Validation Plan of Turbulence Models for Internal Gas Flow Analysis in a Heated Rectangular Riser Duct

Sin-Yeob Kim [a], Dong-Ho Shin [a], Chan-Soo Kim [a], 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), one of the concepts of GEN-IV reactor, is characterized by high operating temperature and stepped-up inherent safety given by passive safety systems [1]. VHTR being developed at Korea Atomic Energy Research Institute adopts an air-cooled Reactor Cavity Cooling System (RCCS) incorporating rectangular riser channels to remove the afterheat emitted from the reactor vessel. Because the performance of RCCS is determined by heat removal rate through the RCCS riser, it is important to understand the heat transfer phenomena in the RCCS riser to ensure the safety of the reactor.

In the RCCS riser channel, mixed convection heat transfer may occur in some circumstances and it was reported that its heat transfer mechanism becomes complicated with the effect of thermo-physical properties variation [2]. In the mixed convection, due to the buoyance 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 free 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. This became the motivation of the present study to investigate the heat transfer phenomena and to find a proper turbulence model for a mixed convection in a rectangular riser channel.

In this study, the CFD analysis for a strongly heated internal gas flow in a tube at low Reynolds numbers was conducted to confirm the prediction capability of two turbulence models, V2F and k-w models, in a commercial CFD code STAR-CCM+. Then, the calculation was extended to the rectangular riser channel geometry. It deduced that a validation experiment with local velocity measurement is required to find an adequate turbulence model for the mixed convection heat transfer analysis. For this reason, a visualization test facility for the gas flow in the RCCS riser was constructed and a preliminary test was conducted.

This paper presents the calculation results for a heated tube and the riser duct geometries and then, the preliminary test result performed in the visualization test facility is followed. Finally, the planned activities with the visualization test facility were introduced.

2. Benchmark Calculations

In previous studies, many turbulence models and their extensions have been suggested and investigated to predict the flow of gas at low Reynolds numbers with intense heating [4, 5]. 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 [6, 7]. V2F turbulence model is the three-equation model which uses a velocity scale v^2, instead of using turbulent kinetic energy k, or introducing wall functions, for the evaluation of the eddy viscosity close to the wall [6]. Spall et al. [7] showed that the V2F turbulence model was able to capture the experimental data of Shehata and McEligot [8] which were obtained under strongly heated internal gas flows. Figure 1 shows the schematic diagram of the test section of Shehata's experiment [8].

In order to confirm the Spall's statements, we performed benchmark calculations against the experimental data of Shehata and McEligot. They conducted experiments for air with strong heating in the vertical tube whose diameter, total length and heating length are 0.0274 m, 2.2468 m and 0.8768 m, respectively. The test section has the unheated section of 50D length to obtain a fully developed flow before

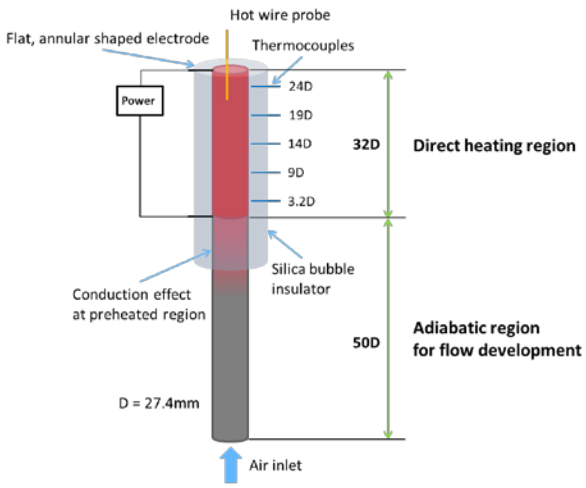

Fig. 1. Schematic diagram of the test section of Shehata's experiment

the heated tube. The experimental case of the selected benchmark data is an air flow with inlet Reynolds number of about 6,000 and non-dimensional heating rate, q^+, of about 0.0018.

The generated meshes for the heated tube consist of 200 points in the axial direction and the y^+ values are less than 0.5 at the center of the control volume adjacent to the wall. 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.

In this paper, benchmark calculations were performed by matching the total enthalpy of air at z/D = 24.54 from the entry of heated tube. Because of thermal conduction to the tube in the upstream direction, there was some preheating to the flowing air before the direct heating. In order to consider the thermal conduction effect with heat balance, two bounding boundary conditions were analyzed in this benchmark calculations; one is the heat flux distribution from the Shehata's data with a preheating heat flux and the latter one is a constant heat flux through the heated wall. The values of preheating heat flux and constant heat flux were derived from the heat balance equation to match the total enthalpy of air at z/D = 24.54.

Benchmark calculations were performed with two different turbulence models; V2F turbulence model and standard k-w turbulence model which were confirmed in the paper of Spall et al [7]. Figure 2 shows wall temperature distributions along the axial direction from CFD analysis and experimental data. Calculation results with V2F model bounds the wall temperature profile, while both calculation results with k-w model underestimated the wall temperature at 24.54D. It is a consistent result reported in the paper of Spall et al.

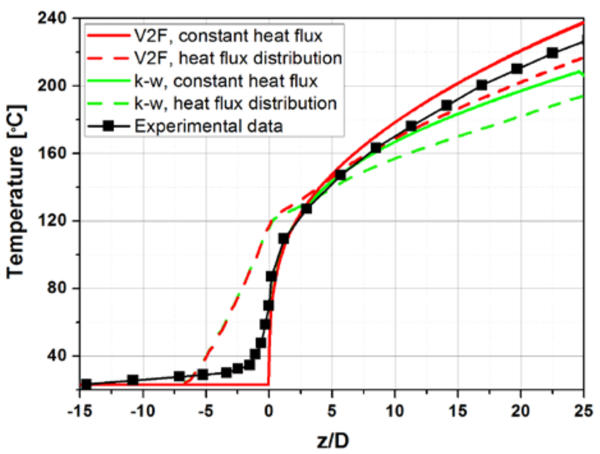

Fig. 2. Wall temperature distribution along the axial direction from the entry of the heated tube for the two boundary conditions with two turbulence model

3. CFD Analysis for Heated Rectangular Riser Duct

In previous section, the mixed convection heat transfer phenomenon was well predicted by the V2F turbulence model against the Shehata's experimental data. However, additional validations for the turbulence models are needed, because the RCCS riser is characterized by larger cross-section area and narrow rectangular geometry different from the cylindrical one.

To compare the prediction result of V2F and k-w turbulence models for a heated rectangular riser duct, CFD calculations were conducted by modeling the geometry of the prototype RCCS riser duct [3]. The y^+ values are less than 2 at the center of the mesh cells adjacent to the wall and the property variations are identical to those of benchmark calculations above.

Figure 3 shows the local vertical velocity profiles at the mid-plane (x = 0 mm), x = 7 mm and x = 14 mm in the heated rectangular riser duct. In Figure 3, calculation results from the V2F and k-w turbulence model show the steep increase of velocity profiles close to the wall region but have different inclination along the x-direction and the y-direction both. It was confirmed that the values of turbulence kinetic energy in the prototype RCCS channel were calculated quite differently by the two turbulence models. It resulted from the combination of the different evaluations of the turbulence kinetic energy near the wall by the V2F and k-w models and the high aspect ratio of RCCS riser duct.

Since the velocity difference between the center and the viscous layer is the primary parameter that determines the heat transfer in the mixed convection phenomena [2], it should be verified which turbulence model predicted the realistic mixed convection phenomena in the heated rectangular riser duct.

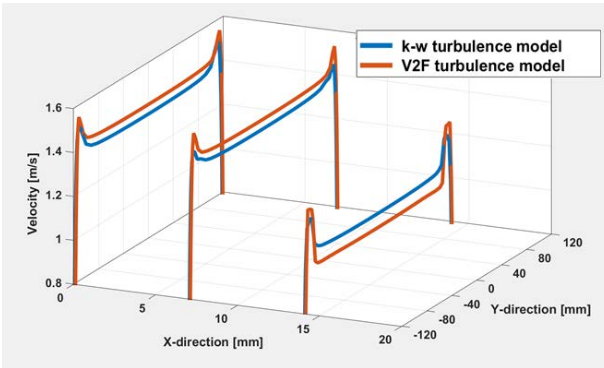

Fig. 3. Local vertical velocity profiles at the mid-plane (x = 0 mm), x = 7 mm and x = 14 mm of certain height in the heated rectangular riser duct

4. Model validation plan

For validation of the turbulence models in a rectangular channel, the following experiments are planned.

4.1 RCCS riser experimental facility

A heat transfer analysis test facility for the RCCS riser, a 1/4 reduced height test facility, named Riser Heat transfer Experimental Facility (RHEF), was constructed to measure the overall heat transfer coefficient under various flow and heat flux conditions [9]. The experiment is currently in operation and the overall heat transfer coefficient of the riser channel with various flow and heat flux conditions have been obtained. The data from this experiment can be used to assess the CFD code and turbulence models by comparing the test section outlet temperature and overall heat transfer coefficient. However, this facility is devoted for the heat transfer coefficient measurement rather than local velocity field and thus, the other two test results will be used for more comprehensive assessment of the code.

4.2 Visualization experiment with a preserved riser duct cross-section geometry

A visualization test facility for the mixed convection heat transfer in a riser was constructed with cross-section geometries identical with the RCCS riser duct. The height of the test section was reduced to 1/16 and the flow characteristics in the developing region can be investigated with this test facility. In the next section of the present paper, the test facility specification and a preliminary test result are briefly introduced.

4.3 Visualization experiment with an area reduced riser duct cross-section geometry

The above mentioned test facility has reduced height and a fully developed velocity profile cannot be obtained from it. Therefore, a test facility of which aspect ratio in the vertical direction is identical with the RHEF riser will be constructed. Considering the fabrication cost required for the test section made of transparent heaters, the scaling ratio was selected to 1/4. With a scaling method preserving important dimensionless number for the mixed convection heat transfer, the visualization of the fully developed flow conditions is expected to be covered with this test facility. A scale effect can be investigated as well.

5. Experimental Setup and Preliminary Test

Previous CFD analysis results showed opposing velocity profiles for the identical mixed convection gas flow. Therefore, in order to verify the suitability of the turbulence models in the RCCS riser channel, local flow structure and turbulence quantities especially near the wall region should be investigated. In this study, the visualization test facility was constructed with preserved riser duct cross-section geometry and this section introduces a preliminary test conducted to obtain the information of local flow structure.

5.1 Experimental apparatus and instrumentation

Figure 4 shows the design of the flow visualization test facility. It has a 1 m-height test section and the width and depth of inner test section are 240mm and 40mm, respectively, which are same with those of the prototype RCCS riser [3]. Four walls of the test section consist of four sheets of transparent heat-resistant glasses to apply Particle Image Velocimetry (PIV) method and visual access. For the resistive heating on the glass wall, transparent conducting material, FTO (Fluorine doped Tin Oxide), was coated on the inner surface of the narrow-side glasses. Figure 5 shows the test section of the facility and heat-resistant glass with FTO coating and its picture. The width and height of six side heat-resistant glasses are 120mm and 300mm and the width and height of FTO coated area on the glass are 40mm and 280mm, respectively. Power was supplied through the 20mm-wide electrodes attached on the heat-resistant glasses.

Air blower was installed for the pressure driven air flow and flow rate of air into the test section was measured by mass flowmeter. DEHS (Di-Ethyl-Hexyl-Sebacat) aerosol which is non-toxic and volatile oil was injected to the test section as seed particles for PIV method. DEHS aerosol was generated to 1 micro-meter

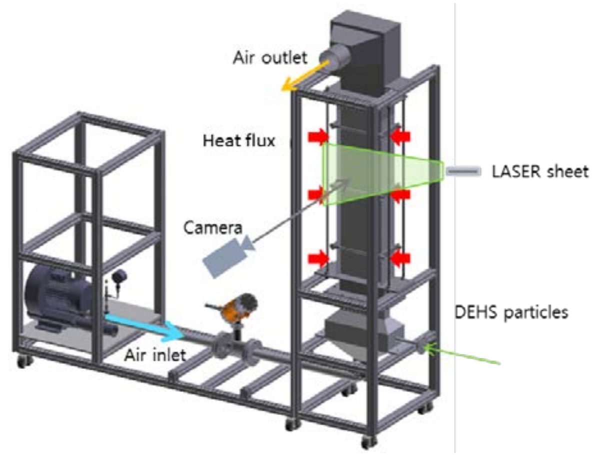

Fig. 4. Design of the flow visualization test facility

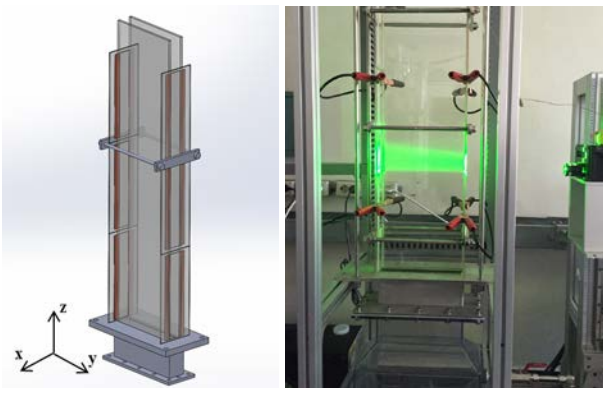

Fig. 5. Design of test section with FTO coated heat-resistant glass (left) and picture of the test section of the facility (right)

droplet size and its volatility keeps the transparent glass from contamination of FTO coated inner surfaces for the PIV method. The lower plenum upstream of the test section was designed to mix up the air flow and DEHS aerosol injected to the side of lower plenum. Flow straightening structure was installed between the lower plenum and the test section to reduce the turbulence intensity of the flowing air. Upper plenum after the test section was designed to prevent the influence of reverse flow to the test section.

Temperature and pressure of air at the inlet and outlet of the test section were measured with thermocouples and pressure transducers. Applying the PIV method, the velocity field and turbulence quantities inside of riser can be measured at different elevations and depths. Continuous laser and high speed camera were equipped for PIV method. Infrared (IR) camera will be used for obtaining thermal boundary conditions of the test section. With wall temperature distribution on the outer surface of test section, the wall temperature distribution of inner surface can be evaluated with radiation balance equations.

5.2 Preliminary test

A preliminary test was conducted in the flow visualization test facility to acquire the local velocity profiles with intense heating. The total mass flow rate of injected air and DEHS aerosol was about 0.0042 kg/s and the electrical power applied to the FTO coating was about 296.2 W. Velocity fields were observed by high speed camera at the mid-plane in the depth direction and z = 0.5 m from the entry of heating region.

Figure 6 shows the mean velocity field which was obtained from the acquired experimental images with PIV method. In Figure 6, the velocity field shows a steep increase of velocity vectors in vertical direction near the wall region as the CFD calculation in Figure 3.

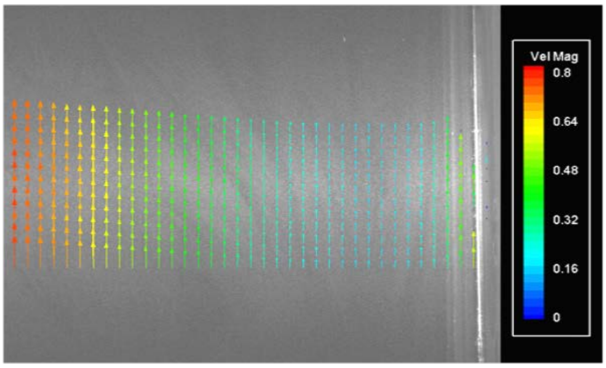

Fig. 6. Mean velocity field obtained with PIV method at the mid-plane in the depth direction

6. Summary

In this study, benchmark calculation was conducted to reproduce the previous statements that V2F turbulence model can capture the mixed convection phenomena with the Shehata's experimental data. Then, the necessity of the model validation for the mixed convection phenomena was confirmed with the CFD analyses for the geometry of the prototype RCCS riser. For the purpose of validating the turbulence models for mixed convection phenomena in the heated rectangular riser duct, validation plan with three experimental tests was introduced. Among them, the flow visualization test facility with preserved cross-section geometry was introduced and a preliminary test result was shown. Further investigations for model validation will be conducted to enhance a comprehension of the heat transfer phenomena in the RCCS riser duct.

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-2015M2A8A2076525)

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] 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.

[5] 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.

[6] P. A. Durbin, Near-wall Turbulence Closure Modeling Without Damping Functions, Thoeret. Comput. Fluid Dynam., Vol. 3, pp.1-13. 1991.

[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.

[8] A. M. Shehata and D. M. McEligot, Turbulence Structure in the Viscous Layer of Strongly Heat Gas Flows, Tech. Rep. INEL-95/0223, Idaho National Engineering Laboratory, Idaho Falls, IO, 1995.

[9] D. H. Shin et al., Preliminary analysis on the mixed convection phenomena in the scaled-down VHTR RCCS riser experiment, Transactions of the Korean Nuclear Society Spring Meeting, Vol. 1, 2016.