A STUDY OF STREAM LINE AND NUMERICAL MODELING AND HEAT TRANSFER PROBLEMS IN FLUID DYNAMICS.

[1]S.BHUVANESWARI [2] R. PADMAPRIYA .

[1] A sst .Prof.. and Head Dept. of Mathematics, Kamban college of Arts and Science for women

[2] Reasearch Scholar., Dept. of Mathematics, Kamban college of Arts and Science for women .

ABSTRACT

In this paper, we using A numerical investigation of laminar flow in a two-dimensional, Cartesian flow that exits from a short channel with a backward-facing step is carried out in this work for the Reynolds number range of $0.00054 \leq Re \leq 54$. We studied the steady state fluid dynamics, phase change and heat transfer of the flow. We analyzed the flow behavior occurring for different outflow velocities and three different configuration of the step . This paper is organized as follows in section we adopt some usual terminology, notations and conventions which will be used later in the section we establish In order to obtain a better understanding of the step angle influence on the fluid dynamics, we obtained the heat transfer flux rates and the axial velocity profiles have also been presented.

***Keywords*:** Numerical simulations; Fluid Dynamics; Heat transfer; Glass; Finite element.

1.1 INTRODUCTION

In mathematics, fluid dynamics is a sub discipline of fluid mechanics that deals with fluid flow—the natural science of fluids (liquids and gases) in motion. It has several sub disciplines itself, including aerodynamics (the study of air and other gases in motion) and hydrodynamics (the study of liquids in motion). Fluid dynamics has a wide range of applications, including calculating forces and moments on aircraft, determining the mass flow rate of petroleum through pipelines, predicting weather patterns, understanding nebulae in installer space and modeling fission weapon detonation. Some of its principles are even used in traffic engineering, where traffic is treated as a continuous fluid, and crowd dynamics. Fluid dynamics offers a systematic structure—which underlies these practical disciplines that embraces empirical and semi-empirical laws derived from flow measurement and used to solve practical problems. The solution to a fluid dynamics problem typically involves calculating various properties of the fluid, such as flow velocity, pressure, density, and temperature, as functions of space and time. Before the twentieth century, hydrodynamics was synonymous with fluid dynamics. This is still reflected in names of some fluid dynamics topics, like magneto hydrodynamics and hydrodynamic stability, both of which can also be applied to gases. The theoretical study of the motion of the fluid is the most successful, fascinating and useful application of mathematics.

An important issue in fluid dynamics is the study of fluid flow and heat transfer in channels with phase change, this being the topic of a range of studies because it is present in many technical applications and natural processes. Giessler and Schlegelm (2008) compared a two-dimensional axisymmetric simulation of a glass melt flowing through a pipe with a one-dimensional model. The fluid was supposed to be heated by internal electromagnetic heating and cooled at the wall by convection. It was considered to be a laminar and steady flow of a viscous electrically-conducting fluid in a pipe with circular cross-section driven by a pressure difference between the inlet and the outlet of the pipe. The highly viscous fluid with constant density was supposed to have strongly temperature-dependent viscosity and electrical conductivity. The steady state flow was governed by the Navier-Stokes equation, the incompressibility condition and the energy equation. The dependence of the mean velocity on the applied pressure difference and the dependence of the mean outlet temperature on the applied pressure difference were analyzed for heating without cooling and cooling without heating. Mirbagheri et al. (2003) simulated the flow of an incompressible molten metal in a mould cavity. They developed a numerical code and proposed a new algorithm with free and solid boundary condition to investigate the heat transfer and the effect of the coating permeability during the mould filling using finite differences. This problem was modeled with the momentum equations, the continuity equation and the heat transfer equation for the liquid or solid zone; the heat transfer equation was used for the mushy region, while the free surface function and the backpressure function were utilized for low melting point metals. Velocity profile, temperature field and filling sequence were obtained from the simulations.

To this end, the governing equations were discretized with the finite difference method and a semi-explicit solution of this approximation was obtained to compute the first guess for new time-level velocities using the initial conditions or previous time-level values for all advective, pressure and viscous accelerations. To satisfy the continuity, pressure was iteratively adjusted in each cell and the velocity changes induced by each pressure change are added to the velocity computed previously. In most cases, an over-relaxation factor is used to accelerate the convergence of the iteration. Shmueli et al. (2008) investigated the process of melting of a phase change material in a vertical tube. The numerical simulations were done with fluent and the flow patterns and a detailed distribution of heat transfer on the inner wall of the tube were obtained. The results of the simulation and the analysis of digital pictures, obtained in previous studies, reveal that the effect of convection in the liquid phase provokes non-radial heat fluxes from the wall to the melting zone. The conservation equations for air were solved in the domain bounded by the perimeter of the inner walls, phase change melting from below, and the upper boundary of the tube from above. Boundary conditions for the momentum equation were no-slip at all solid surfaces and slip at the upper boundary of the tube, while for the energy equation constant and uniform temperature on the outside wall of the tube, zero heat flux at the bottom, and ambient air temperature at the upper boundary of the tube were considered. Alexiades et al. (2003) studied the effect of grid and scheme on the numerical solution of a tin melting flow. Simulations were carried out in a square cavity filled with tin (pure metal) initially at freezing temperature (solid tin). The top and bottom boundaries were considered to be adiabatic, while the right boundary was maintained at constant temperature. At the beginning of the simulation, the left boundary was suddenly brought to a hot temperature higher than the melting temperature. Thermal gradients generated density gradients, which provoked convection in the liquid tin. The shape of the solid-liquid interface, the flow pattern in the liquid and the heat transfer process were analyzed. Giannopapa (2006) developed a computational model to simulate the blow-blow forming process of glass containers that characterized the glass flow and the heat transfer. He was able to track the geometry of the deforming interface of glass applying structured and unstructured fixed meshes. The Navier-Stokes equations were used to describe the motion of the hot glass, while the energy equation was used to describe the heat transfer between the glass and the mould. The governing equations were discretized by the finite element method. In this work, the propagation of glass during the forming process and the temperature profiles were analyzed. Salinas (2006) studied numerically the solidification of aluminum alloy in a square cavity considering the flow of an incompressible Newtonian fluid with isotropic thermal properties where the phase change and the heat transfer by conduction and convection were considered. The mathematical model was based on a set of partial differential equations given by momentum, continuity and heat transfer equations.

The discretization of the governing equations was done by the finite volume method combined with a SIMPLER scheme to join pressure and velocities. Transient results of the non-isothermal solidification were obtained and the streamlines and isotherms analyzed, the results showing a complex flow with secondary vorticity. The goal of the present work is to extend the one-dimensional study reported by Jimenéz (2002), providing insight into the velocity, the phase change and the temperature fields, as well as to analyze the heat fluxes and axial velocity profiles. The flow is assumed to be two-dimensional because the major changes occur only in the axial and transversal directions. The present research is of particular relevance for the glass industry because we have greatly extended current knowledge on how vortices and heat transfer interact with the geometry of the step channel. Furthermore, this fluid thermal interaction of glass flow underpins much of the behavior of the material properties.

1.2 LITERATURE REVIEW

The convection driven by two different density gradients with differing rates of diffusion is widely known to as ―double-diffusive convection‖ and is an important fluid dynamics phenomenon. The study of double-diffusive convection has attracted attention of many researchers during the recent past due to its occurrence in nature and industry. The problems of time-dependent double-diffusive (mass and heat) flow over a semi-infinite vertical plate have been studied extensively by many researchers. [12] investigated transient free convective flow past a semi-infinite vertical plate with mass transfer, and analyzed the finite difference of transient free convection with mass transfer of an isothermal vertical flat plate. [7] Examined the effect of radiation in convective flow over a cone. [5] studied the effects of varying viscosity and thermal conductivity on steady free convective flow and heat transfer along an isothermal vertical plate in the presence of heat sink. [8] investigated similarity solution for laminar thermal boundary layer over a flat plate with a convective surface boundary condition. One-dimensional time-dependent double-diffusive mass & heat flow over a semi-infinite vertical plate, under a convective surface boundary condition has been studied in this chapter. Using Similarity Transforms, the governing nonlinear partial differential equations have been transformed into a set of coupled nonlinear ordinary differential equations, which are solved by Homotopy Perturbation Method.

1.3 STREAM LINE FLOW AND HEAT TRANSFER PROBLEM

Two-dimensional flow of a viscous incompressible fluid of variable thermal conductivity in neighborhood of a stagnation point on a non-conducting stretching sheet is considered. The stretching sheet is located in the plane *y* = 0 and *x*-axis is taken along the sheet the fluid occupies the upper half plane i.e. *y*> 0. The governing equations of flow are:

The second derivatives of u and T with respect to x have been eliminated on the basis of magnitude analysis considering that Reynolds number is high. Hence the Navier-Stokes equation modifies into Prandtl's boundary layer equation.

In the free stream u = U(x) = bx, the equation (2) reduces to

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = U\frac{dU}{dx} + \upsilon\frac{\partial^2 u}{\partial y^2}. \qquad (1.1)$$

The corresponding boundary conditions are.

$$y \to 0: \quad u = u_w\ x \to c \quad v \to 0, \quad T \to T_w, \qquad (1.2)$$

$$y \to \infty: \quad u = U\ x \to bx, \quad T \to T_\infty \qquad (1.3)$$

Method of Solution

Introducing the stream function ψ (x, y) as defined by

$$u = \ , \ \frac{\partial \psi}{\partial y} \qquad (1.4)$$

$$v = -\frac{\partial \psi}{\partial x} \qquad (1.5)$$

And the similarity variable

$$\varepsilon = (c / \nu)^{1/2}\, y \qquad (1.6)$$

$$\Psi(x, y) = (c\,\nu)^{1/2}\ x\, f(\varepsilon), \qquad (1.7)$$

into the equations (4) and (5), we get

$$f''' + ff'' - (f')^2 + \lambda^2 = 0, \qquad (1.8)$$

And

$$(1+\varepsilon)\,\zeta'' + \varepsilon(\zeta')^2 + \Pr\zeta' f + \Pr S\,\zeta = 0. \qquad (1.9)$$

where ε-perturbation parameter , ε-similarity parameter { = $(c/\nu)^{1/2}$ y} , ε_∞-value of ε at which boundary conditions is achieved ,κ-uniform thermal conductivity ,κ*- variable thermal conductivity ,ν-kinematic viscosity ,ρ-density of fluid ,ψ-stream function , ζ-dimensionless temperature{ = ($T - T_\infty$) / ($T_w - T_\infty$)} , η_w-shear stress ,S- heat source/sink parameter {= Q/ρ C_p c } , T- fluid temperature

The governing boundary layer and thermal boundary layer equations (1.8) and (1.9) with the boundary conditions (1.6) and (1.7) are solved using Homotopy perturbation method. Introducing the following

Homotopy : boundary conditions, we get:

$$\mathbf{f_0(\eta) = \lambda\eta - e^{-\eta} + \lambda e^{-\eta} - \lambda + 1}, \tag{1.10}$$

$$\mathbf{f_1(\eta) = \eta e^{-\eta} + 4e^{-\eta} + 3\eta - 4}\ , \tag{1.11}$$

$$\theta_0(\eta) = \mathbf{e^{-\eta}}, \tag{1.12}$$

$$\theta_1(\eta) = \frac{A_2 e^{-\eta}}{(A_1)^2 + 1} + \frac{A_3 e^{-2\eta}}{(A_1)^2 + 4} + \frac{A_4 e^{-\eta}}{(A_1)^2}\eta - \frac{\eta - 2}{A_1} - \left(\frac{2A_4}{A_1^{\,4}} + \frac{A_3}{A_1^{2} + 4} + \frac{A_2}{A_1^{2} + 1}\right) e^{-\eta}. \tag{1.13}$$

Finally by summing up the results, and letting $p \rightarrow 1$ we have the $\mathbf{f(\eta)}$, $\boldsymbol{\theta(\eta)}$ as:

$$\mathbf{f(\eta) = \lambda\eta - e^{-\eta} + \lambda e^{-\eta} - \lambda + 1 + (\eta e^{-\eta} + 4e^{-\eta} + 3\eta - 4)}\ , \tag{1.14}$$

$$\theta(\eta) = e^{-\eta} + \frac{A_2 e^{-\eta}}{(A_1)^2 + 1} + \frac{A_3 e^{-2\eta}}{(A_1)^2 + 4} + \frac{A_4 e^{-\eta}}{(A_1)^2}\eta - \frac{\eta - 2}{(A_1)^2} - \left(\frac{2A_4}{A_1^{\,4}} + \frac{A_3}{A_1^{2} + 4} + \frac{A_2}{A_1^{2} + 1}\right) e^{-\eta}. \tag{1.15}$$

Where $A_1 = Pr\,S$, $A_2 = Pr\,S + Pr - \lambda\,\varepsilon\,Pr - 1$, $A_3 = \lambda P_r - P_r - 2$, $A_4 = \lambda P_r$.

1.4 PROBLEM FORMULATION

We present numerical simulations of the out flowing fluid from a two dimensional axially symmetric short channel with an inclined backward- facing step, as shown in Figure 1. ϕ is the step inclination angle and the fluid enters at the superior boundary of the channel and exits at the inferior boundary of the domain at the outflow velocity denoted by U_{out}. This velocity is also imposed at the left inferior lateral boundary of the domain to simulate the casting velocity when the fluid is solidified or the jet velocity when the fluid experiments no phase change. No-slip boundary conditions were imposed at all solid walls.

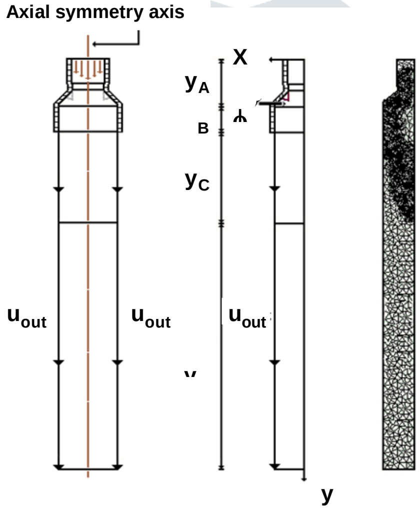

Figure 1: Geometry of the cannel with inclined backward- facing step and computational adaptive mesh

The range of outflow velocity explored in this investigation was 8.65×10^{-5} m/s $\leq U_{out} \leq 8.65\times10^{0}$ which for glass as the working fluid corresponds to the Reynolds number ($Re=U_{out}\ L\ /\ v$) range of $0.00054 \leq Re \leq 54$, where L is the distance from the entrance of the channel to the lower boundary of the domain. The walls of the short channel from the superior part of the channel to the step are assumed to be adiabatic. After the step, convection heat transfer coefficients are imposed along the domain boundary in order to take into account the heat exchange by convection of the glass. with the surroundings, which are kept at T_0=300K At the entrance of the channel, the glass temperature is T_{in}=1373K and the viscous stress vanishes along this boundary. The flow region **Ω** chosen for the numerical simulations is a two-dimensional channel, where the flow, heat transfer and phase change are analyzed. The conservation equations that describe this problem for a steady state incompressible fluid in this region are the momentum, continuity and energy equations.

$$-\nu\Delta\vec{u}+\vec{u}.\nabla\vec{u}+\frac{1}{\rho}\nabla=\vec{f}\ in\ \Omega, \quad (1.16)$$

$$\Delta.\vec{u}=0\ in\ \Omega, \quad (1.17)$$

$$\nabla.(-k\nabla T)+\rho C_p\vec{u}.\nabla T=Q\ in\ \Omega, \quad (1.18)$$

Where $\vec{u}=(u_1,u_2)$is the velocity vector, u_1 and u_2 being the transversal and axial velocity components respectively; ν is the kinematic viscosity, p is the pressure, k is the thermal conductivity, C_p is the specific heat, Q is the heating power per unit volume (heat source term) and T is the temperature Because we are interested in the flow behaviour at the phase change interface, the only body force taken into account in this model is given by a Darcian approximation that is a damping force, which ensures zero velocity of the solid phase and is given by the following body force

$$\vec{f}=\frac{(1-c_a)^2}{c_a^3+\varepsilon}c_b\left(u-u_{out}\,j\right), \quad (1.19)$$

Where c_a is the fraction of liquid phase, given by:

$$c_a=\begin{cases}1 & ,(T>T_m+\Delta T)\\ \dfrac{(T-T_m+\Delta T)}{2\Delta T} & ,(T_m-\Delta T)\le(T_m+\Delta T),\\ o & ,\ (T<T_m-\Delta T)\end{cases} \quad (1.20)$$

c_b is the permeability constant and ε is a small number to prevent division by zero in calculations. This term vanishes in the liquid phase region (c_a=1). T_m=1223K is the melting temperature and ⊗T=20K is the temperature interval where the liquid-solid mixture coexists.

Physically, in the mushy region when the fluid temperature decreases the glass solidifies; as a consequence, the specific heat capacity in the energy equation should be:

$$C_p=C_p+\delta\Delta H. \quad (1.21)$$

in order to incorporate the effect of the significant amount of latent heat released in the phase change region. On the other hand, most of the released heat occurs near the melting temperature, like a peak of heat due to phase change. Thus, in order to capture this physical phenomenon, we introduce the exponential parameter δ, which allows for a pulse type latent heat in the mushy region. δ is defined below:

.

$$\delta=\frac{e^{\left(-(T-T_m)^2/(\Delta T)^2\right)}}{\Delta T\sqrt{\pi}} \quad (1.22)$$

While the enthalpy change is given by :

$$\Delta H=T\Delta C_p \quad (1.23)$$

The velocity boundary conditions of the channel are:

$$u_1=0,\ \frac{\partial u_2}{\partial x}=0,\qquad for\ x=0$$

$$u_1=0,\ u_2=0,\qquad for\quad y_A,y_b \quad (1.24)$$

$$u_1=0,\ u_2=u_{out},\qquad for\ \ y_c,y_D,y_E$$

$u_1 = 0,\ u_2 = u_{out}, \qquad for \quad y = L.$

The temperature boundary conditions are given by.

$$\frac{\partial T}{\partial x} = 0 \qquad for \quad y_A,$$

$$h_B = 35\ W/m^2 \cdot K,\ \text{for}\ y_B,$$
$$h_C = 105 W/m^2 \cdot K,\ \text{for}\ y_C,$$
$$h_D = 60\ W/m^2 \cdot K, \quad \text{for}\ y_D, \qquad (1.25)$$
$$h_E = 40\ W/m^2 \cdot K, \quad \text{for}\ y_E$$

The thermophysical properties of glass used in the simulation are independent of temperature, see Table 1. The data for the glass flow simulated in this work, including the melting temperature and the temperature interval where the liquid-solid mixture coexists, are those for an actual glass and were published by a glass bottle manufacturer (Silices de Veracruz S.A. de C.V) located at Veracruz, Mexico.

Table 1: Physical properties of glass.

Property	Glass
Density	2380 kg/m3
Specific heat	1235.08 J/kg·K
Thermal conductivity	2.1 W/m·K
Dynamic viscosity	600 Pa.s
Latent heat	205 kJ/kg

1.5 CONCLUSION

In this paper free convective flow of a viscous Fluid past a vertical plate with constant porosity and span-wise co-sinusoidal temperature by using Regular Perturbation along with Homotopy Perturbation method. One-dimensional time-dependent double-diffusive mass & heat flow over a semi-infinite vertical plate, under a convective surface boundary condition using Homotopy perturbation method and Similarity Transforms. The property of variable thermal conductivity on flow of a viscous incompressible fluid in variable free stream near a stagnation point on a nonconducting stretching sheet using Similarity transformation and Homotopy perturbation method. The effect of variable thermal conductivity and heat source/sink on flow of a viscous incompressible electrically conducting fluid in the presence of uniform transverse magnetic field and variable free stream near a stagnation point on a non-conducting stretching sheet by Similarity Transformation & Homotopy perturbation method. The problem of laminar, incompressible and viscous flow between two moving porous walls, which enable the fluid to enter or exit during successive expansions or contractions by using Laplace transform and Homotopy Perturbation Method (HPM). The effects of heat generation and chemical reaction on unsteady MHD flow heat and mass transfer near a stagnation point of a three dimensional porous body in the presence of a uniform magnetic field is studied by similarity transformation and Homotopy Perturbation Method. A rotating disk in an infinite viscous fluid is extended to the case where the disk surface admits partial slip and the flow due to a rough rotating disk solved by Similarity Transformation & Homotopy Perturbation Method.

The velocity profile, temperature profile, skin friction coefficient and rate of heat transfer of the flow have been presented through graphs and tables. The numerical data has been derived through computer programming software MATLAB. The problem has been solved by using Homotopy perturbation method along with one of the methods mentioned as follows (i) Regular perturbation method (ii) Similarity Transform and (iii) Laplace Transform. The numerical results obtained have been mentioned in the end chapters in detail. It has been observed that Homotopy perturbation method when employed to solve governing equations of flow and heat and mass transfer along with traditional method after reducing the problem in ordinary differential equations (whether linear of non-linear, coupled or uncoupled), gives the solution very quickly, accurate and at a great ease as campared to tranditional methods when they employed alone. The beauty of HPM lies that it handles the non linearity of the equations in a very smooth way. Although it has been observed that an initial solution is to be assumed satisfying the boundary conditions of the problem. The velocity and temperature distributions in all the problems studied in the thesis using HPM have the same patterns which have been obtained by the other researchers satisfying the corresponding initial and boundary conditions of the respective problems. Numerical results obtained by HPM method in all the problems have good agreement with the numerical results obtained by other researchers. So it may be concluded that HPM method is very fast accurate and simple method to solve the problems in fluid dynamics if it is used along with some traditional method such as regular perturbation or similarity transforms.

This study was carried out by the numerical solution of the Navier-Stokes equations coupled with the energy equation, using finite element and adaptive meshes. The Darcian approximation was implemented in order to take account of the damping of the velocity when the solidification occurs in a glass flow. We studied the interaction between fluid dynamics, heat transfer and phase change in an out flowing fluid from a short channel with hot temperature at the entrance of the channel. The outflow velocity of the fluid was imposed at the inferior boundary of the domain and on the left inferior lateral boundary of the domain to simulate the casting velocity when the fluid is solidified or the jet velocity when the fluid experiments no phase change. No-slip boundary conditions were imposed at all solid walls is also easy to apply and understand.

FUTURE SCOPE

Since study of fluid dynamics has great importance and significance in the field of science and technology. So efforts should be made to do more research to find the problems of fluid dynamics which occurs in other fields like biotechnology, nano technology, chemical industry etc. and to find new methods and techniques which make the solution obtaining procedure more accurate, exact, fast, easy and compatible to modern computing machine. HPM is such a method. It may be employed to above mentioned problems and areas and problems of turbulent flows also. Further research is require to carry out to employ HPM method without using any traditional methods.

NOMENCLATURE

c_a - Liquid phase fraction

c_b - Permeability constant

h - Spatial step **m**

k_t - hermal conductivity **W/m·K**

L - Length m

L_{oh} - Space of trial function for pressure **Pa**

L_h - Space of finite element trial function for pressure **Pa**

p - Pressure **Pa**

P_1 - Polynomial space of degree one

p_h Approximation of pressure finite element **Pa**

qh - Trial function of pressure **Pa**

REFERENCES

1) **Ahmed S.E.** and Chamka, A.J. ―Similarity Solution for Unsteady MHD Flow Near a Stagnation Point of a Three-Dimensional Porous Body with Heat and Mass Transfer‖, Heat Generation/Absorption and Chemical Reaction.Int. Journal of applied fluid mechanics **(2011),**
2) **Mojtabi, A** and Charrier-Mojtabi, M. C. ―Double diffusive convection in porous media,‖ in Handbook of Porous Media, 2nd edition, **2005 pp. 269– 320.**

3) **Abah, S.O.,** Eletta, B.E. and Omale, S.O. ―The Numerical analysis of the effect of free convection heat and transfer on the unsteady boundary layer Flow Past a Vertical Plate‖, International Journal of Theoretical and Mathematical Physics, vol. 2, **(2012), pp. 33-36.**
4) **Abbas, Z**., Hayat, T., Sajid, M. & Asghar, S. Unsteady flow of a second grade fluid film over an unsteady stretching sheet. Mathematical and Computer Modeling 48: **2008, 518-526.**
5) **Aiyesimi, Y.M**., Abah, S.O. and Okedayo, G.T. The analysis of hydromagnetic free convection heat and mass transfer flow over a stretching vertical plate with suction. American Journal of Computational and Applied Mathematics, 1, (**2011), 20-26.**
6) **Arunachalam, M**. and Rajappa, N.R. ―Forced convection in liquid metals with variable thermal conductivity and capacity‖, Acta Mechanica 31, **(1978),** pp**. 25-31.**
7) **Ayaz, F**. ―Solutions of the systems of differential equations by differential transform method‖, Appl. Math. Comput., **147: 2004 547-567.**
8) **Aziz, A.** ―A similarity solution for laminar thermal boundary layer over a flat plate with a convective surface boundary condition‖, Communications in Nonlinear Science and Numerical Simulation, 14, (**2009) ,1064-1068.**
9) **Bansal, J.L**. ―Magneto-fluid-dynamics of Viscous Fluids.‖,Jaipur Pub. House, **(1994).** Jaipur, India.
10) **Bansal, J.L**. Viscous Fluid Dynamics. Oxford & IBH Pub. Co.(**1977),** New Delhi.
11) **Beavers G.S**. and Joseph D.D., Boundary condition at a naturally permeable wall, J. Fluid Mech. **30(1967),** pp. **197-207**.
12) **Bechert D.W**. & Bartenwerfer M. The viscous flow on surfaces with longitudinal ribs. J. Fluid Mech. 206, (**1989) 105-129**.
13) **Biazar J.,** Ayati Z. and Ebrahimi H. ‖Homotopy perturbation method for general form of porous medium equation,‖ Journal of Porous Media, 12, **(2009) 1121-1127.**
14) **Boutros, Y.Z**. and Abd-el-Malek, M.B. and Badran, N.A. and Hassan, H.S. ―Lie-group method for unsteady flows in a semi-infinite expanding or contracting pipe with injection or suction through a porous wall‖, J. Comput. Appl. Math., 197 (2): **2006 465-494.**
15) **Bararnia, H**., and Ghotbi, A.R. and Domairry, G. ―On the analytical solution for MHD natural convection Àow and heat generation Àuid in porous medium‖, Communication in Nonlinear Science and Numerical Simulation 14, (2009), pp. **2689–2701.**
16) **C. Ping,** ―Similarity solutions for mixed convection from horizontal impermeable surfaces in saturated porous media,‖ International Journal of Heat and Mass Transfer, vol. 20, no. 9, 1977, pp. **893–898.**
17) **Chaim, T.C**. ―Heat transfer in a fluid with variable thermal conductivity over stretching sheet‖, Acta Mechanica 129, (**1998), 63-72.**
18) **Chakrabarti, A**. and Gupta A.S. Hydro magnetic flow and heat transfer over a stretching sheet. Quarterly Journal of Applied Mathematics 37, (1979), **73- 78.**
19) **Chamka, A.J.** and Khaled, A.R.A. ―Similarity solution for hydro magnetic mixed convection and mass transfer for Hiemenz flow though porous media‖, Int. Journal of Numerical Methods for Heat and Fluid Flow 10, **(2000), 94-115.**
20) **Chamkha, A.J**. ―MHD flow of a uniformally stretched vertical permeable surface in the presence of heat generation/absorption and chemical reaction‖, International Communication in Heat and Mass Transfer 30, (2003), pp. **413- 422.**