HEFAT2008
6th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
30 June to 2 July 2008
Pretoria, South Africa
Paper number: CL1

THE NUMERICAL AND EXPERIMENTAL ANALYSIS OF FORCED CONVECTION ON THE SURFACES OF A RECTANGULAR BOX WITH HEAT GENERATION

Colak L.,* Ipek N.E., and Demir G.
*Author for correspondence
Department of Mechanical Engineering,
Baskent University,
Ankara, 06530,
Turkey,
E-mail: lcolak@baskent.edu.tr

ABSTRACT

The system development with experimental studies that is accomplished by trial and error method is neither practical nor economical since it is difficult and expensive. So that, creating the mathematical model and determining the most appropriate design data by using numerical solutions is a more effective and cheaper approach for the system development. In this study numerical and experimental analysis of forced convection heat transfer from the horizontal and lateral surfaces of a 0.3m x 0.3m x 0.52m rectangular box in a 0.55m x 0.55m x 3m wind tunnel, which is heated by a 0.5m length cylindrical 450 W power source with a constant surface temperature of 200ºC, was performed. The temperature distribution on all exterior sides of rectangular box subjected to forced convection, were calculated by finite volume numerical analysis and compared with the results obtained from the experimental measurements. The main objective of this study is to verify the mathematical model which was constituted for the numerical solution. The results obtained from the numerical analysis were approximately similar to the ones obtained in experimental analysis, so the mathematical model was verified.

INTRODUCTION

Rapid improvements and facilities in computer technology make the use of computer technology common in system design. Computational fluid dynamics (CFD) is intensively used for analysis of fluid and heat transfer problems. This study was created as a term project in Baskent University Mechanical Engineering Department. In this project the aim was to verify a mathematical model by making numerical and experimental analysis of forced convection heat transfer from the horizontal and lateral surfaces of a rectangular box in a wind tunnel, which was heated by a cylindrical power source with a constant surface temperature and verify the mathematical model which was constituted for the numerical solution. The generated model can be used for industrial purposes, as a sealed electronic box cooling.

Before starting numerical and experimental analysis, early studies in these subjects were searched in literature.

A numerical study of natural convection around a square, horizontal, heated cylinder placed in an enclosure [1], unsteady thermal entrance forced convection heat transfer in laminar flow with a periodic variation of inlet temperature [2], numerical and experimental analysis of unsteady heat transfer with periodic variation of inlet temperature in circular ducts [3], in which temperature variations along the centreline of the circular duct were observed to have thermal oscillations with the same frequency as the inlet periodic heat input and amplitudes that decayed exponentially with distance along the duct, were analysed first.

After careful consideration of the papers about, unsteady turbulent forced convection in a parallel-plate channel with time wise variation of inlet temperature [4], the effects of fin spacing, fin height and fin length on the free convection heat transfer from the horizontal fin arrays numerically [5], in which the three-dimensional elliptic governing equations were solved using a finite volume based computational fluid dynamics (CFD) code and by analysing the paper about the effects of obstacles of various diameters, located upstream of a rectangular channel partially heated from the bottom and so providing constant heat flux, on heat transfer experimentally [6], the mathematical model, which was used in this study, was performed.

To prove the accuracy of the mathematical model embodied in this project, the results obtained from the finite volume method and numerical analysis should be well-matched at an acceptable rate with the experimental results. For numerical analysis Fluent was used and 0.3 m x 0.3 m x 0.52 m rectangular box which was heated by a cylindrical rod type power source with a constant surface temperature was located in the centre of wind tunnel for the experimental studies.

NUMERICAL METHOD

Experimental work in research-development studies are expensive and time consuming although they may provide very useful data. This is mostly because of setting the whole experimental set-up and repeating some measurements several times due to the design and assembly errors of the set-up, the device calibrations and the measurement errors resulted from incorrect reading of the obtained values.

For those reasons numerical methods are commonly used at the first phase of the complex problems' analysis instead of the experimental methods. For the numerical computations, at first, system should be modelled mathematically. After that, the most suitable numerical method should be picked and the necessary data should be determined for the correct solution.

The finite volume method was decided for the analysis of the problem as it is very commonly used for the nonlinear heat transfer and fluid dynamics problems which had the most suitable characteristics of this study. In this context, the commonly used computational fluent dynamics software, Fluent, was used in the numerical analysis at Baskent University Engineering Faculty.

The solid model of the system was created and boundary types were given at Gambit which is the pre-processor of Fluent. The following process was defining the correct size and property of the mesh used for the temperature distribution on the model. Mesh type is important for the numerical solution stability and accuracy. The computational domain is divided into control volumes. The smaller the control volumes the more accurate the solutions are obtained. However, depending on the type of the solver, the density of mesh may lead to instability. Thus, the solution should be dense enough to be grid-independent. At that point, an optimization was done to have such mesh density in those regions where the temperature gradients were high. As the surface of the cylindrical heater and the surfaces of the rectangular box were critical regions, smaller control volumes were preferred there. The "hexahedral" elements seem to be the most suitable mesh type for the heat transfer problems with finite volume methods in the literature. Six-faced hexahedral elements were selected as the mesh type, since the numerical errors and diffusion in models done with hexahedral meshes was lower.

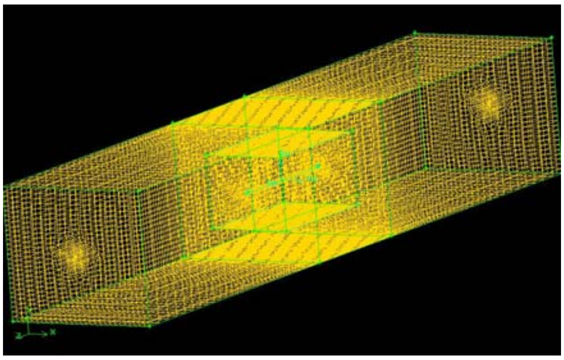

Figure 1 Discretization of the geometry into smaller pieces with hexahedral elements

The fact that the air flow inside the wind tunnel around the rectangular surface was turbulent; "the realizable k-epsilon" turbulence model and "the standard-wall" function approach were selected as they were the most common approaches for these kinds of problems in the literature [7]. The air temperature at the inlet of the wind tunnel was measured as 7.4°C and the velocity of the air was measured as 1.9 m/s. These values were used as boundary conditions in the solution of the model. 176484 "hexahedral" finite volume elements and 192178 nodes were obtained from the hexahedral mesh model, which can be seen in Figure 1, created by "Gambit" pre-processor. The numerical model was solved with FLUENT software and after 300 iterations residuals were below 10^{-3}, by assuming a constant surface temperature of the cylindrical heater at 200°C and by assuming steady state heat transfer during the experimental measurements.

For the mathematical model defined in this study, three dimensional Navier-Stokes equations, continuity equation, conservation of energy equation and time-averaged turbulence equations were solved numerically with finite volume method. In this study, the density change with temperature was also taken into consideration. The details of the solution method are provided in the FLUENT manuals [8].

Throughout the iterations, the residual history graph in Figure 2 and the temperature values of every three points from each surface of the rectangular box were obtained. In model, the coordinate axis was located at the centre of the rectangular box and the air flow was from the –z direction. Eighteen points, where the temperature values were recorded. The places of these eighteen points are;

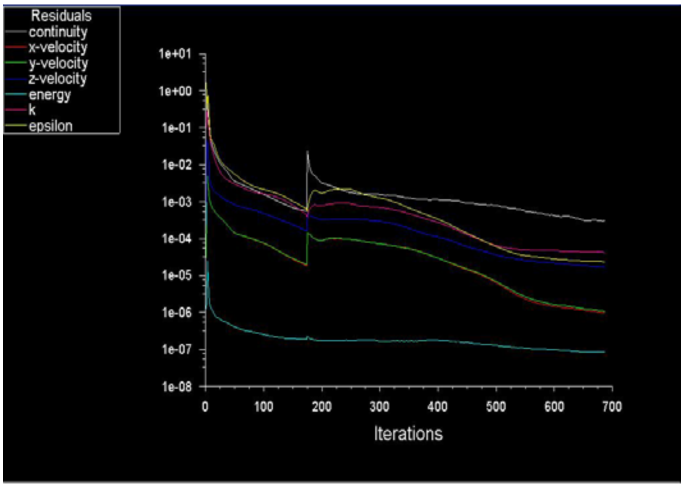

Figure 2 Decreasing of the residuals in the finite volume numerical analysis with hexahedral finite element meshes

1) At the two square-shaped surfaces, inlet and exit surfaces, which had a size of 0.30m x 0.30m on the rectangular box. The locations of these points can be seen in Figure 3.

2) At the surfaces at the top, bottom and also the lateral sides of the rectangular box which had a size of 0.30m x 0.52m. The points on these surfaces are given in Figure 3.

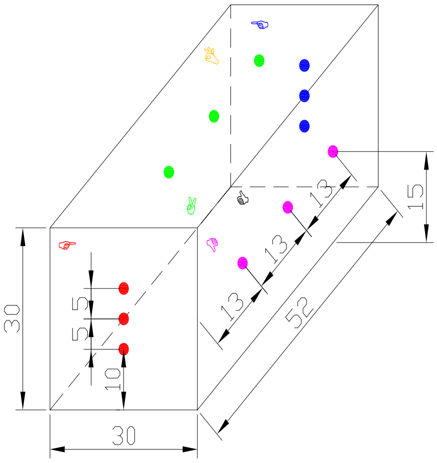

Figure 3 Schematic view of the points on the surface of the box which were used to obtain the temperature values

EXPERIMENTAL ANALYSIS

After the numerical analysis had been done, the experimental set-up in Figure 4 was set up. A cylindrical rod type electrical heater with the power of 450 W was located in the centre of the rectangular box with thickness of 1.0mm and manufactured by using Cr-Ni stainless steel.

The eighteen stations which were used to obtain temperature values from the "FLUENT" model were marked on the surfaces of rectangular box and thermocouple connections were set on these marked points to measure the temperature. During this study, DIN 43710 and IEC 584 standardized Fe-Const, Type-J thermocouples which are capable of measuring the temperature between -200 $^{\circ}$C and +800 $^{\circ}$C were used. During the connection process of the thermocouples, the marked points on the rectangular box was perforated to get holes which have a diameter of 4 mm and the rubber cable tips which were also had 4 mm diameters, were used in order to bond the connection cables of the thermocouples with a spot welding machine to read the temperature values.

Figure 4 The Experimental set-up

During the experimental measurement process, in order to achieve constant temperature from the cylindrical heater which was located at the centre axis of the rectangular box, the heater was connected to a digital control device by using a relay. Also a separate safety thermostat was attached to the circuit to prevent high temperature condition which may cause damage on heater in case of an uncontrolled temperature situation. By using the digital control device, the temperature sensitivity over the heater surface was between 200±5°C.

The air flow velocity was measured by a flow-meter with the sensitivity of ±2% that was placed at the inlet of the wind tunnel at the centre axis.

After the experimental set-up had been set and the measurement devices had been calibrated, the experimental measurements were started. The velocity of the air in the wind tunnel was set at 1.9 m/s by fixing the torque of the frequency controlled fan motor and the air inlet temperature was recorded as 7.4°C. After the thermocouple readings had reached to steady state, the temperature values from eighteen points on the surfaces of rectangular box were recorded.

RESULT AND DISCUSSION

In this study, the numerical and the experimental analysis of forced convection heat transfer from the horizontal and lateral surfaces of a rectangular box, which was heated by a cylindrical rod type heater with a constant surface temperature, was performed. Path-lines around the rectangular box and three dimensional temperature distributions on the rectangular box surfaces were shown in Figure 5 and Figure 6, respectively. In numerical analysis six-faced hexahedral elements were selected as the mesh type, since the numerical errors and diffusion in models done with hexahedral meshes was less than the other mesh types. The temperature values obtained from the eighteen stations on the six surfaces of the box were calculated separately by using finite volume method and experimental analysis.

The analysis results from FLUENT finite volume software indicate that with the hexahedral meshes, temperature values were obtained from three different points on every surface of the box. After examining the temperature distribution on every surface by using numerical analysis, the temperature distribution was observed as symmetrical, just as it was expected. The fact that the results obtained from the hexahedral meshes and the results from the experiments were close to each other, the grid-independency test was not necessary.

Additionally, when the results of the numerical analysis for each corresponding surfaces were compared to each other, the temperature values obtained from the above plate (A) was higher than the below plate's (C) temperature values as can be seen in Figure 7 and Figure 8, respectively. The main reason of this situation was that the density of the heated-air decreases and the remaining cold air moves to the below part of the box. As it was also expected for the inlet and the outlet section of the rectangular box, the frontal plate (E) which first contacted to the cold air coming from the wind tunnel also had a lower temperature value than the backside plate (F) as can be seen in Figure 9 and Figure 10. Furthermore, due to the dead zones and the vortex at the backside of the box, it was seen that there was a wake region at the back of the box, where the temperature values on that plate were also higher than the ones on the frontal face. The lateral plates B and D had similar temperature distributions as it was expected because of the symmetry as can be seen in Figure 11 and Figure 12.

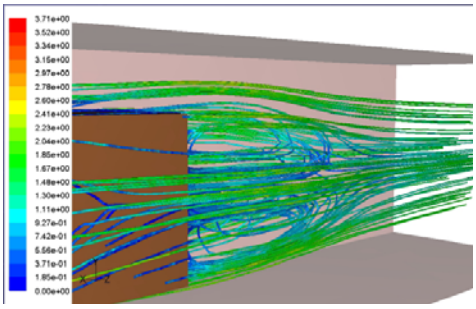

Figure 5 Path-lines colored by velocity magnitude (m/s)

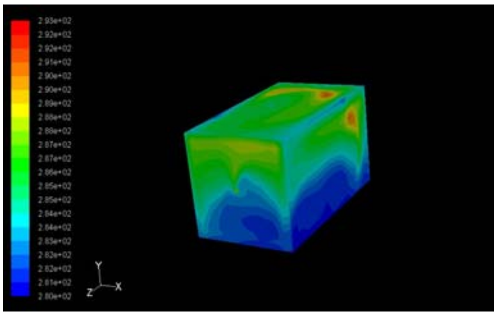

Figure 6 Three dimensional view of the temperature distribution

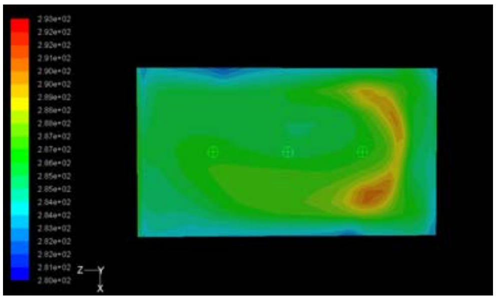

Figure 7 Temperature distribution of the above (A) surface of the rectangular box

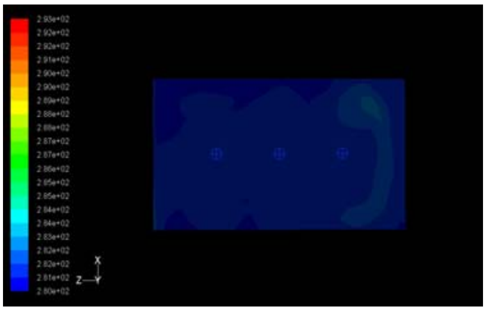

Figure 8 Temperature distribution of the bottom (C) surface of the rectangular box

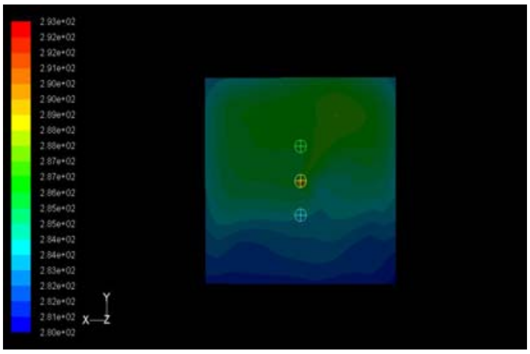

Figure 9 Temperature distribution of the frontal (E) surface of the rectangular box

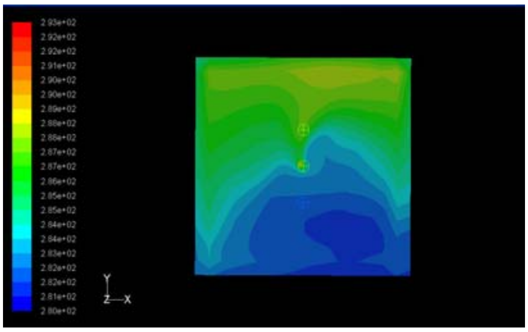

Figure 10 Temperature distribution of the backside (F) of the rectangular box

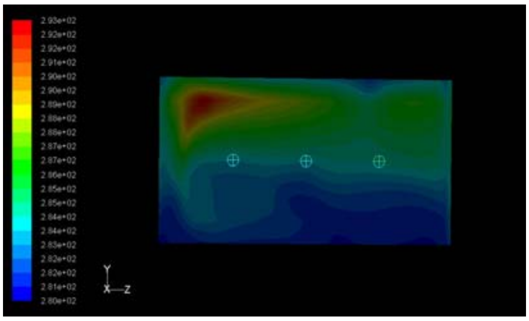

Figure 11 Temperature distribution of the lateral (B) surface (+X)

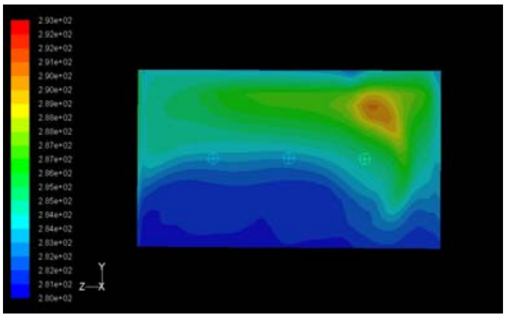

Figure 12 Temperature distributions of the lateral (D) surface (-X)

In this study, which was mainly focused on proving the validity of the model designed for numerical analysis, although most of the temperature values at eighteen stations were close to each other as can be seen in Table 1, the values were not as close as expected due to number of iterations, the turbulence model, errors related with the steady state assumption and the other experimental factors.

Table 1 Numerical and experimental temperature values (K)

SURFACES	COORDINATES			NUMERICAL ANALYSIS(K)	EXPERIMENTAL ANALYSIS(K)	ERROR (%)
	X	Y	Z			
Upper Surface (A)	0,00	0,15	-0,13	286,41	289,10	0,93
	0,00	0,15	0,00	285,76	290,80	1,73
	0,00	0,15	0,13	286,32	291,00	1,61
Lower Surface (C)	0,00	-0,15	-0,13	281,28	285,40	1,44
	0,00	-0,15	0,00	281,54	285,70	1,46
	0,00	-0,15	0,13	281,52	286,80	1,84
Right Lateral Surface (D)	0,15	0,00	-0,13	284,65	286,70	0,72
	0,15	0,00	0,00	282,93	286,80	1,35
	0,15	0,00	0,13	283,06	287,40	1,51
Left Lateral Surface (B)	-0,15	0,00	-0,13	283,59	286,30	0,95
	-0,15	0,00	0,00	283,70	285,80	0,73
	-0,15	0,00	0,13	284,19	286,90	0,94
Inlet Square Surface (E)	0,00	0,05	-0,26	285,98	286,30	0,11
	0,00	0,00	-0,26	288,85	285,50	1,17
	0,00	-0,05	-0,26	283,69	285,90	0,77
Outlet Square Surface (F)	0,00	0,05	0,26	287,25	287,60	0,12
	0,00	0,00	0,26	286,85	287,10	0,09
	0,00	-0,05	0,26	282,04	288,00	2,07

The results of experimental and numerical analysis given in Table 1 were also compared graphically in Figures 13-18 for better clarification of the results.

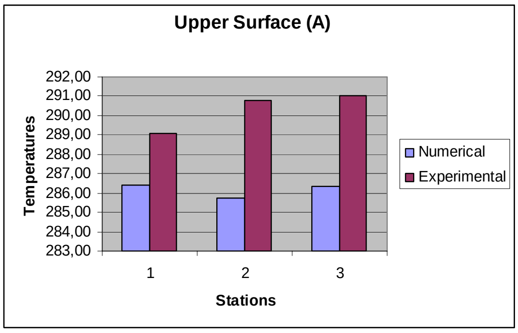

Figure 13 The comparison of numerical and experimental temperature values at upper surface (A)

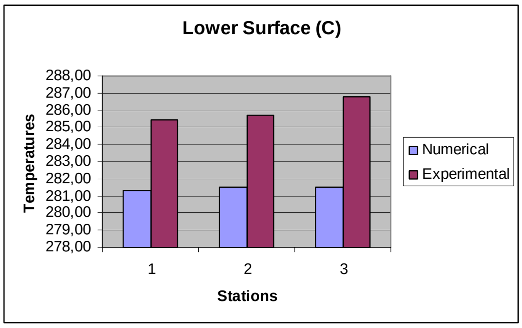

Figure 14 The comparison of numerical and experimental temperature values at lower surface (C)

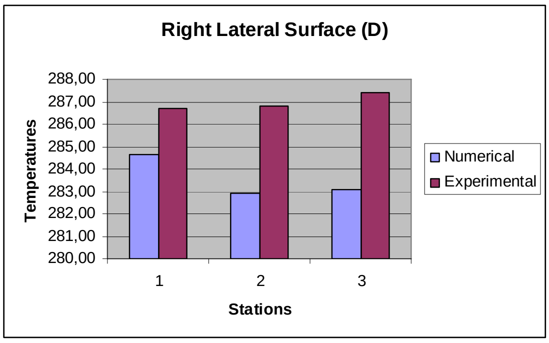

Figure 15 The comparison of numerical and experimental temperature values at right lateral surface (D)

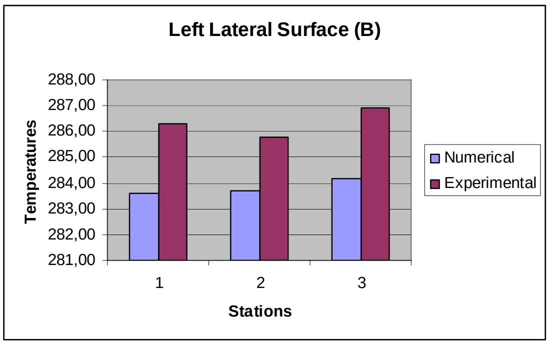

Figure 16 The comparison of numerical and experimental temperature values at left lateral surface (B)

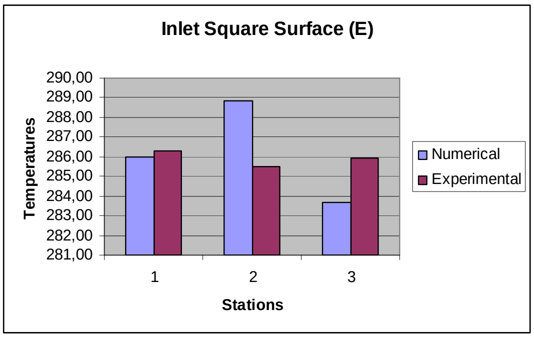

Figure 17 The comparison of numerical and experimental temperature values at inlet square surface (B)

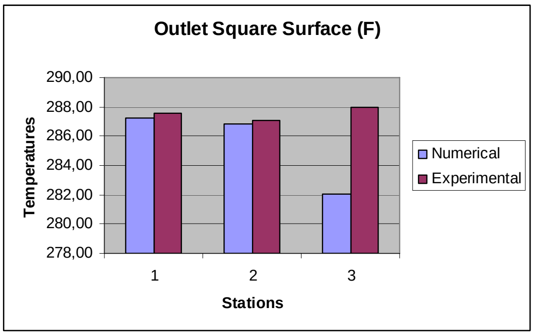

Figure 18 The comparison of numerical and experimental temperature values at outlet square surface (F)

The comparison of the numerical and experimental results for eighteen nodes at six surfaces in Table 1 showed that there was approximately 1% error between the values. These results were also clarified in Figures 13-18. The differences on the height of columns on graphs were because of the scale of vertical axis of graphs. There was at most 6 K difference on some of the graphs due to the numerical errors discussed before and due to experimental errors explained below in detail.

The main error factors, which affected the experimental analysis, were;

1) Not fully developed air flow and unstable air velocity inside the tunnel due to the 4 meters distance between the blower fan of the wind tunnel and experimental analysis area,

2) The absence of air diverter elements inside the existing wind tunnel, that causes deviations in air velocity,

3) The assumption of 1.9 m/s air velocity at the centre of the inlet of wind tunnel as same across the inlet cross-section,

4) The assumption of 200°C constant surface temperature of cylindrical rod heater, although it oscillates between 195°C and 205°C,

5) Inaccurate experimental temperature data collection due to the extra contact resistance caused by bonding the thermocouple cables with the cable tips to the surface of experiment setup, despite the accurate calculation of surface temperature values in the numerical analysis,

6) Thermocouple cables and the trivets used for the assembling of the box inside wind tunnel caused non-uniform air flow around the rectangular box.

As a result, the numerical results of created mathematical model, which can also be used for industrial purposes as a sealed electronic box cooling, by using the computational fluid dynamics software FLUENT and the experimental results were close to each other with an approximate error of 1% for hexahedral mesh, so the mathematical model was verified. For obtaining better results, doing more research and development works for the numerical model and for the experimental set-up and the usage of high accuracy measuring devices suggested as future works.

REFERENCES

[1] De, A. K., and Dalal A., A numerical study of natural convection around a square, horizontal, heated cylinder placed in an enclosure, *International Journal of Heat and Mass Transfer*, Vol. 49(23), 2006, pp. 4608-4623

[2] Li W., and Kakac S., Unsteady thermal entrance heat transfer in laminar flow with a periodic variation of inlet temperature, *International Journal of Heat and Mass Transfer*, Vol. 34, 1991, pp. 2581–2592

[3] Brown D., Li W., and Kakac S, Numerical and experimental analysis of unsteady heat transfer with periodic variation of inlet temperature in circular ducts, *International Journal of Heat and Mass Transfer,* Vol. 20, 1993, pp.883–899

[4] Kakac S., and Li W., Unsteady turbulent forced convection in a parallel-plate channel with time wise variation of inlet temperature, *International Journal of Heat and Mass Transfer*, Vol. 37, 1994, pp. 447–456

[5] Baskaya S., Sivrioglu M., and Ozek M., Numerical flow visualization of natural convection from horizontal rectangular fin arrays, *Journal of Heat Science and Technic*, Vol. 25, 2005, pp.22-28

[6] Evin D., and Tanyildizi F., The effect of obstacles normal to the flow in a partially bottom heated horizontal channel on heat transfer, *Journal of Firat University Science and Mechanical*, Vol. 18, 2006, pp. 249-255

[7] Versteeg H.K., and Malalasekera W., An introduction to computational fluid dynamics, *The Finite Volume Method,* 2nd ed. Pearson Education Limited, 2007

[8] Fluent Incorporated, FLUENT User’s guide, Version 6.2.16, 2004

[9] Incropera F.P., Dewitt D.P., Bergman T.L., and Lavine A.S., *Fundamentals of Heat and Mass Transfer,* 6th ed. Wiley, 2007