Applied Thermal Engineering 137 (2018) 123–133

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Research Paper

Numerical study on the effect of different hole locations in the fan case on the thermal performance inside a gas oven range

Seong Hyun Park[a], Yang Ho Kim[c], Young Soo Kim[c], Yong Gap Park[b,*], Man Yeong Ha[a,*]

[a] School of Mechanical Engineering, Pusan National University, Jang Jeon 2-Dong, Geum Jeong Gu, Busan 609-735, Republic of Korea
[b] Rolls-Royce and Pusan National University Technology Centre in Thermal Management, Jang Jeon 2-Dong, Geum Jeong Gu, Busan 609-735, Republic of Korea
[c] Home Appliance and Air Solution Company, LG Electronics, Gaeumjeong-Dong, Seong San Gu, Changwon, Republic of Korea

HIGHLIGHTS

- A numerical study of a gas oven range was carried out using an actual 3D geometry and ANSYS FLUENT.
- A test geometry was developed by referencing a real product.
- The flow pattern and temperature distribution inside the oven cavity changed dramatically.
- Thermal performance was evaluated based on the average temperature and temperature uniformity.

ARTICLE INFO

Keywords:
Computational Fluid Dynamics (CFD)
Gas oven range
Thermal performance

ABSTRACT

This paper discusses the effect of the hole location in the fan case on the thermal performance of a gas oven range. A computational fluid dynamics (CFD) study was carried out in ANSYS FLUENT. A DO model was used to include the effect of thermal radiation in the oven cavity. A test geometry was developed by referencing a real product, including the oven cavity, external walls, fan cases, fans, and burners. The simulation was validated with experimental data and showed that the maximum difference in temperature is 2.5%, while the difference in average temperature is 0.44%. A total of 15 cases were examined using different hole locations in the fan case. The direction of the velocity vector at the holes was changed by the different hole locations, and the flow pattern and temperature distribution inside the oven cavity also changed dramatically as a result. The thermal performance was evaluated based on the average temperature and temperature uniformity inside the oven cavity.

1. Introduction

Domestic ovens use electrical coil heating and gaseous fuel to provide thermal energy to an enclosed cavity [1]. Ovens heat food to cook it by conduction, convection, and radiation. These three heat transfer modes should be included when modeling the temperature and velocity fields in domestic ovens. Radiation is often predominant at low air speeds, while convection is much more important at higher air speeds [2]. Computational fluid dynamics (CFD) is useful for predicting the temperature and velocity fields in the oven cavity while considering all three heat transfer modes. Many researchers have developed CFD methods to analyze the flow patterns and temperature fields.

Earlier studies conducted 2D CFD simulations because 3D calculation is very computationally expensive for calculating the temperature and velocity fields in an oven [3–6]. Wong et al. [3] developed a 2D CFD modeling method for a continuous baking process using sliding mesh techniques and a segregated unsteady state solver. They assumed that the burners are circular object with a fixed wall temperature. Therdthai et al. [4] established a two-axis CFD model and varied several oven operating parameters, including the heat supply, fan volume, and heat distribution in the oven.

With rapid advances in parallel computing technologies, numerous 3D CFD studies have been carried out [7–16]. Numerical simulations have been conducted using commercial code and have considered radiation models involving both steady and unsteady calculation to evaluate and improve the thermal performance of commercial prototypes. Some studies have considered different geometries change to increase the thermal performance of products.

Mistry et al. [7] developed a three-dimensional transient CFD model to simulate natural convection heat transfer in an oven for two different cooking cycles. Their model of an electric oven included a three-dimensional, unsteady, natural convective flow-thermal field coupled

* Corresponding authors.
E-mail addresses: pyg777@pusan.ac.kr (Y.G. Park), myha@pusan.ac.kr (M.Y. Ha).

https://doi.org/10.1016/j.applthermaleng.2018.03.087
Received 30 December 2016; Received in revised form 30 August 2017; Accepted 26 March 2018
Available online 27 March 2018
1359-4311/

Nomenclature

ρ	air density (kg/m^3)
$\vec{v}$	velocity vector (m/s)
p	pressure (Pa)
$\vec{g}$	gravitational acceleration (m/s^2)
μ	dynamic viscosity (kg/m s)
$\bar{\bar{\tau}}$	stress tensor (Pa)
E	energy (J/kg)
k	thermal conductivity (W/m K)
h	sensible enthalpy (J/kg)
σ	Stefan–Boltzmann constant ($W/m^2 K^4$)
a	absorption coefficient
T	temperature
$T_{reference}$	reference temperature
θ	dimensionless temperature
θ_{cal}	calculated dimensionless temperature
θ_{exp}	measured dimensionless temperature
$\Delta\theta$	dimensionless temperature discrepancy
$\theta_{vol,avg}$	volume-averaged temperature inside the oven cavity
$V_{chamber}$	volume of oven cavity
RMS_{vol}	temperature uniformity inside the oven cavity

with radiative heat transfer. They applied a suction pressure at the top vent to obtain a physically reasonable flow pattern through the vent openings. The results showed good agreement with the experimental results. They also presented a comparative analysis of the thermal fields inside the oven for bake and broil cycles. Boulet et al. [8] developed a 3D CFD model to describe the transient heat transfer in a pilot plant oven with radiation coupled with a mixed convection regime. The radiation model was simulated by a surface to surface (S2S) model, whereas a k-e realizable model was used for turbulence. The CFD model was validated with experimental data for transient temperature.

Chhanwal et al. [9] conducted a CFD calculation that considered three different radiation models: the discrete transfer radiation model (DTRM), discrete ordinates (DO) model, and S2S model. The simulation results showed good agreement with the experimental results. They simulated a bread-baking process with bread in the center of the oven to investigate the profiles of temperature and starch gelatinization of the crust and crumb of the product. The bread temperature was validated with experimental measurements. Rek et al. [10] numerically simulated the heat transfer in a new generation of ovens by changing the heater shape, fan cover shape, heater temperature, and ventilation system flow rate. They carried out steady-state 3D CFD calculation with a DO model for radiation. They determined the influence of changes in the geometry and boundary conditions on the velocity and temperature distribution inside the oven cavity.

Smolka et al. [11,12] experimentally validated a 3D CFD analysis of thermal and flow fields in a laboratory drying oven with varying rotational speed of the device fan, the effectiveness of the distribution gaps, and the rate of heat generated in the oven. The temperature uniformity was improved by adjusting the device configurations. An experiment was carried out to look at the thermal performance of the improved design, which showed good agreement with the CFD results. They concluded that the proposed CFD methodology can be applied to designing an oven.

Many researchers have conducted CFD studies with various parameters, including oven geometries, operating conditions, and boundary conditions. However, there are few studies on the effect of the location of holes in the fan case, which could influence the temperature value and temperature uniformity of air inside the oven cavity. Therefore, a CFD study was conducted with different locations of four holes on sides of the fan case to quantitatively estimate the effect on the thermal performance. The calculations were conducted with full factorial design. The thermal performance was evaluated based on the average temperature and temperature uniformity. The simulations were solved using the commercial code ANSYS FLUENT 13.0, and the numerical results were validated with experimental results.

2. Numerical methodology

A three-dimensional geometry was created, including the oven cavity, fan case, and burner. The volume mesh was generated using ANSYS workbench 13.0. The structured cut-cell mesh was distributed in the computational domain to generate a fine mesh around the small holes. The total number of mesh faces was 18,174,889 with 5,950,913 cells. A segregated steady state solver was used to solve the governing equations of momentum, mass, energy conservation, and the turbulence kinetic energy equation. For the turbulent flow, a k-ε model was used with the standard wall function near the wall boundaries. The radiation heat transfer was taken into account using the DO model in FLUENT, which solves the radiative transfer equation for a finite number of discrete solid angles, each associated with a vector direction. All of the governing equations solved in the current study are shown below:

Continuity equation

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\vec{v}) = 0 \tag{1}$$

Momentum conservation equation

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla\cdot(\rho\vec{v}\vec{v}) = -\nabla p + \nabla\cdot(\bar{\bar{\tau}}) + \rho\vec{g} \tag{2}$$

$$\bar{\bar{\tau}} = \mu[(\nabla\vec{v} + \nabla\vec{v}^{T}) - \frac{2}{3}\nabla\cdot\vec{v}I] \tag{3}$$

Energy equation

$$\frac{\partial}{\partial t}(\rho E) + \nabla\cdot(\vec{v}(\rho E + p)) = \nabla\cdot kT \tag{4}$$

$$E = h - \frac{p}{\rho} + \frac{v^2}{2} \tag{5}$$

Radiation model equation (DO)

$$\nabla\cdot(I(\vec{r},\vec{s})\vec{s}) + (a + \sigma_s)I(\vec{r},\vec{s}) = an^2\frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\vec{r},\vec{s}')\Phi(\vec{s},\vec{s}')d\Omega' \tag{6}$$

where ρ is the density, $\vec{v}$ is the velocity vector, μ is the viscosity, $\bar{\bar{\tau}}$ is the stress tensor, E is the energy, p is the pressure, h is the enthalpy, $I(\vec{r},\vec{s})$ is the radiation intensity, a is the absorption coefficient, $\vec{r}$ is the position vector, $\vec{s}$ is the direction vector, $\vec{s}'$ is the scattering direction vector, and $\Phi(\vec{r},\vec{s})$ is the phase function. Because the air temperature inside the oven cavity varies widely, variation of the thermophysical properties of air such as density, viscosity, thermal conductivity, and specific heat were considered as polynomial functions of temperature. To create the polynomial functions, the data were extracted with reference to the NIST database [17]. All data were fitted with a polynomial equation. The oven walls were made from stainless steel, and the external walls were made from wool. The emissivity of stainless steel and wool were supplied by the manufacturers. Table 1 presents the thermophysical properties used, including the emissivity of each solid.

The 3D CFD geometry was developed with reference to a real product geometry taken from the manufacturer. This gas oven consists of an oven cavity, outer wall, fan case, fan, insulation wall, and burner, as shown in Fig. 1. The oven cavity includes a glass wall, inner stainless wall, fan case, and burner. The thermal and flow fields were calculated for an oven cavity by varying the fan geometry.