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

The Numerical Study for Chimney Heights on Laminar Natural Convection Heat Transfer

Chul-Kyu Lim[a*], Sang-Koo Han[a] and Bum-Jin Chung[b]
[a]*Atomic Creative Technology Co., Ltd., #204, IT Venture Town, 35, Techno 9 Ro, Yuseong-gu, Daejeon 305-510, Korea*
[b] *Department of Nuclear Engineering, Kyung Hee University, 1732 Deokyoung-daero, Giheung-gu, Yongin-si, Gyeonggi-do 446-701, Korea*
[*]*Corresponding author: happiness0902@actbest.com*

Many researchers have studied natural convection heat transfer inside a chimney which can be used as an important heat sink because it provides effective and stable means of heat removal [1~5].

A thermally insulated chimney attached to a heated section induces an increase in the flow rate and leads to a higher transfer rate. A chimney accelerates flow through buoyancy; the chimney acts as a shroud for the plume emanating from the heated section of the furnace [6]. And the heat transfer is enhanced with an increase in the chimney height since this increase the flow rate (Fig. 1).

This study investigated numerically the influence of heights of a chimney on the velocity and temperature fields. Numerical simulations using FLUENT 6.2 software were performed.

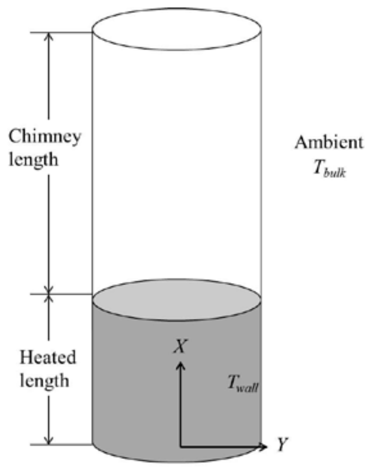


Fig. 1. Schematic diagram of a chimney system.

2. Numerical methods

2.1 Numerical Analysis

Numerical analysis was carried out using FLUENT 6.2 software [7]. GAMBIT software was used to generate the two-dimensional (2D) mesh of the chimney system. The overall domain had a width of 0.035 m and a height from 0.17 m to 1.07 m, depending on the chimney height. The height of the heated section was 0.07 m. Figure 2 shows the domain used for the calculations, which comprised heated walls, adiabatic walls.

The number of cells and faces was about 5,000. The right and left sides of the domain were taken as a pressure inlet and outlet, respectively. Since the boundary layer was confined in an extremely thin region along the wall, concentrated grids were used for this region while coarse grids were used for the rest of the domain. The simulations were carried out using the Boussinesq approximation with material properties corresponding to those in the experiments.

The temperature of the heated wall was maintained at a constant temperature of 400 K, and the cold rod was kept at 200 K. All other boundary conditions were imposed as adiabatic walls. The bulk temperature of fluid remained constant at 300 K.

A steady segregated solver was used, with a second-order upwind algorithm for momentum and energy and a laminar flow model. The PRESTO algorithm was adopted for pressure discretization, while the SIMPLE algorithm was used for pressure–velocity coupling. The residual momentum and energy values were taken as 10^{-6}.

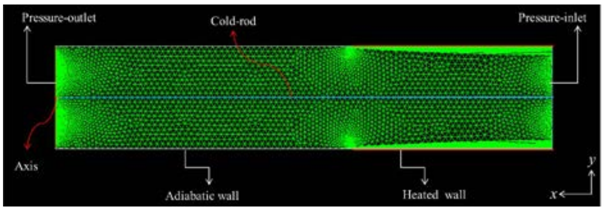


Fig. 2. Solution grid of the 2D axisymmetric model.

2.2 Test Matrix

Table 1 presents the test matrix and geometrical arrangements of the FLUENT simulations were. The diameter and length of the heated section were 0.035 m and 0.07 m, respectively. The diameter of the chimney was 0.035 m, but the heights were varied from 0 m to 1.00 m. The Prandtl number was 2,094.

Table 1: Test matrix for the numerical analyses.

D (m)	L (m)	Ra_D	Pr	Chimney heights (m)
0.035	0.07	7.23×10^9	2,094	0.00, 0.20, 0.40, 0.60, 0.80, 1.00

3. Results and discussion

3.1 Comparison with Experiments

The current numerical results using the FLUENT 6.2 software were compared with the existing experiments of Lim and Chung [5]. They carried out the natural convection heat transfer experiments of a chimney of Ra_D 7.23×10^9 for various exit lengths using the analogy concept. As shown in Fig. 3, the two Nu_D's show good agreement. As the exit length increases, the heat transfer enhances, but up to a certain length. Further extension of the exit length (chimney height) does not vary the heat transfer.

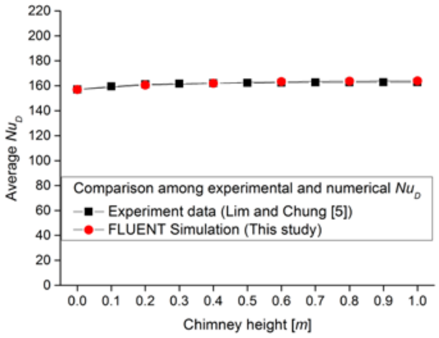
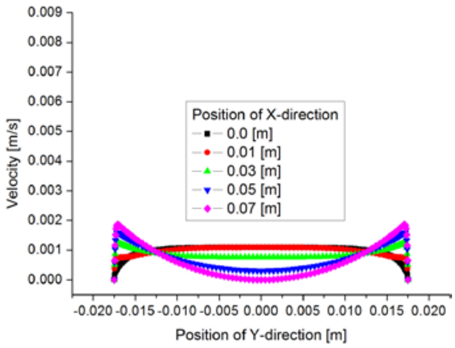


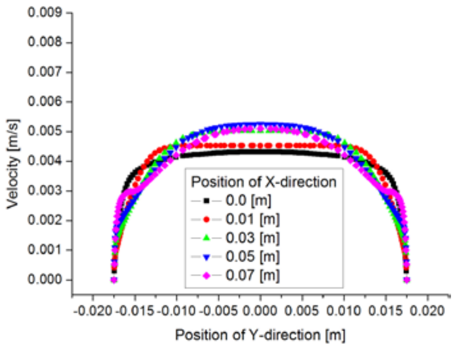
Fig. 3. Comparison between experimental and numerical average Nu_D values.

3.2 Chimney Phenomena

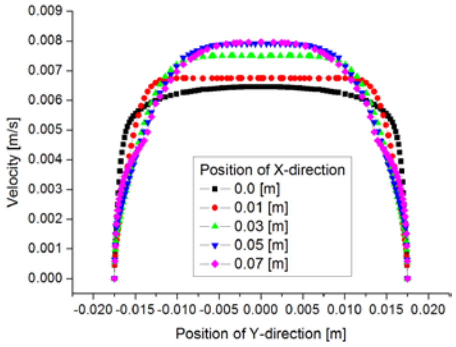
Figure 4 presents the velocity profiles calculated for the heated section for three different chimney heights. The velocity increased with increasing chimney height. Flow development along the flow direction was also observed. In the absence of a chimney, the velocity peaked near the wall, indicating natural convection. However, in the presence of a chimney, the velocity peak shifted to the centerline of the domain, indicating forced convection.



(a) No chimney



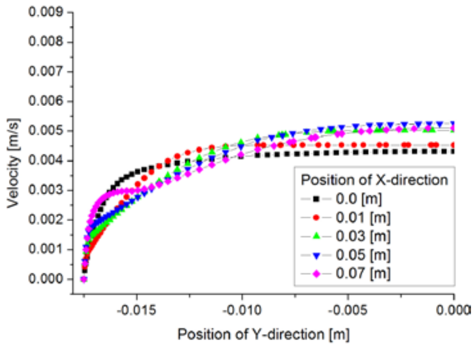
(b) 0.2 m chimney height



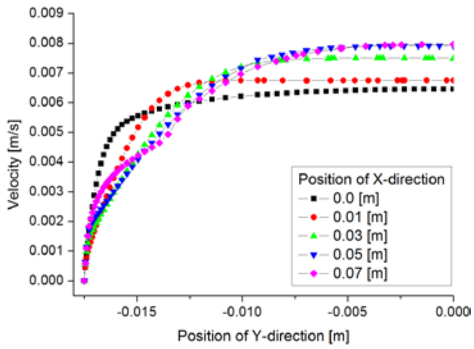
(c) 0.8 m chimney height

Fig. 4. Velocity profiles in the heated section.

Figure 5 shows the velocity profiles from the heated wall to the centerline of the chimney. The velocity profiles developed as the fluid proceeded to the top. A notable observation is that the velocity peaks appeared near the heated wall at the 0.7 m location. This result may be attributable to the combined influence of forced and natural convection: the former prevails near the entrance and the latter near the exit.



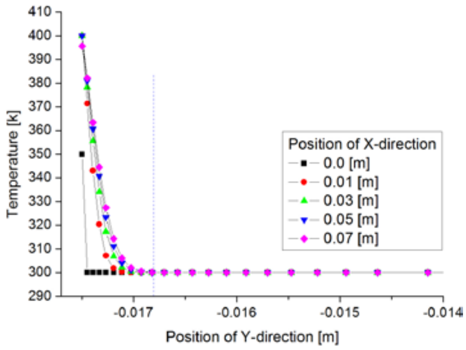
(a) 0.2 m chimney height



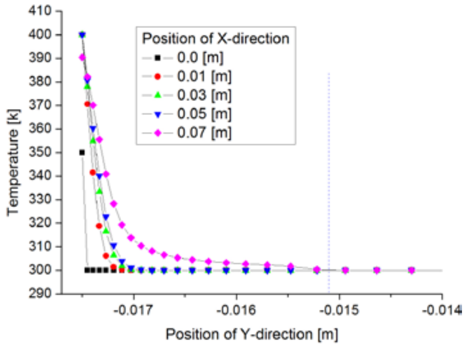
(b) 0.8 m chimney height

Fig. 5. Velocity profiles near the heated wall.

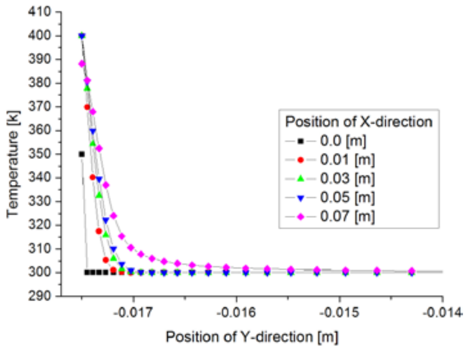
Figure 6 presents the temperature profiles along the heated cylinder for three different chimney heights. The thermal boundary layer developed along the flow direction. Because of the very high Prandtl number in this study, the thermal boundary layers were much thinner than the velocity boundary layers.



(a) No chimney



(b) 0.2 m chimney height



(c) 0.8 m chimney height

Fig. 6. Temperature profiles near the heated wall.

4. Conclusions

This study investigated numerically the natural convection heat transfer according to the different chimney heights. The Nusselt numbers from numerical simulations using FLUENT 6.2 software agreed well with the existing experiments of Lim and Chung [5].

Both numerical and experimental results confirmed that an increase in the chimney height enhances the heat transfer up to a certain height due to the extended buoyant acceleration inside the chimney. And that further increases do not affect the heat transfer because the height was determined by the balance of acceleration driven by buoyancy and deceleration caused by friction between the fluid and the wall of the chimney [8].

The velocity and temperature profiles near the heated wall were calculated. The velocity increased with increasing chimney height as the velocity profiles developed as the fluid proceeded to the top. And the temperature profiles along the heated cylinder for three different chimney heights. The thermal boundary layer developed along the flow direction.

REFERENCES

[1] S. E. Haaland and E. M. Sparrow, Solutions for the channel plume and the parallel-walled chimney, Numerical Heat Transfer, Vol. 6, pp. 155-172, 1983.
[2] T. S. Fisher and K. E. Torrance, Experiments on chimney enhanced free convection, Journal of Heat Transfer, Vol. 121, pp. 603-609, 1999.
[3] S. Kazansky, V. Dubovsky, G. Ziskind, R. Letan, Chimney-enhanced natural convection from a vertical plate: experiments and numerical simulations, Int. J. Heat Mass Transfer, Vol. 46, pp. 497-512, 2003.
[4] A. Andreozzi, B. Buonomo, O. Manca, Thermal management of a symmetrically heated channel-chimney system, Int. J. Thermal Siences, Vol. 48, pp. 475-487, 2009.
[5] C. K. Lim and B. J. Chung, Influence of the Entrance and Exit Lengths on the Natural Convection Heat Transfer of a Cylinder in a Duct, Journal of Energy Engineering, Vol. 21, No. 1, pp. 18-25, 2012.
[6] A. Auletta, O. Manca, B. Morrone, V. Naso, Heat transfer enhancement by the chimney effect in a vertical isoflux channel, Int. J. of Heat and Mass Transfer 44, pp. 4345-4357, 2001.
[7] FLUENT User’s Guide, release 6.3 FLUENT Incorporated (2006).
[8] A. Auletta, O. Manca, B. Morrone, V. Naso, Heat transfer enhancement by the chimney effect in a vertical isoflux channel, Int. J. of Heat and Mass Transfer 44, pp. 4345-4357, 2001.