# CFD Study of Deteriorated Turbulent Heat Transfer in Upward Flow

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

KAIST MMR is a Gas-cooled Fast Reactor (GFR) using supercritical $CO_2$ as a working fluid of reactor core and power cycle without intermediate heat exchanger [1]. KAIST MMR needs to rely on a passive Decay Heat Removal (DHR) system with natural circulation of gas to remove the heat when a Lost Of Coolant Accident occurs since passive decay heat removal using conduction and radiation is not enough to provide sufficient decay heat removal rates.

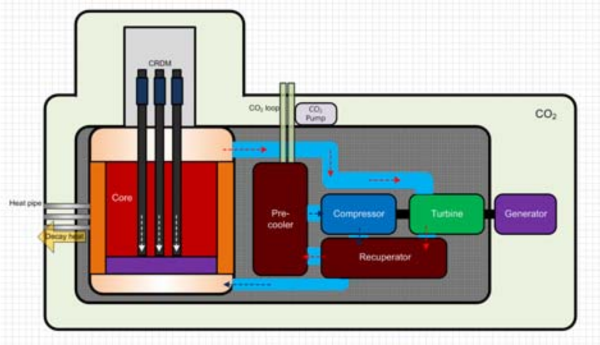


Fig. 1 KAIST MMR concept design [1]

During an accident phase, a gas cooled system might operate in the deteriorated turbulent heat transfer (DTHT) regime under high heat flux and low cooling flow environment. Once the gas flow is in the DTHT regime, the flow will show a unique behavior and this effect will significantly affect temperature, velocity, turbulent momentum and heat transfer. DTHT regime can be induced by two effects: (1) buoyancy and (2) acceleration.

Apart from these two deteriorating effects, another unique behavior of fluid in the DTHT regime is that the convective heat transfer rate will continue to deteriorate until it reaches certain point. The downstream of this point, is known as the recovery region, where the convective heat transfer rate returns back to the high values by recovering turbulence. We called this phenomena as re-turbulization.

The map of the DTHT regime can be seen from fig. 2, where the x-axis is the buoyancy parameter and y-axis is the acceleration parameter which is the agreed governing non-dimensional numbers among the researchers to illustrate the phenomena. The Buoyancy parameter is defined in Eq. (1) and the acceleration parameter is defined in Eq. (2), respectively. The threshold value for both effects to move from the forced turbulent heat transfer to the DTHT regime are found to be $Bo^* \geq 2\text{x}10^{-6}$ and $K_v \geq 2.5\text{x}10^{-6}$ [2] in the previous works.

$$Bo^* = \frac{Gr_q}{\mathrm{Re}^{3.425}\,\mathrm{Pr}^{0.8}} \quad (1)$$

$$K_v = \frac{4q^+}{\mathrm{Re}} \quad (2)$$

Many experiments and simulation have been done to investigate this phenomenon and the boundary of the regime. However, very limited number of experiment was conducted in the regime where buoyancy effect and acceleration effect are in the same order of magnitude and high enough to cause DTHT (mixed DTHT).

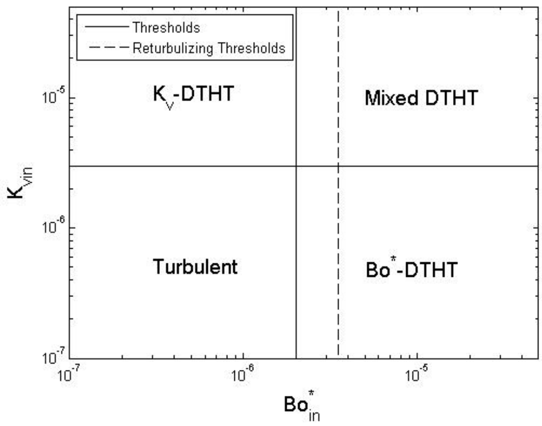


Fig. 2 Map of DTHT regime

Some important experimental researches that have been done in the gas DTHT regime is Lee et al. [3] who investigated the heat transfer of gas flow in the range of buoyancy parameter from $3\text{x}10^{-9}$ to $10^{-5}$ and acceleration parameter span from $6\text{x}10^{-8}$ to $5\text{x}10^{-6}$ and presented the behavior of Nusselt number ratio from the experiment as fig. 3 and fig. 4.

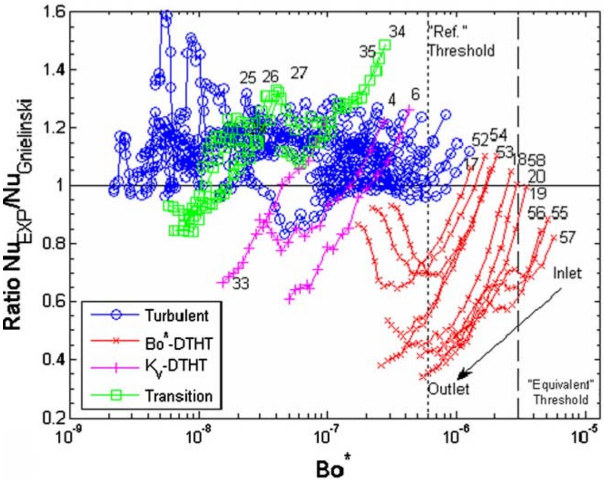

Fig. 3 Nu ratio-Bo plot [2]

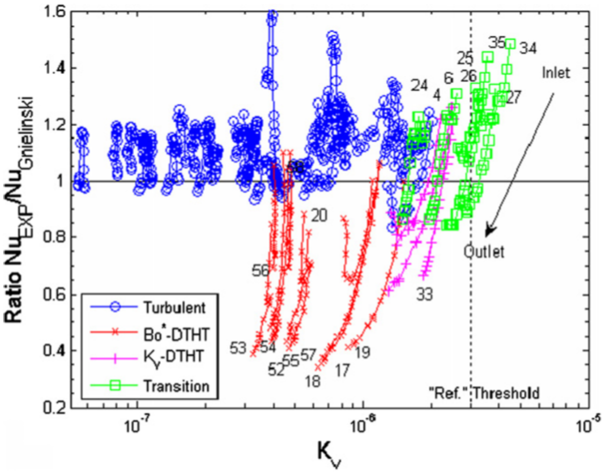

Fig. 4 Nu ratio-$K_v$ plot [2]

This paper will discuss a Computational Fluid Dynamics analysis on DTHT by assuming hypothetical boundary conditions especially on the mixed DTHT regime.

## 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 ($\varepsilon$), velocity variance scale ( $\overline{v^2}$ ), and elliptic relaxation function ($f$) shown in Eq. (3) to Eq. (6). 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 $k$-$\varepsilon$ model well. The anisotropic wall effects are modeled through the elliptic relaxation function $f$.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=P-\rho\varepsilon+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+S_k \tag{3}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{C'_{\varepsilon 1}P-C_{\varepsilon 2}\rho\varepsilon}{T}+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+S_\varepsilon \tag{4}$$

$$\frac{\partial}{\partial t}\left(\rho\overline{v^2}\right)+\frac{\partial}{\partial x_i}\left(\rho\overline{v^2}u_i\right)=\rho k f-6\rho\overline{v^2}\frac{\varepsilon}{k}+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial \overline{v^2}}{\partial x_j}\right]+S_{\overline{v^2}} \tag{5}$$

$$f-L^2\frac{\partial f}{\partial x_j}=(C_l-1)\frac{\frac{2}{3}-\overline{v^2}/k}{T}+C_2\frac{P}{\rho k}+\frac{5\overline{v^2}/k}{T}+S_f \tag{6}$$

Where

$$P=2\mu_t S^2,\ S^2\equiv S_{ij}S_{ij},\ S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \tag{7}$$

$$T'=\max\left[\frac{k}{\varepsilon},6\sqrt{\frac{v}{\varepsilon}}\right] \tag{8}$$

$$T=\min\left[T',\frac{\alpha}{\sqrt{3}}\frac{k}{\overline{v^2}C_\mu\sqrt{2S^2}}\right] \tag{9}$$

$$L'=\min\left[\frac{k^{3/2}}{\varepsilon},\frac{1}{\sqrt{3}}\frac{k^{3/2}}{\overline{v^2}C_\mu\sqrt{2S^2}}\right] \tag{10}$$

$$L=C_L\max\left[L',C_\eta\left(\frac{v^3}{\varepsilon}\right)^{1/4}\right] \tag{11}$$

### *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. 5. 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 after the test section in order to allow an outflow boundary condition and remove boundary condition effect on the upstream. There are variations of inner diameter for some cases to produce the conditions of buoyancy and acceleration parameters. The thermo-physical properties of gases were provided to FLUENT by using NIST real gas model mode.

Table I. Measurement Location

| 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 |

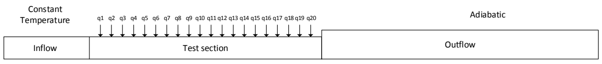


Fig. 5 Problem Domain

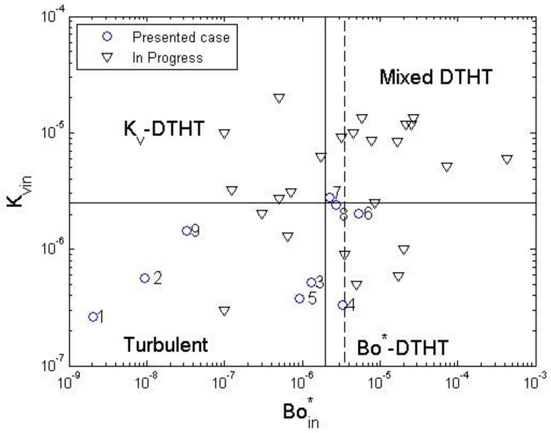


Fig. 6 Case Map

5 boundary conditions were employed: (1) uniform profiles for all physical variables at the inlet; (2) The Neumann condition of zero was imposed for all physical variables at the outlet; (3) A constant 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. 7 and Fig. 8, 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.

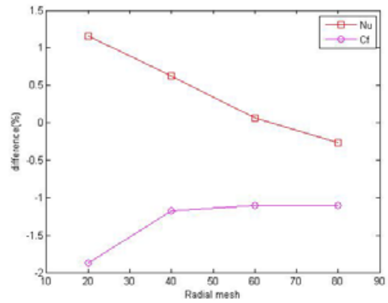


Fig. 7. Grid convergence test result.

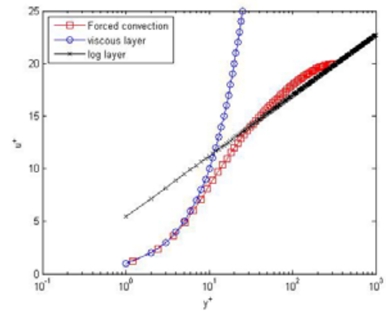


Fig. 8. Forced convection test result.

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. (12). As for the friction factor assessment, numerically obtained skin friction coefficient is compared to Blasius correlation shown in Eq. (11).

$$Nu_{Gnielinski} = \frac{\left(f/8\right)\left(\mathrm{Re}-1000\right)\mathrm{Pr}}{1+12.7\sqrt{f/8}\left(\mathrm{Pr}^{2/3}-1\right)}\left(\frac{T_w}{T_b}\right)^{-0.45}\left(1+\left(\frac{L}{D}\right)^{\frac{2}{3}}\right) \quad (12)$$

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

$$c_{f0} = 0.079\,\mathrm{Re}^{-0.25} \quad (13)$$

*2.4 CFD result in DTHT regime and discussion*

This paper will review the numerical analysis results to predict what will happen when the fluid flow enters both buoyancy induced DTHT and acceleration induced DTHT regimes.

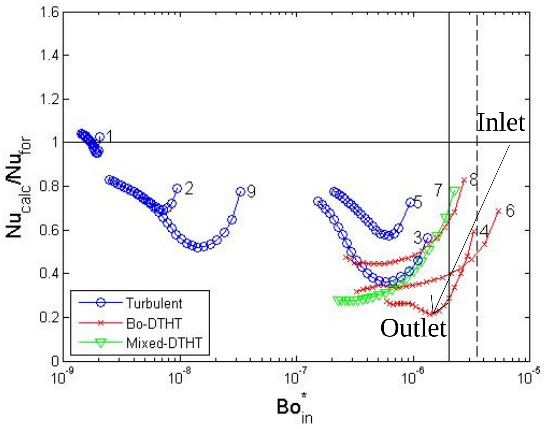


Fig. 9 Nu ratio vs. Buoyancy Parameter

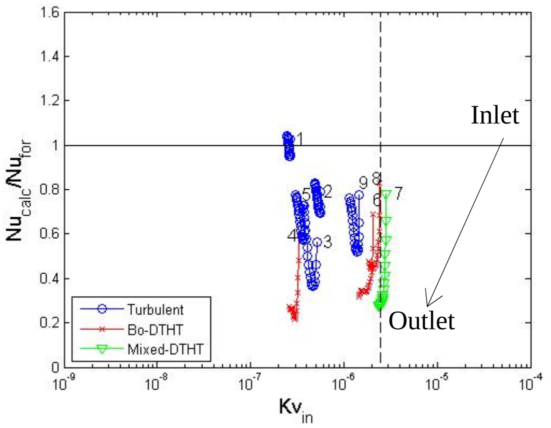


Fig. 10 Nu ratio vs. Acceleration Parameter

So far 20% of the planned cases were calculated and even at this stage some interesting results are obtained. The comparison between numerical result (Fig. 9 and Fig. 10) and the experimental result that we refer (Fig. 3 and Fig. 4), the trend that shown from the result is Nusselt number ratio from buoyancy induced DTHT and mixed DTHT behave as expected, mixed DTHT Nusselt number ratio is steeper than buoyancy induced DTHT, but when the case is in turbulence regime, $v^2$-$f$ numerical results show that it was in the re-turbulization. These results show us that $v^2$-$f$ turbulence model should be modified so it can reflect the actual physical phenomenon correctly.

### 3. Summary and Further Works

It has been found that a gas cooled fast reactor has a tendency to operate in the Deteriorated Turbulent Heat Transfer (DTHT) regime as heat flux becomes higher under low cooling flow environment such as natural circulation operation in the past research works. Therefore, the unique behavior of the gas properties in the DTHT regime should be investigated. Previous researches have been done in either focusing on the buoyancy induced DTHT regime only or in the acceleration DTHT only. Very limited researches were conducted in the regime where both occur at the same time and in the same order of magnitude.

Numerical analysis is done with the $v^2$-$f$ turbulence model to predict the physical phenomena for future experimental work. The effects of buoyancy and acceleration were studied with CFD by designed cases to distinguish the dominant effect in the mixed DTHT regime. Numerical results of the $v^2$-$f$ turbulence model show us that it can predict quite well in the buoyancy induced DTHT regime but the prediction of the returbulization is not satisfactory so far. The returbulization occurs before the observed condition in the experiment, which indicates that the $v^2$-$f$ turbulence model may require further improvement.

More numerical results in the Mixed-DTHT regime will be obtained and studied to provide clearer view on the strongly heated turbulent flow and its heat transfer deteriorating mechanism.

### REFERENCES

[1] S. G. Kim, S. Cho, Y. Kim, Y. H. Jeong, J. I. Lee, System Design of a Supercritical $CO_2$ cooled Micro Modular Reactor, Proceedings of the HTR, Oct. 27-31, 2014, Weihai,China.

[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, Gas heat transfer in a heated vertical channel under deteriorated turbulent heat transfer regime, Ph.D. thesis, MIT, Cambridge, MA, USA.