Turbulence Model Assessment for a Heated Rectangular Riser of Air-cooled RCCS in Turbulent Forced and Mixed Convection Heat Transfer

Sin-Yeob Kim [a], Chan-Soo Kim [b], Hyoung Kyu Cho [a*]
[a]Department of Nuclear Eng., Seoul National Univ., 1 Gwanak-ro, Gwanak-gu, Seoul 08826
*[b]Nuclear Hydrogen Reactor Technology Division, Korea Atomic Energy Research Institute, 111 Daedeok-daero 989
beon-gil, Yuseong-gu, Daejeon 34057*
[*]Corresponding author: chohk@snu.ac.kr

1. Introduction

Reactor Cavity Cooling System (RCCS) is a passive cooling system of Very High Temperature gas-cooled Reactor (VHTR), and it uses natural circulation of outside air to remove decay heat emitted from the reactor vessel [1]. Korea Atomic Energy Research Institute (KAERI) designed air-cooled RCCS incorporating rectangular riser channels [2], whose normal operation condition is in turbulent force convection condition. However, turbulent mixed convection can occur in emergency operation conditions due to the decrease of the chimney effect and pressure difference inducing lower flow rate of air circulation. Therefore, the exact prediction of RCCS performance is of great importance to ensure the safety of the reactor vessel of VHTR. Furthermore, experimental study and research are insufficient about the heat transfer phenomena inside a rectangular riser.

Several researches on the performance of RCCS adopted rectangular riser channels have been conducted with reduced-scale experiment facilities, at KAERI, Argonne National Laboratory (ANL), University of Wisconsin [2, 3, 4]. At Seoul National University (SNU), two experimental studies for the single RCCS riser were conducted; one is for the measurement of local heat transfer coefficient of the single riser and the latter one is for the measurement of local flow structure and turbulence quantities with flow visualization [5, 6]. From the results of these researches, heat transfer deterioration was identified in some experimental conditions whose air flow rate is relatively low, and predictions of the experimental data using CFD analysis showed different calculation results depending on the selection of turbulence models.

In this study, using measurement data from the previous flow visualization experiment [6], CFD analysis was conducted in turbulent forced and mixed convection conditions with various turbulence models. By comparing the results of CFD and visualization experiment, including local flow rate and turbulence quantities, the prediction capabilities of turbulence models were assessed. In the end, the relationships between the heat removal through a riser and the flow characteristics were investigated for the further improvement of the prediction of CFD analysis in turbulent forced and mixed convection conditions.

2. CFD Analysis for a Heated Rectangular Riser

2.1 Calculation Conditions

In the previous research at SNU, flow visualization experiment obtaining local velocity fields in turbulent forced and mixed convection conditions were conducted, whose 2m-height rectangular test section consists of transparent heat resistant glass and FTO material for resistive heating on the inner surface [6]. Because heat losses through the outer wall of the test section cannot be controlled or measured in the visualization experiment, heat transfer quantification methodology for visualization experiment was newly established [7]. According to the methodology, outer wall temperature distributions of each visualization experimental conditions were obtained by infrared thermometry, and Fig. 1 shows one of the captured temperature distributions.

Fig. 2 shows the concept of boundary conditions in CFD analysis, which is modelled on the test section of the visualization experiment. Glass was modelled by 4 mm-thickness solid structure for the consideration of thermal conduction, and FTO coating is modelled by 1 μm-thickness film to impose volumetric heat source whose heating power is same with the imposed power in the experiment. Inner part of the test section is for the airflow whose width, depth and height are 120 mm, 20 mm and 2000 mm, respectively, same with the heated test section of the experiment facility.
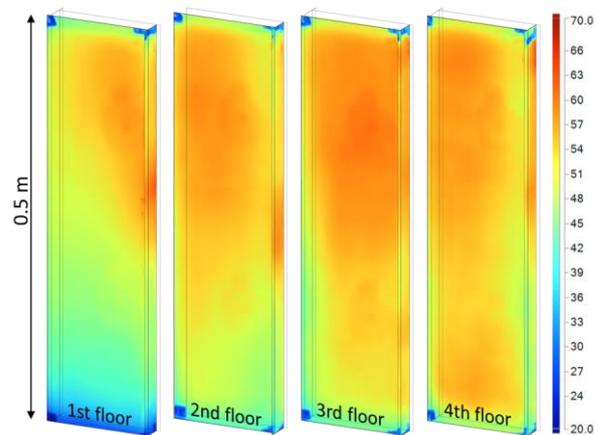
Fig. 1. One of the captured temperature distributions on the outer surface of test section.
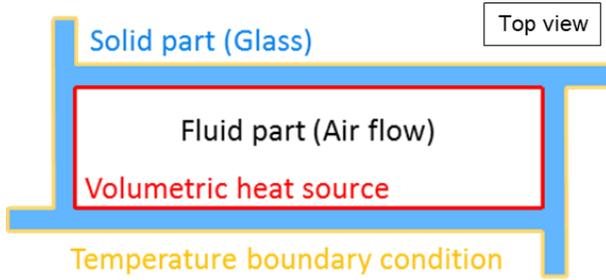
Fig. 2. The concept of boundary conditions in CFD analysis modelled on the test section of the visualization experiment.

Developed distributions of velocity and turbulence quantities were imposed as inlet boundary conditions.

In this paper, among the various experimental conditions in the experiment, 3 different convective heat transfer conditions were selected for the turbulent forced and mixed convection conditions, and CFD analysis and comparison of calculation results with the experimental data were conducted as shown in Table I.

Table I: Experimental Conditions

Case	Inlet Re	T_{out}-T_{in}	Heat removal
A	5500	32.7 K	234 W
B	5500	70.4 K	508 W
C	5000	81.5 K	536 W

2.2 Grid Validation

STAR-CCM+ (Ver. 13.02), one of the commercial CFD codes, was used for the CFD calculation and prediction capabilities of turbulence models were assessed. Before the CFD analysis for the experimental conditions, grid validation was performed to ensure the suitability of generated mesh, for the fluid part of the test section. Inlet Reynolds number was 4500, temperature difference between the inlet and outlet of the test section was about 80 °C, from 20 °C at the inlet, and V2F k-ε turbulence model was used for the analysis. According to the Richardson extrapolation [8], three grid base sizes of 2.0 mm, 1.4 mm and 0.8 mm were chosen, and Table II shows the calculation results of average velocity and temperature at the outlet of the test section and grid convergence index (GCI) under 0.2%. From this grid validation, a mesh generated by

Table II: Analysis Results depending on the Grid Base Size

	Mean velocity	Mean temperature
Grid size of 2.0 mm	2.7807 m/s	100.76 °C
Grid size of 1.4 mm	2.7813 m/s	100.38 °C
Grid size of 0.8 mm	2.7837 m/s	100.48 °C
Extrapolated value	2.7803 m/s	100.51 °C
GCI (95%)	-0.15%	0.031%
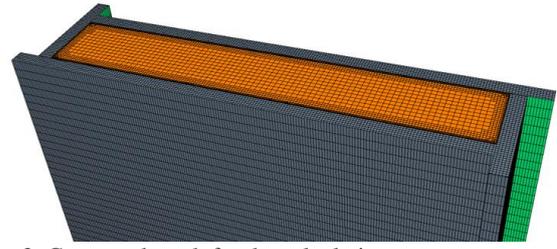
Fig. 3. Generated mesh for the calculation geometry.

the grid size of 1.4 mm was used for the CFD calculation and it is described in Fig. 3.

2.3 Turbulence Models

According to the STAR-CCM+ user guide and previous researches, four different turbulence models were selected as target turbulence models of prediction capability assessment for the heat transfer phenomena inside a rectangular riser in turbulent forced and mixed convection conditions [9, 10, 11]. Known to be suitable for the calculations in convective heat transfer conditions with intense heating, SST k-ω and V2F k-ε turbulence models were selected as the target turbulence models [10]. According to the previous researches, V2F model shows good prediction performances for the calculations of convective heat transfer with intense wall heating [10, 12]. Assessment for realizable k-ε two-layer turbulence model, which is one of the most famous turbulence models, was also conducted [13]. The last one is Reynolds stress transport (RST) model, which directly calculates the components of the specific Reynolds shear stress tensor, so naturally account for the effects of turbulence anisotropy, swirl rotation, and so on [14].

The calculation was performed in steady-state condition, the property variations for the density were defined by incompressible ideal gas law, the specific heat of air was calculated by gas kinetics option, and Sutherland's law was applied for the thermal conductivity and specific heat of air [9].

3. Results of Turbulence Model Assessment

Table III shows the temperature difference between the inlet and outlet of the test section in three different experimental conditions, and CFD analysis results for each experimental condition using four different turbulence models. Depending on the turbulence models, the results show significant differences, and the V2F k-ε turbulence model predicted the results of experiment best among the four turbulence models.

Fig. 4 shows the measurement locations of velocity fields and system coordinate of the visualization experiment, and Fig. 5 presents the nondimensionalized local temperature distribution at

Table III: Results of Inlet/Outlet Temperature Difference

Case	Tout-Tin = ΔT				
	Exp.	Realizable k-ε	SST k-ω	V2F k-ε	RST
A	32.7 K	37.9 K	35.1 K	34.4 K	37.2 K
B	70.4 K	90.3 K	82.0 K	77.9 K	86.1 K
C	81.5 K	110.0 K	99.2 K	92.0 K	102.6 K
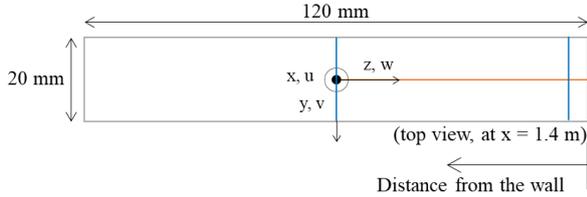
Fig. 4. Measurement locations of velocity fields and system coordinate in the experiment.
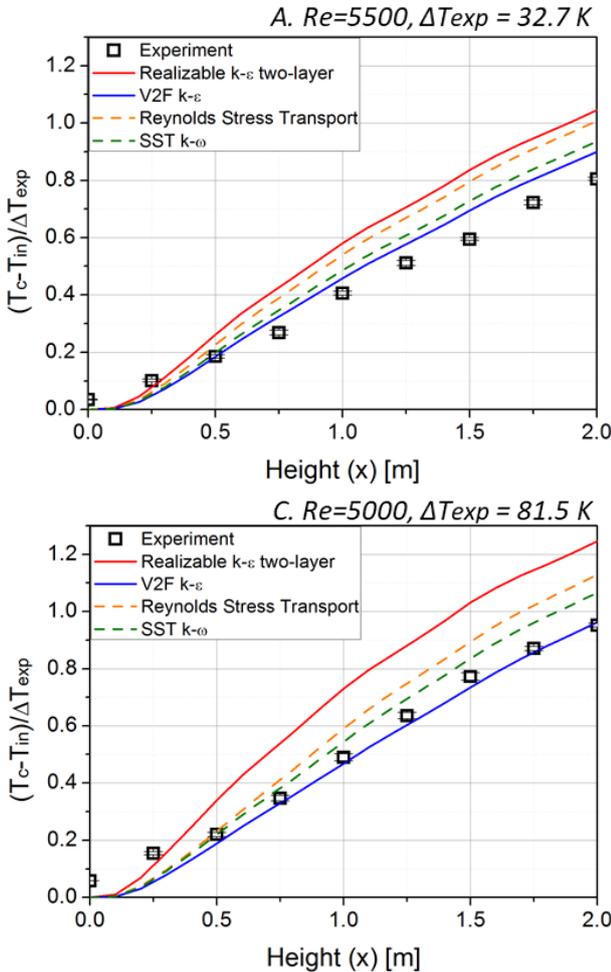
Fig. 5. Local temperature at the center of the test section along the elevation for the case A (up) and C (down).

the center of the test section along the elevation from the experiment and CFD analysis for the case A and C. In both cases, similar to the results of inlet/outlet temperature difference, the V2F k-ε turbulence model reproduced the experimental results most closely, while other turbulence models overestimated the experimental data.

In Fig. 6, vertical velocity profiles at y = 0 mm from the results of experiment and CFD analysis were compared for the case C. In the case of the results of experiment, because laminarization of air flow at near the corner precedes that of the center, and vertical velocity near the corner has relatively higher distribution. However, CFD calculation results with four turbulence models cannot predict the increase of vertical velocity near the corner.
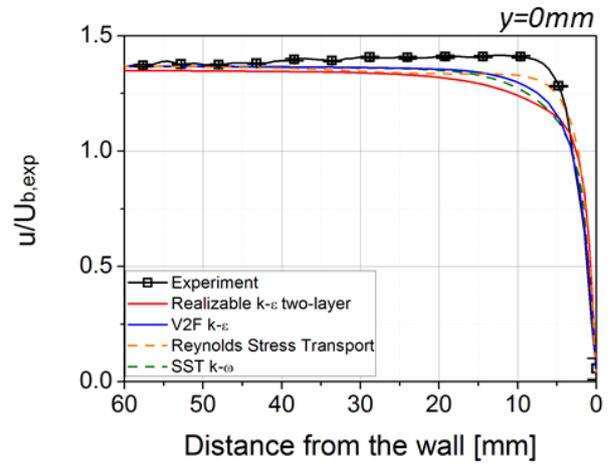
Fig. 6. Vertical velocity profiles at the mid-plane of the test section for the case C.

In Fig. 7, Reynolds shear stress profiles at three different measurement locations from the experiment and CFD analysis were presented for the case C. At y = 0 mm, Reynolds shear stress of the experiment shows a zero distribution near the corner, while turbulence models cannot reproduce the larminarization, as in the case of vertical velocity.

At z = 0 mm and 56 mm described in Fig. 7, if the results of the Reynolds shear stress are compared with the results of the temperature difference in Table III, the closer the CFD calculation predicts the Reynolds shear stress distribution to zero, the smaller the inlet/outlet temperature difference. Therefore, the prediction of heat removal through the rectangular riser using turbulence model would be related to the overall distribution of Reynolds shear stress, and V2F k-ε turbulence model can be selected as the optimum turbulence model for the prediction of convective heat transfer phenomena inside a heated rectangular riser of RCCS, because it can reproduce the Reynolds shear stress distribution in the mixed convection region most closely with the experimental results.

However, the four target turbulence models, even the V2F k-ε turbulence model, cannot reproduce the complex Reynolds shear stress distributions inside the rectangular test section and secondary flows in turbulent forced convection conditions. Therefore, to

predict the heat transfer phenomena in the rectangular duct accurately further researches are needed on the distribution of Reynolds shear stress, especially near the corner [6, 15].
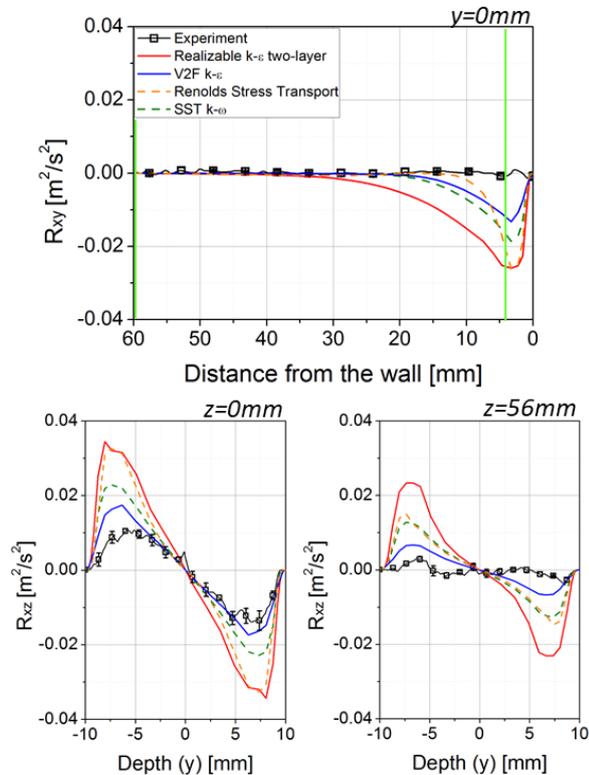
Fig. 7. Reynolds shear stress profiles at y = 0 mm, z = 0 mm and z = 56 mm for the case C.

4. Conclusions

Turbulence model assessment was conducted inside a rectangular riser of RCCS in turbulent forced and mixed convection conditions by comparing the results of CFD analysis to the data of visualization experiment. Four target turbulence models were selected, and the calculation results using these models were compared to assess the predictabilities of the turbulence models. Among them, V2F *k-ε* turbulence model shows the best prediction of heat removal through the riser and distributions of Reynolds shear stress. However, local laminarization preceded at the corner of the test section in the experiment was not reproduced by the calculation results. For the improvement of prediction of heat transfer phenomena inside a heated rectangular riser of RCCS, further researches are required to enhance the understandings of distribution of Reynolds shear stress, especially at the corner region.

ACKNOWLEDGEMENT

REFERENCES

[1] J. H. Chang, Y. W. Kim, K. Y. Lee et al., A Study of a Nuclear Hydrogen Production Demonstration Plant, Nucl. Eng. Technol., Vol. 39, p. 111, 2007.
[2] J. H. Kim, Y. Y. Bae, S.D. Hong., et al., Results of the Preliminary Test in the 1/4-Scale RCCS of the PMR200 VHTR, Trans. of the KNS autumn meeting, Oct.30-31, 2014, Pyeongchang, Korea.
[3] S. Lomperski, W. D. Pointer, C. P. Tzanos, et al., Generation IV Nuclear Energy System Initiative. Air-Cooled Option RCCS Studies and NSTF Preparation (ANL-GenIV-179), Argonne National Laboratory, Argonne, IL, 2011.
[4] D. Lisowski, M. Moses, M. H. Anderson, M. L. Corradini, Design Considerations for a Scaled RCCS with Air for the VHTR, Proceedings of NURETH 15, May 12-15, 2013, Pisa, Italy.
[5] D. H. Shin, C. S. Kim, G. G. Park, H. K. Cho, Experimental Analysis on Mixed Convection in Reactor Cavity Cooling System of HTGR for hydrogen production, Int. J. Hydrogen Energy, Vol. 42(34), pp. 22046-22053, 2017.
[6] S. Y. Kim, C. S. Kim, G. C. Park, H. K. Cho, Experimental Study on Local Flow Structure and Turbulence Quantities in a Heated Rectangular Riser of Air-cooled Reactor Cavity Cooling System, Trans. Of the KNS autumn meeting, Oct. 24-25, 2019, Goyang, Korea.
[7] S. Y. Kim, D. H. Shin, C. S. Kim, G. C. Park, H. K. Cho, Flow Visualization Experiment in a Two-side Wall Heated Rectangular Duct for Turbulence Model Assessment in Natural Convection Heat Transfer, Nuclear Engineering and Design, Vol. 341, pp. 284-296, 2019.
[8] V 20 Committee, Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer, American Society of Mechanical Engineers, New York, NY, USA, 2009.
[9] PLM Siemens, STAR-CCM+ User Guide Version 13.04, Siemens PLM software Inc., 2018.
[10] R. E. Spall et al., An Assessment of k-w and v2-f turbulence models for strongly heated internal gas flows, Numer. Heat Transfer, Part A 46, pp.831-849, 2004.
[11] D. P. Mikielewicz, A. M. Shehata, J. D. Jackson, Temperature, Velocity and Mean Turbulence Structure in Strongly Heated Internal Gas Flows: Comparison of Numerical Predictions with Data, Int. J. Heat Mass Transfer, Vol. 45, pp. 4333-4352, 2002.
[12] L. EI Moutaouakil et al., Performance of various RANS eddy-viscosity models for turbulent natural convection in tall vertical cavities, Heat and Mass Transfer, Vol. 50, pp. 1103-1113, 2014.
[13] T. H. Shih et al., A new k-e eddy viscosity model for high Reynolds number turbulent flows, Comput. Fluids Vol. 24 (3), pp. 227-238, 1995.
[14] S. Sarkar et al., Application of a Reynolds-stress turbulence model to the compressible shear layer, ICASE Report 90-18, NASA CR 182002, 1990.
[15] L. C. Hoagland, Fully Developed Turbulent Flow in Straight Rectangular Ducts – Secondary Flow, Its Cause and Effect on the Primary Flow, Ph.D. Thesis, Massachusetts Institute of Technology, Cambridge, MA, USA, 1962.