Research
Volume 10:3, 2021

Journal of Applied & Computational Mathematics

ISSN: 2168-9679 Open Access

Numerical Investigation of Mixed Convection Flows with Variable Viscosity and Prandtl Number and Validation Using ANSYS FLUENT

Abhishek Kumar Singha[1*], A. Royb[2], and A. K. Singh[3]

[1]Department of Mathematics, Indian Institute of Technology Delhi, Delhi, India

[2]Department of Applied Mechanics, Indian Institute of Technology Madras, Chennai, India

[3] Department of Mathematics, VIT University, Chennai, India

Abstract

Purpose: The purpose of present study is to analyze the mixed convection impact for an exponentially decreasing free stream in laminar water boundary layer flow with variable viscosity and Prandtl number. The analysis includes the effect of suction/blowing and heat generation/absorption.

Design/methodology/approach: The non-linear partial differential equation governing the flow and thermal fields are presented in non-dimensional form by using appropriate transformations. The Quasilinearization technique in combination with implicit finite difference scheme has been adopted to solve the non- linear coupled partial differential equations.

Findings: It was found that the influence of The numerical results are displayed graphically to illustrate the influence of various non-dimensional physical parameters on velocity and temperature buoyancy assisting force on both velocity as well as temperature profile is significant. Skin friction coefficient increases with Richardson number and deceleration parameter. The heat transfer coefficient decrease with the increase of heat generation/absorption parameter and increases with an increase in wall suction/injection parameter.

Originality/value: The present investigation deals with the solution of steady laminar water boundary layer flows with an exponentially decreasing free stream velocity and temperature-dependent viscosity and Prandtl number applicable to water using practical data.

Keywords: Exponentially decreasing free stream • Mixed convection • Water boundary layers • Quasilinearization technique • ANSYS FLUENT

Introduction

The study of boundary layer flow over an exponentially decreasing free stream velocity, involves the effect of suction/blowing and heat generation/absorption within the fluid. Many practical applications of such technique can be found in various engineering fields, for example, hot rolling, thermal design of thrust bearing, continuous casting etc.[1,2]. In boundary layer control theory, the variations and temperature dependence of the viscosity and thermal conductivity of the fluid can easily be accomplished by maintaining a temperature difference between the solid wall and the fluid. As a result, boundary layer flow separation may occur due to wall heating and decrease in the fluid viscosity with an increasing temperature. In literature, very few studies based on steady laminar boundary layer flows over heated bodies and variable fluid properties are reported [3,4]. The mixed convection case is especially pronounced in heat transfer problem where the forced flow is low and/or the temperature difference is large. Some of the recent studies on mixed convection flows with applications can be found [5,6].

The introduction of suction/blowing to the fluid in the boundary layer has made the understanding of fluid flow and heat transfer characteristics much more rigid and complex. Since the last few decades, a significant amount of effort is being put by many researchers regarding this issue. In the presence of adverse pressure gradient, the boundary layer begins to separate from the wall surface and ultimately breaks away from the bounding wall. The properties of

---

[*]**Address for Correspondence:** Abhishek Kumar Singha, Department of Mathematics, Indian Institute of Technology Delhi, Delhi, India, Tel: +91443993155; E-mail: abhishekkumar.singh@vit.ac.in

Received: 10 February, 2021; **Accepted:** 24 February, 2021; **Published:** 03 March, 2021

boundary layer flow separation are significant and have gained much attention due to its technological applications [7-9]. However, it has certain limitations. For instance, it diminishes pressure recovery; also it causes increase of drag, loss of lift etc. Several techniques which already exist can be employed to control the boundary layer separation. Boundary layer suction, blowing and introduction of a transverse magnetic field are one of them [10]. To account for this, Curle introduced a steady, laminar velocity boundary layer flow, where external flow velocity was approximated by ue=u0 (1 seξ), 0<s<1, where u0 is constant, s is a small parameter and ξ is a scaled stream wise coordinate [11]. Further Chiam has analyzed time dependent solution of laminar exponentially decreasing velocity boundary layer for flow past a surface [12].

The objective of the present investigation is to deal with the solution of laminar water boundary layer flows over an exponentially decreasing free stream velocity with temperature dependent fluid viscosity and Prandtl number. The fluid considered here is water as it is one of the most common fluids found in engineering applications. The nonlinear partial differential equations governing the flow and thermal fields are solved numerically using the implicit finite difference scheme in combination with the quasilinearization technique [13]. The numerical results are presented in terms of skin friction and heat transfer rate which are useful in determining the surface heat requirements for stabilizing the laminar boundary layer flow over an exponentially decreasing free stream velocity.

Mathematical Analysis

Consider a 2D laminar, incompressible, mixed convection water boundary flow with an exponentially decreasing free-stream velocity distribution. Mass transfer vw(x) (suction/injection) is allowed along its vertical surface. Further simplification is made through the assumption that the rate of blowing of fluid is negligible and has no impact on the in viscid flow at the edge of the boundary layer. Also, it is supposed that both the injected and the boundary layer fluid preserve the same physical properties. The x-axis is taken along the surface in the vertically upward direction and the y-axis is taken normal to it (Figure 1). Due to the temperature difference in the fluid there is a rise in the buoyancy force. It is presumed that the fluid flows with moderate velocities and the temperature difference between the wall and the free stream is minute (<50ºC). Since the change in the density and specific heat (Cp) of water with respect to the temperature is less than one percent, both are treated as constants. On the other hand, there is a significant change in the thermal conductivity and viscosity (μ) [and hence Prandtl number (Pr)] with respect to the temperature. Thus, it is assumed that the viscosity and Prandtl number vary as an inverse linear function of temperature (T) [13,14].

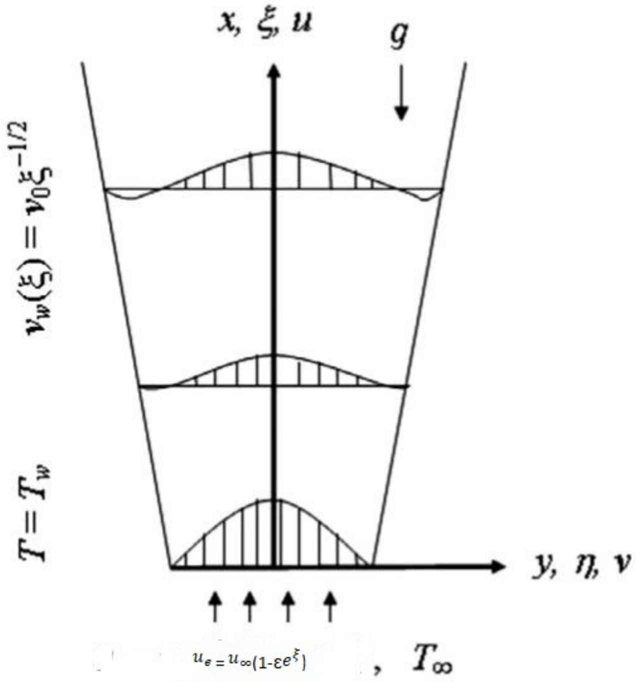

Figure 1. Schematic diagram of the Physical model and coordinate system with the boundary conditions.

The fluid at the edge of the boundary layer is maintained at a constant temperature T∞ and the body has a uniform temperature Tw. Here viscosity and Prandtl number are variable. Under these assumptions, the conservation of mass, momentum and energy equations governing the mixed convection boundary layer flow within a diverging channel are given by Schlichting and Patil et al. [10,14].

Method of Solution

The procedure is to numerically solve the non-linear partial differential equations subject to the boundary conditions, using an implicit finite difference scheme jointly with the Quasi-linearization technique [13]. For accomplishing the quadratic convergence and monotonicity, carefully construct an iterative sequence of linear equations so as to approximate the non-linear equations. The non-linear coupled partial differential equations with the boundary conditions.

The detailed description of the present method is provided in recent study by Patil et al. [10]. At each iteration step, the sequence of linear partial differential equations were expressed in difference form using central difference scheme in the η- direction and backward difference scheme in ξ- direction. Thus in each step, the resulting equations were reduced to a system of linear algebraic equations with block tridiagonal matrix, which is solved by Varga's algorithm [15]. The step size in η and ξ direction has been chosen as Oη = 0.01 and Oξ = 0.01, respectively, throughout the computations. Also it has been found that a further decrease in Oη and Oξ do not change the results upto third decimal place. Based on the relative difference between the current and previous iteration values a convergence criterion is used. As soon as, the difference approaches 10−4, the solution is supposed to have converged and the iteration process is stopped.

Results and Discussion

The numerical results are presented in terms of velocity profile (F), temperature pro- file (G), skin friction coefficient (Cf) and heat transfer rate (Nu) for various values of Q(−1 ≤ Q ≤ 1), Ri(−3 ≤ Ri ≤ 10), Re (10 ≤ Re ≤ 250), s(0.000001 ≤ s ≤ 0.1), A(1 A 1), and ξ(0 ξ 1), where Q is the heat source parameter. η∞, the edge of boundary layer is taken between 6.0 to 8.0.

The influence of Reynolds number on velocity and temperature profiles (F,G) are plot- ted in Figures 2 and 3 for s=0.01. From the figures it is clear that, with the increase of Reynolds number, the velocity and temperature profiles are also increased within the boundary layer. Further, in this case, the dimensionless velocity overshoots is observed in the neighborhood of the wall. Figure 2 also shows that the overshoot in the dimensionless velocity increases with an increase in Reynolds number. Increment of momentum within the exponentially decreasing velocity boundary layer is closely associated with increase of Reynolds number which in turn accelerates fluids within boundary layer fluid. Temperature overshoot is also observed near the wall as in Figure 3 for higher Reynolds number (Re 300). This is due to the fact that, the increase of Reynolds number enhances the velocity within the boundary layer and temperature overshoot appears in presence of viscous dissipation for higher Re (Re ≥ 100).

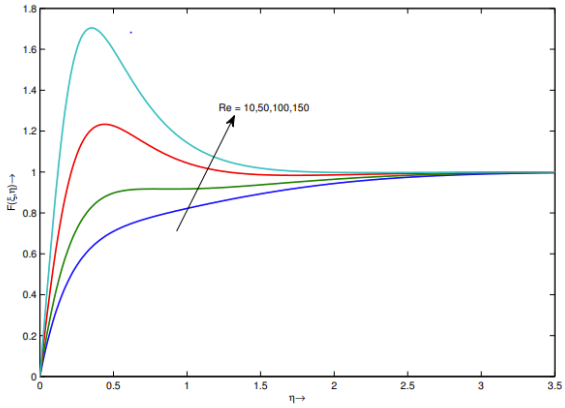

Figure 2. Effect of Reynolds number Re on velocity profiles for A=1, Q=0.04, Ri=10 and ξ=1.

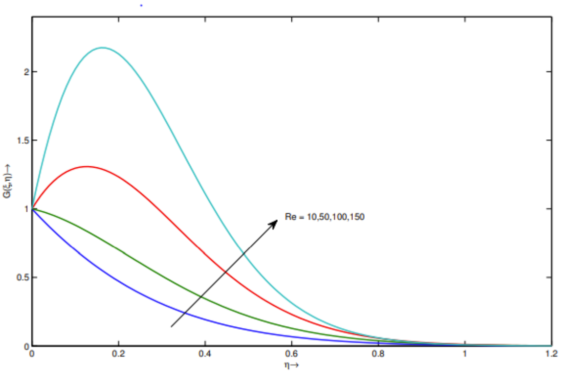

Figure 3. Effect of Reynolds number Re on temperature profiles for A=1, Q=0.04, Ri=10 and ξ = 1.

Figures 4 and 5 show the velocity and temperature profiles (F,G) for s=0.01, A=1, Q=0.1, Re=50 and ξ=1 which depends on Richardson number (Ri). The velocity overshoot is observed close to the wall region in Figure 4.

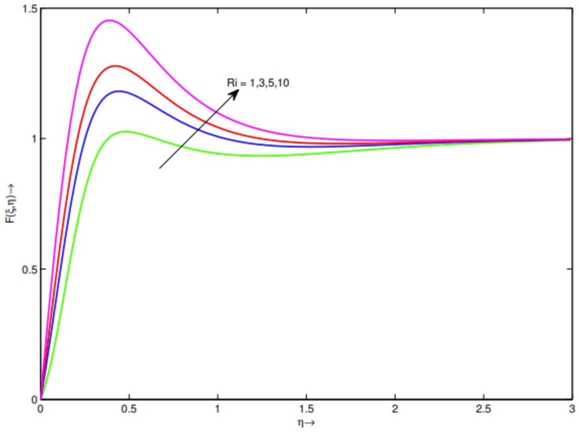

Figure 4. Effect of Richardson number Ri on velocity profiles for A=1, Q=0.1, Re=50 and ξ=1.

Since Richardson number is considered as a buoyancy assisting parameter for mixed convection flow, the increase of Ri results in more pronounced velocity overshoots near the wall region. However, Figure 5 reveals that, overshoots in temperature profiles are comparatively less in magnitude which means that the effect of Ri on temperature profiles are less significant. In particular, for instance, for s=0.01, A=1, Q=0.1, Re=50 and ξ=1 , at η=0.5, the velocity profiles raise approximately about 15% whereas a reduction of 54% is observed in temperature profiles, as Ri increases from 1 to 3.

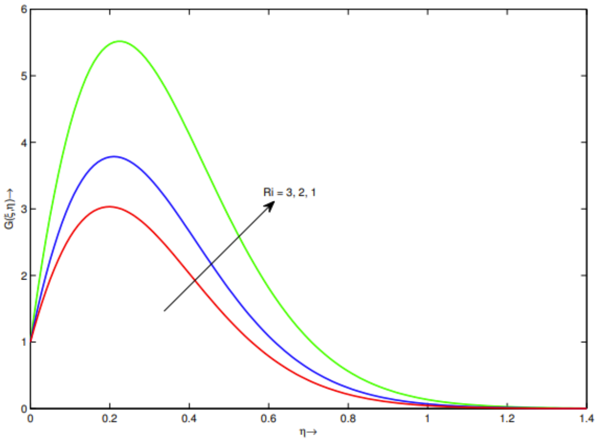

Figure 5. Effect of Richardson number Ri on temperature profiles for A=1, Q=0.1, Re=50 and ξ=1.

The effects of wall suction/blowing and stream-wise coordinate ξ on velocity figurations for s=0.01, Re=10, Q=0.1 and Ri=10 are displayed in Figure 6. Results indicate that both injection (A<0) as well as stream-wise co-ordinate (ξ) enhance the magnitude of velocity overshoot. The increase of velocity overshoot near the boundary layer profiles near the boundary for injection (A<0) is justified as more fluids are pushed into boundary layer. In particular, for instance, for s=0.01, A=0, Q=0.1, Re=10 and Ri=10 at η=1, the velocity profiles overshoot is observed around 40%, as stream wise coordinate ξ increases from 0 to 1. This velocity overshoot is further increased upto 90% as A changes from A=0 to A=-0.5 while other parameters remaining the same.

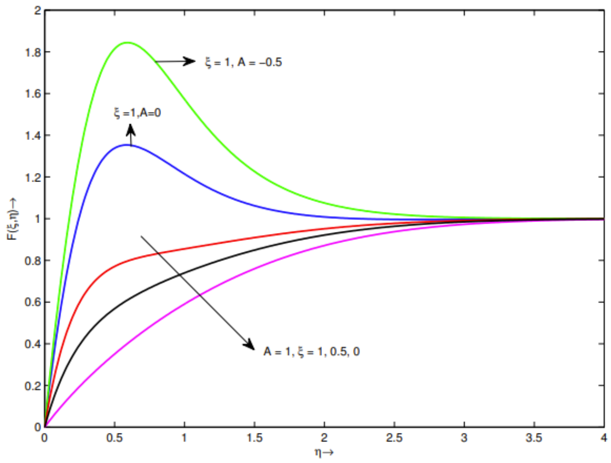

Figure 6. Effects of wall suction/injection and ξ on velocity profiles for Re=10, Q= 0.1 and Ri=10.

Figures 7 and 8 explain the effects of Q and wall suction/blowing parameter A on velocity and as well as temperature profiles for s=0.01, Ri=10, Re=10 and ξ=1. For the increase in heat source parameter (Q>0), increment of overshooting on velocity as well as temperature boundary layer profiles is observed which confirms that the heat source parameter influences on the exponential decreasing free stream flow. The velocity and temperature overshoot is more pronounced for injection (A<0) than suction (A>0). In particular, for instance, ξ=1, s=0.01, A = -0.25, Re=10 and Ri=10 at η=1, the velocity profiles increase around 250%, as Q increases from -0.25 to 0.25.

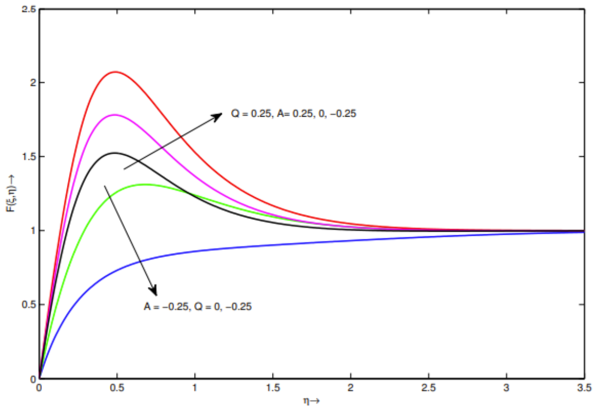

Figure 7. Effects of heat source parameter Q and wall suction/injection parameter A on velocity profiles for Ri=10, Re=10 and ξ=1.

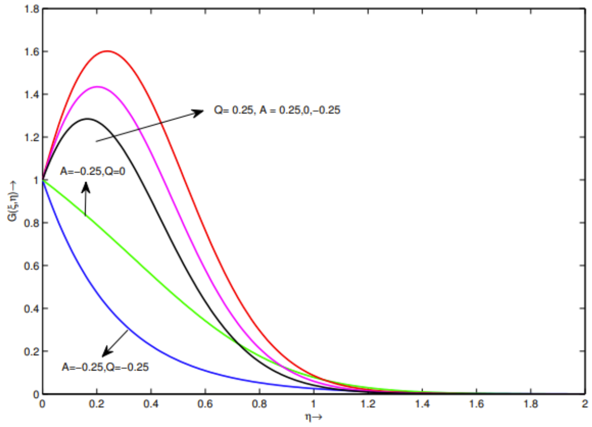

Figure 8. Effects of heat source parameter Q and wall suction/injection parameter A on temperature profiles for Ri=10, Re=10 and ξ=1.

The effects of the Ri and s on the skin friction coefficient Re1/2Cf for A=1, Re=10 and Q=0.1 are shown in Figure 9. It is perceived that an increase in the Ri leads to an increase in the skin friction coefficient Re1/2Cf. As a result, an increase in the buoyancy force which in turn increases skin friction Re1/2Cf. Physically, ξ increases the external flow velocity outside the laminar boundary layer thereby enhancing the skin friction coefficient. In particular, at ξ=1, A=1, Re=10 and Q=0.1, Re1/2Cf increases approximately by 139% to 195%. This happens when the Ri increases from 2 to 10 corresponding to the ξ=0.001 and ξ=0.1 respectively. For ξ=0.01, Re=10 and Ri=10, the effects of heat source/sink parameter Q and the wall suction/injection parameter A on the heat transfer coefficient Re−1/2Nu, are shown in Figure 10. Also from Figure 10, it is noted that the heat transfer coefficient Re−1/2Nu decreases with an increase of Q while Re−1/2Nu increases with an increase in A. In particular, at ξ =1, Re=10, Ri=10 and s=0.01, Re−1/2Nu decreases by 20% to 68% approximately. The reason is that the source/sink parameter Q increases from -0.1 to 0.1 corresponding to A =-0.5 and A=0.5 respectively.

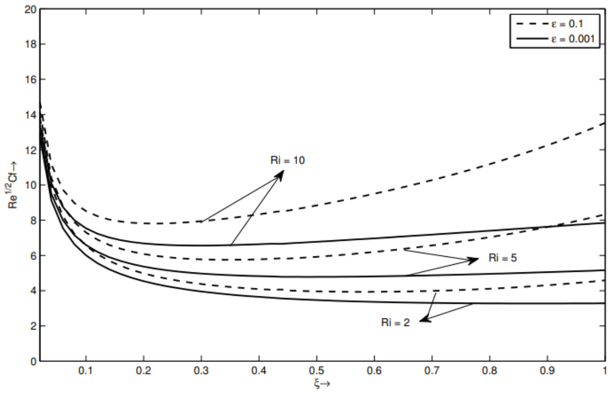

Figure 9. Effects of small parameter Ri on skin-friction for A=1, Re=10 and Q=0.1.

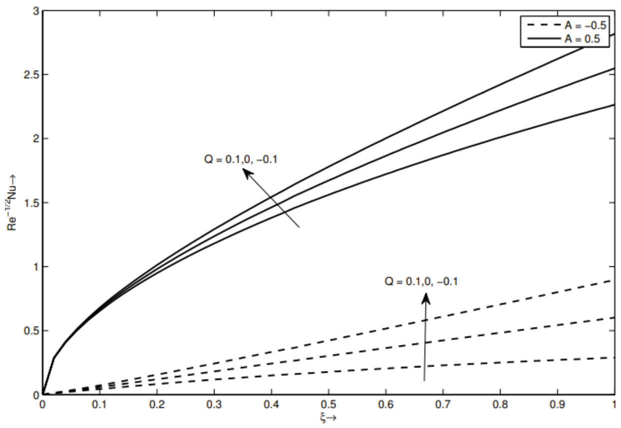

Figure 10. Effects of A and Q on Nusselt number for Re=10 and Ri=10

Flow Simulation Using ANSYS FLUENT and Validation

To validate the presented numerical approach and results, flow simulation is performed with the help of ANSYS FLUENT- a commercial CFD package. The simulated results are compared with the numerical results subsequently. Heat transfer in the laminar regime is simulated with ANSYS for laminar flow with constant wall temperature and exponentially decreasing inlet velocity. Exponentially decreasing inlet velocity and variable viscosity and Prandtl number are defined in ANSYS by user defined function [17]. In order to analyze fluid flow, flow domains are split into smaller sub-domains to get possible solution of partial differential equations which govern the fluid flow. The governing equations are then discretized and solved inside each of these sub domains. The 2D geometry is divided into discrete volumes using a structured grid. Mesh sizes (values) are rearranged near wall appropriately to resolve the boundary layer velocity profile. Total 12000 elements are created in mesh grid. k-s flow modelling is used to simulate this problem [18]. Pressure velocity coupling method is used to solve this model and second order upwind spatial discretization is applied for momentum. Exponentially decreasing velocity profile with free stream velocity of 1 m/sec is used as inlet boundary condition. No-slip wall boundary condition is applied to the model surfaces without specifying any wall roughness. A convergence criterion of 10−6 is selected for the residuals.

A comparison on velocity profiles is done to validate the flow modelling and numerical approach. Figures 11 and 12 show the comparisons on velocity profiles at ξ=0.75 and 1.0 with s=0.01. A slight overshooting is observed in velocity profiles by ANSYS prediction away from the wall. One of the major assumptions of boundary layer theory is, if Reynolds number tends to infinity, then boundary layer is infinitely thin. Due to these assumptions, it considers the outer flow to be an in viscid region outside the boundary layer for flow past a flat plate. But it turns to be a real effect that at a finite Reynolds number, boundary layer thickness displaces the outer flow which causes overshoot observed in FLUENT solution. In a real scenario, viscosity can't able to manifest shear stress outside the boundary layer due to lack of velocity gradient which is accounted by the FLUENT solution [19]. Otherwise, numerical solutions are in good agreement with the simulated results. For further downstream direction along the plate, the velocity profile near the plate overshoots over the free stream value. This can be explained as, with an increase in the viscosity index, the fluid becomes lighter with a gain in temperature that is the reason of overshooting of velocity profiles near the wall.

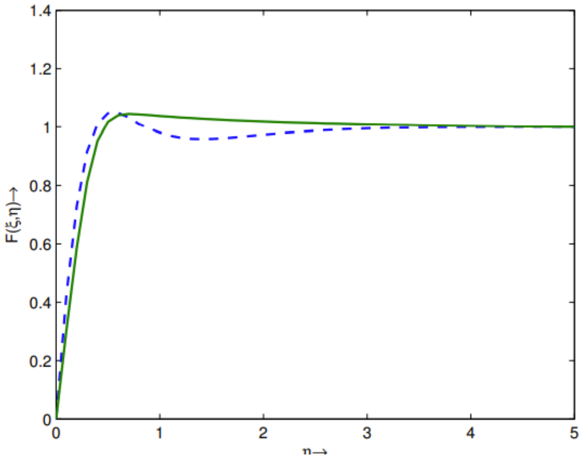

Figure 11. Comparison of results of F (ξ,η) with ANSYS FLUENT at ξ=0.75 for A=0, Q=0, Ri=0.

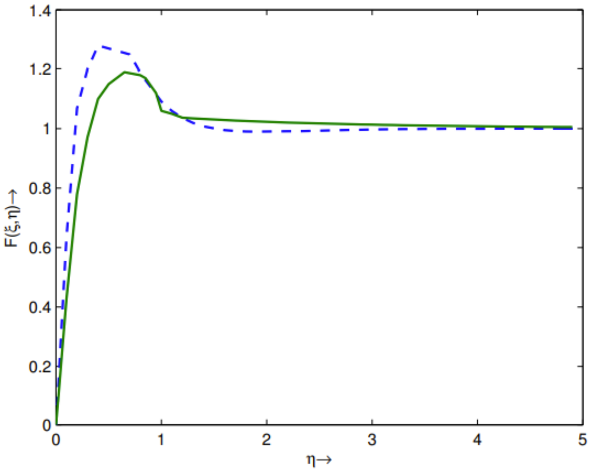

Figure 12. Comparison of results of F (ξ, η) with ANSYS FLUENT at ξ=1 for A=0, Q=0, Ri=0.

Results of present numerical approach is validated and compared with those of reported results in literature for particular cases by Patil et al. [10. Comparisons of Fη(ξ, 0) and Gη(ξ, 0) for various dimensionless parameter in presence of A and Q show excellent agreement.

Conclusions

The following conclusions can be drawn from these detailed computational and numerical results:

The influence of buoyancy aiding force (Ri>0) on both velocity as well as temperature profile is significant. Near the wall region as Ri increases from 1 to 10, the increment in velocity profile enhance from 103% to 145%.

An overshoot in the velocity and temperature field near the wall is observed due to higher value of Re. Near the wall as Re increases from 100 to 200, the increment in velocity and temperature profile enhance from 123% to 228% and 31% to 251% respectively.

The skin friction coefficient Re1/2Cf increases with s and buoyancy parameter Ri. Especially, like, A=1, Re=10, Q=0.1 at ξ=1, Re1/2Cf increase around 139% and 195% as Ri increases from 2 to 10 corresponding to the s=0.001 and s=0.1 respectively.

It is observed that the heat transfer coefficient Re−1/2Nu reduces with the increase of Q and increases with the increase of A. In particular, for instance, Ri=10, Re=10, s=0.01 at ξ=1, Re−1/2Nu decreases approximately about 68% and 20% as hear source/sink parameter Q increases from -0.1 to 0.1 corresponding to the wall suction/blowing parameter A=-0.5 and A=0.5 respectively.

References

1. Labropulu, F, JM Dorrepaal, and OP Chandna. “Oblique Flow Impinging on a Wall with Suction or Blowing.” Acta Mechanica 115 (1996): 15-25.
2. Al-Sanea, Sami A. “Mixed Convection Heat Transfer along a Continuously Moving Heated Vertical Plate with Suction or Injection.” Int J Heat Mass Transfer 47 (2004): 1445-1465.
3. Pantokratoras, Asterios. “Forced and Mixed Convection Boundary Layer Flow Along a Flat Plate with Variable Viscosity and Variable Prandtl Number: New Results.” Heat Mass Transfer 41 (2005): 1085-1094
4. Chin, KE, Roslinda Nazar, Norihan M Arifin, and Ioan Pop. “Effect of Variable viscosity on Mixed Convection Boundary Layer Flow over a Vertical Surface Embedded in a Porous Medium.” Int Comm Heat Mass Transfer 34 (2007): 464–473.
5. Patil, Prabhugouda Mallanagouda, Satakshi Roy and IoanPop. “Unsteady Effects on Mixed Convection Boundary Layer Flow from a Permeable Slender Cylinder Due to Non-Linearly Power Law Stretching.” Computers & Fluids 56 (2012): 17-23.
6. Rashad, AM, AJ Chamkha, and M Modather. “Mixed Convection Boundary-Layer Flow Past a Horizontal Circular Cylinder Embedded in a Porous Medium Filled with a Nano Fluid under Convective Boundary Condition.” Computers & Fluids 86 (2013): 380–388.
7. Brown, SN and K Stewartson. “Laminar Separation.” Annual Review of Fluid Mechanics 1 (1969): 45-72.
8. Williams III, James C. “Incompressible Boundary-Layer Separation.” Annual Review of Fluid Mechanics 9 (1977): 113-144.
9. Patil, Prabhugouda M. “Effects of Surface Mass Transfer on Steady Mixed Convection Flow from Vertical Stretching Sheet with Variable Wall Temperature and Concentration.” Int J Numeri Methods Heat & Fluid Flow 22 (2012): 287-305.
10. Patil, PM, HS Ramane, S Roy and Hindasageri, Vijaykumar, et al. “Influence of Mixed Convection in an Exponentially Decreasing External Flow Velocity.” Int J Heat Mass Transfer 104 (2017): 392-399.
11. Curle, N. “Development and Separation of a Laminar Boundary Layer with an Exponentially Increasing Pressure Gradient.” Q J Mech Appl Math 34 (1981): 383-396.
12. Chiam, TC. “A Numerical Solution for the Laminar Boundary Layer Flow with an Exponentially Decreasing Velocity Distribution.” Acta Mechanica 129 (1998) 255–261.
13. Roy , Satyajit, and Saikrishnan Ponnaiah. “Non-Uniform Slot Injection (Suction) into Steady Laminar Water Boundary Layer Flow over a Rotating Sphere.” Int J Heat Mass Transfer 46 (2003): 3389–3396.
14. Schlichting, Hermann and Klaus Gersten. Boundary-Layer Theory, Germany (2016).
15. Varga, Richard S. Matrix Iterative Analysis 27, USA (2009).
16. Lide, David R. CRC Handbook of Chemistry and Physics, 71st Edition, USA, (1990).
17. ANSYS, ANSYS Fluent UDF Manual, USA (2011).
18. Pantokratoras, Asterios. “The Forced Convection Flow over a Flat Plate with Finite Length with a Constant Convective Boundary Condition.” J Mech Sci Technol 28 (2014): 1909–1915.
19. Khaled, Chaib, Nehari Driss, and Sad Chemloul Nouredine. CFD Simulation of Turbulent Flow and Heat Transfer over Rough Surfaces.” Energy Procedia 74 (2015): 909-918.

How to cite this article: . "Numerical Investigation of Mixed Convection Flows with Variable Viscosity and Prandtl Number and Validation Using ANSYS FLUENT ." *J Appl Computat Math*10 (2021) : 145