HEFAT2012
9th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
16 – 18 July 2012
Malta

Developing a New Design to Improve Radiation Heat Transfer in an Oven-Like Cavity

Temel O., Isik O., Onbasioglu S.U.
Department of Mechanical Engineering,
Istanbul Technical University,
Istanbul, 34437
Turkey
E-mail: otemel@itu.edu.tr, isikozg@itu.edu.tr, onbasiogl@itu.edu.tr

ABSTRACT

This study concerning a built-in oven is intended to reduce the energy consumption by using reflective surfaces which increase the effect of radiation. Therefore, positions of reflective surfaces that are thought to increase the effect of radiation were decided primarily. The Monte Carlo method was used in order to calculate shape factors belonging to the surfaces placed in different positions.

On the framework of investigating the effects of radiation, CFD analysis was carried out only for the cases without brick, which necessitates the usage of models valid for participating mediums. Thus, radiative heat transfer is modeled by the means of S2S model.

By changing the positions of reflective surfaces in the cavity, energy consumption and the duration of the cooking were measured. In designing the minimum energy consuming model, the value of the view factors had also taken into account. Indeed, the decided model with the least energy consumption had surfaces with shape factors greater than the others.

NOMENCLATURE

T — $temperature\ [K]$
y^+ — $dimensionless\ wall\ distance\ \left(y^+ = \frac{y u_\tau}{\upsilon}\right)$
u_τ — $frictional\ velocity\ [ms^{-1}]$
υ — kinematic viscosity of air $[m^2 s^{-1}]$
TI — $turbulence\ intensity\ [\%]$
d_h — $hydraulic\ diameter\ [m]$
k — $turbulence\ kinetic\ energy\ [m^2 s^{-2}]$
ε — $turbulence\ dissipation\ rate\ [m^2 s^{-3}]$
SWF — $standart\ wall\ functions$
EWF — $enhanced\ wall\ functions$
$S2S$ — $surface\ to\ surface\ model$
CFL — $\Delta t \sum_{i=1}^{n} \frac{u_i}{\Delta x_i}$
Δt — $step\ size\ in\ time\ [s]$
Δx_i — $step\ size\ in\ spatial\ directions\ [m]$
u_i — $velocity\ in\ any\ spatial\ directions\ [ms^{-1}]$
g — $gravitational\ acceralation\ [ms^{-2}]$
L — $characteristic\ length\ [m]$
β — $thermal\ expansion\ coefficient\ [K^{-1}]$
Gr — $Grashof\ Number\ \left(Gr = \frac{g L^3 \beta \Delta T}{\upsilon^2}\right)$

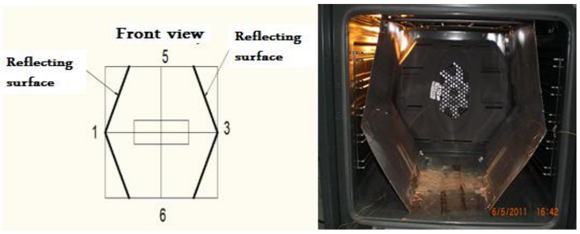
INTRODUCTION

The complexity of analytical solution of radiation problems have pave the way for numerical solutions beside them. Radiosity Irradiosity Method (RIM), Discrete Transfer Method (DTM), Discrete Ordinate Method (DOM), Finite Volume Method and Monte Carlo Method are some of the most popular numerical methods in radiation problems [1]. Monte Carlo Method, which is used in this study, is a statistical method that models a physical problem. Markhov chain and random numbers set a base for this method which gives rapid results [2]. Monte Carlo aprroach has been adapted to radiation problems by Howell [3]. These models have been used oftenly in industrial cases. These studies and a large literature summary are present in [4]. On the other hand, numerical studies in domestic ovens have been conducted by various studies including the effects of radiation in addition to natural convection [5,6,7].

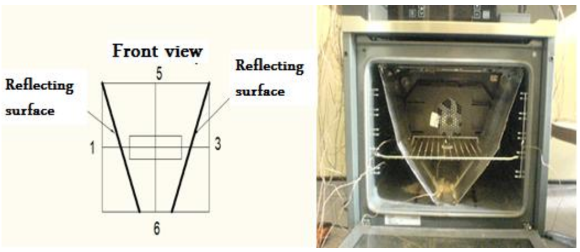
In the present work, such an analysis including the effects of radiation and convection in domestic ovens has been performed by the means of S2S radiation model and k-ε turbulence model. Three different enclosure models have been designed in a built-in oven in order to improve the efficiency of radiation. The main goal was to improve efficiency by improving the contribution of view factor effect. The experimental study has been carried out in order to compare the models from the view points of energy consumption and cooking time. By the way, the computational study focused on the mechanism of heat transfer in cavity.

MONTE CARLO CALCULATIONS

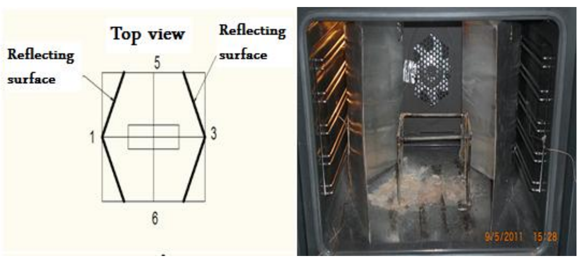
Various models formed by reflective surfaces (Figure 1) have been tested.



(a)



(b)



(c)

Figure 1 Models to improve radiative heat transfer a) Model I, b) Model II, c) Model III

The reflecting surfaces improve the radiation interaction between surfaces and the brick. Due to shadow affect caused by the brick in the center of the cavity, calculating view factors analytically is a though manner. Monte Carlo method has been used with generated grid indicated in Figure 2.

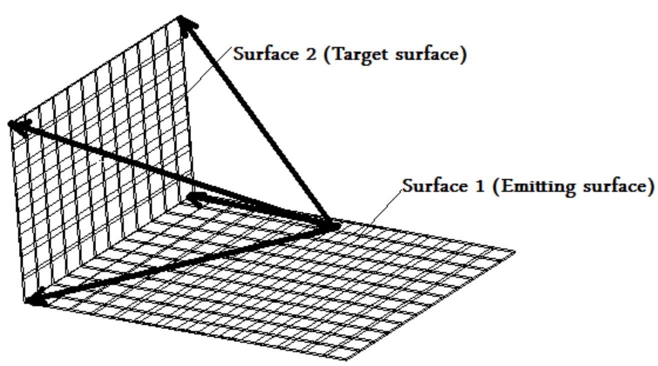


Figure 2 Grid generated for Monte Carlo approach

The generated grid shown in Figure 2 provides a random approach to calculate view factors. A random path from the grid points can be created using randomly generated numbers. By examining the limitations of the geometry, it is understood whether the random path reaches the target or not. The algorithm uses this approach for all the grid points and finally gives the result.

By this algorithm view factors between the reflecting surfaces and the brick have been calculated for the three models tested (Figure 1) and compared in Table 1.

Table 1 View factor interactions between reflecting surfaces and the brick

Model	I	II	III
Summation of the view factors between reflecting surfaces and the brick	0.13	0.12	0.16

As it can be seen from the results, Model III has the leading interaction with the brick.

EXPERIMENTAL STUDY

In order to improve radiation efficiency between the heaters and the brick, representing the food, different reflecting surfaces have been set in the enclosure.

Experimental Setup

The energy consumption experiments of the oven have been conducted according to EN 50304 standards [8] on a built-in oven in Figure 3.

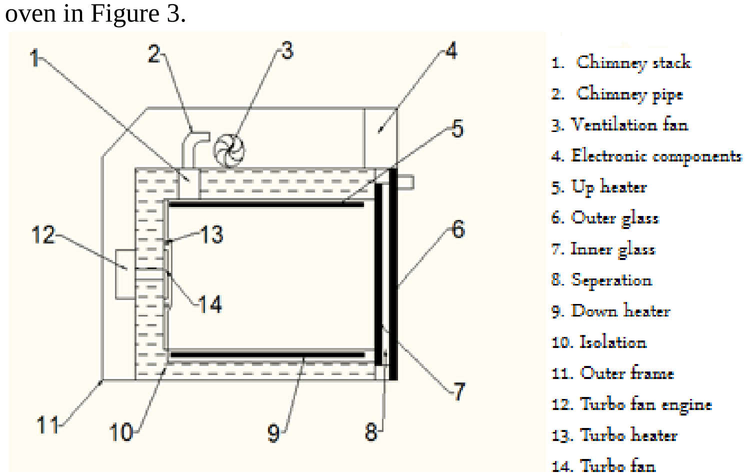


Figure 3 The built-in oven used in study

Experimental Results

Experiments were carried according to 8.3 item in EN 50304 [8] in three different temperature set; 140K, 180K, 220K in the static mode. In the static mode of the oven, only upper and bottom heaters are on, whereas the effect of the radiation can be examined clearly. The results of the three different models are compared with the original case and tabulated below (Table 2).

Table 2 Results of the energy consumption experiments

Experiment	**Cooking time** (min):	**Energy consumption** (Wh):
140K Static	47.0	757
140K Static	49.6	775
140K Static Model I	48.1	768
140K Static Model II	50.2	774
140K Static Model III	44.3	734
180K Static	41.8	894
180K Static	42.0	866
180K Static	43.8	939
180K Static Model I	42.3	902
180K Static Model II	40.9	903
180K Static Model III	39.3	884
220K Static	38.8	1064
220K Static	38.6	1069
220K Static Model I	38.8	1071
220K Static Model II	38.5	1089
220K Static Model III	36.0	1027

As it can be seen from the results, only Model III has a significant positive effect on energy consumption and cooking time. Model III provides an improvement of 3 %, 6 % and 7 % for 140K, 180K, 220K respectively. For the cooking time, Model III has provided respectively 8 %, 7.7 % and 7 % time consumption according to the original case. Thus, the improvement prediction by the Monte Carlo has been validated.

NUMERICAL MODELING

CFD analysis stands as a good tool to understand the role of convection and radiation in the enhancement achieved by the experimental study. Numerical study has been conducted only for the best-responded model to the enhancement trials namely Model III and to the original geometry namely Original Model. For the improved model, only-convection concerning analysis has been conducted in addition to the case with taking radiative heat transfer into account by the means of S2S model. It is expected to observe that after a time duration, radiative heat transfer should be the dominant mode of heat transfer [6]. In further sections, streamlines, temperature, Courant number and y^+ distributions, will be presented for two planes named as Plane 1 and Plane 2. The computational domain with the referred planes is shown in Figure 4. The virtual duct is inserted in the computational domain to initiate the driving force.

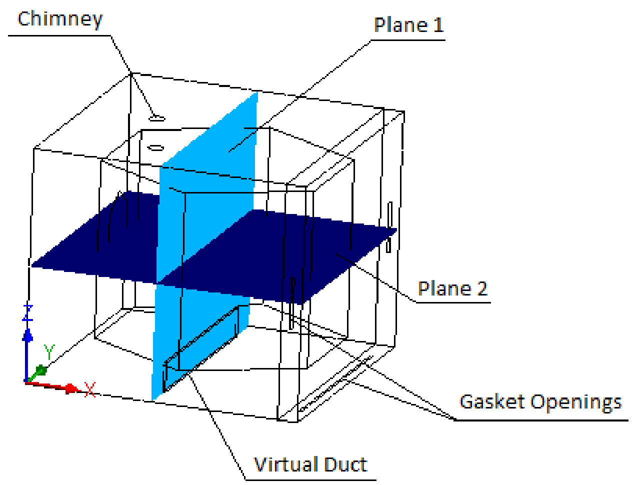


Figure 4 Computational Domain and Planes

Main criteria for creating the grid, is to provide the maximum y^+ value under the proper limits of chosen turbulence model [9] and the step sizes in spatial directions should be in the optimum order of range due to stability of solution which has been investigated by Courant number [10]. In Figure 5, Courant Number distribution for Plane 2 is given.

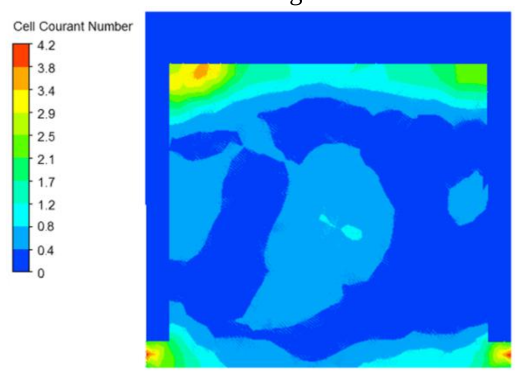


Figure 5 Courant Number Distribution at t=600s (Original Model)

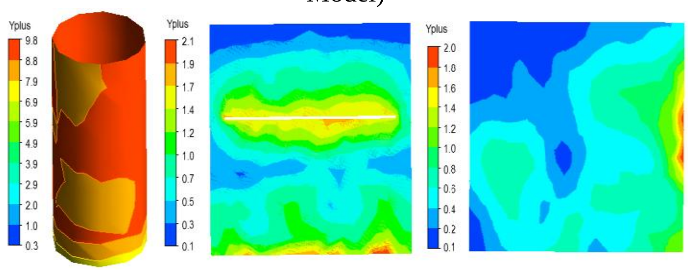


Figure 6 y^+ distributions at t=600s (Original Model – chimney & bottom of cavity & right wall of cavity)

Four different time steps sizes have been tried for the original case, and on the framework of grid independency, distributions of turbulent viscosity ratio along a line passing through on the centre of Plane 1 are plotted.

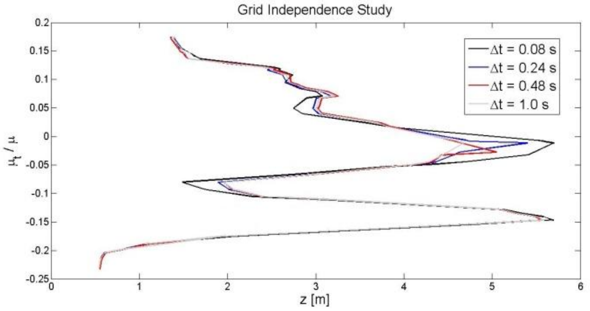


Figure 7 Grid Independence Study: Turbulent Viscosity Ratio Distributions

As a result of independence study, the grid with time step size of 0.08 second is chosen. For the validation of the numerical results, transient temperature distributions on the center of cavity are compared with the experimental results.

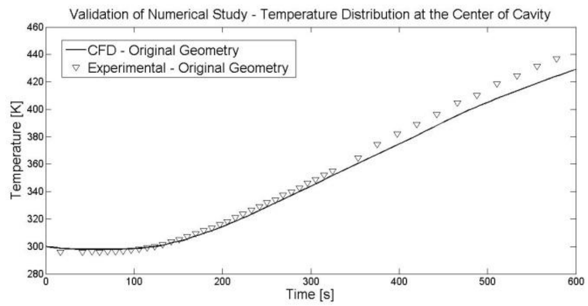


Figure 8 Validation of Numerical Results: Comparison of Temperature Distributions at the Centre of Cavity acquired by numerical and experimental studies

Validating the numerical results of improved model with the experimental ones is not appropriate because the computational domain and the experimental setup are different.

Material Properties

Ideal gas formulation has been used for the variation of density due to buoyancy forces, suction in chimney and the radiative heat transfer through the fluid within the cavity. Emissivity values for walls and the glass are chosen as 0.85, 0.7, respectively. Finally, the emissivities of heaters are specified as 0.3.

Boundary Conditions

Table 3: Turbulent Boundary Conditions

Zone	Type	TI [%]	Hyd. Diameter [m]
Chimney	Pressure – BC	8	0.03
Virtual Duct	Pressure – BC	11	0.01
Gasket Openings	Pressure - BC	12	0.008

Suction along the chimney is modelled with a constant mass-flow rate which is 0.00064 kg/s. For modelling transient heat transfer, it is essential to use varying temperature boundary conditions for heaters and walls [7]. Thus thermal boundary conditions for walls and heaters are specified by user defined functions, with polynomial fitting to experimental temperature distributions. Air temperature at gasket openings and the virtual duct, which are shown in Figure 4, are specified as ambient temperature which is 296 K.

Modelling of Turbulence

It is predicted that viscous forces would be overweighed by the buoyancy forces due to the low viscosity of air and the high temperature differences within the cavity because of heating process $Gr = O(10^7)$. Hence, modelling of turbulence becomes a vital part of this study and the additional stress tensor has been modelled by a turbulent-viscosity model (k-ε Turbulence Model) with full buoyancy effect option. The boundary conditions related to turbulence are listed in Table 3. For the near-wall treatment, SWF and EWF are shortlisted, SWF are chosen since there is no gain in solving viscous sub-layer by using EWF. Considering all of above, during mesh creation, it is aimed that, the first point must be placed within the buffer layer which corresponds to y^+plus range of 5 $\sim$ 30.

As a result of low velocity values, turbulent boundary layer existing on walls consists of non-negligible buffer layer thickness. Thus it is important to obey that limitation. As it is shown in Figure 6, this limitation is satisfied.

Radiation Model

The current computational analysis does not include the brick. Thus, the medium in the cavity is not participating. In this case, S2S which considers only the radiative heat fluxes from surfaces to surfaces [9], is chosen for modelling of radiation. It should also be noted that, air in the cavity for this level of temperature can be considered as non-participating medium.

Results

As can be seen from the streamlines of planes perpendicular to the gravitational direction and at the altitude of cooking process (Figure 7, 8, 9), during the time period of 0 – 400s, convection takes the important part of the heat and momentum transport within the cavity.

The radiative heat transfer dominates convective effects at the rest of the simulation since the temperature difference rises.

These statements can be proved by observing the streamline patterns at the intersection of two inclined surfaces for the enhanced model. In Figure 7 and Figure 8, still a boundary layer existence is observed in opposite to Figure 9, where the fluid particles adjacent to the walls have noticeable motion as a result of radiative heat transfer. It must be also noted that by the generation of circulations in the intersection region of cavity, heat transfer from cavity to the insulation decreases because of low velocity value. This mechanism of transport brought about a same cooking performance with a lower heat loss from cavity when it is compared to the original model which is a satisfactory for the aim of the study.

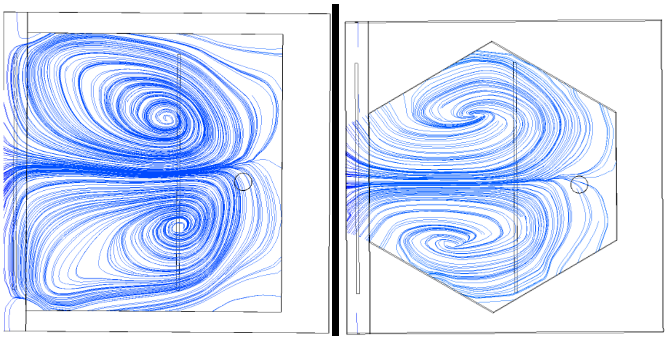


Figure 9 Streamlines at t=200s (Original Model-left vs Model III - right) – Plane 2

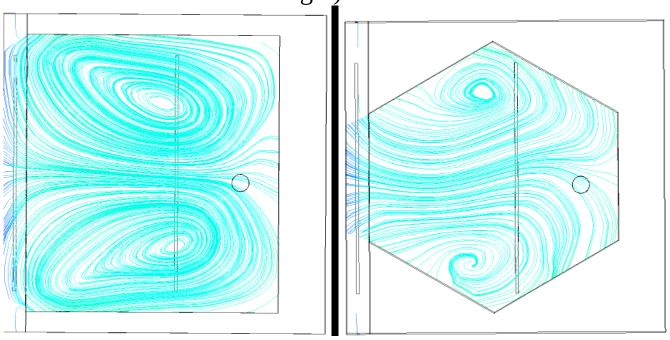


Figure 10 Streamlines at t=400s (Original Model-left vs Model III - right) – Plane 2

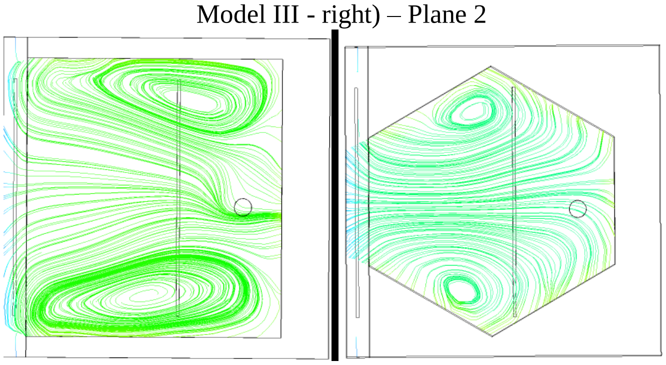


Figure 11 Streamlines at t=600s (Original Model-left vs Model III - right) – Plane 2

In the matter of deciding whether radiative heat transfer or convection provides the referred enhancement, only-convection concerning CFD analysis has been conducted for the improved model (Figure 10).

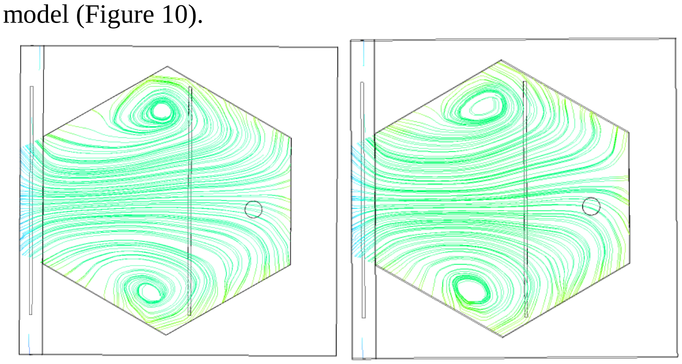


Figure 12 : Streamlines at t=600s (Convection-Model III-left vs Model III-right) – Plane 2

Numerical results have certainly showed that effect of radiation increases the scale of circulations and decreases the heat loss from cavity expectedly.

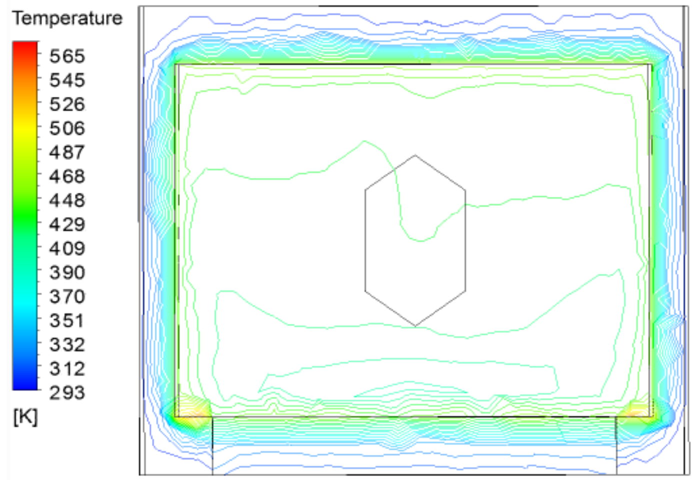


Figure 13 : Temperature Contour at t=600s (Original Model)

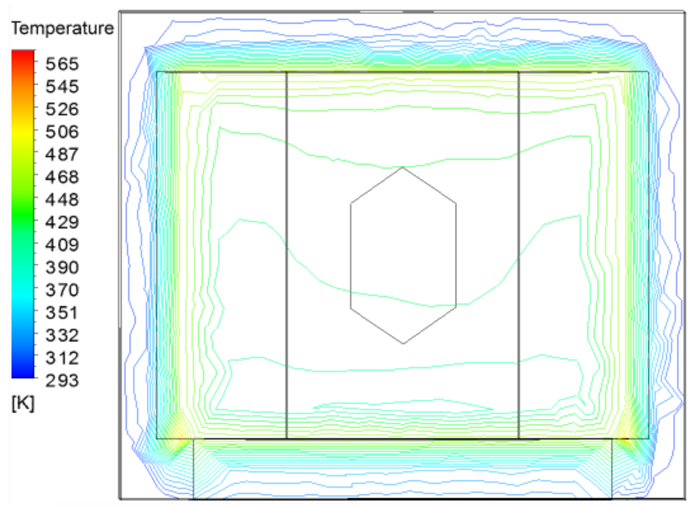


Figure 14 : Temperature Contour at t=600s (Model III)

Temperature distribution along the vertical spatial direction (Plane 1) on the center of cavity where the cooking process is being conducted is another important performance parameter. As can be seen in Figure 11 and Figure 12 , for the improved model, with the higher view factor, heat has been transferred to the cooking position from heaters faster than in the original model.

This improvement is also related to the convective heat transfer within the cavity, since the fluid is confined with inclined surfaces, boundary layer generation at the intersection of surfaces, provides the flow-circulation to be closer to the center of cavity.

CONCLUSION

Experimental study has showed that replacing reflecting surfaces brought about an improvement in energy efficiency and in cooking time. The prediction of best model by Monte

Carlo method has been validated by the experimental results. On the other hand, using the computational tools, it is concluded that reflecting surfaces have positive effects on improvements in terms of not only radiation but also convection.

ACKNOWLEDGEMENT

The authors would like to thank The Scientific and Technological Research Council of Turkey and Arcelik R&D Department personnel and managers for their valuable contributions and financial support.

REFERENCES

[1] Guillem Colomer Rey, Numerical Methods for Radiative Heat Transfer, *Doctoral Thesis*, Terrassa, July 2006

[2] Siegel Robert, Howell John R. ; *Thermal Radiation Transfer*, New York, 2002, s. 390- 417

[3] Howell, John R., 1968, "Application of Monte Carlo to Heat Transfer Problems ," *Advances in Heat Transfer*, Vol. 5, J. P. Harnett and T. Irvine, eds., Academic Press, San Diego, pp. 1-54.

[4] Howell John R., The Monte Carlo Method in Radiative Heat Transfer, *Journal of Heat Transfer*, Vol 120, August 1998

[5] E. M. Sparrow , J. P. Abraham, A computational analysis of the radiative and convective proceses that take place in preheated and non-preheated ovens, *Heat Transfer Engineering* Vol 24 (5) (2003) 25-37

[6] N. Chhanwal, A. Anishaparvin, D. Indrani, K.S.M.S. Raghavarao, C. Anandharamakrishnan**,**Computational fluid dynamics (CFD) modeling of an electrical heating oven for bread-baking process, *Journal of Food Engineering* 100 (2010) 452–460,

[7] Micael Boulet, Bernard Marcos, Michel Dostie, Christine Moresoli, CFD modeling of heat transfer and flow field in a bakery pilot oven, *Journal of Food Engineering* Vol 97 (2010) 393–402

[8] European Committee for Electrotechnical Standardization (CELENEC), 2001. Electric ovens for household use – Methods for measuring energy consumption, EN 50304 2001

[9] Fluent 6.1 User's Guide, Fluent Inc. 2003-01-25

[10] An introduction to computational fluid dynamics, The finite volume method, H.K. Versteeg and W. Malalasekera, copublished in the United States with John Wiley & Sons Inc., New York