Study of Strongly Heated Upward Turbulent Gas Flow using v^2-f turbulence model

Yohanes Setiawan Nietiadi [a], Jeong Ik Lee [a*], Yacine Addad [b]
[a] *Department of Nuclear and Quantum Engineering, Korea Advanced Institute of Science and Technology*
291 Daehak-ro, Yuseong-gu, Daejeon, 305-701, Republic of Korea
Tel: 82-42-350-3829, Fax: 82-42-350-3810
[b] *Department of Nuclear Engineering, Khalifa University of Science, Technology & Research (KUSTAR)*
P.O.Box 127788, Abu Dhabi, UAE
Tel: 971-(0)2-5018515, Fax: 971-(0)2-4472442
[*] *Corresponding author: jeongiklee@kaist.ac.kr*

1. Introduction

Gas-cooled Fast Reactor (GFR) is one of the six Generation IV nuclear systems which operates in high pressure and high temperature. To meet the Generation IV nuclear system requirement for enhanced safety, GFR needs to rely on passive Decay Heat Removal (DHR) system with natural circulation of gas to remove the heat when Lost Of Coolant Accident occurs. Since the power density is high with GFR, conduction and radiation heat transport is not enough to cool the core. In the situation where high heat flux and low pressure happens at the same time, gas heat transfer coefficient has tendency to become deteriorated. This heat transfer regime is known as Deteriorated Turbulent Heat Transfer (DTHT) regime. DTHT can be induced by two effects: (1) buoyancy and (2) acceleration. The threshold value for both effects to move from the forced turbulent heat transfer to the DTHT regime are found to be $Bo^*_{th} \sim 6\times10^{-7}$ and $K_{vth} \sim 3\times10^{-6}$ in the previous works [1]. Many experiments and simulations have been done to investigate this phenomenon and the boundary of the regime. However, very limited number of experiments was conducted in the regime where buoyancy effect and acceleration effect are in the same order of magnitude and high enough to cause DTHT. Lee et al. [2] has done the experiment using gas natural circulation system in that regime and this paper will refer to that data and result. To have a better knowledge of v^2-f turbulence model and gas heat transfer, simulation has been conducted and representative results of the simulation will be presented.

2. Methods and Results

Numerical analysis was performed using a commercial computational fluid dynamics code ANSYS FLUENT 14.5 to model the mixed convection flow in a gas system. Coupled algorithm is applied to solve the flow in a vertically oriented round tube by applying two-dimensional axi-symmetric model in a cylindrical coordinate system. In this section turbulence model, problem domain, and the results will be described

2.1 Turbulence Model

The v^2-f turbulence model is consisted of 4 equation model based on transport equation for turbulence kinetic energy (k), dissipation rate (ε), velocity variance scale ($\overline{v^2}$), and elliptic relaxation function (f) shown in Eq. (1) to Eq. (4). Eddy viscosity was evaluated by using velocity variance scale and it has shown to provide the right scaling to represent the damping of turbulent transport near the wall which is not represented in the k-ε model well. The anisotropic wall effects are modeled through the elliptic relaxation function f.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=P-\rho\varepsilon+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+S_k \tag{1}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{C'_{\varepsilon1}P-C_{\varepsilon2}\rho\varepsilon}{T}+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+S_\varepsilon \tag{2}$$

$$\frac{\partial}{\partial t}\left(\rho\overline{v^2}\right)+\frac{\partial}{\partial x_i}\left(\rho\overline{v^2}u_i\right)=\rho kf-6\rho\overline{v^2}\frac{\varepsilon}{k}+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial \overline{v^2}}{\partial x_j}\right]+S_{\overline{v^2}} \tag{3}$$

$$f-L^2\frac{\partial f}{\partial x_j}=(C_l-1)\frac{\frac{2}{3}-\overline{v^2}/k}{T}+C_2\frac{P}{\rho k}+\frac{5\overline{v^2}/k}{T}+S_f \tag{4}$$

Where

$$P=2\mu_t S^2,\ S^2\equiv S_{ij}S_{ij},\ S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \tag{5}$$

$$T'=\max\left[\frac{k}{\varepsilon},6\sqrt{\frac{\nu}{\varepsilon}}\right] \tag{6}$$

$$T=\min\left[T',\frac{\alpha}{\sqrt{3}}\frac{k}{\overline{v^2}C_\mu\sqrt{2S^2}}\right] \tag{7}$$

$$L'=\min\left[\frac{k^{3/2}}{\varepsilon},\frac{1}{\sqrt{3}}\frac{k^{3/2}}{\overline{v^2}C_\mu\sqrt{2S^2}}\right] \tag{8}$$

$$L=C_L\max\left[L',C_\eta\left(\frac{\nu^3}{\varepsilon}\right)^{1/4}\right] \tag{9}$$

2.2 Problem Domain

The problem domain is designed to match the operational geometry of GFR which is our domain of interest. The geometry of the problem domain was constructed as shown in Fig. 1. A developing length was provided to match the fully developed flow condition at the entrance of the test section. The test section was instrumented to report heat flux at 20 different locations, tabulated in Table I. An adiabatic outlet section was attached to the test section in order to allow an outflow boundary condition. The inner diameter for developing length and test section is around 16mm and for the outlet region is around 32mm. The operating condition was selected to match J. I. Lee [3] experimental set up and J. I. Lee experimental data on natural circulation will be used to validate the numerical analysis result. The thermophysical properties of gases were provided to FLUENT by using real gas model equation of state mode.

Table I: Thermal couple position

Thermal Couple	L/D
TC01	2
TC02	8.1
TC03	14.1
TC04	20.2
TC05	26.2
TC06	32.3
TC07	38.3
TC08	44.4
TC09	50.4
TC10	56.5
TC11	62.5
TC12	68.5
TC13	74.6
TC14	80.6
TC15	86.7
TC16	92.7
TC17	98.8
TC18	104.8
TC19	110.9
TC20	116.9

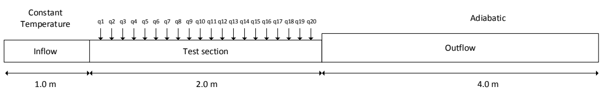

Fig. 1. Problem domain.

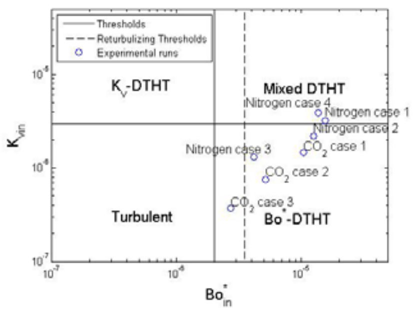

Fig. 2. Problem domain.

Grid convergence was investigated and it was found that mesh system with 60 radial mesh was good enough to ensure good resolution of the near wall flow (wall y^+<1). For axial direction, axial mesh of 400 control volumes was used in the test section. 5 boundary conditions were employed: (1) uniform of all physical variables at the inlet; (2) The Neumann condition of zero was imposed for all physical variables at the outlet; (3) heat flux value at 20 different locations was set in the test section wall; (4) A constant temperature of 300 K was assumed for the developing region while adiabatic wall condition was applied to surrounding wall region; (5) an axisymmetric boundary condition was selected at the centerline of the test section.

2.3 CFD validation

Before the validation is started, the mesh system convergence was checked. For this purpose, validation of mesh systems has done and shown in Fig. 3 and Fig. 4, hence radial mesh 60 was chosen as the mesh system and the number of axial number was 100 for developing section, 400 for test section, and 1,000 for chimney section.

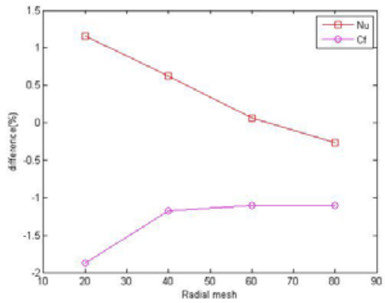

Fig. 3. Grid convergence test result.

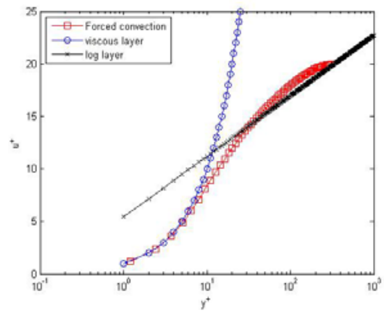

Fig. 4. Forced convection test result.

The v^2-f model performance in modeling forced convection flow is assessed by its heat transfer and friction factor prediction. Heat transfer criteria assessed by comparing numerically obtained Nusselt number to Gnielinski correlation shown in Eq. (10). As for the friction factor assessment, numerically obtained skin friction coefficient is compared to Blasius correlation shown in Eq. (11).

$$Nu_{Gnielinski} = \frac{\left(f/8\right)\left(\mathrm{Re}-1000\right)\mathrm{Pr}}{1+12.7\sqrt{f/8}\left(\mathrm{Pr}^{2/3}-1\right)}\left(\frac{T_w}{T_b}\right)^{-0.45}\left(1+\left(\frac{L}{D}\right)^{\frac{2}{3}}\right) \quad (10)$$

$$f = (1.82\log_{10}\mathrm{Re}-1.64)^{-2}$$

$$c_{f0} = 0.079\,\mathrm{Re}^{-0.25} \quad (11)$$

2.4 Comparison with experiment data

J. I. Lee has several cases which is the case that are suspected in the regime which buoyancy and acceleration effect has the same order of importance and this paper will review the numerical analysis using CFD to investigate what will happen in the case which buoyancy induced DTHT and acceleration induced DTHT has the same order.

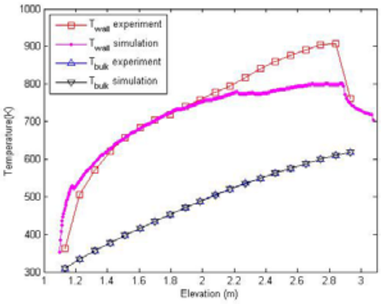

Fig. 5. Comparison of Temperature between experimental data and simulation of nitrogen case 1.

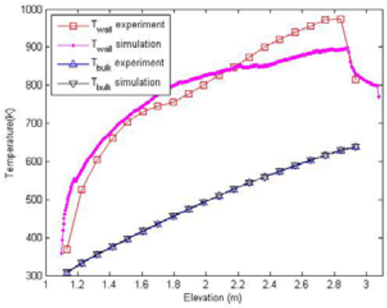

Fig. 6. Comparison of Temperature between experimental data and simulation of nitrogen case 2.

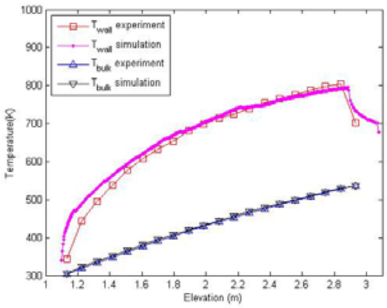

Fig. 7. Comparison of Temperature between experimental data and simulation of nitrogen case 3.

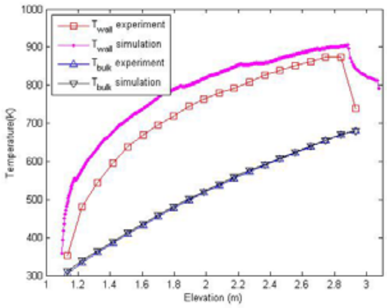

Fig. 8. Comparison of Temperature between experimental data and simulation of nitrogen case 4.

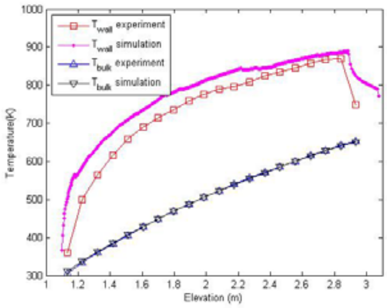

Fig. 9. Comparison of Temperature between experimental data and simulation of CO_2 case 1.

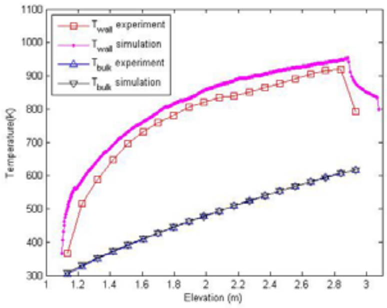

Fig. 10. Comparison of Temperature between experimental data and simulation of CO_2 case 2.

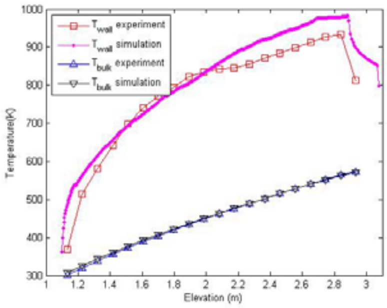

Fig. 11. Comparison of Temperature between experimental data and simulation of CO_2 case 3.

2.5 Discussion

As it can be observed from Fig. 2, the 7 cases can be divided into 3 different kinds of cases which are (1) Buoyancy induced DTHT, (2) Re-turbulizing Buoyancy induced DTHT, and (3) Mixed DTHT. In Buoyancy induced DTHT regime (Nitrogen case 2, CO_2 case 1, and CO_2 case 2), v^2-f turbulence model can predict the heat transfer coefficient quite well even though the simulation results overpredicted the deterioration of heat transfer coefficient of the gas. In re-turbulizing regime (CO_2 case 3), v^2-f turbulence model can predict the wall temperature quite well but does not show re-turbulization as experimental data does. In mixed DTHT regime, which is the regime that still uncharted, v^2-f turbulence model can actually predicts well like in the Buoyancy induced DTHT when the inlet acceleration parameter is higher than 1.567 times the inlet acceleration parameter thresholds and buoyancy parameter higher than 22.685 times the inlet buoyancy parameter thresholds, but it shows returbulizing phenomena when the inlet acceleration parameter is higher than 0.879 times the inlet acceleration parameter thresholds and buoyancy parameter higher than 20.895 times the inlet buoyancy parameter thresholds.

3. Conclusions

It has been found that gas cooled fast reactor has a tendency to operate in Deteriorated Turbulent Heat Transfer (DTHT) regime as heat flux becomes higher under low cooling flow environment such as natural circulation operation. Therefore, the unique behavior of the gas properties in the DTHT regime should be investigated. Previous researches have been done in Buoyancy induced DTHT regime only or in acceleration DTHT only but only a few researches conducted in the regime that both occur at the same time and in the same order of effect.

Numerical analysis is done with v^2-f turbulence model to observe the performance and the effect of buoyancy and acceleration to v^2-f model performance. The numerical results show that the v^2-f model performs reasonably well to match the fully developed flow condition in forced convection cases. Among seven cases that had been simulated, we can divide the cases to three groups, (i) returbulizing Buoyancy induced-DTHT with $2\times10^{-6} \le Bo_{in}^* \le 3.5\times10^{-6}$, (ii) Buoyancy induced-DTHT with $3.5\times10^{-6} \le Bo_{in}^*$, and (iii) mixed-DTHT. In case of returbulizing Buoyancy induced-DTHT with $2\times10^{-6} \le Bo_{in}^* \le 3.5\times10^{-6}$, the wall temperature from experimental data is underpredicted by v^2-f model and this is the region of returbulization that was defined by J. I. Lee[1]. It was found that the v^2-f model can predict the heat transfer coefficient quite well even though the simulation results overpredicted the deterioration of heat transfer coefficient of the gas. In re-turbulizing regime (CO_2 case 3), v^2-f turbulence model can predict the wall temperature quite well but does not show re-turbulization as experimental data does. In mixed DTHT regime, which is the regime that still uncharted, v^2-f turbulence model can actually predict well like in the Buoyancy induced DTHT when the inlet acceleration parameter is higher than 1.567 times the inlet acceleration parameter thresholds and buoyancy parameter higher than 22.685 times the inlet buoyancy parameter thresholds, but it shows returbulizing phenomena when the inlet acceleration parameter is higher than 0.879 times the inlet acceleration parameter thresholds and buoyancy parameter higher than 20.895 times the inlet buoyancy parameter thresholds.

More experimental data on Mixed-DTHT regime is needed to confirm these results and to make clear the thresholds of these regimes.

ACKNOWLEDGMENTS

The authors gratefully acknowledge that this research was supported by the Korean Ministry of Trade, Industry, and Energy and also supported by Korea Advanced Institute of Science and Technology.

REFERENCES

[1] D. M. McEligot and C. W. Coon and H. C. Perkins, Relaminarization in Tubes, International Journal of Heat & Mass Transfer: Shorter Communications, 13, p. 431-433, 1969.

[2] J. I. Lee, P. Hejzlar, P. Saha, P. Stahle, M.S. Kazimi, D. M. McEligot, Deteriorated turbulent heat transfer (DTHT) of gas up-flow in a circular tube: experimental data. International Journal of Heat and Mass Transfer, 51, p. 3259–3266, 2008.

[3] J. I. Lee, H.C. No, P. Hejzlar, Evaluation of system codes for analyzing naturally circulating gas loop, Evaluation of system codes for analyzing naturally circulating gas loop, Nuclear Engineering and Design 239, p. 2931-2941, 2009