GRD Journal for Engineering | Volume 2 | Issue 4 | March 2017
ISSN: 2455-5703

Computational Fluid Dynamics Analysis of Horizontal Heated Plate for Natural Convection

Aneesh K Johny
PG Scholar
Department of Mechanical Engineering
Mar Athanasius College of Engineering, Kerala, India

Gokul Raj C R
PG Scholar
Department of Mechanical Engineering
Mar Athanasius College of Engineering, Kerala, India

Nandhu Krishna
PG Scholar
Department of Mechanical Engineering
Mar Athanasius College of Engineering, Kerala, India

Abstract

Free or natural convection heat transfer is predominant in many engineering applications such as cooling of electronic equipment, pollution, materials processing, energy systems, and safety in thermal processes and geophysical flows. The present work discusses about the result of a computational fluid dynamics study of Steady, Laminar, Free convection heat transfer in a horizontal plate facing upwards in which two walls are adiabatic and other two ends are open to the ambient .The aim is to simulate the flow using appropriate CFD tools and compare the results with available experimental data. The CFD tool used here is ANSYS-FLUENT.
Keywords- ANSYS Fluent, Computational Fluid Dynamics, Heat Transfer, Natural Convection, Plate

I. INTRODUCTION

A. General Background

Natural convection heat transfer occurs in many engineering applications such as cooling of electronic equipment’s, solar collectors, chimney and furnaces, nuclear reactors and geophysical flows. In these equipment’s, the source of heating, in general is either due to volumetric heat generation or due to surface heat fluxes. For instance, electronic equipment generates heat, which can be expressed in terms of volumetric heat generation. For example, the heat generation in nuclear fuel rods due to nuclear reaction can be considered as volumetric heat generation.

Electronic devices and equipment now permeate virtually every aspect of our daily life. Among the most ubiquitous of these is the electronic computer varying in size from the handheld personal digital assistant to large scale main-frames or servers. In many instances a computer is imbedded within some other device controlling its function and is not even recognizable as such. The applications of computers vary from day-to-day activities to highly complex systems such as health care equipment’s, economic, scientific, and defense devices. In the ever growing scenario of computer applications, failure of computer systems will results in major disruption of many vital services and even have life threatening consequences. As a result, it is very important to improve the reliability of electronic computers as in the same way to improve their speed and storage capacity.

In the present work, natural convection in a typical geometry of an electronic chip is considered. The electronic chip is modelled heat source placed an important role in the natural convection heat transfer in the geometry. The characteristics of heat transfer in the in such geometries are essential for the design of such equipment’s. The objective of the present work is to carry out a computational fluid dynamics analysis of a geometry model which fulfill the requirements of an electronic chip.

B. Cooling Load of Electronic Equipment

The first step in design process is to find out the total cooling load, which is essential for the design of a cooling system. The easiest way for determining the power dissipation of electronic equipment is by measuring the voltage applied V and current I under full load conditions. Then by using the following formula:

$$W = VI = I^2R$$

After determining the cooling load, as a common practice some safety margin or a "cushion", and make some allowance for future growth. We should not go overboard in being conservative, since an oversized cooling system will cost more, occupy more space, be heavier and consume more power. The design of the cooling system for an electronic device must be carried out considering the actual field operating conditions. Device must undergo extensive testing to satisfy stringent requirement for safety and reliability in applications like military, space, etc. The duty cycle is another important consideration in the design and selection of cooling technique. The actual power dissipated by a device can be considerably less than the rated power, depending on its duty cycle.

C. *Objective*

- To study the natural convection phenomena in a heated plate and also in a vertical channel.
- Numerical simulation will be done using ANSYS fluent.

II. LITERATURE REVIEW

Natural Convection is the result of density variation between the solid and fluid surface provided, the surface temperature is higher than fluid temperature (Ts>Ta). The flow is generated due to rise in buoyancy forces by the variation in density. A heated body cooling in ambient air generates such a flow in the region surrounding it (Ts>Ta). The buoyant flow arising from heat or material rejection to the atmosphere, heating and cooling of rooms and buildings, recalculating flow driven by temperature and salinity differences in oceans, and flows generated by fires are other examples of natural convection. In the literature [1] a detailed review of using natural convection and advantages are given.

The interest in buoyancy-induced flows and the associated heat and mass transfer has been increased over the past several decades, due to the vast application of such flows in electronics, pollution, thermal processes, etc. The focus of this study is based on electronic equipment's. A comprehensive review on natural convection in electronic equipment cooling is given in [2]. The electronics components has been miniaturized, but the performance are increasing continuously. This led to the production of high power density chips, which shortens the life time, increased rate of malfunction, etc. Efficient heat removing mechanism is essential for such components to compensate the above mentioned problems. A suitable thermal management strategy is essential for improving the reliability and life time of such electronics devises. Studies shows that for every 2˚C increase in the temperature will decline the reliability by 10% [2]. A study conducted by Yeh [3] in US Air Force concluded with the findings, over 50% of the electronics failure are related to temperature. Subsequently, a large number of researches have been conducted to improve the thermal aspects of electronics equipment's.

Elenbass [4] conducted experiments in laminar natural convection heat transfer in smooth parallel plate in a vertical channel and reported a detailed study of the thermal characteristics of cooling by natural convection. R.A. Wirtz et al. [5] considered a geometry with constant heat sources by placing it over entire length of the wall. A number of discrete heat sources placed over a wall was considered by Chen Linhui et al. [6] since the geometry cannot simulate discrete placement of chips. Osterle [7] conducted numerical analysis on free convection heat transfer for development of boundary layer between parallel isothermal vertical plates and get result for velocity, temperature and pressure variation throughout the flow field .The numerical method used is hybrid finite difference method. Yousef et al. [8] conducted an experimentally study of free convection heat transfer in air from isothermal horizontal surfaces heated and facing upward by using a Mach-Zehnder interferometer. Anu Nair et al. [9] conducted a review on natural convection from horizontal heated plate facing upward. In his work he compared different correlations used for Nusselt numbers in the calculation of heat transfer.

Although the works dealing with natural convection in electronics packaging is in abundance, the literature [10], [11] dealing with the maximization of heat transfer. Study about the optimization of the position of concentrated heat source was recognized in [12], [13]. For natural convection the parameter considered is Reyleigh number, the distance between heat source, and heat dissipation rate [14], [15], [12]. The work of Da Silve et al. [16] point out the location of the heat source as a function of Reyleigh number. They carried out both numerical and theoretical analysis on natural convention in vertical cavity. Their conclusion is contrary to the results presented by Liu et al. [14]. M. Corcione [17] measured the temperature and velocity distribution of the heated square plate on both sides and studies the correlations for free convection. The first theoretical study on the natural convection is given in the literature [18]. The work showed the existence of similarity solutions for a semi-infinite isothermal plate for air as working fluid. Hassan and Mohamed [19] carried out experimental studies on free convection from inclined plates.

Anu Nair et al. [20] conducted new optimization method to find out the value of C in the Nusselt Number. The purpose of this paper is to study the natural convection phenomena in a horizontal heated plate with two side adiabatically sealed. This study is conducted for a two-dimensional, natural convection flow over the surface in a steady state manner, and the analysis is conducted using the ANSYS Fluent software [21]. The finite volume technique and the SIMPLE are used to solve the conversation equations.

III. MATHEMATICAL FORMULATION AND MODELLING

A. *Mathematical Formulation*

The Cartesian coordinate system is used for the modelling, the basic governing equations are follows

Continuity Equation:

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0 \tag{1}$$

Momentum Equations:

$$\rho\left(U\frac{\partial U}{\partial x} + V\frac{\partial V}{\partial y}\right) = -\frac{\partial P}{\partial x} + \mu\nabla^2 U \tag{2}$$

$$\rho\left(U\frac{\partial U}{\partial x} + V\frac{\partial V}{\partial y}\right) = -\frac{\partial P}{\partial y} + \mu\nabla^2 V + \rho g\beta(T - T_0) \quad (3)$$

Energy Equation:

$$\rho C_p\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = k\nabla^2 T \quad (4)$$

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} \quad (5)$$

Reyleigh Number:

$$Ra = \frac{g\beta qH^4}{\alpha vk} \quad (6)$$

Where,

g is the local gravitational acceleration
μ is the dynamic viscosity.
β is the coefficient of thermal expansion,
α is the thermal diffusivity

B. Modelling

The CFD package, ANSYS Fluent 14 was used in the two dimensional simulation of natural convection of the experimental setup. The geometry of the problems are shown in fig 1. It consists of a heat source placing centrally between two adiabatic plates. This whole assembly is placed inside a large enclosure to simulate natural convection. The enclosure is filled with a Newtonian fluid. The walls of the enclosure are kept as adiabatic. The pressure inlet boundary condition is applied to bottom opening of the enclosure and pressure outlet condition is applied to top opening of the enclosure.

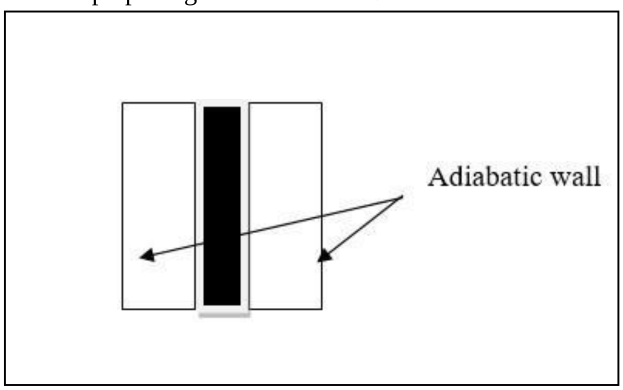

Fig. 1: Problem formulation for the heat transfer from the horizontally heated plate with adiabatic wall

In these inlet and outlet boundaries pressure is given as atmospheric pressure. Coupled boundary condition is given to top, bottom and two major sides of the heat source and adiabatic condition is given to two side end walls. The adiabatic side plates on both sides of the heat source are modelled as a rectangular solid with adiabatic boundary condition to the wall. From heat transfer point of view chip is modelled as heat source. Volumetric heat generation rate Q in W/m3 is given to solid heat source. Fluid is treated as incompressible with constant density. Since the temperature variation is small compared to the absolute temperature and we assume Boussinesq approximation.

C. Numerical Simulation

Numerical solutions for the velocity and temperature fields were generated by using a finite-volume package, FLUENT 14. The finite volume technique and semi implicit method for pressure linked equation (SIMPLE) are used to solve basic conservation equations. FLUENT solvers are based on the control volume method. The finite volume method is used to represent and evaluate partial differential equations as algebraic equations. Similar to the finite difference method, values are calculated at discrete places on a meshed geometry. The small volume surrounding each node point on a mesh is termed as Finite Volume. In this method, using the divergence theorem, volume integrals are converted to surface integrals. In each finite volume, these terms are evaluated as surface fluxes. Because the flux leaving a given volume is equivalent to that entering the adjacent volume, these methods are conservative. Another advantage of the finite volume method is that it is easily formulated to allow for unstructured meshes. Partial differential equations are discretized into a set of algebraic equations. All algebraic solutions are then solved numerically to render the solution field.

IV. RESULT AND DISCUSSION

To ensure that the result obtained by the numerical study are independent of the computational grid, a grid sensitivity analysis was carried out .The tests showed that the maximal temperature reached at any point on the wall is sufficiently insensitive to grid The details of grid independence study for Aspect ratio 0.072 and height ratio 0.2 are given in Table .1. The experimental setup is analyzed using fluent for different aspect ratio and different height ratio. The temperature contours of Aspect ratio for 0.072 and height ratio 0.2 is as shown below

Table 1: Grid Independence Test for Aspect Ratio 0.072 and Height Ration 2

Mesh	*Mesh Volumes*	*T_{max} (K)*
60 x 40 x 6	*17500*	*390*
70 x 50 x 7	*26500*	*395*
90 x 50 x 7	*32500*	*401*
100 x 50 x 8	*44000*	*401.2*

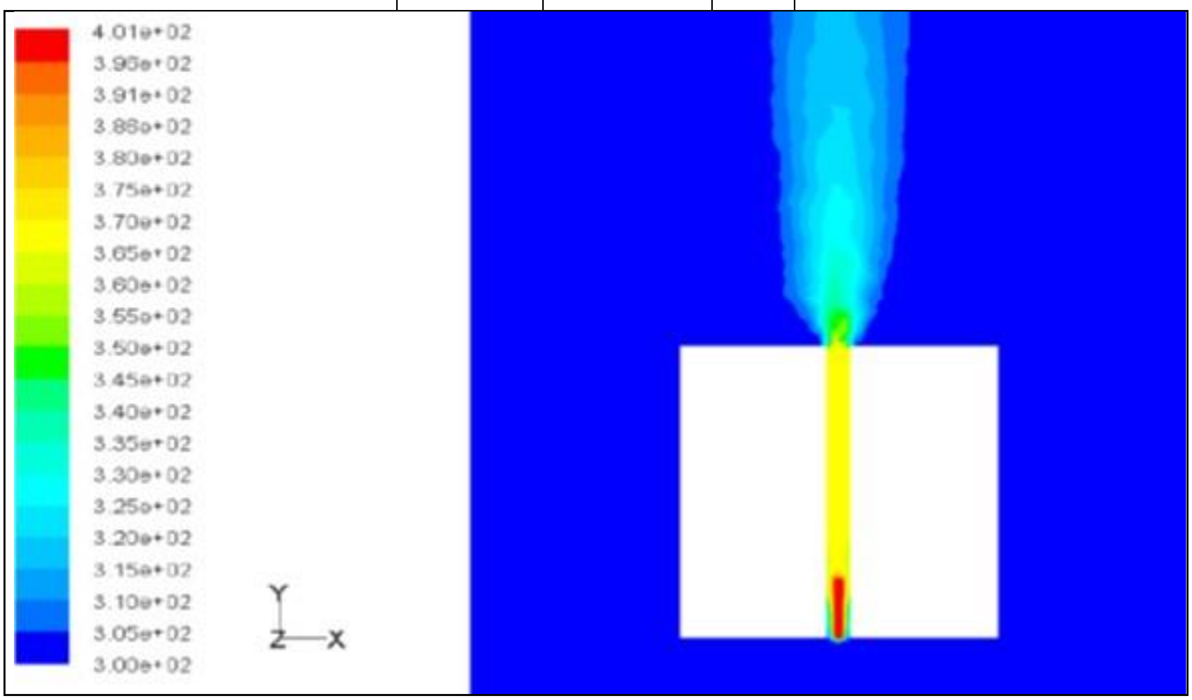

Fig. 2: The variation of Temperature (K)

The contour, fig 2, represents the variation of temperature along the surface of the heated plate. At steady state, the plate temperature is found to be 401 K and the fig 3 represents the variation of flow velocity and maximum flow velocity found to be 70.4 m/s from the simulation.

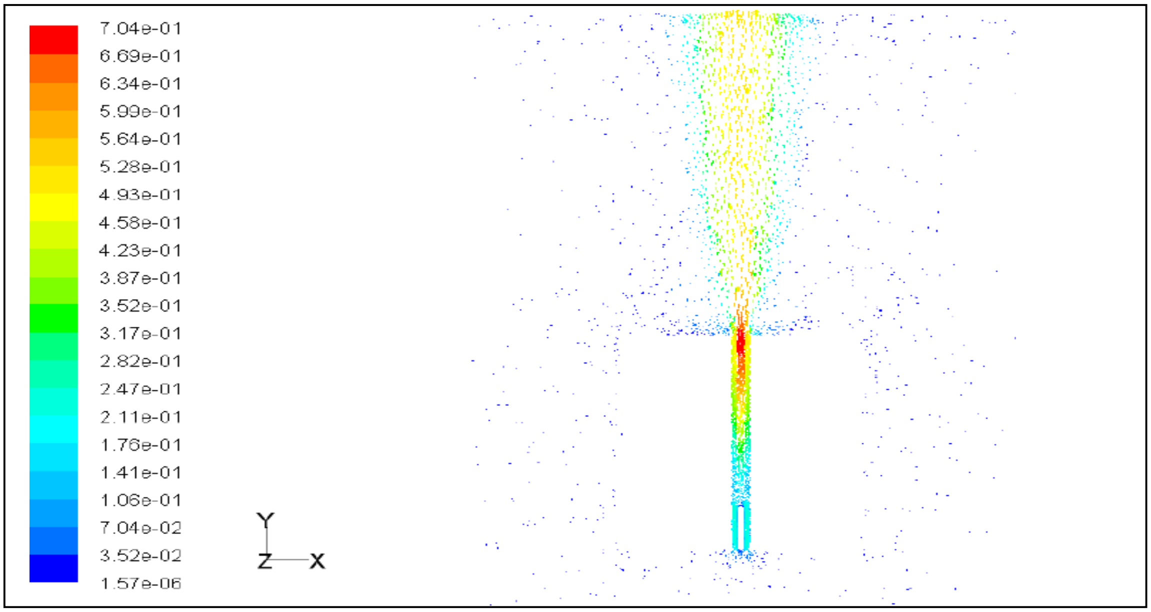

Fig. 3: The vector representation of Velocity (m/s)

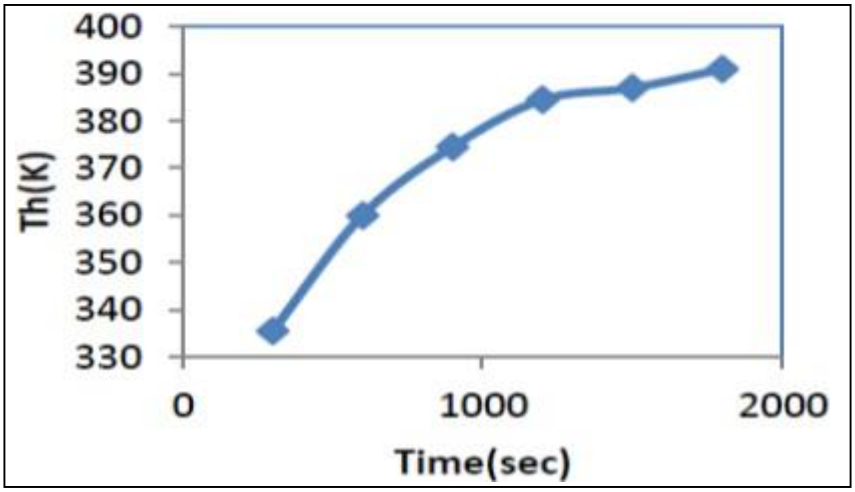

Fig. 4: The variation of surface temperature with respect to the time

The fig 3 represents the variation of surface temperature of the plate with respect to the time. The surface temperature is increases with time and achieves steady state. The steady state will be achieved at 1800s and the stedy state temperature will be 392K by experiment.

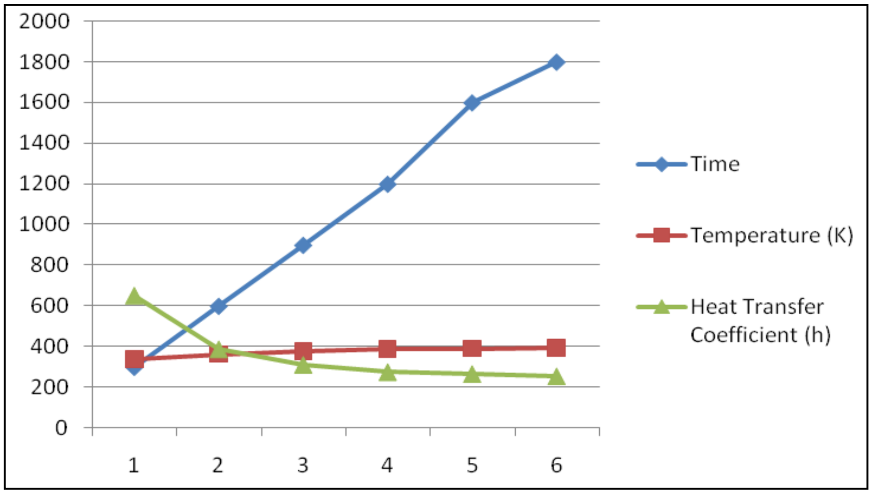

Fig. 5: The variation of surface temperature and heat transfer coefficient with respect to the time

Fig 5 represents the variation of heat transfer coefficient with respect to time. The heat transfer rate decreases with time and achieves an optimum value.

V. CONCLUSION

Natural convection heat transfer can be utilized in many engineering applications such as cooling of electronic equipment's, solar collectors, nuclear reactors, etc. In flow over heated plates, gravity-induced body forces may result from density variations within the fluid. This paper aimed to do a numerical simulation using commercially available CFD software. Here the package ANSYS Fluent-14 used for the analysis. In this work, the heated plate was considered in horizontal orientation. The result obtained were shown that the natural convection result in significant importance in cooling, especially for electronic equipment's. From the analysis the most prominent result obtained is that, the heat transfer between the fluid and solid surface depends upon density variation due to temperature difference. The steady state temperature obtained from the analysis is higher than experimentally obtained value and the variation is 2.2 % of experimental steady state temperature. The phenomenon of natural convention will always present even if other means of cooling, like forced convection, failed to operate properly. So the results highly recommends to consider the effect of natural convection while designing such equipment.

ACKNOWLEDGMENT

The authors wish to thank Prof. Reji Mathew for his support. This work was the result of support from all the staff members and students of PG Studies, Department of Mechanical Engineering, and Mar Athanasius College of Engineering, India. All the equation are taken from Heat and Mass Transfer Fundamentals and Applications 5th Edition, 2015, authored by Yunus A. Cengel.

REFERENCES

Basic

[1] Y. Kitamura, M. Ishizuka, "Chimney Effect On Natural Air Cooling Of Electronic Equipment Under Inclination", IEEE, pp. 77-83.
[2] A. Bar-Cohen, A. D. Kraus, S. F. Davidson, "Thermal Frontier in the Design and Packaging of Micro Electronic Equipments," Journal of mechanical Engineering, vol. 105, no. 6, pp.53-59, 1983
[3] L. T. Yeh,"Review of Heat transfer Technologies in Electronic Equipments" ASME J. Electron. Packaging, vol.117, pp. 333-339, 1995.
[4] W. Elenbaas, "Heat Dissipation of Parallel Plates by Free Convection", Physica, vol.9, pp. 1-28, 1942.
[5] R.A. Wirtz, R.J.Stutzman, "Experiments On Free Convection Between Vertical Plates With Symmetric Heating," ASME, Journal of heat transfer, vol. 104, pp. 501-507, 1982.
[6] Chen Linhui, Tian Huaizhang, Li Yanzhong,Zhang Dongbin, "Experimental Study On Natural Convective Heat Transfer From A Vertical Plate With Discrete Heat Sources Mounted On The Back" ,Energy conservation and management, vol. 47, pp.3447-3455, 2006.
[7] Osterle, "The Development Of Free Convection Between Heated Vertical Plates", ASME Journal of Heat Transfer, vol.84, pp. 40–44, 1962.
[8] W.W.Yousef, J.D.Tarasuk, W.J.P McKeen, "Free Convection Heat Transfer From Upward Facing Isothermal Horizontal Surfaces", Journal of Heat Transfer, vol. 104, pp. 493-500, 1982.
[9] P. Anu Nair, K. Karuppasamy, B. Benziger, P. Balkrishnan, "Natural Convective Heat Transfer From Horizontal Heated Plate Facing Upward in aVertical Channel – A Review," International Journal Mechanical Engineering and Research,vol. 5, no. 1,pp. 27-38, 2015.
[10] Y. Liu, N. Phan-Thein, C. W. Leung, T. L. K, "An Optimum Spacing Problem For Five Chips On A Horizontal Substrate in a Vertically Insulated Enclosure," Comput. Mech., vol. 24 pp. 310-317, 1999.
[11] M. D. Landon, A. Campo, "Optimal Shape For Laminar Natural Convective Cavities Containing Air And Heated From The Side," International Commun. Heat Mass Transfer, vol. 26, pp. 389-398, 1999.
[12] I. Sezai, A. A. Mohamad, "Natural Convection From A Discrete Heat Sourse On The Bottom Of A Horizontal Enclosure," International Journal of Heat and Mass Transfer, vol. 43, pp. 2257-2266, 2000.
[13] H. Y. Wang, F. Penot, J. B Saulnier, "Numerical study of a buoyancy – induced flow along a vertical plate with discretely heated integrated circuit packages," International Journal of Heat and Mass Transfer, vol. 40, pp. 1509-1520, 1997.
[14] Y. Liu, N. Phan-Thien, "An Optimum Spacing Problem for Three Chips Mounted On A Vertical Substrate In An Enclosure," Numer. Heat Transfer, Part A, vol. 37, pp. 613-630, and 2000.
[15] S. Chen, Y. Liu, "An Optimum Spacing Problem For Three-By-Three Heated Element Mounted On A Substrate," Heat and Mass Transfer, vol. 39, pp. 830-842, 2000.
[16] A. K. da Silva, S. Lorente, A. Bejan, "Optimal distribution of discreteheat source on a wall with natural convection," International Journal for Heat and Mass Transfer, vol. 47, pp. 203-214, 2004.
[17] M. Corcione, "Heat Transfer Correlations For Free Convections From Upward Facing Horizontal Rectangular Surface", 5 th WSEAS Int. Conf. on Heat and Mass Transfer, vol. 2, pp. 48-60, 2007.
[18] K. Stewartson, "On Free Convection from A Horizontal Plate", ZAMP, vol. 9, pp. 276-282, 1958.
[19] Hassan, K. E. Mohamed, "Natural convection from iso thermal flat surfaces," Int. Jou. Of Heat and Mass Transfer, vol. 13, pp. 1873-1886, 1970.
[20] P. Anu Nair, K. Karuppasamy, "Comparative Study Of Bayesian Approach And Laast Sqare Residual Optimization Method In Horizontal Heated Plate Facing Upward – An Experimental Approach," International Journal of Research in Aeronautical and Mechanical Engineering, vol.3, no. 4, pp.7-18,2015
[21] ANSYS Inc, Fluent, 2014, Computational Fluid Dynamics Software, Canonsburg, USA

Book References

[22] Yunus A. Cengel - Heat and Mass Transfer Fundamentals and Applications 5th Edition, 2015