I-PRO-O-169-05

Study of air circulation inside the oven by computational fluid dynamics.

Siravit Laocharoenpongsang[1], Prayut Jiamrittiwong[1], Karn Pana-Suppamassadu[2] and Thanarak Srisurat [1,*]

[1] *Faculty of Science, Energy and Environment, King Mongkut's University of Technology North Bangkok, Rayong Campus, Rayong, Thailand 21120*

[2] *Department of Chemical Engineering, King Mongkut's University of Technology North Bangkok, Bangsue, Bangkok, Thailand 10800*

*Corresponding author's e-mail address: thanarak.s@sciee.kmutnb.ac.th

Abstract

In industry, variety of hot air ovens has been utilized and trays usually have been installed in order to handle a large amount of specimens or raw products. The pattern of trays installation; however, could cause the non-uniform hot air circulation and temperature distribution which in turn degrade the quality of final products. By using a computational approach, this research investigated the flow behavior of hot air within the adopted design oven in which the blower rotated at normal speed of 1,155 rpm to induce desire air circulation. From the simulation results, the tray installation influenced flow field and decreased flow speed in the space among trays. Pressure drop was also higher with trays installed as compared to that of the oven without tray. The simulation results have been served as a guideline in improvement of the industrial oven design.

Keywords: Simulation, Air circulation, Oven

1. Introduction

Nowadays, hot air circulation ovens have been widely adopted in heating rubber pipe and metal, surface coating, food production etc. and, within the ovens, trays need to be installed for placing specimens. Patterns of tray installation could induce the non-uniform flow and temperature distributions of hot air within and trend to affect the final quality of products [1-3]. Generally and limitedly, flow and temperature measurement devices have been used to measure point-wise the velocity and temperature of hot air [4]. Later on, an alternative like computational approach has gained popularity in predicting two-dimensional flow behavior of hot air in circulation [5-7]. However, recent studies have shown that the three-dimensional simulation could offer more accurate and detailed information of hot air circulation [8,9]. [10] reported that the corners of oven accumulate heat because of low air circulation. In the present study, one of the oven designs proposed by the industry was investigated through the Computational Fluid Dynamics (CFD) in order to comprehend the behavior of hot air circulation within in both cases of with and without trays.

2. Simulation Approach

2.1 Oven Geometry and Computational Domain

In three-dimensional simulation, the air circulation within the oven without and with trays installed were simulated by Finite Element Scheme (COMSOL MULTIPHYSICS 3.5a). The oven structure had a rectangular volume i.e. 8,000 mm wide, 3,410.5 mm long and 1,000 mm high. On the far end, there were left and right exits where actually air could flow out and recirculate within the oven. The exits were 660 mm wide and 1,000 mm high as shown in Figure 1.

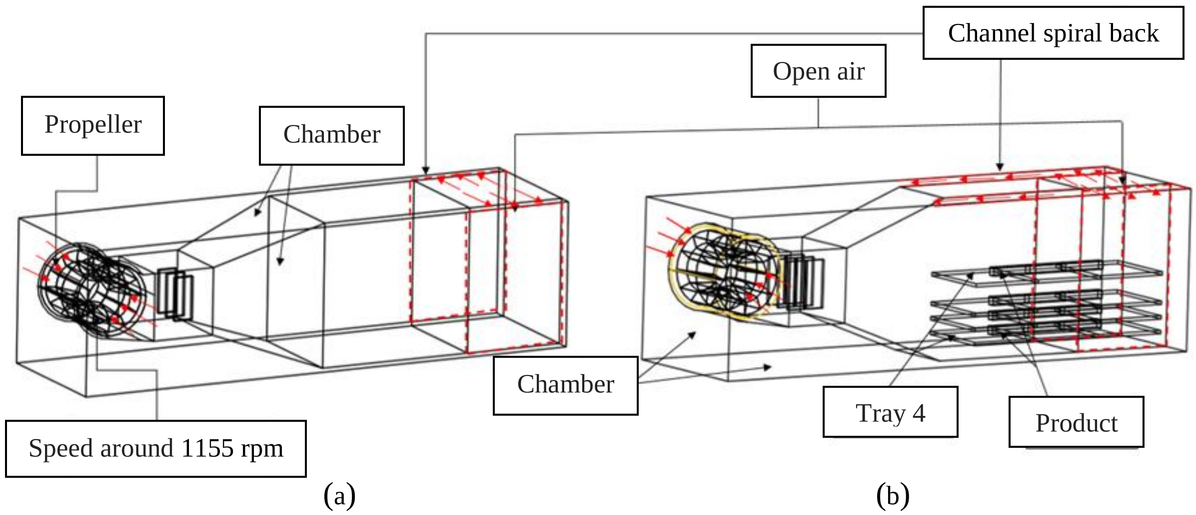


Figure 1 Geometry of oven and control boundaries (a) without trays (b) with 4-row trays; velocity arrows indicating flow circulation.

2.2 Governing Equations

Air movement within the designed oven was induced by a rotation of the blower propeller at 1,155 rpm which was simulated by Moving Mesh technique and Incompressible Navier-Stokes equations as follow:

$$dx; \cos(-2 \times pi \times rpm \times t) \times X - \sin(-2 \times pi \times rpm \times t) \times \mathrm{Y} - \mathrm{X} \tag{1-1}$$

$$dy; \sin(-2 \times pi \times rpm \times t) \times X - cos(-2 \times pi \times rpm \times t) \times \mathrm{Y} - \mathrm{Y} \tag{1-2}$$

$$dz\,; z - z \tag{1-3}$$

$$\rho \frac{\partial u}{\partial t} - \nabla \cdot \eta(\nabla \mathrm{u} + (\nabla \mathrm{u})^T) + \rho(u \cdot \nabla)u + \nabla p = F \tag{1-4}$$

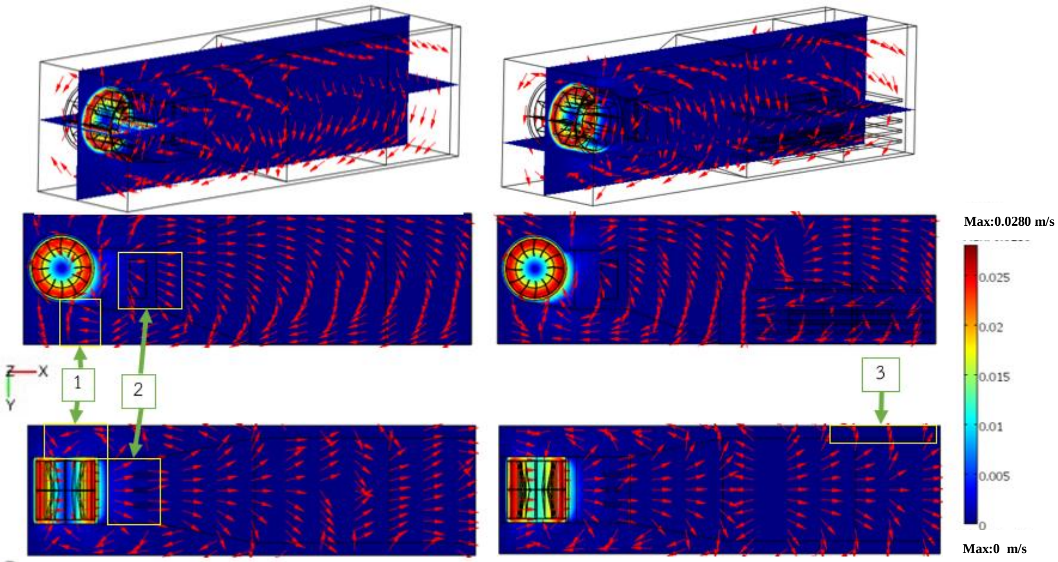
In the above equations**,** ρ (kg/m^3) denotes the density, u is the dynamic viscosity (m/s), t equal time (s), T is the temperature (°C), p is the pressure (Pa), η denotes the viscosity (Pa$\cdot s$) and surface tension force components, denoted by F (N/m^3).

The air flow speed was controlled by the angular speed of the propeller.

3. Simulation Results

3.1 Flow Circulation

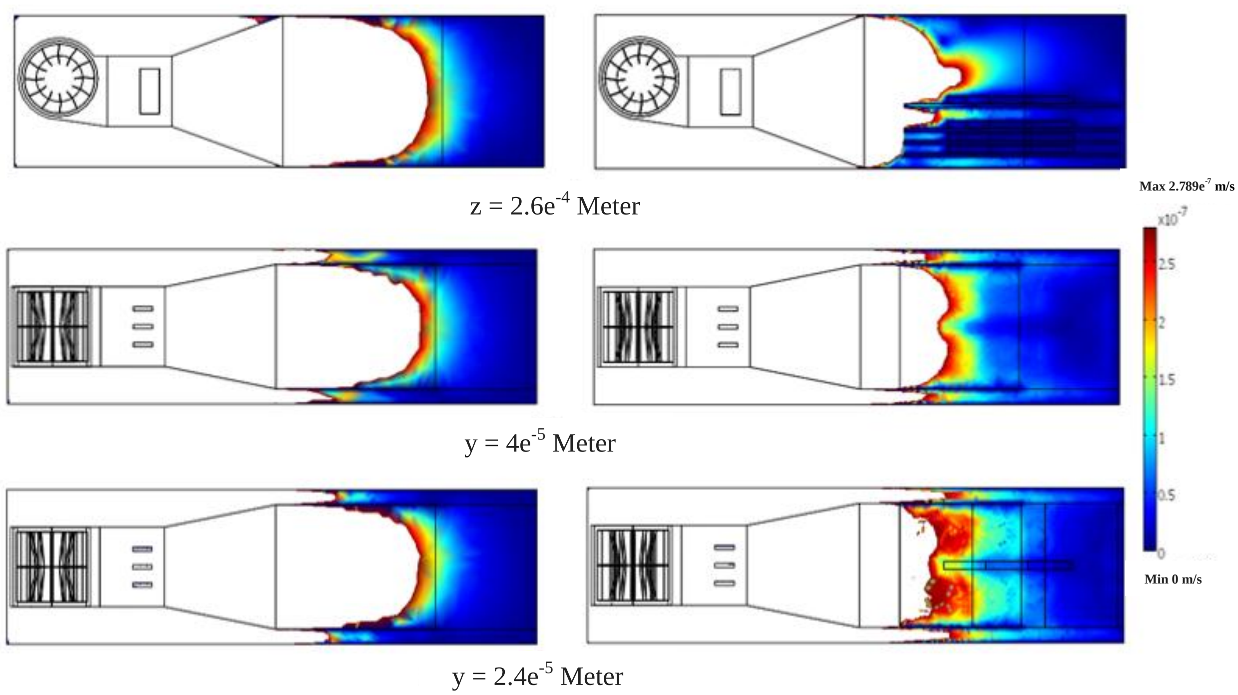
Three dimensional flow fields of hot air in cases of without tray and with 4-row trays were illustrated in Figure 2. From flow contour, air speed was high in the vicinity of propeller tip due to the angular momentum transport. Air was drawn by propeller from the sides (location 1) as indicated by velocity vectors (arrows) and forced out through the front (location 2). Afterwards, the high speed bulk flow continued to the outlets (location 3) where swirl air flow became discernible. Obviously, air speed close to the outlets decreased due to flow diversion and increase in flow area.



a) Without trays b) With 4-row trays

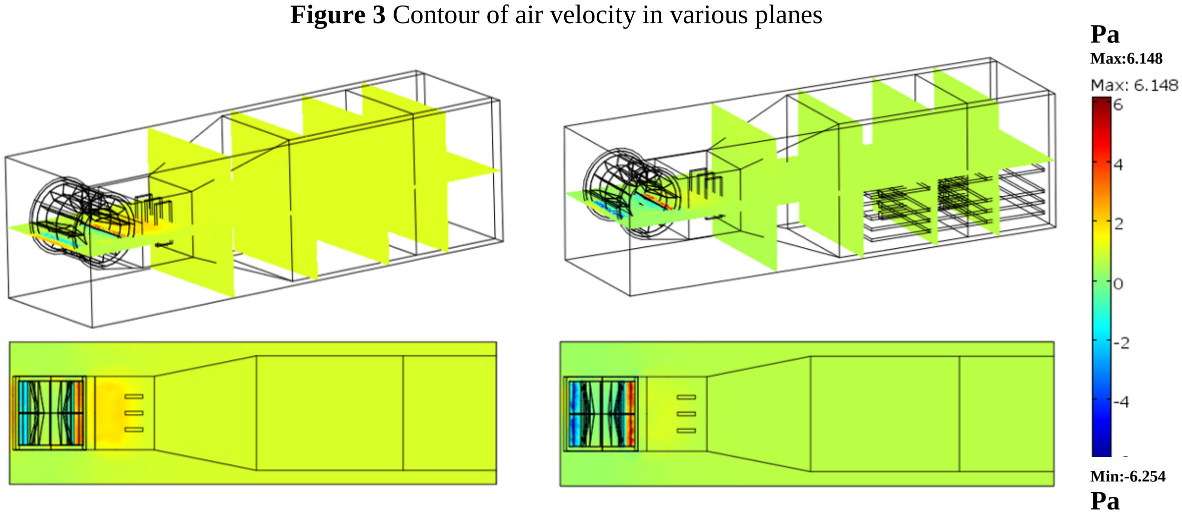
Figure 2 Hot air circulation i.e. velocity vectors and contours

Without trays, flow of air nearly became fully-developed at the straight end of the oven and air flew uniformly through the exit as shown in Figure 2a and 3a. However, the installation of trays introduced flow obstacles and prolonged flow development. From side view of Figure 3, since the bottom three trays were closer together as compared to that between the first and second trays, the flow resistances were higher in those vicinities and so the averaged velocities were lower. The air flow could not circulate downward through the impermeable walls of trays so it flew straight and split up at the side exits. However, from Figure 4, the existence of trays introduced the higher pressure drop and the air pressure distributions were rather uniform in both cases.



a) Without trays b) With 4-row trays

Figure 3 Contour of air velocity in various planes



a) Without trays b) With 4-row trays

Figure 4 Distribution of air pressure

Conclusion

The installation of trays within the oven affected the flow orientation and also reduced the flow speed between adjacent tray rows. A corresponding pressure drop was therefore larger than that in case of no trays. The simulation results provided essential information serving a better industrial oven design.

Acknowledgement

This research and the graduate's master thesis of King's Mongkut University of Technology North Bangkok had been partly supported by the Aeroflex, Company Limited and Miss Kulyarat Rosmod.

References

[1] Verboven, P. S. (1999). Computational fluid dynamics modelling and validation of the isothermal air flow in a forced convection oven. *Computational fluid dynamics modelling and validation of the isothermal air flow in a forced convection oven*, 41-53.

[2] Lijun Wang, D.-W. S. (2003). Recent developments in numerical modelling of heating and cooling processes in the food industry - a review. *Trends in Food Science & Technology*, 408-423.

[3] Illés, B. (2010). Measuring heat transfer coefficient in convection reflow ovens. *Measurement*, 1134-1141.

[4] Pieter Verboven *, N. S. (2000). Computational fluid dynamics modelling and validation of the temperature distribution in a forced convection oven. *Journal of Food Engineering 43*, 61-73.

[5] Sundaram Gunasekaran *, H.-W. Y. (2007). Effect of experimental parameters on temperature distribution during continuous and pulsed microwave heating. *Journal of Food Engineering*, 1452–1456.

[6] P NAVANEETHAKRISHNAN∗, P. S. (2007). Heat transfer and heating rate of food stuffs in commercial shop ovens. *Department of Mechanical Engineering, Kongu Engineering College, Perundurai 638 052*, 535–544.

[7] M.A. Delele a, *. A. (2009). Optimization of the humidification of cold stores by pressurized water atomizers based on a multiscale CFD model. *Journal of Food Engineering*, 228–239.

[8] Stamou, A. I. (2008). Improving the hydraulic efficiency of water process tanks using CFD models. *Chemical Engineering and Processing*, 1179–1189.

[9] A. Guardo a, *. M. (2009). A CFD approach to evaluate the influence of construction and operation parameters on the performance of Active Transparent Facades in Mediterranean climates . *Energy and Buildings*, 534–542.

[10] Zinedine Khatir a, ⇑. J. (2012). Computational fluid dynamics (CFD) investigation of air flow and temperature distribution in a small scale bread-baking oven. *Applied Energy*, 89–96.