Iranica Journal of Energy & Environment 3 (3): 241-245, 2012
ISSN 2079-2115
IJEE an Official Peer Reviewed Journal of Babol Noshirvani University of Technology
DOI: 10.5829/idosi.ijee.2012.03.03.0548

BUT

CFD Application on Food Industry; Energy Saving on the Bread Oven

[1]*Ali Motamedzadegan and* [2]*Shahla Khoddam*

[1]Sari Agricultural Science and Neutral Resources University, P.O. Box 578, Sari, Iran
[2]Faculty of Food industry, Sari Agricultural Science and Neutral Resources University, Sari, Iran

(Received: February 26, 2012; Accepted: May 28, 2012)

Abstract: In this paper, application of computational fluid dynamics simulation of the cooking process of bread and energy saving inside an oven is presented. A two dimensional CFD model is applied to investigate the hot air distribution and bread cooking conditions inside the stove. Results are shown as velocity contours, turbulence intensity, static temperature and pressure distribution. It was found that the turbulence intensity was the highest at part of the stove, which means which part of bread would cook the fastest as it receives larger quantity of energy compared to other parts. The arrangement of the stove is discussed and some improvement offered to find a desire temperature distribution that leads to a suitable cooking process. In addition, the current results can be useful for next improvements on the existing stove for energy saving.

Key words: CFD; Cooking Process; Air Flow Pattern; Numerical Method

INTRODUCTION

Computational fluid dynamics (CFD) is a simulation tool which uses numerical methods and algorithms to solve and analyze problems associated with fluid flow [1]. CFD was originally developed from the pioneering accomplishments of enthusiasts such as Richardson [2] and Courant *et al*. [3], have obtained insight into fluid motion investigated the development of influential numerical techniques [4]. Recent adoption of CFD has been implemented as a method of choice to investigate the progression of the high costs and time consumption associated with experimentation to produce an efficient method of choice of experiment. CFD is matured into a powerful tool in many industries such as food technology. The demonstration and powerfulness of CFD are presented by number of researchers as cited in ligatures [5-8]. In these studies, CFD was used as a useful tool for calculation of the fluid flow, heat and mass transfer parameters [5, 7]. Furthermore, CFD has been used for modeling the processes involved in food processing and products especially in heating stoves [4].

Numerical and CFD methods have been used to simulate a bakery pilot oven in order to investigate three dimensional heat transfer mechanisms combined with the effects of turbulency introduced by Boulet *et al*. [9]. Hitesh *et al*. modeled 3-D CFD to simulate free convective heat flow in a thermal field coupled with radiative heat transfer inside an electric stove [10]. The aim of present was to investigate heat flow pattern in bread cooking oven. In addition more knowledge about air flow pattern through oven and more understanding about bread condition cooking phenomena were obtained. This leads to design a new stove or to develop on existing ovens so that a high quality of food product is obtained [11]. The purpose of the CFD simulation in this paper is to understand the air flow profile within the bread stove and hence the heat distribution.

MATERIAL AND METHOD

General Governing Equations: The governing equations of fluid flow and heat transfer can be considered as mathematical model for the laws of conservation of fluid flow and often referred to as Naviere-Stokes equations. When these laws are applied to a fluid flow, these conservation laws relate the rate of change of a desired fluid property to external forces which are considered as follows:

- The law of conservation of mass (continuity), which states the mass flows into a fluid element, exactly balanced with outlet mass flow [1].

Correspond Author: Shahla Khoddam, Sari Agricultural Science and Neutral Resources University, P.O. Box 578, Sari, Iran.

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \quad (1)$$

- The law of conservation of momentum (Newton's second law of motion), which states that the sum of the external forces acting on a fluid particle is equal to the rate of change of linear momentum. The momentum equation is stated as follows:

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=\frac{\partial}{\partial x_j}\left[-p\delta ij+\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_i}{\partial x_i}\right)\right]+\rho g_i \quad (2)$$

- The law of conservation of energy (the first law of thermodynamics), which states that the rate of change of energy of a fluid particle is equal to heat transfer and the work done by the system (on particles). The resulted equation is written as follows:

$$\frac{\partial}{\partial t}(\rho C_a T)+\frac{\partial}{\partial x_j}(\rho u_j C_a T)-\frac{\partial}{\partial x_j}\left(\lambda\frac{\partial T}{\partial x_j}\right)=S_T \quad (3)$$

Turbulence Equation: The present problem includes a fan which generates convective heat transfer as a dominated heat sources. Cause of these circumstances and with attention to high speed of the fan, the flow regime through the oven assumed as turbulent. As the flow is turbulent, additional equations (standard K-ε model) for turbulent viscosity are also solved [12].

CFD Description and Implementation

Fluent Software: FLUENT Inc. offers three software packages inside the CFD framework that are appropriate for food engineer's modeling requirements [1]. The three packages are FLUENT® (general purpose with metaphysics capabilities), FIDAP® (modeling complex physics) and POLYFLOW ® (polymer modeling). FLUENT® Inc. is presently one of the best leading suppliers of CFD software in the world. The most interesting features of the FLUENT® software include models for heat exchangers, discrete phase models for multiphase flows, food industry such as ventilation, drying, sterilization, refrigeration, cold display and storage and mixing numerous high quality reaction models and the phase change model which tracks the melting and freezing points in the bulk fluid [1]. In this paper, the commercial CFD software Fluent (FLUENT Inc.) was used to simulate the flow field and heat transfer through the stove including breads.

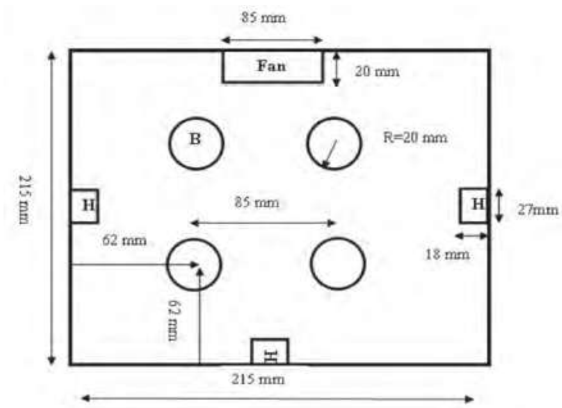

Fig. 1: Geometrical overview of bread stove

Solution Procedure by CFD: In order to solve for a flow field a general CFD code must take the mathematical statements inputted. The structured CFD model incorporated into a suitable arrangement and solved for the specified boundary conditions [1]. Iterative methods are commonly used by CFD codes to solve a whole set of discredited equations so that they can be applied to a single dependent variable.

Simulation Procedure

Geometry and Computational Domain: In any modeling procedure the first step is to define the primary physics of the modeling system. A physical model stands for the scientific features of a system which is modeled. In current study, the two dimensional model is used to view the flow field from the side and above. The dimensions and location of the elements like fan, heating element and 4 breads are shown in Figure 1. The fan located on the wall for the goal of generating forced convection through the oven.

Mesh Generation: Second step of the CFD analysis of bread cooking model within a stove is designed for the system geometry and its discretization into a computational grid of finite volumes. The mesh or grid quality plays an important role in the accuracy and stability of the CFD numerical computation [1]. For a 2D geometry, the mesh generator partitions the sub-domains into triangular or quadrilateral mesh elements which contain nodes. If the boundary is curved (like the bread section area), these elements represent only an approximation of the original geometry. Each of the element nodes has specific thermal properties of apparent density, thermal conductivity, specific heat capacity, etc. The generated quadrilateral meshes are shown in Figure 2.

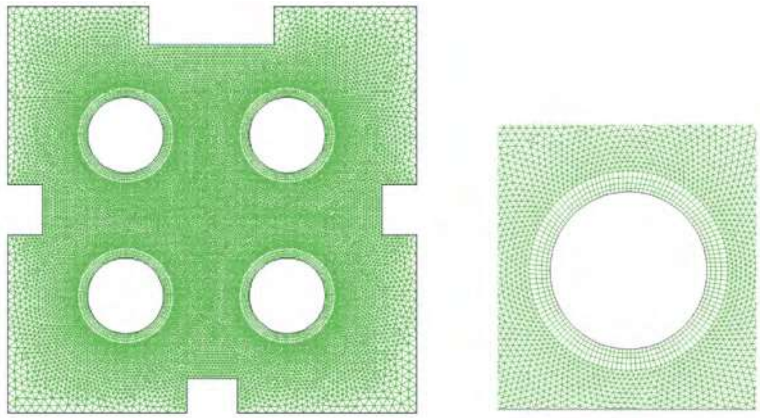

Fig. 2: Mesh Generation a) Whole Computational Domain and b) Enlargement of Bread's mesh

Table 1: Boundary Conditions

Fan Velocity	12 m/s (Normal to boundary)
Total Heat Generation	1300 W/m^3
Initial Bread Temperature	Ambient
Wall Temperature	Ambient

Boundary Conditions: Boundary conditions are the physical assumptions that implement on the boundaries of the modeling cases. It means that CFD methods need these conditions to be solved. The relevant boundary conditions with respect to oven condition in the model are supplied depending on the required output of the model. Therefore, in present case, the boundary conditions are as following. The fan located at the top boundary of computational domain (Figure 1); that stands for the fan used in the real stove. This element is modeled as a velocity inlet source which means that air enters the oven from the top part. The heating elements are set constant at wall temperature together with the heat generation source. The rest of the domain are modeled as wall. The values of the boundary conditions are presented in Table 1.

RESULTS AND DISCUSSIONS

In this section results are presented in two categories. Flow field and thermal heat transfer are the important parts of cooking stove. The CFD investigation is stated as follows.

Flow Field: The velocity contour and velocity vector of the whole stove domain are shown in Figures 3 and 4. The velocity of the hot air on the top of the bread cross section was higher than the bottom of the breads.

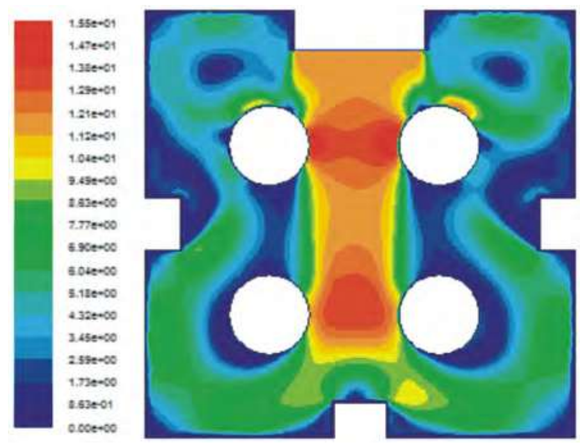

Fig. 3: Velocity Contours

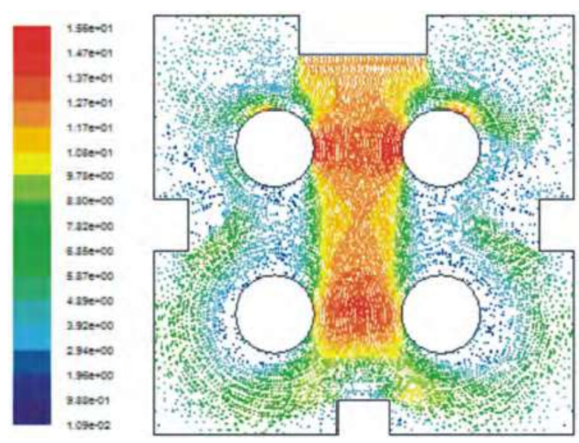

Fig. 4: Velocity Vectors

The velocity on the top was higher since the heat was contributed from both forced convection from the fan on one side and natural convection from the heating elements on the other sides.

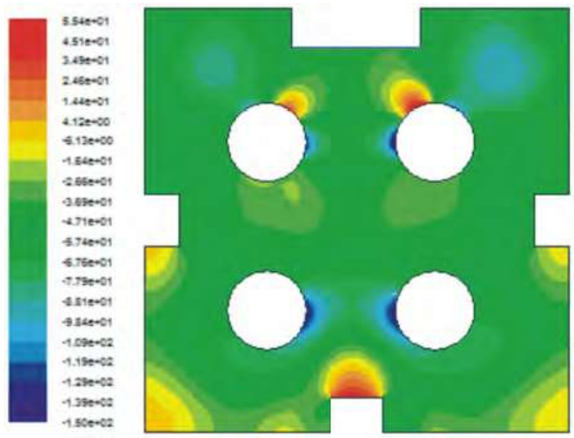

Fig. 5: Contour of Static Pressure

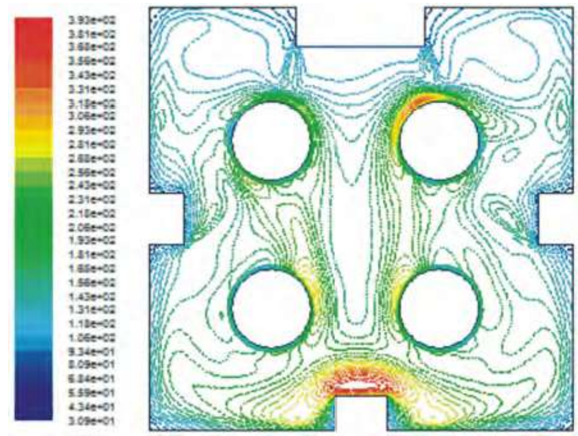

Fig. 7: Contour of Static Temperature

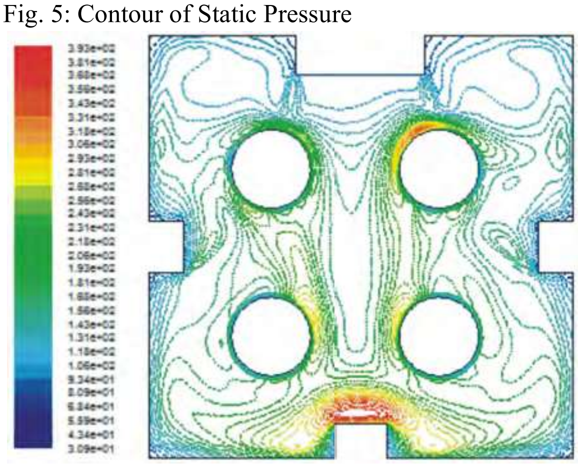

Fig. 6: Contour of turbulent intensity

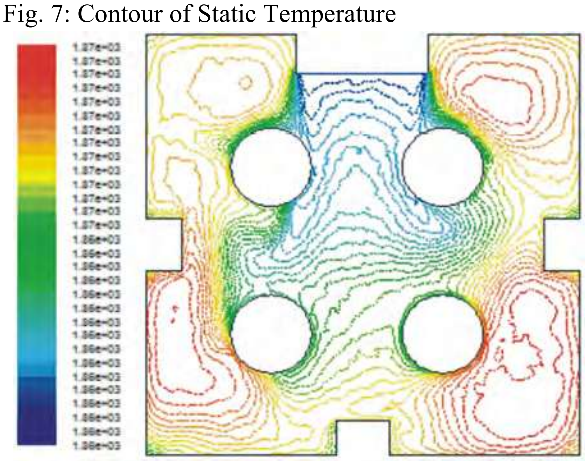

Fig. 8: Contour of Enthalpy

The contour of static pressure is shown in Figure 5. It was observed that the high pressure is occurred at the top of two upon bread and at the bottom heating element.

Figure 6 illustrates the result of turbulent intensity through the oven. Based on the turbulence results, large amount energy is concentrated at the top of the bread and the middle of the two bottom breads. In addition, presentation of results suggests that the sides of the bread which are facing the middle section of the bread stove will be cooked fast; that was due to the high concentration of turbulence kinetic energy in this region.

Heat Transfer: Contour of static temperature is shown in Figure 7. It was observed that the high temperature is happened near by the heating elements at the left, right and bottom sides. Because of the forced convection created by fan velocity, the heat from heating elements moves and rotates on the other sides of the oven. As a consequence, the regions near the top breads catch the heat from the sources. Moreover, it can be seen that the temperature is not homogenous in the whole regions of the stove. One of the options for setting a better mixing is that the displacement of the fan's stove. It is desired to place the fan near by the heating element and thus the forced convection and natural convection make the better blend of heat transfer through the oven.

The results of the enthalpy calculation show similar trends of static temperature (Figure 8).

CONCLUSION

In this paper, a two-dimensional hot air flow CFD model within a bread stove is presented. The purpose of the CFD simulation is to determine the conditions through the oven during the food preparation procedure in terms of heat distribution. The results showed that the CFD is a reliable method for the investigation in food processing industries. In addition, the results showed that the air flow pattern inside the stove with the mentioned configuration may not the best arrangements and it is possible the flow pattern to be improved. One of the suggested procedure is change of the fan position inside the oven.

On the other hands the fan position can be changed and a desired temperature distribution obtained. In findings of the best fan position can be simulated by CFD model in the future investigations.

REFERENCES

1. Norton, T. and D.W. Sun, 2006. Computational fluid dynamics (CFD) e an effective and efficient design and analysis tool for the food industry: A review, Trends in Food Science and Technology, 17: 600-620.
2. Richardson, L.F., 1910. The approximate arithmetical solution by finite differences of physical problems involving differential equations, with an application to the stresses in a masonry dam, Philosophical Transactions of the Royal Society of London A, 210: 307-357.
3. Courant, R., K. Friedrichs and H. Lewy, 1928. Die partiellen differenzengleichungen der mathematischen Physik, Mathematische Annalen (Historical Archive), 100: 32-74.
4. Shang, J.S., 2004. Three decades of accomplishments in computational fluid dynamics, Progress in Aerospace Sciences, 40: 173-197.
5. Sajjadi, B., M.K. Moraveji and R. Davarnejad, 2011. Investigation of Temperature and Influent Load on Nitrifying Treatment of Using Wastewater CFD, Iranica Journal of Energy and Environment, 2: 08-17.
6. Rafiee, S., A. Keyhani and A. Mohammadi, 2008. Soybean Seeds Mass Transfer Simulation during Drying Using Finite Element Method, World Applied Sciences Journal, 4: 284-288.
7. Xia, B. and D.W. Sun, 2002. Applications of Computational Fluid Dynamics (CFD) In the Food Industry: A Review, Computers and Electronics in Agriculture, 34: 5-24.
8. Sebti, S.S., S.H. Khalilarya, I. Mirzaee, S.F. Hosseinizadeh, S. Kashani and M. Abdollahzadeh, 2011. A Numerical Investigation of Solidification in Horizontal Concentric Annuli Filled with Nano-Enhanced Phase Change Material (NEPCM), World Applied Sciences Journal, 13: 09-15.
9. Boulet, M., M. Bernard, M. Dostie and C. Moresoli, 2010. CFD Modeling of Heat Transfer and Flow Field in a Bakery Pilot Oven, Journal of Food Engineering, 97: 393-402.
10. Mistry, H., G. Subbu, S. Dey, P. Bishnori and J.L. Castillo, 2006. Modeling of Transient Natural Convection Heat Transfer in Electric Ovens, Applied Thermal Engineering, 26: 2448-2456.
11. Saxena, D.C., P. Haridas Rao and K.S.M. S Rao, 1995. Analysis of Modes of Heat Transfer in a Tandoor Oven, Journal of Food Engineering, 26: 209-217.
12. Mondal, A. and A.K. Datta, 2010. Two-dimensional CFD modeling and simulation of crustless bread baking process, Journal of Food Engineering, 99: 166-174.