TWO-DIMENSIONAL FINITE ELEMENT ANALYSIS OF FORCED CONVECTION FLOW AND HEAT TRANSFER IN A LAMINAR CHANNEL FLOW

Rajesh Khatri[1],

[1]M.Tech Scholar, Department of Mechanical Engineering, S.A.T.I., vidisha (M.P.) 464001, India.
khatri_rajesh234@rediffmail.com

Pankaj Agarwal[2] ,

[2]Professor in Department of Mechanical Engineering, S.A.T.I., vidisha (M.P.) 464001, India.
dr_pankajagarwal@rediffmail.com

Abstract:
In this paper heat transfer and fluid flow characteristics in a channel has been theoretically investigated. In this study, FEM is employed to analyze a fluid flow inside a channel and then solve for the heat flow transfer through the same channel. The fluid flow is expressed by partial differential equation (Poisson's equation). While, heat transfer is analyzed using the energy equation. The Navier Stokes equations along with the energy equation have been solved by using simple technique. The domain is discretized using 2626 elements and that corresponds to a total number of nodes 2842. The channel has a constant heat flux at the two walls and the three dimensional numerical simulations. Numerical solutions were obtained using commercial software Ansys Fluent. The working fluid was air (Pr=0.7). The local Nusselt numbers are obtained, which can be used in estimation of flow and heat transfer performance in a channel In addition, local Nusselt numbers, velocity magnitude and temperature profiles, and pressure profiles are analyzed. Results showed that both fluid flow and temperature flow are influenced significantly with changing entrance velocity. The overall objective of this paper is to study the flow characteristics and heat transfer analysis inside a channel while increasing entrance velocity.

Keywords: Laminar flow, Forced Convection, Finite Element Method, Fluid Flow, heat transfer

1. INTRODUCTION

Finite element method techniques are widely used in problems that require solution of partial differential equations. In this method of analysis, a complex region defining a continuum is discretized into simple geometric shapes called finite elements. The material properties and the governing relationships are considered over these elements and expressed in terms of unknown values at element corner. An assembly process duly considering the loading and constraints, results in a set of equations. Solution of these equations gives us the approximate behaviour of the continuum. Thompson [1] discusses solution of partial differential equations involved in areas such as Fluid Mechanics, Elasticity and Electromagnetic Field by using FEM. Details about fluid mechanics and heat transfer problems and their solution can be found in [2]. A more complex transient heat conduction equation is discussed in Winget and Hughes [3]. Similarly Johan et al. [4] and Jacob and Ebecken [5] develop step size selection schemes based on heuristic rules for compressible Navier-Stokes equations and structural dynamics problems respectively. H. Shokouhmand, S.M.A. Noori Rahim Abadi [6-7] numerically studied the mixed convection heat transfer through a vertical wavy isothermal channel. W. Liu, A.Y. Shehata, S. Ali present finite element modeling of non-viscous in-compressible fluid flow and convective temperature flow. Many researchers have worked on heat transfer and flow field with bluff bodies. Convective heat transfer or, simply, convection is the study of heat transport processes by the flow of fluids. Problems related to convective heat transfer rest on basic thermodynamics and fluid mechanics principles, which essentially involved with partial differential equations.

Within the scope of thermal engineering, energy conservation and sustainable development demands have been driving research efforts towards more energy efficient equipments and processes. The petroleum and process industries have been quite active in progressively incorporating heat transfer enhancement solutions to the efficiency increase requirements along the years. More recently, heat exchangers employing micro-channels with characteristic dimensions below 500 microns have been calling the attention of researchers and practitioners, towards applications that require high heat removal demands and/or space and weight limitations.

In this paper, a two-dimensional steady flow problem is solved using the finite element method through solving partial differential equations of the fluid flow. The flow field of that fluid is then employed to solve the partial differential equations of temperature flow. For both fluid flow and temperature flow, boundary conditions are applied. The domain of the problem is discretized to a large number of elements to assure the accuracy of the solution. Four-node isoparametric quadrilateral elements are used to model this problem. Analyses are done by studying both flow field and temperature field for various values of entrance velocity. Analyses show very important results, such as the temperature field is decreasing by increasing the entrance velocity.

2. NUMERICAL MODELLING

The numerical simulation is performed in a two dimensional laminar channel flow. A fluid passes through a domain represented by a channel of height (H = 6cm) and length (L=2500cm) in a two dimensional plan as shown in figure.1. The fluid flow enters from the left with a uniform velocity V, as illustrated. The channel has two heat fluxes of intensity 37.5w/m^2 and 68.6w/m^2. As the fluid passes through the domain, the velocity of the fluid changes due to the presence of the small pipe at the middle. Also, due to convection phenomenon, heat energy is carried out by the fluid flow through the pipe domain. Thus, the temperature flow field changes by changing the velocity of the fluid flow. By assuming that the fluid is incompressible with constant properties, both velocity and temperature fields are independent of each other. As such, the governing partial differential equations that describe the fluid flow and the temperature flow can be solved independently. Air has been taken as working fluid for which Prandlt number is 0.71.

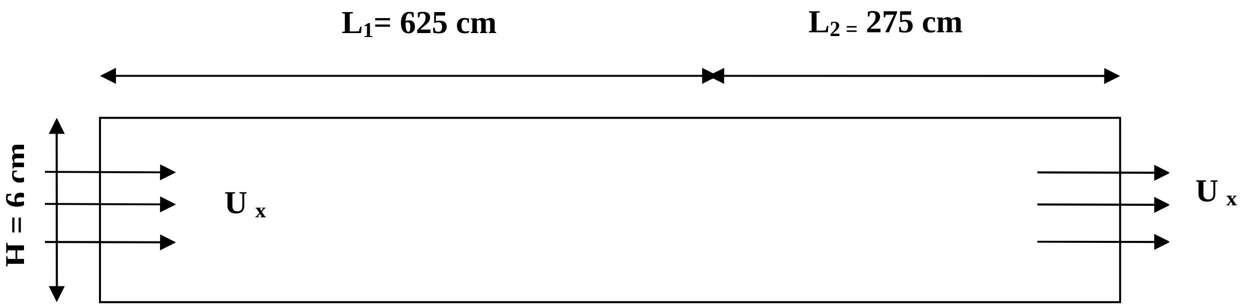

Fig. 1 Geometry of Channel

The governing equations i.e. continuity, momentum, and energy equations are used to simulate the incompressible steady flow in a given computational domain.
The continuity equation in two dimensional for incompressible flow is:

$$\frac{\partial U}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (1)$$

The momentum equations are:

$$\frac{\partial u}{\partial x}+\frac{\partial u^2}{\partial x}+\frac{\partial (uv)}{\partial y}=\frac{\partial p}{\partial x}+\frac{1}{Re}\left(\frac{\partial u}{\partial x}+\frac{\partial u}{\partial y}\right) \qquad (2)$$

$$\frac{\partial u}{\partial x}+\frac{\partial v^2}{\partial x}+\frac{\partial (uv)}{\partial x}=\frac{\partial p}{\partial x}+\frac{1}{Re}\left(\frac{\partial v}{\partial x}+\frac{\partial v}{\partial y}\right) \qquad (3)$$

The energy equations are:

$$\frac{\partial \theta}{\partial \tau} + \frac{\partial u\theta}{\partial x} + \frac{\partial (v\theta)}{\partial y} = \frac{1}{Pr\ Re}\left(\frac{\partial^2 \theta}{\partial x^2} + \frac{\partial^2 \theta}{\partial y^2}\right) \quad (4)$$

3. Boundary Conditions

A uniform one dimensional velocity is applied at the inlet of computational domain. The pressure at the outlet of the computational domain is taken equal to zero gauge. The inlet temperature of air is considered to be uniform at 300K. On the walls, no slip boundary conditions are applied for momentum equations. A constant heat flux of 37.5w/m^2 and 68.6w/m^2 is applied on two surfaces.

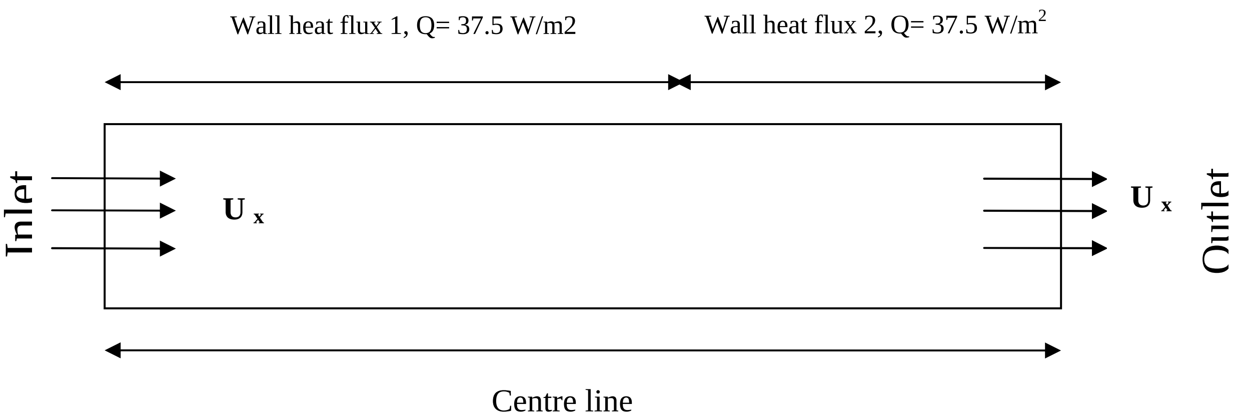

Fig.2 Boundary conditions of the fluid flow

The commercial CFD software Fluent 12.0 is used to simulate the fluid flow and temperature field. A preprocessor GAMBIT is used to generate the required mesh for the solver. The governing equations are discretized using the finite volume method. The simple algorithm is used for the convective terms in the solution equations. The second order up winding scheme is used to calculate the flow variables. The solver iterates the equations till the convergence is obtained for the set residuals.

4. Results and Discussion

The computations are carried out in a parallel plate channel of length 2500 cm and height 6 cm. The flow structure and heat transfer characteristics are obtained for different values of entrance velocity U_x= 0.1, 0.5, 0.8, 1.1 m/s

4.1. *Flow Characteristics and Heat Transfer Characteristics for U_x= 0.1, 0.5, 0.8, 1.1 m/s*

4.1.1. Flow Characteristics

The time averaged flow structure is seen by looking at velocity vector plots. Figure 2-5 shows the velocity vector plot for entrance velocity U_x = 0.1, 0.5, 0.8, 1.1 m/s

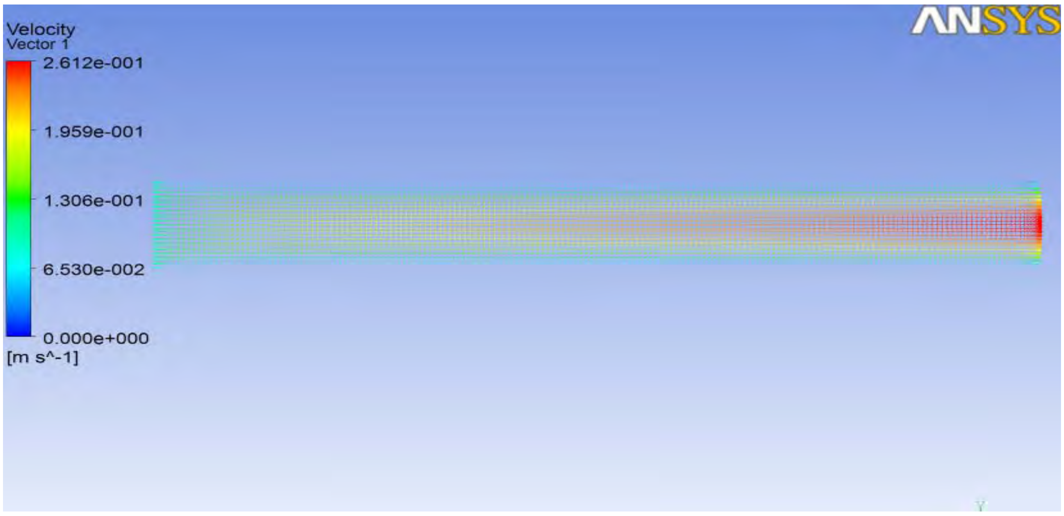

Fig.3 Velocity vector plot for U x = 0.1 m/s

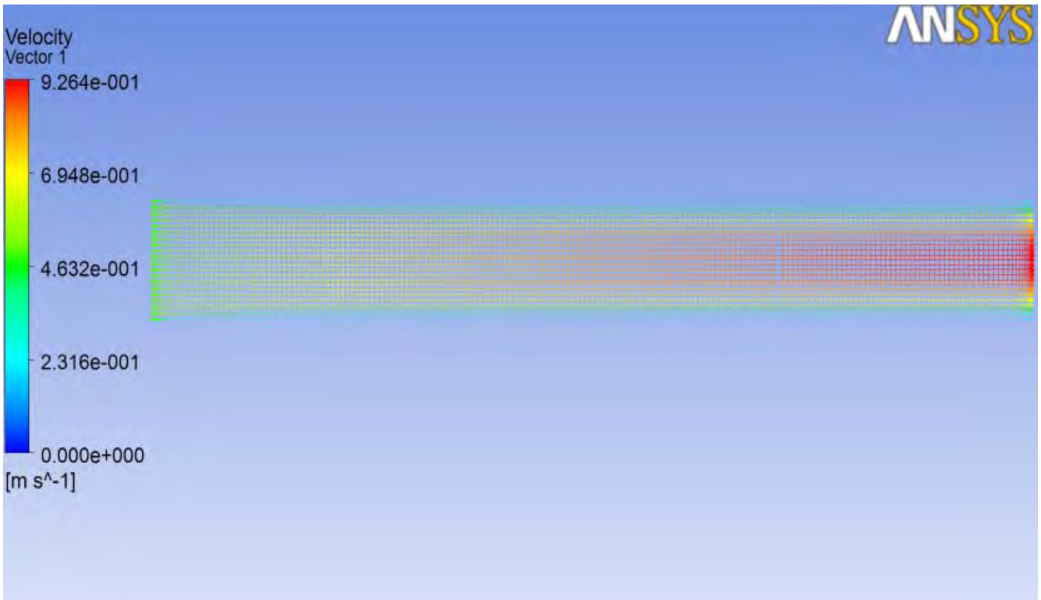

Fig.4 Velocity vector plot for U x = 0.5 m/s

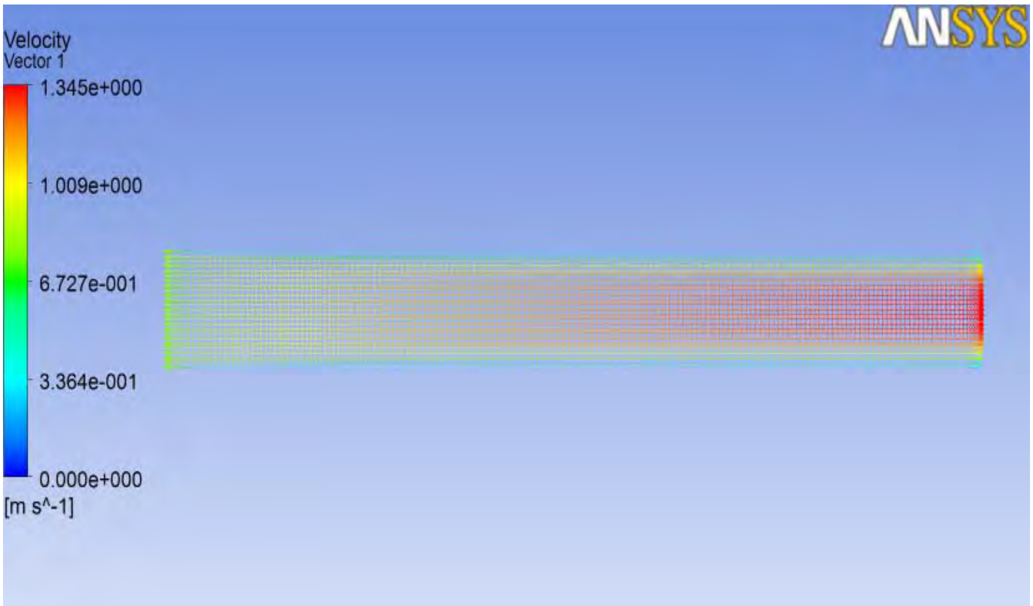

Fig.5 Velocity vector plot for U x = 0.8 m/s

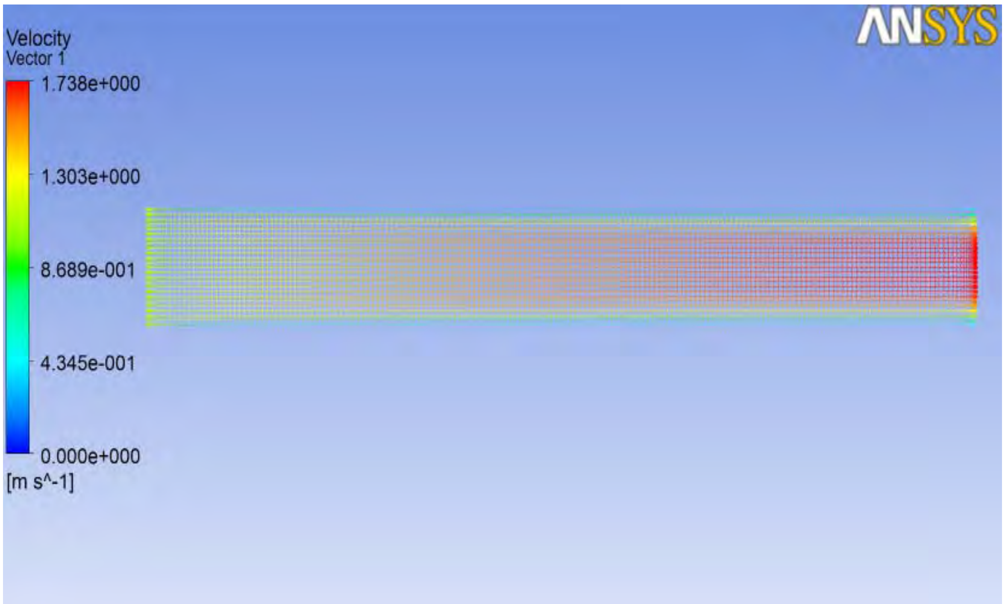

Fig.6 Velocity vector plot for U x = 1.1 m/s

4.1.2. Temperature contours and Heat Transfer Characteristics when U x= 0.1m/s.

Figure 6 shows the temperature contours of the computation domain of plane channel. Plot shows that the fluid temperature is increasing in x- direction.
Figure 7 shows the variation of temperature along the channel length for U_x= 0.1, 0.5, 0.8, 1.1 m/s. Plots shows that temperature is increasing along the channel length and is maximum when entrance velocity is U_x= 0.1 m/s.
Figure 8 shows the Surface Nusselt number variation on the top and bottom wall of the channel.

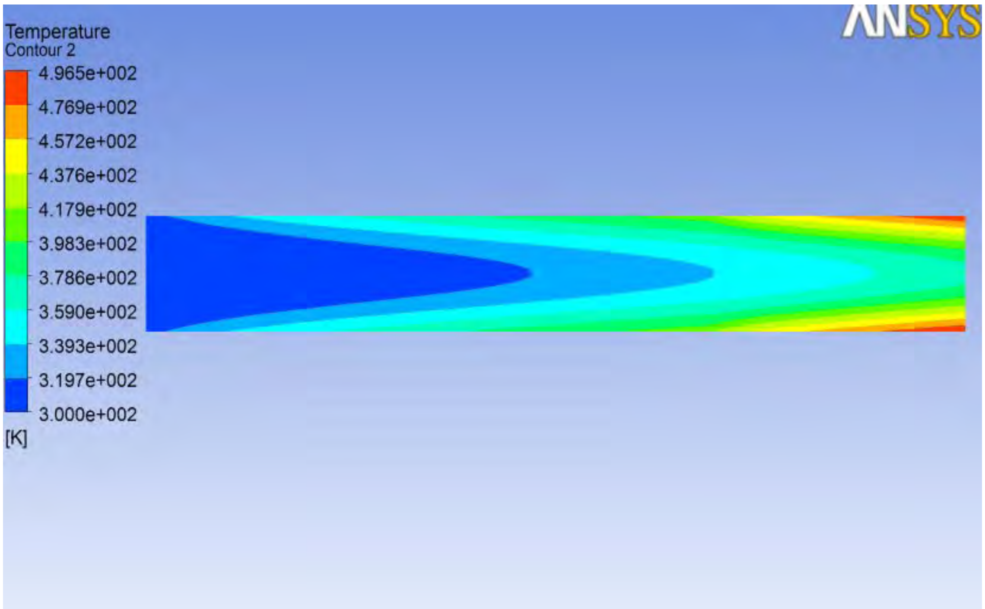

Fig.7 Temperature contours of computation domain for U x = 0.1 m/s

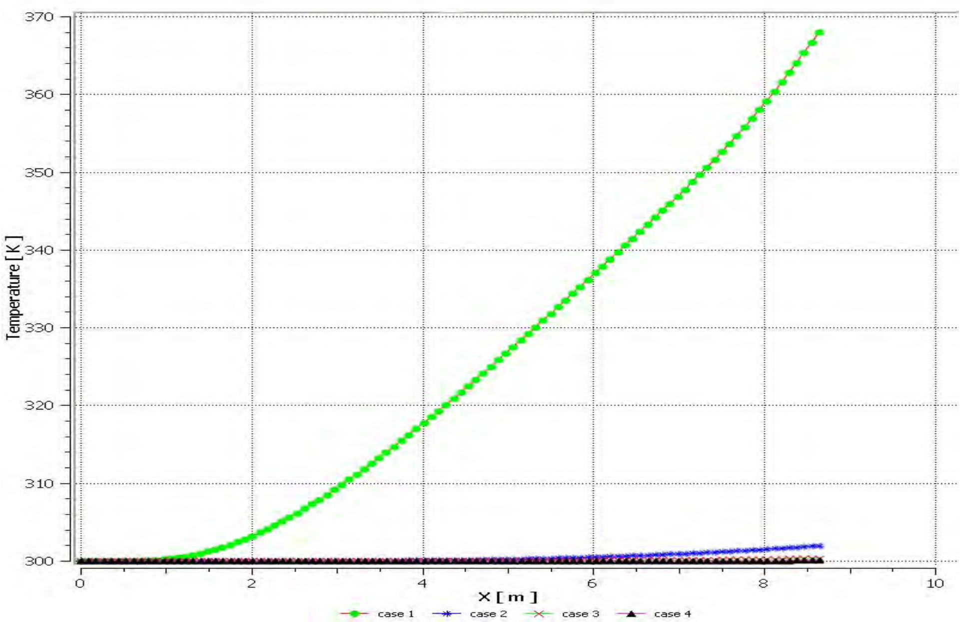

Fig.8 Variation of temperature along channel length for U_x= 0.1, 0.5, 0.8, 1.1 m/s

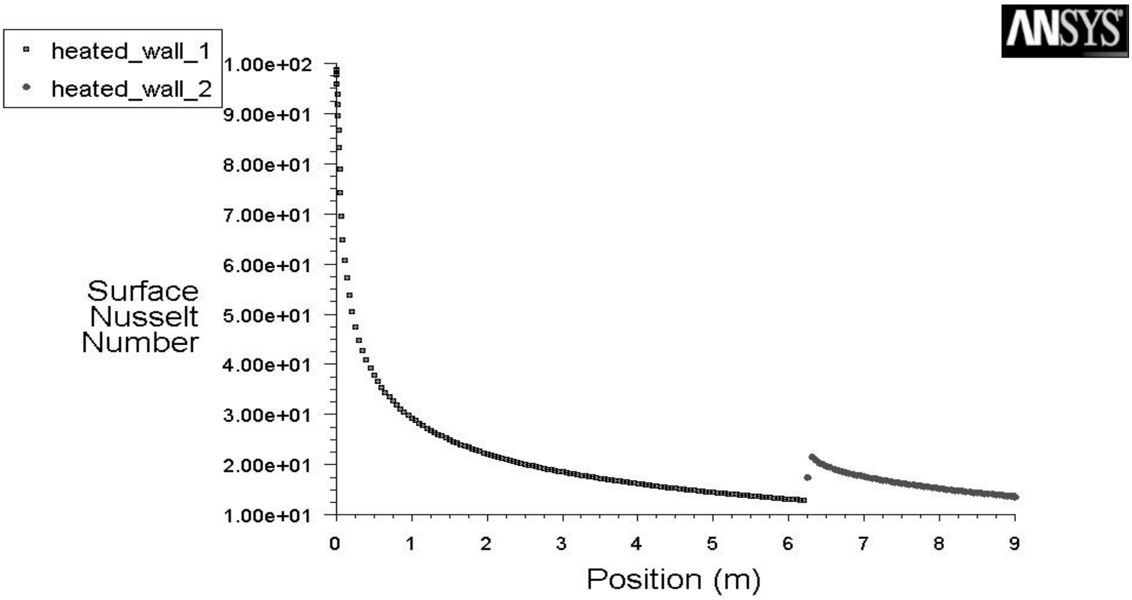

Fig. 9 Variation of Nusselt Number along channel length

4.1.3. Temperature contours and Heat Transfer Characteristics when U x= 0.5m/s.

Figure 9 shows the temperature contours of the computation domain of plane channel. Plot shows that the fluid temperature is increasing in x- direction.
Figure 10 shows the Surface Nusselt number variation on the top and bottom wall of the channel.

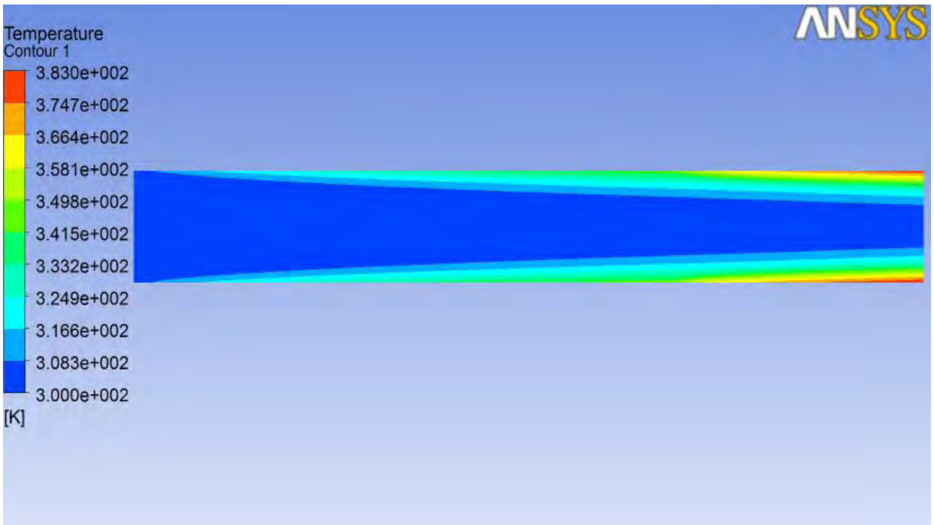

Fig.10 Temperature contours of computation domain for U x = 0.5 m/s

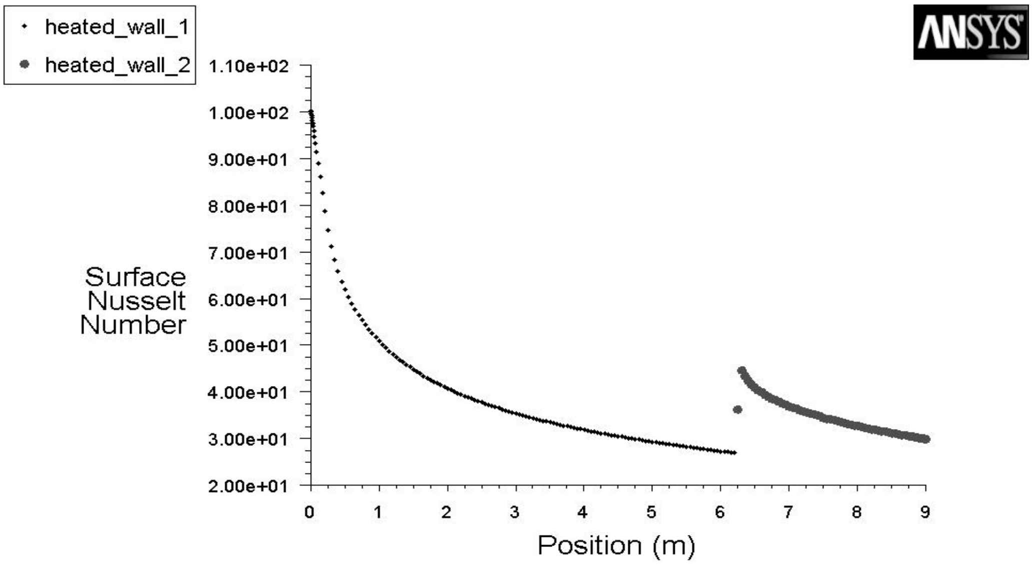

Fig. 11 Variation of Nusselt Number along channel length

4.1.4. Temperature contours and Heat Transfer Characteristics when U x= 0.8m/s.

Figure 11 shows the temperature contours of the computation domain of plane channel. Plot shows that the fluid temperature is increasing in x- direction.
Figure 12 shows the Surface Nusselt number variation on the top and bottom wall of the channel.

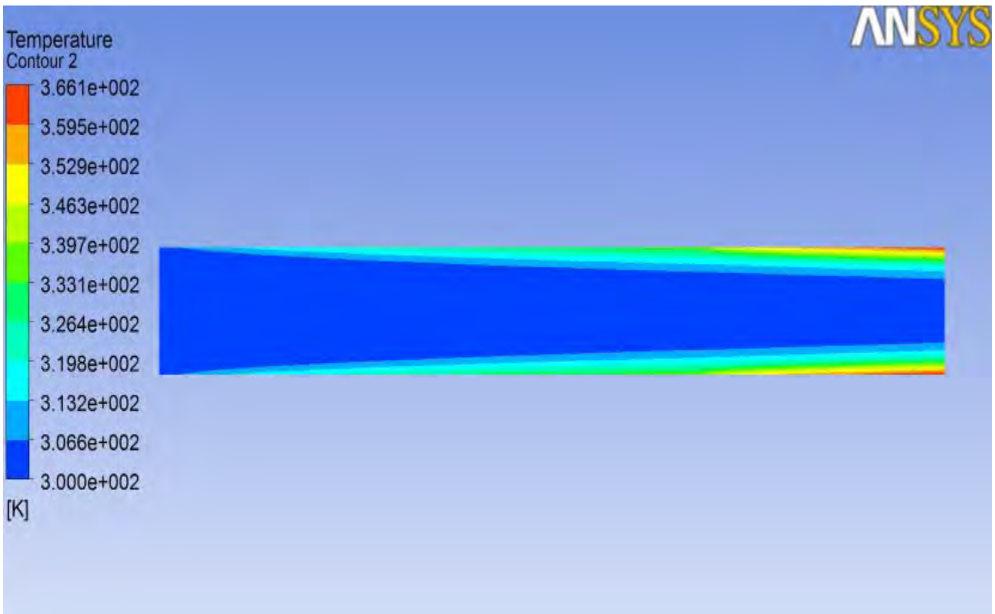

Fig.12 Temperature contours of computation domain for U x = 0.8 m/s

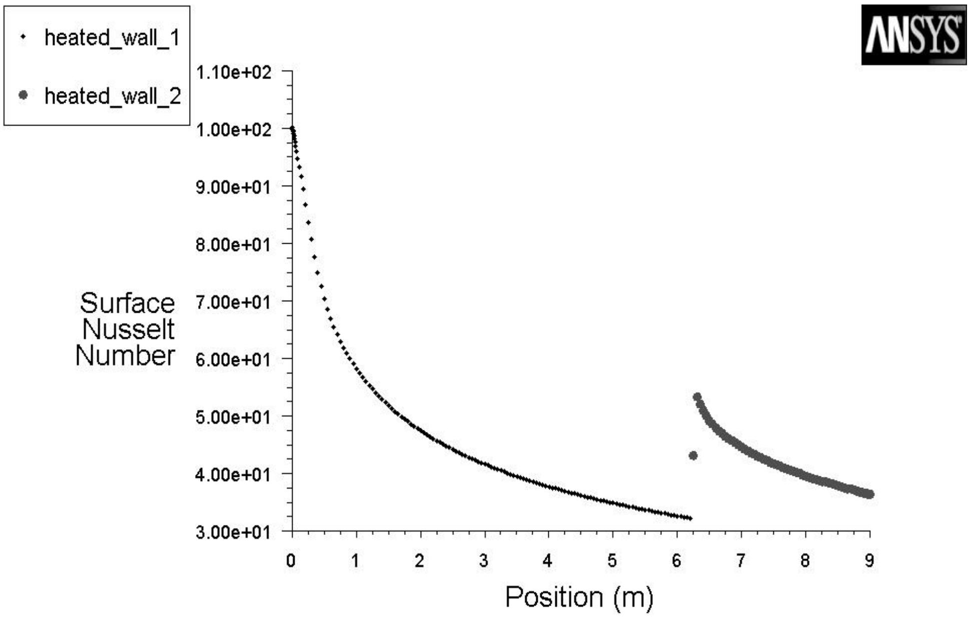

Fig. 13 Variation of Nusselt Number along channel length

4.1.5. Temperature contours and Heat Transfer Characteristics when U x= 1.1m/s.

Figure 13 shows the temperature contours of the computation domain of plane channel. Plot shows that the fluid temperature is increasing in x- direction.
Figure 14 shows the Surface Nusselt number variation on the top and bottom wall of the channel.

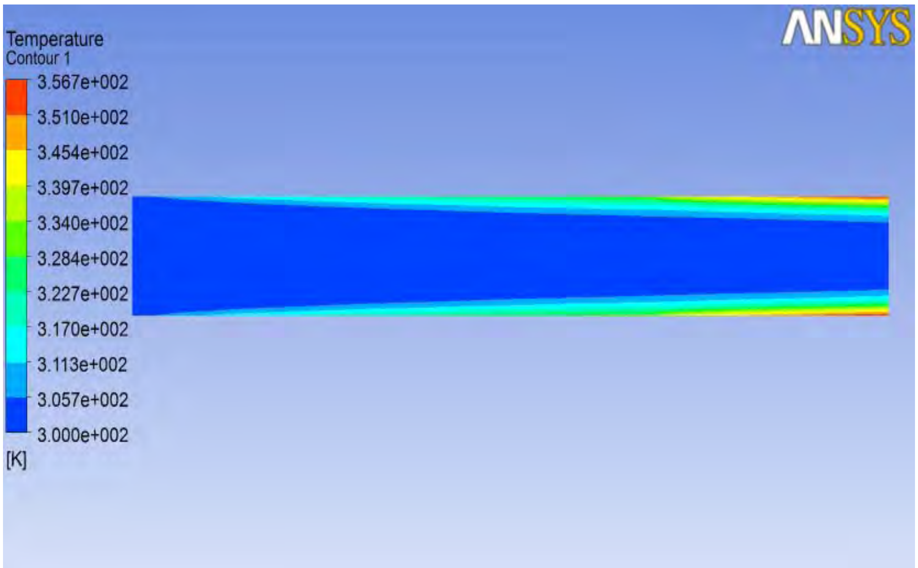

Fig.14 Temperature contours of computation domain for U x = 1.1 m/s

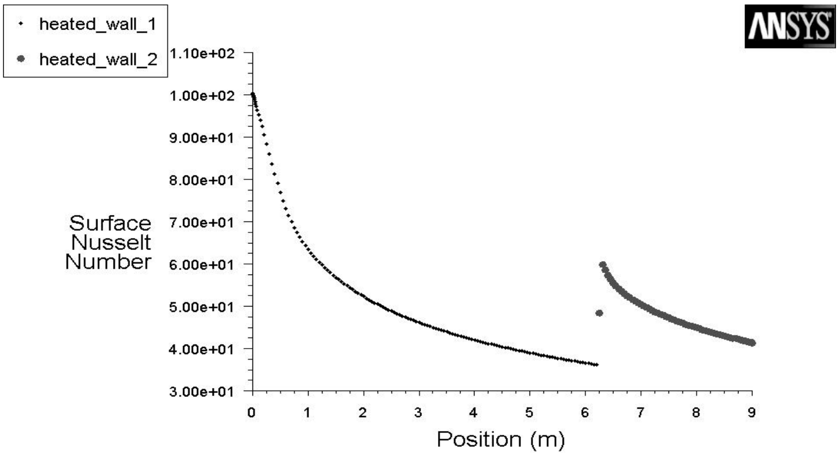

Fig.15 Variation of Nusselt Number along channel length

4.1.6. Pressure Characteristics when U x= 0.1, 0.5, 0.8 1.1 m/s.

The heat transfer enhancement is achieved at the cost of pressure loss. Figure 15 shows the pressure variation along the channel length. Plot shows that maximum pressure loss is achieved in case IV and minimum in case I.

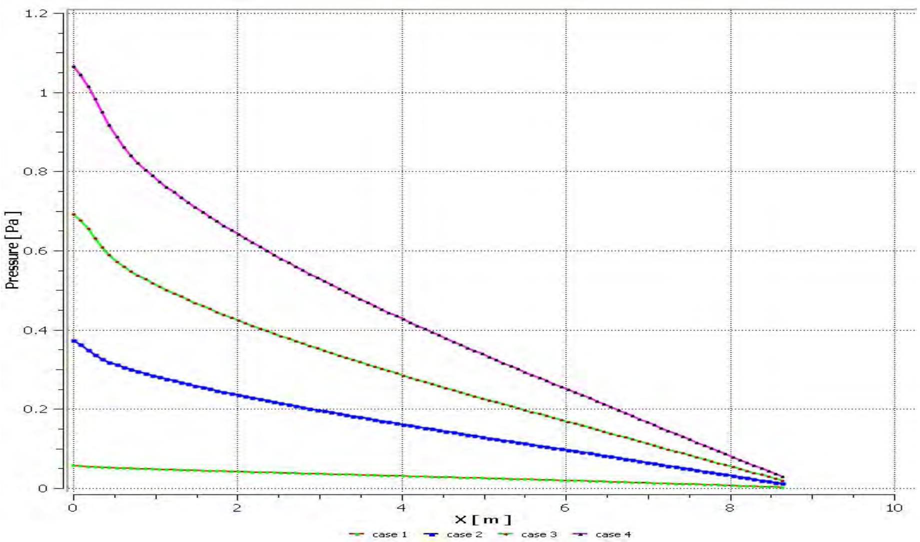

Fig. 16 Pressure variation along the channel length

5. Conclusions

In the present problem the numerical simulation of laminar flow in a parallel plate channel for different entrance velocity is analyzed using the finite element method through solving partial differential equations of the fluid flow. The fluid flow is expressed by partial differential equation (Poisson's equation). While, heat transfer is analyzed using the energy equation. The flow field of that fluid is then used to solve the partial differential equations of temperature flow. The influence of the presence of heat flux outside the domain on the temperature flow field is also investigated. Analyses are done by studying flow field, temperature field and pressure for various values of entrance velocity. Results showed that fluid flow, temperature flow and pressure are influenced considerably by changing entrance velocity.

The flow structure and heat transfer characteristics are studied in detail.

On the basis of the results obtained the following conclusions are made.

(1) The effect of entrance velocity on flow and thermal filed is presented graphically. Results show that with increasing velocity flow pattern and temperature changes.

(2) Local Nusselt number of wall is decreases gradually through the channel length.

(3) Pressure is also decreases with increasing entrance velocity.

References

[1] Thompson, Erik G.(2004): Introduction to the Finite Element Method: Theory Programming and Applications, John Wiley & Sons Inc.
[2] Bejan A(1984): Convection Heat transfer, John Wiley and Sons Inc.
[3] Winget J. M, Hughes T.J.R. (1985): Solution algorithms for nonlinear transient heat conduction analysis employing element by-element iterative strategies, Computer Methods in Applied Mechanics and Engineering, 52:711–815.
[4] Johan Z, Hughes T.J.R, Shakib F. (1991): A globally convergent matrix-free algorithm for implicit time-marching schemes arising in Finite element analysis in Fluids, Computer Methods in Applied Mechanics and Engineering ,87:281–304.
[5] Jacob BP, Ebecken NFF (1993): Adaptive time integration of nonlinear structural dynamic problems. European Journal of Mechanics and Solids; 12(2):277–298.
[6] Shokouhmand H, Abadi Rahim Noori S.M.A, (2010): Proceedings of the world congress on Engineering Vol. II, pp. 1–6.
[7] Shokouhmand , H (1980) : Local Heat Transfer Coefficient in Natural Convection Transition Region " Reg .J.Energy ,Heat & Mass Transfer ,Vol. 2 No ,2.pp.113-125.