Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 18-19, 2017

A Study on the Local Flow Structure in a Strongly Heated Rectangular Riser Duct with a Flow Visualization Experiment and CFD Analysis

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

Reactor Cavity Cooling System (RCCS) is a passive safety system of Very High Temperature gas-cooled Reactor (VHTR) being developed at Korea Atomic Energy Research Institute (KAERI) [1]. RCCS incorporates rectangular riser channels to remove the afterheat emitted from the reactor vessel. Because an accurate prediction of heat removal rate through the RCCS riser is important to ensure the safety of the reactor, understanding of the heat transfer phenomena in the RCCS riser is needed.

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 the mixed convection, due to the buoyancy force induced by temperature and density differences, local flow structure and heat transfer mode near the heated wall have significantly dissimilar characteristics from both forced convection and natural convection [3]. However, this phenomenon, especially in a rectangular duct has not been investigated sufficiently due to the lack of experimental data and insufficient understanding of the local flow structure.

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 of experiment facility was measured with IR thermometry to be applied to the boundary condition of CFD calculations. With the measured experimental conditions, CFD analysis was conducted using two turbulence models, V2F and realizable k-ε model, in a commercial CFD code STAR-CCM+. Finally, the local flow structures from the experiment and CFD analysis were compared to validate the prediction capability of the turbulence models.

2. Experiment and CFD analysis

2.1 Experiment Facility

Fig. 1 shows a design of flow visualization test facility and its test section [4]. The height of test section is 1m and the width and depth of inner test section are 240mm and 40mm, respectively, which are same with the width and depth of the prototype RCCS riser. The test section consists of four sheets of heat-resistive glasses to obtain internal flow structure with PIV method and visual access. On the inner surface of the two narrow-side glasses, transparent conductive material, Fluorine doped Tin Oxide (FTO), is coated for the resistive heating of test section, and the power for resistive heating is controlled by power supply. Di-Ethyl-Hexyl-Sebacat (DEHS) aerosol, which is non-toxic and volatile 1 micrometer size aerosol, is injected as seed particles for PIV method. DEHS aerosol is injected into the lower plenum where the air flow and DEHS aerosol are mixed up. To blow air flow and measure the air flow rate, air blower and mass flowmeter are installed before the lower plenum.

To obtain thermal boundary conditions of the outer surface of test section, infrared (IR) camera was equipped. Emissivity calibration of the outer surface was conducted by comparing the maximum wall temperature measured by IR camera and thermocouple. To obtain local flow structure and turbulence quantities, continuous planar laser and high speed camera were equipped for PIV method. Linear traverses for the laser and high speed camera are installed to move the measurement position back and forth maintaining the focal length of camera lens.

2.2 Experimental Conditions

In this study, local flow structure in natural convection heat transfer condition was measured. Inlet flow rate was about $0.00325 m^3/s$ and Reynolds number at the test section inlet is about 1500. Power for resistive heating through each narrow-side test section was about 145W. Maximum outer wall temperature was

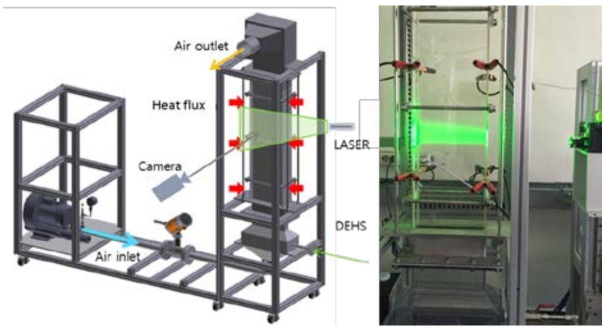

Fig. 1. The design of the flow visualization test facility (left) and its test section (right) constructed in Seoul National University.

about 440K and the emissivity of the outer surface of glass was calibrated as 0.85. Outer wall temperature distributions for the narrow-side and wide-side of test section are captured with IR thermometry and obtained temperature distributions are used for the thermal boundary condition in CFD calculations.

Measurement position of the high speed camera was about 0.8m from the entrance of the test section and velocity fields were captured at the 0mm, 10mm and 15mm from the mid-plane in the depth direction. Velocity fields were captured for 60 seconds and time-averaged local flow structure were obtained by averaging 1500 velocity vector fields using PIV method.

2.3 CFD analysis

Many turbulence models and their extensions have been suggested and investigated to predict the gas flow at low Reynolds number with intense heating [5, 6]. 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 of mixed convection. Spall et al. [7] showed that the V2F turbulence model was able to capture the experimental data which were 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. With these two turbulence models, CFD analysis for air flow in the flow visualization test facility was conducted to confirm the prediction capability of the turbulence models in natural convection heat transfer.

Fig. 2 shows a schematic diagram of calculation geometry and its thermal boundary condition for the test section of experiment facility. Fluid geometry is modeled as half of the real test section using symmetry plane and its width, depth and height are 120mm, 40mm and 900mm, respectively. Solid geometry was used to model the glass and FTO coating of narrow-side test section. FTO coating layer is 0.2mm thickness for volumetric heat source and glass is 3.8mm thickness. Outer wall temperature distributions measured from the experimental facility were imposed as the temperature boundary conditions of the calculation.

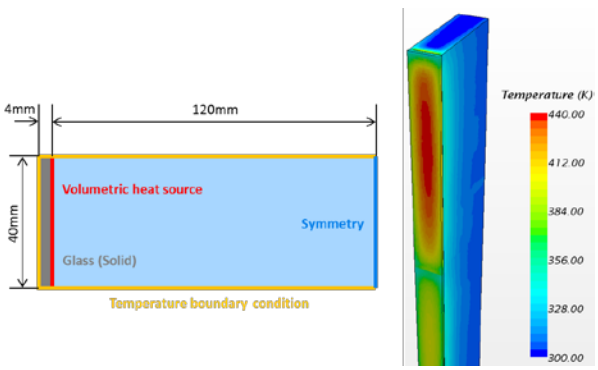

Fig. 2. Schematic diagram of calculation geometry (Left) and its temperature boundary condition (Right) imposed in the CFD calculations.

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 simulation, the property variations for density and viscosity were defined by incompressible ideal gas law and Sutherland's law, respectively [7]. For the thermal conductivity and specific heat, polynomial fits from the Spall's paper [7] were applied.

3. Results

Fig. 3 and 4 show the local flow structure near the outlet of test section at the mid-plane of test section. Vertical velocity profile shows sharp peak of velocity near the wall region due to the buoyancy effect induced by temperature and density differences. Near the wall region, in Fig. 3 indicate steep increase of velocity fluctuation near the wall region. For the horizontal velocity, the flow direction is positive for the right-hand side direction and the magnitude of the symmetrical distribution of horizontal velocity in Fig. 4 implies the air flow from the center of the test section toward the both of left and right side wall region. These phenomena enhance velocity fluctuations near the wall region, which would affect the heat transfer rate inside the RCCS riser duct. Fig. 5 shows the local vertical velocity profiles at x = 0mm, 10mm and 15mm from the mid-plane of test section.

Since the velocity distribution at the viscous layer is the primary parameter that determines the heat transfer

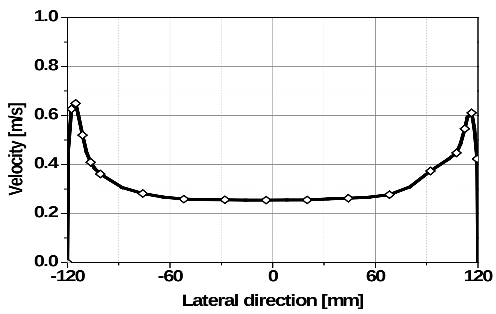

Fig. 3. Time-averaged local vertical velocity near the outlet of test section at the mid-plane of test section.

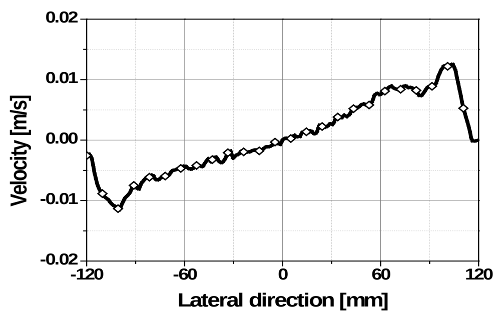

Fig. 4. Time-averaged local horizontal velocity near the outlet of test section at the mid-plane of test section.

in the natural and mixed convection phenomena [2], it should be verified which turbulence model can predict realistic heat transfer phenomena with buoyancy effect. Fig. 6 and 7 show the local vertical velocity profiles from CFD analysis results with experimental data at x = 0mm and x = 15mm. In Fig. 6, at the mid-plane of test section, velocity profile from CFD analysis shows higher velocity distribution than the velocity profile obtained from experiment, and peak velocity near the wall region was over-estimated by both of the turbulence models. In Fig. 7, at x = 15mm from the mid-plane, CFD analysis results predict the experimental data well, not only velocity distribution at the center of fluid but also peak velocity near the wall region.

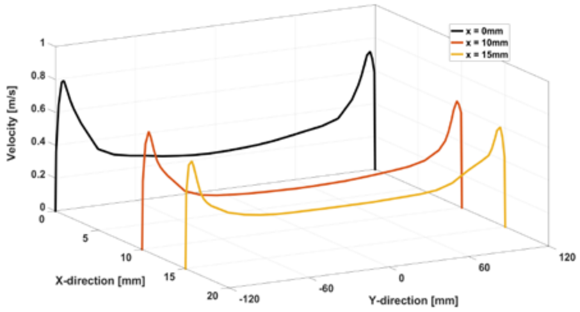

Fig. 5. Time-averaged local vertical velocity profiles at x = 0mm, 10mm and 15mm from the mid-plane of test section.

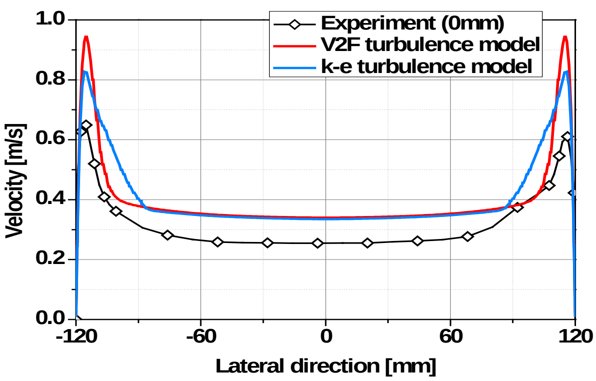

Fig. 6. Local vertical velocity profiles from the CFD analysis results and experiment at the mid-plane.

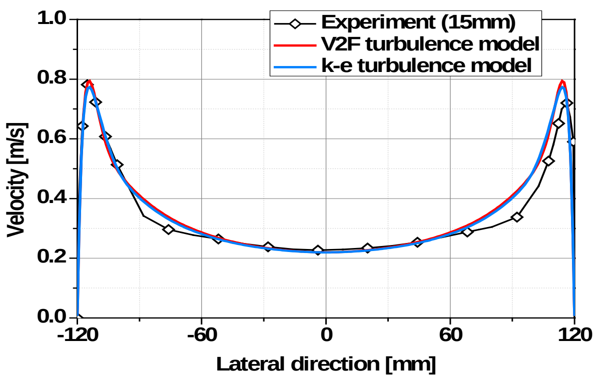

Fig. 7. Local vertical velocity profiles from the CFD analysis results and experiment at x = 15mm.

4. Conclusion

Local flow structures in a strongly heated rectangular riser duct were investigated with the flow visualization test facility and CFD analysis in natural convection condition. To predict accurate heat removal rate in RCCS riser duct, the local flow structure and turbulence quantities should be investigated in the rectangular riser duct. Velocity peak near the wall region was observed from the experiment caused by the buoyancy effect. CFD analysis was conducted to validate the prediction capability of turbulence models with V2F and realizable k- ε turbulence model. The peak velocity near the wall region was over-estimated at the mid-plane by both of the turbulence models. Further investigations will be conducted for the local flow structure and turbulence quantities in mixed convection condition as well as natural convection condition.

ACKNOWLEDGEMENT

This research was supported by the National Nuclear R&D Program through the National Research Foundation of Korea (NRF) funded by MSIP; Ministry of Science ICT & Future Planning. (No. NRF-2016M2A8A2953030)

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] 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.
[5] 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.
[6] 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.
[7] R. E. Spall et al., An Assessment of *k-w* and v_2-f Turbulence Models for Strongly Heated Internal Gas Flows, Numerical Heat Transfer, Part A, 46, pp.831-849, 2004.