Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 26-27, 2017

A Study on the Local Flow Structure and Turbulence Quantities inside a Heated Rectangular Riser Duct for Turbulence Model Assessment

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

1. Introduction

Very High Temperature gas-cooled Reactor (VHTR) that is one of the concepts of GEN-IV reactor adopts Reactor Cavity Cooling System (RCCS) as a passive safety system removing the decay heat from the reactor vessel [1]. RCCS is comprised of vertical rectangular riser channels, which surround the reactor vessel to remove the afterheat emitted from it. To ensure the integrity of the reactor vessel, an accurate prediction of heat removal rate through the RCCS riser is essential, and understanding of the heat transfer phenomena in the RCCS riser is required.

In the RCCS riser channel, mixed convection heat transfer may occur in some circumstances whose heat transfer mechanism becomes complicated with the effect of thermo-physical properties variation [2]. In mixed convection conditions, the heat transfer and local flow structure have significantly dissimilar characteristics from both forced convection and natural convection due to the buoyancy force induced by temperature and density differences [3]. But this phenomenon, especially in a vertical rectangular duct, has not been sufficiently investigated due to the lack of experimental data and insufficient understanding of the local flow structure.

Several researches on the performance of RCCS riser have been conducted with reduced-scale experiment facilities [4, 5, 6], and the results implied that the overall heat transfer rate through the experiment facilities and prototype RCCS riser would be different depending on the thermal boundary conditions and convective heat transfer regime [6]. To enhance the understanding of the heat transfer mechanism in the rectangular riser duct, an investigation on the local flow structure is required for the different thermal boundary conditions.

In this study, local flow structure was measured in a strongly heated rectangular riser duct with Particle Image Velocimetry (PIV) method. Outer wall temperature distribution of the test section was obtained by infrared (IR) thermometry for the boundary condition of CFD calculation. By comparing the experimental data and CFD analysis results, prediction capabilities of turbulence models were assessed in a heated rectangular riser duct for natural convection. Finally, planned activities for further investigation were introduced.

2. Experiment and CFD analysis

2.1 Experiment Facility

Fig. 1 shows the design of flow visualization test facility and the picture of its test section. The height of test section is 1 m, and the width and depth of inner test section are 240 mm and 40 mm, respectively, which are same with those of the prototype RCCS riser [7]. The test section consists of transparent heat-resistant glass to obtain internal flow structure with PIV method. On the inner surface of the two narrow-side glass faces, transparent conductive material, FTO (Fluorine doped Tin Oxide), is coated for the resistive heating of test section, and the power for resistive heating is controlled by a power supply. DEHS (Di-Ethyl-Hexyl-Sebacat), which is non-toxic and volatile oil, is injected to the test section as seed particles of 1 μm droplet size for PIV method. DEHS aerosol is injected before the lower plenum where the air flow and DEHS aerosol are mixed up and flow straightening structures are installed to reduce the turbulence intensity of the flowing air. To blow air flow and measure the air flow rate, an air blower and a mass flowmeter were installed.

To obtain thermal boundary conditions of the outer surface of test section, an IR camera was equipped. Emissivity calibration on the outer surface was conducted by comparing the maximum wall temperature measured by the IR camera and thermocouple. To obtain local flow structure and turbulence quantities, a continuous planar laser system and a high speed camera were equipped for PIV method. Linear traverses for the laser system and high speed camera were installed to move the measurement position back and forth for the purpose of measuring the local flow structure at different positions.

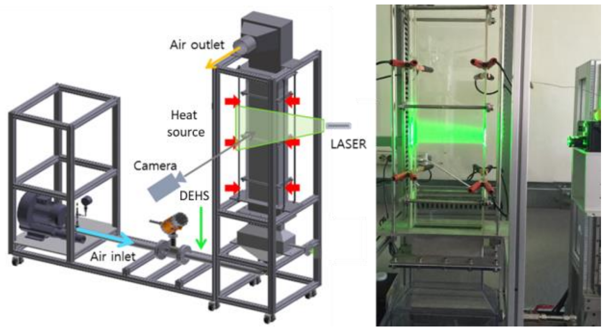

Fig. 1. The design of the flow visualization test facility (left) and the picture of its test section (right) [8]

2.2 Experiment Conditions

In this study, local flow structure in natural convection heat transfer condition was measured, where $Gr/Re^2 \approx 14$ at 0.8 m from the entrance of the test section. Inlet flow rate was about 0.0039 kg/s, which is correspondent with 0.34 m/s at the inlet of test section, and its inlet Reynolds number is about 1500. Electrical power for resistive heating through each narrow-side FTO coating was about 140 W. Maximum outer wall temperature was about 150 °C and the emissivity of the outer surface of glass was calibrated as 0.86. Outer wall temperature distributions of the test section were captured with the IR thermometry and obtained temperature distributions were used for the thermal boundary condition in CFD calculations.

The High speed camera captured internal flow at 0.2 m and 0.8 m from the entrance of the test section. For the depth direction, velocity fields at the -18 mm, -15 mm, -10 mm, 0 mm, 10 mm, 15 mm and 18 mm from the mid-plane were obtained. Images were captured for 60 seconds and time-averaged local flow structure were obtained by averaging 1500 velocity vector fields analyzed by PIV method.

2.3 CFD Analysis

The test facility of the present study was simulated with a commercial CFD code, STAR-CCM+, to evaluate the prediction capabilities of two turbulence models; V2F turbulence model [9] and realizable k-ε turbulence model [10]. To predict the flow behavior and heat transfer at low Reynolds number with wall heating, many turbulence models and their extensions have been suggested and investigated [11, 12]. Among the Reynolds-Averaged Navier-Stokes (RANS) turbulence models, the V2F turbulence model was recommended as the most adequate turbulence model for predicting the flow with intense wall heating. Spall et al. [13] showed that the V2F turbulence model was able to capture the experimental data obtained from the internal gas flows in a strongly heated riser tube. The other model, realizable k-ε turbulence model, is one of the well-established turbulence models capable of resolving fluid behavior through the boundary layer.

Fig. 2 shows the design of the test section and the schematic diagram of its calculation geometry from above in CFD analysis. Width, depth and height of the fluid geometry are 240mm, 40mm and 900mm, respectively. Solid geometry was used to model the glass and FTO coating of the narrow-sides of test section. FTO coating layer is 0.2mm thickness for volumetric heat source and glass is 3.8mm thickness. In Fig. 3, outer wall temperature distributions measured by the IR thermometry were imposed as the temperature boundary conditions of CFD calculations.

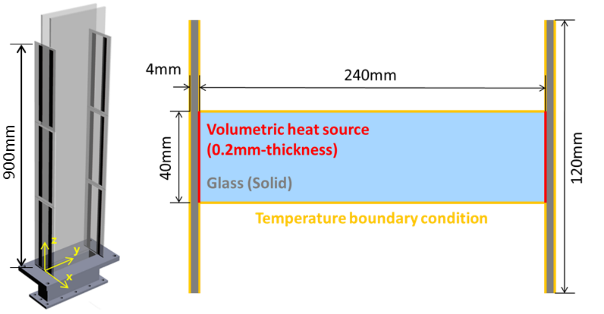

Fig. 2. The design of the test section (left) and the top view of calculation geometry for CFD analysis (right)

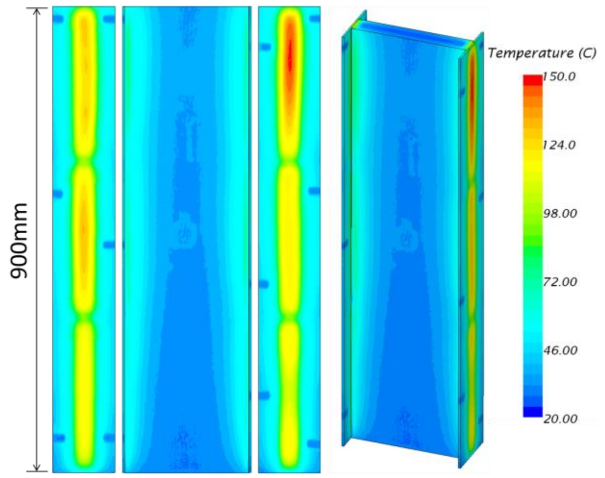

Fig. 3. Measured outer wall temperature distributions (left) and temperature boundary conditions imposed on the calculation geometry of the CFD analysis (right)

Hexahedral meshes of about 2 million cells were generated with the grid size of 1 mm. Generated meshes of the fluid geometry consist of 300 points in the axial direction and the y+ values are less than 0.2 at the center of the control volume adjacent to the solid interface. In the CFD calculations, the property variations for density and viscosity were defined by incompressible ideal gas law and Sutherland's law, respectively [13]. For the thermal conductivity and specific heat of air, polynomial fits from the Spall's paper [13] were applied.

3. Results and Discussions

3.1 Results of the Experiment

Fig. 4 and 5 show the vertical velocity profiles of local flow structure near the inlet (z = 0.2 m) and outlet (z = 0.8 m) of test section. In Fig. 4, velocity profiles near the inlet show velocity peaks at the near-wall region and a little uneven flow distributions at the center. In Fig. 5, velocity profiles at the outlet of the test section show relatively much steeper velocity peaks and have faster velocity distributions due to the thermal expansion of the air near the wall region.

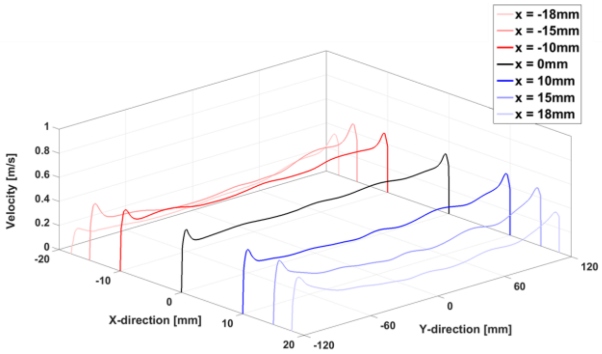

Fig. 4. Vertical velocity components of the obtained local flow structure at z = 0.2 m from the entrance of the test section at the 7 different positions in the depth direction

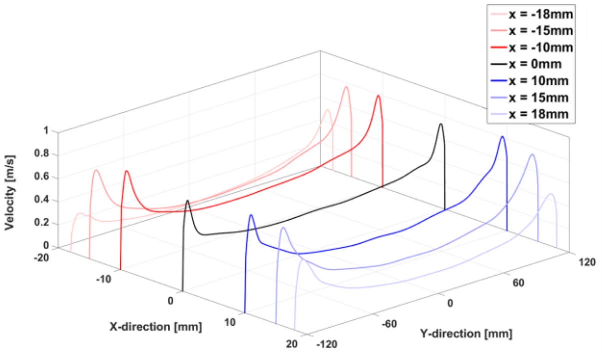

Fig. 5. Vertical velocity components of the obtained local flow structure at z = 0.8 m from the entrance of the test section at the 7 different positions in the depth direction

Fig. 6, 7 and 8 respectively show vertical velocity components, horizontal velocity components and vertical velocity fluctuations of the local flow structure at z = 0.8 m. In Fig. 6, for the vertical velocity profiles, velocity peaks at x = 10 mm and 15 mm are higher and thicker than the velocity peaks at the mid-plane. In Fig. 7, the symmetrical distribution of horizontal velocity profile at the mid-plane (x = 0 mm) means the air flow from the center of the test section toward the both of left and right side wall region. However, at x = 15 mm and 18 mm, reverse flows are observed at the near-wall region. These behaviors of the vertical and horizontal flow imply that some air flows toward the corner of test section at the near-wall region, and then returns to the center of rectangular riser duct. The reverse directional flow of horizontal velocity profiles at x = 15 mm and 18 mm is deemed to be attributed to these secondary flows swirling at the corner of the test section.

Fig. 8 shows the fluctuation of the vertical velocity components and fluctuation profiles have much higher fluctuation values at the near-wall region than those of the center due to the buoyancy effect. In Fig. 8, unlike the fluctuation peaks of the mid-plane, the fluctuation peaks around the corner of test section show much thicker peaks near the wall, and the positions of the peaks around the corner are farther from the wall than peak position of the mid-plane. These also imply the existence of the secondary flow at the corner.

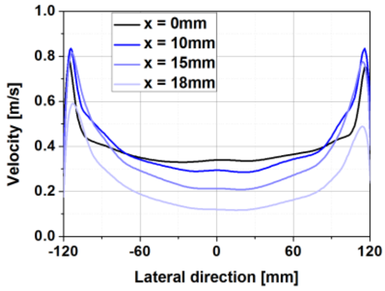

Fig. 6. Vertical component velocity profiles at z = 0.8 m and at x = 0 mm, 10 mm, 15 mm and 18mm from the mid-plane

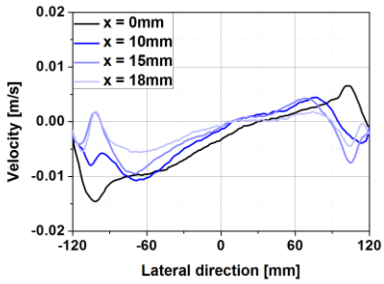

Fig. 7. Horizontal component velocity profiles at z = 0.8 m and at x = 0 mm, 10 mm, 15 mm and 18mm

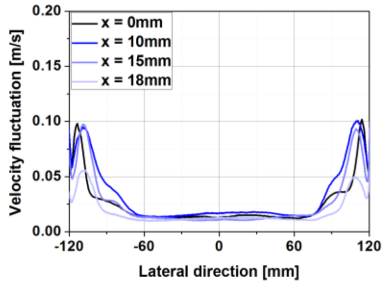

Fig. 8. Fluctuation profiles of the vertical velocity components at z = 0.8 m and at x = 0 mm, 10 mm, 15 mm and 18mm

3.2 Results of the CFD Analysis

Since the velocity distribution at the viscous layer is the primary parameter that determines the heat transfer of natural and mixed convection phenomena [2], it should be verified which turbulence model can predict realistic flow structure with buoyancy effect. Local flow structure obtained from the CFD analysis was compared to that of the experiment results. Fig. 9 shows the vertical velocity profiles from the CFD results with experimental data at the mid-plane. Unlike the local flow structure from the experiment, V2F turbulence model predicted much higher velocity peaks and realizable k-ε turbulence model predicted much thicker velocity peaks at the near-wall region. Fig. 10 shows the horizontal velocity profiles from the CFD results with experimental data at x = 15 mm, where the effect of secondary flow was observed. In Fig. 10, both of the turbulence models did not present the flow heading to the center of test section which would affect the heat transfer rate inside of the test section.

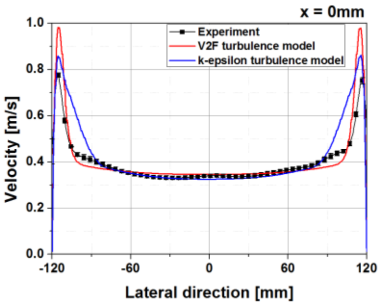

Fig. 9. Vertical velocity profiles from the experiment and CFD analysis at the mid-plane

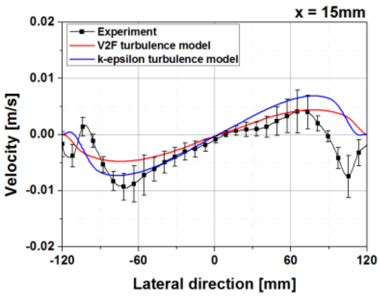

Fig. 10. Horizontal velocity profiles from the experiment and CFD analysis at x = 15mm from the mid-plane

3. Conclusions and Planned Activities

Local flow structure and turbulence quantities were obtained in a heated rectangular riser duct in natural convection condition using PIV method, and the data was compared with the CFD analysis results to assess the prediction capabilities of two turbulence models; V2F turbulence model and realizable k-ε turbulence model. From the experimental data, the vertical velocity peaks induced by the buoyancy effect were observed near the wall region, and the existence of secondary flow at the corner of test section was identified. However, the two turbulence models seem to have limitations in reproducing the secondary flow around the corner of test section and it is regarded as the reason that they predict higher velocity peaks at the near-wall region unlike the experimental data.

For further investigation on heat transfer phenomena in the RCCS riser duct, newly designed experiment facility is shown in Fig. 11. Its width and depth of inner test section are 120 mm and 20 mm, respectively, which are the half of the prototype RCCS riser's. With this reduced scale of the cross-sectional area, the height of the heated test section is approximately 60 hydraulic diameters of the test section for fully developed flow. Four inner surface of the heated test section will be entirely coated with FTO coating to investigate more realistic thermal boundary conditions of the RCCS riser duct. With this experiment facility, local flow structure and overall heat transfer rate will be investigated and the same turbulence model assessment will be conducted for various thermal boundary conditions.

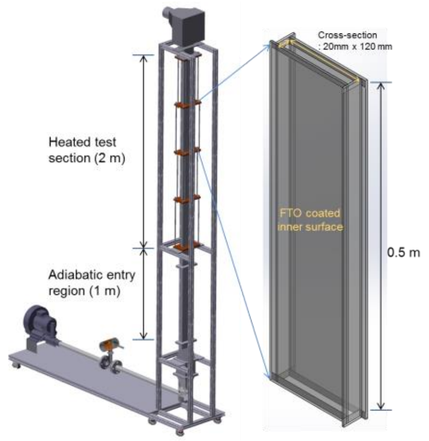

Fig. 11. Schematics of the newly designed experiment facility (left) and its design of the heated test section (right)

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] T. Aicher and H. Martin, New Correlations for Mixed Turbulent Natural and Forced Convection Heat Transfer in Vertical Tubes, Int. J. Heat Mass Transfer. Vol. 40, No. 15, pp. 3617-3626, 1997.

[3] J. I. Lee, P. Hejzlar, P. Saha, et al., Studies of the Deteriorated Turbulent Heat Transfer Regime for the Gas-Cooled Fast Reactor Decay Heat Removal System, Nucl. Eng. Des., Vol. 237, p. 1033, 2007.

[4] 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.

[5] 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.

[6] D. H. Shin at al., Preliminary analysis on the mixed convection phenomena in the scaled-down VHTR RCCS riser experiment, Trans. of the KNS spring meeting, May. 12-13, 2016, Jeju, Korea.

[7] Y. Y. Bae, B. H. Cho, Scaling Analysis of PMR200 RCCS Natural Cooling Performance Test (KAERI/GP-415/2015), Korea Atomic Energy Research Institute, Daejeon, 2015.

[8] S.Y. Kim et al., Validation Plan of Turbulence Models for Internal Gas Flow Analysis in a Heated Rectangular Riser Duct, Transactions of the Korean Nuclear Society Autumn Meeting, Vol. 2, 2016.

[9] P. A. Durbin, Separated Flow Computations with the k-ε-v^2 Model, AIAA JOURNAL, Vol.33, No.4, pp.659-664, 1995

[10] T. H. Shih et al., A New k-ε Eddy Viscosity Model for High Reynolds Number Turbulent Flows, Computers Fluids, Vol. 24, No. 3, pp. 227-238, 1995.

[11] V. C. Patel, W. R. Rodi, G. Scheuerer, Turbulence Models for Near-Wall and Low Reynolds Number Flows: A Review, AIAA JOURNAL, Vol.23, No.9, pp.1308-1319, 1984.

[12] D. P. Mikielewicz at al., Temperature, Velocity and Mean Turbulence Structure in Strongly Heated Internal Gas Flows Comparison of Numerical Predictions with Data, Int. J. Heat Mass Transfer 45, pp. 4333-4352, 2002.

[13] R. E. Spall et al., An Assessment of *k-w* and *v2-f* Turbulence Models for Strongly Heated Internal Gas Flows, Numerical Heat Transfer, Part A, 46, pp. 831-849, 2004.