Middle-East Journal of Scientific Research 13 (8): 1095-1100, 2013
ISSN 1990-9233

DOI: 10.5829/idosi.mejsr.2013.13.8.548

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

Shahla Khoddam

Faculty of Food Industry, Sari Agricultural Science and Neutral Resources University, Sari, Iran

Abstract: In this paper the 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 the bread cooking conditions inside the stove. The Results are shown as velocity contours, turbulence intensity, static temperature and pressure distribution. It is found that the turbulence intensity is highest at the which part of the stove and it would mean 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 I offered to find a better temperature distribution that leads to a better cooking process. Also the current results can be useful for next improvements on the existing stove for energy saving issue.

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 that involve fluid flow [1]. CFD was originally developed from the pioneering accomplishments of enthusiasts such as Richardson [2] and Courant *et al*. [3], who in their actions to obtain insight into fluid motion instigated the development of influential numerical techniques that have advanced the numerical description of all types of fluid flow [4]. The recent adoption of CFD has been both inevitable and progressive, as the high costs and time consumption associated with experimentation has often precluded the desire to produce efficient in-depth results. CFD is now maturing into a powerful and enveloping tool in many industries such as food technology with each solution representing a rich drapery of mathematical physics, numerical methods, user interfaces. Also CFD has been used by many researchers to modeling the processes involved in food products especially stoves [4].

The CFD method has been used to simulate a bakery pilot oven in order to study three heat transfer mechanisms combined with the effects of turbulence by Boulet *et al*. [5]. Hitesh et al modeled 3-D CFD to simulate free convective flow thermal field coupled with radiation heat transfer inside an electric stove [6]. The main aim of the researches on bread cooking industry is to have more knowledge about air flow pattern through oven and thus have more understanding about bread condition cooking phenomena. This leads to design a new stove or to develop on existing ovens so that a better quality of food is obtained [7]. The main purpose of the CFD simulation of the present paper is to understand the air flow profile within the bread stove and hence the heat distribution.

MATERIALS AND METHODS

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

- The law of conservation of mass (continuity), which states that the mass flows entering a fluid element, should balance exactly with those leaving [1].

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_j) = 0 \qquad (1)$$

- 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 its rate of change of linear momentum.

Corresponding Author: Shahla Khoddam, Student of Faculty of Food industry, P.O. Box: 578, Sari, Iran.

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

- 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 the heat addition and the work done on the particle. The resulting equations can be written as:

$$\frac{\partial}{\partial t}(\rho C_{\mathrm{a}} T)+\frac{\partial}{\partial x_j}(\rho u_j C_{\mathrm{a}} T)-\frac{\partial}{\partial x_j}\left(\lambda\frac{\partial T}{\partial x_j}\right)=s_{\mathrm{T}} \quad (3)$$

Turbulence Equation: The present problem includes a fan which makes the convection heat transfer as a dominated mechanism. Cause of these circumstances and with attention to the high speed of the fan, the regime of flow through the oven can be assumed turbulence. As the problem also involves turbulence, the additional two equations (standard K--ε model) for turbulent viscosity a length scales are also solved [8].

CFD Description and Implementation

Fluent Software: FLUENT Inc. offers three software packages inside the CFD framework that are appropriate for the food engineer's modeling needs [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 in the bulk fluid [1]. In this paper, the commercial CFD software Fluent (FLUENT Inc.) is used to simulate the flow field and heat transfer through the stove which includes breads.

Solution Procedure by CFD: In order to solve for a flow field a general CFD code must take the mathematical statements inputted by the user, structure them into a suitable arrangement and solve them 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 may 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.

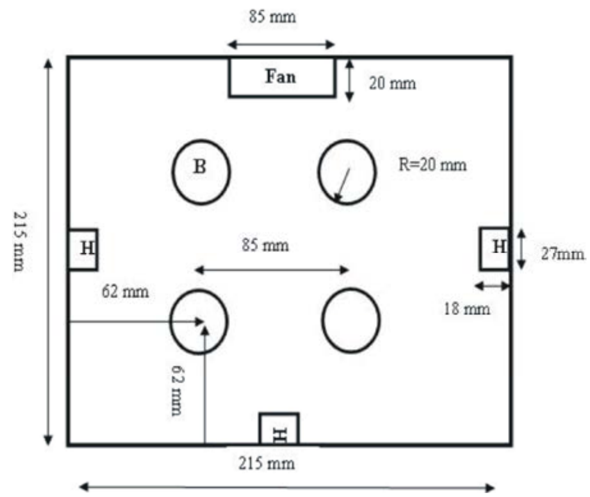

Fig. 1: Geometrical overview of bread stove

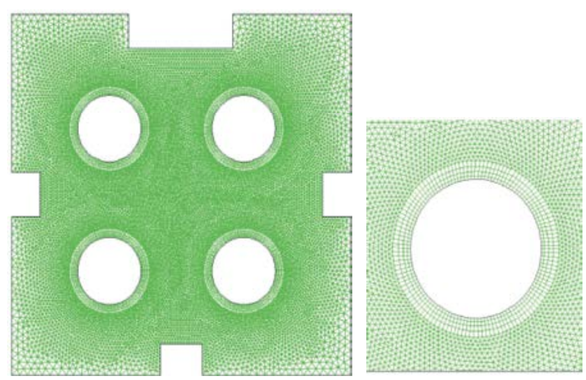

Fig. 2: Mesh Generation

a) Whole Computational Domain and b) Enlargement of Bread's mesh

Mesh Generation: Second step of the CFD analysis of bread cooking modeling within an stove is the design of 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 a CFD numerical computation [1]. For a 2D

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

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 by Figure 2.

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 solve a problem. The relevant boundary conditions with respect to oven condition in the model are supplied depending on the required output of the model. Therefore in the present case, the boundary conditions are as following. The fan was located at the top boundary of computational domain (Figure 1) is stands for the fan that used in the real stove. This element is modeled as a velocity inlet source which means that air enters the oven from this part. The heating elements are set as the constant wall temperature together with the heat generation source. The rest parts of the domain are modeled as wall. The values of the boundary conditions are presented in Table 1.

RESULTS AND DISCUSSIONS

In this section the results are presented on the two categories. Flow field and thermal heat transfer are the important parts of cooking stove CFD investigation that mentioned as following.

Flow Field: The velocity contour and velocity vector of the whole stove domain is shown via Figures 3 and 4. The velocity of the hot air is higher near the top of the two upon bread cross section and near the bottom of the rest of breads. The velocity is higher there as it has the contribution of both forced convection from the fan on one side and natural convection from the heating elements on the other sides.

Contour of static pressure is shown in Figure 3. It can be seen 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 one bread and at the middle of two bottom breads. Also this Figure suggests that the sides of the bread which are facing the middle section of the bread stove will be cooked faster due to the high concentration of turbulence kinetic energy in this region.

Heat Transfer: Contour of static temperature is shown in Figure 7. It is obvious that the high temperature is happened at the near of heating element at the left, right and bottom sides. Cause of the forced convection of fan velocity, the heat from heating elements moves and

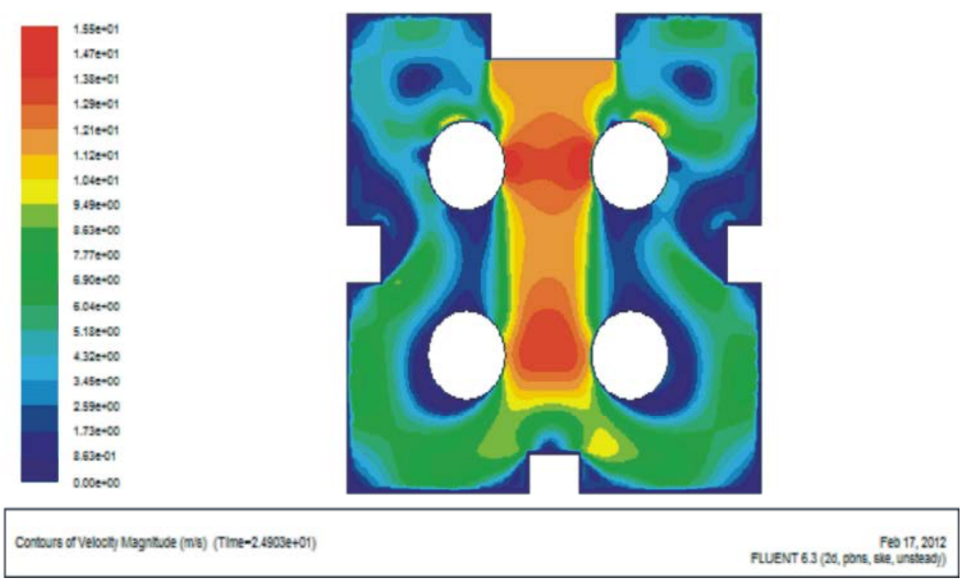

Fig. 3: Contour of Velocity Contour

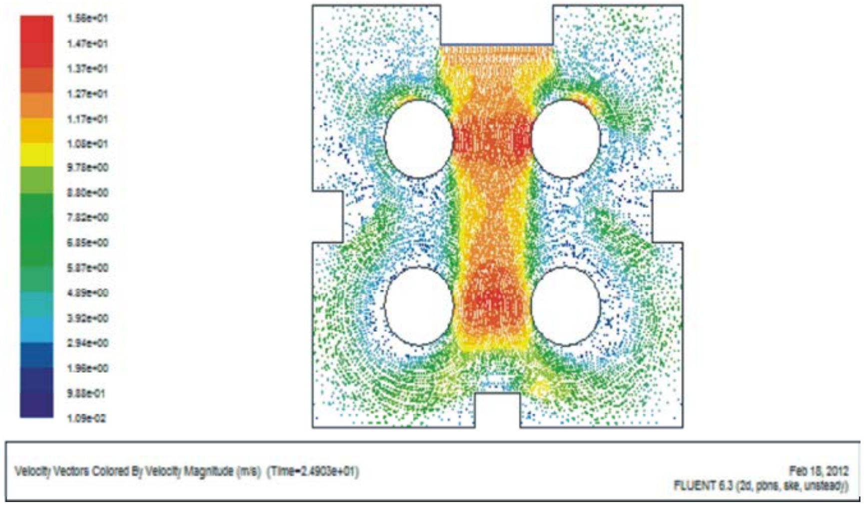

Fig. 4: Velocity Vectors

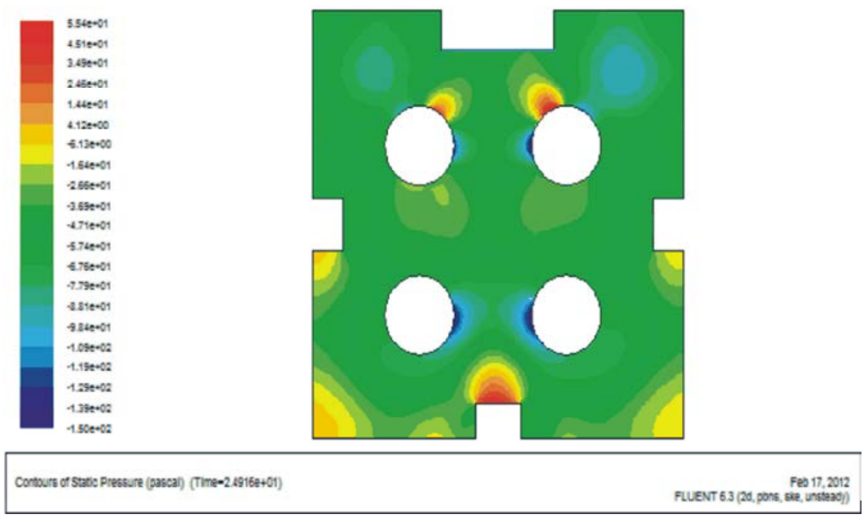

Fig. 5: Contour of Static Pressure

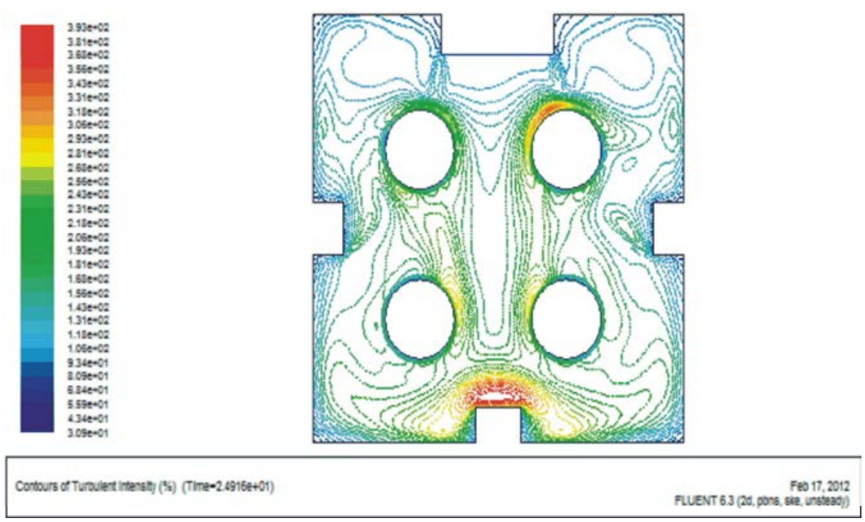

Fig. 6: Contour of turbulent intensity

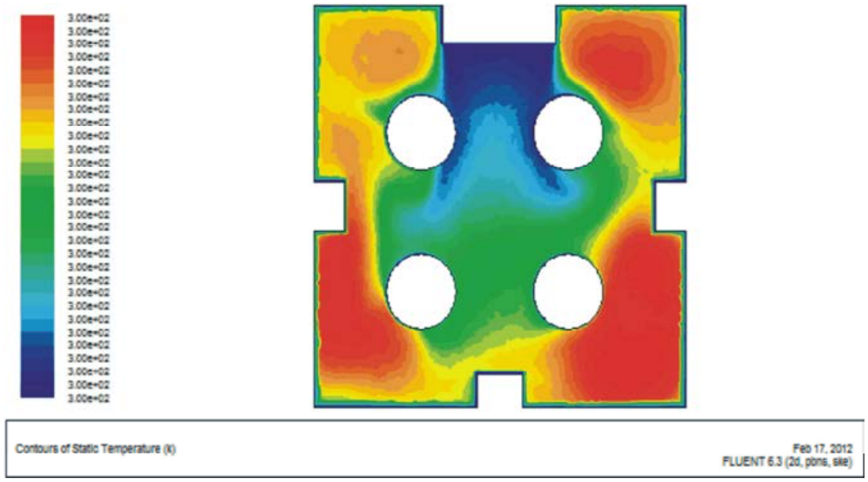

Fig. 7: Contour of Static Temperature

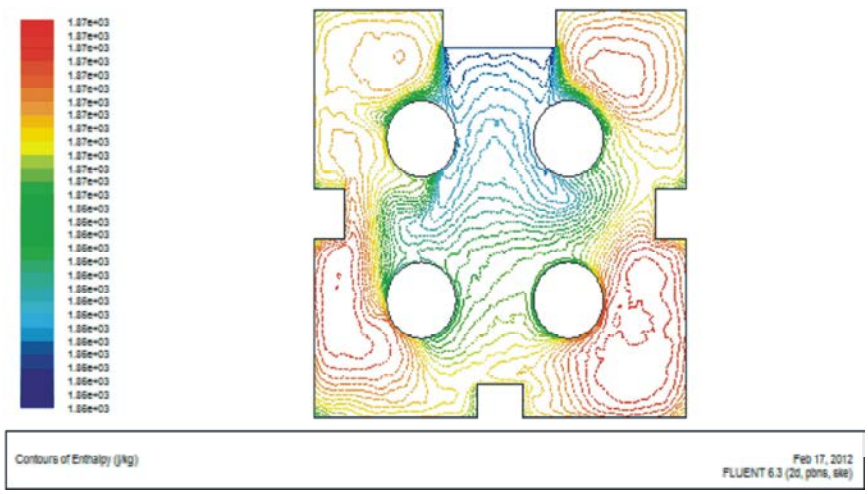

Fig. 8: Contour of Enthalpy

rotates on the other sides of the oven. As a consequence, the regions near the top breads catch the heat from the sources. Also it can be seen that the temperature is not homogenous on the whole regions of the stove. One of the options for setting a better mixing is that the place of fan's stove can be changed. It can be placed near 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 a similar trends of static temperature (Figure 8).

CONCLUSION

In this paper, a two-dimensional hot air flow CFD modeling 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 show that the CFD is a capable method for investigation of process in food industry. Also the results show that the air flow pattern inside the stove with the mentioned configuration is not the best arrangement and can be improved. One of the offers is the change of the fan position thorough the oven. On the other hands the fan position can be changed and a better temperature distribution happened. The finding of best position of fan can be called as a next part of this CFD simulation on 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 Technol., 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 Sci., 40: 173-197.
5. 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.
6. 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.
7. Saxena, D.C., P. Haridas Rao and K.S.M.S. Raghava Rao, 1995. Analysis of Modes of Heat Transfer in a Tandoor Oven. Journal of Food Engineering, 26: 209-217.
8. 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.