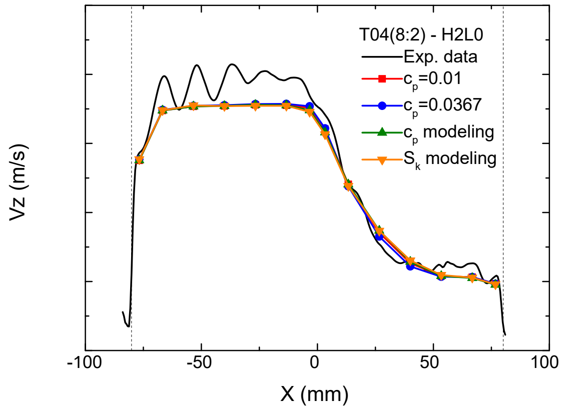


(c) Transverse distribution of the TKE

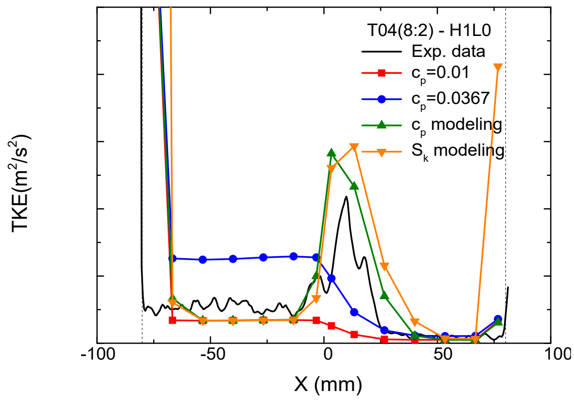


(d) Transverse distribution of the turbulence intensity

Fig. 1. Application of the new source term models: Test T01(8:2, Re 12000) – H1L0



(a) Axial-velocity distribution

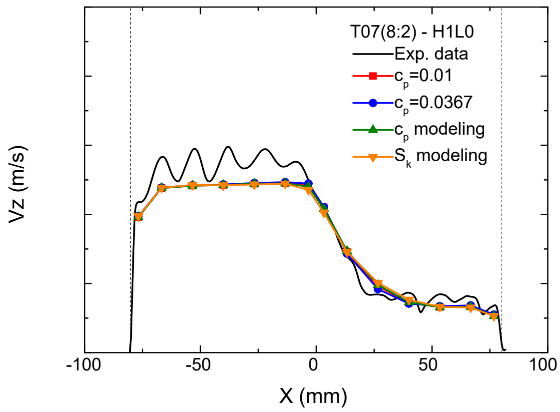


(b) Transverse distribution of the TKE

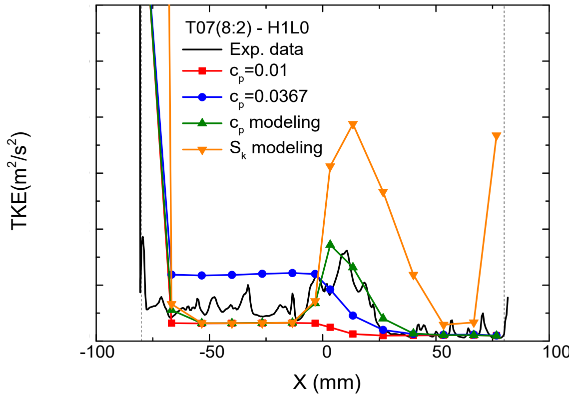
Fig. 2. Application of the new source term models: Test T04(8:2, Re 12000) – H1L0

4. Conclusion

The Improvements to the source term model were proposed to properly predict the TKE in the region with the velocity gradient using PRIUS test results. The basic concept of improvement is that the velocity gradient will affect the generation or dissipation of turbulence. the two models considering the axial-velocity gradient had little effect on the axial and lateral velocity predictions,



(a) Axial-velocity distribution



(b) Transverse distribution of the TKE

Fig. 3. Application of the new source term models: Test T07(8:2, Re 12000) – H1L0

and only affected the prediction of the TKE and turbulent intensity in the region with the velocity gradient. The new models predicted the trend of TKE under asymmetric flow conditions better than the previous model. The experimental database in a rod-bundle geometry will be addressed the modeling and validation of sub-channel analysis. It can also be useful for CFD in open medium validation.

ACKNOWLEDGEMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP).

REFERENCES

[1] Bestion, D., Valette, M., Fillion, P., Gaillard, P., 2017. 3D Core thermalhydraulic phenomena in PWR SBLOCAs and IBLOCAs, NURETH-17.
[2] Jeong, J.J., Yoon, H.Y., Park, I.K., and Cho, H.K., 2010. The CUPID code development and assessment strategy. Nuclear Engineering and Technology, 42(6), 636-655.
[3] Yoon, H.Y., Lee, J.R., Kim, H., Park, I.K., Song, C.H., Cho, H.K., and Jeong, J.J., 2014. Recent improvements in the CUPID code for a multi-dimensional two-phase flow analysis

of nuclear reactor components. Nuclear Engineering and Technology, 46(5), 655-666.

[4] Nakayama, A., and Kuwahara, F., 2008. A general macroscopic turbulence model for flows in packed beds, channels, pipes, and rod bundles, Journal of Fluids Engineering, 130(10), 101205-1-101205-7.

[5] Chandesris, M., Serre, G., and Sagaut, P., 2006. A macroscopic turbulence model for flow in porous media suited for channel, pipe and rod bundle flows. International journal of heat and mass transfer, 49(15-16), 2739-2750.