Study of buoyant force acting on different fluids moving over a horizontal plate due to forced convection

J Singh¹ and C Singh²

¹ Mechanical Engineering Department, Thapar Institute of Engineering and Technology, Patiala 147004, Punjab, India
² Department of Mechanical Engineering, University College of Engineering, Punjabi University Patiala, 147002, Punjab, India

jashanpreet.singh@thapar.edu

Abstract. In this paper, the effect of buoyancy force on heat transfer over flat plate due to forced convection is investigated numerically. The governing equations namely continuity, Navier-Stokes and energy equations were discretized by using an explicit finite difference technique. Code generation has been done in FORTRAN-95 programming language. The testing of FORTRAN code has been done for different fluids at different grid points. The length of horizontal flat plate (x) was taken as 0.5 m. The inlet velocity of fluid (u) at the edge of plate was taken as 3 m/s. The kinematic viscosity (υ) of air and water at 323 K were taken as 17×10⁻⁶ m²/s and 66×10⁻⁷ m²/s respectively. The surface temperature (Ts) of plate and flow temperature (To) of fluid were taken as 300 K and 323 K respectively. The grid was discretized by continuous method into 1001×101 (m×n) mesh points. The results show the boundary layer thickness, velocity and temperature evolutions, and heat transfer coefficient formed by different fluids. Also, the heat transfer coefficient to surrounding fluid across flat plate was found higher for air as compared to water. Results showed that buoyancy force acts from downward position so it was observed maximum at lower distance of plate. However, buoyancy effect reduces as the distance of x increases.

1. Introduction

In present study, the computational fluid dynamics (CFD) was employed to solve the heat transfer due to forced convection across flat plate. Forced convection heat transfer across flat plate is universally involved in many engineering applications such as cooling coils, radiators, cooling circuits, refrigerators, steam generators, etc. [1]. CFD is globally accepted to mathematically solve the heat transfer problems [2]. In fluid dynamics, the governing equations such as Navier-Stokes equation, continuity equation and additional conservation equations (like example energy or species concentrations), are solved by using CFD tools/software packages or programming languages [3]. Buoyancy plays a significant role in heat transfer problems related to fluid flow. Buoyancy is generated due to the temperature difference between a fluid and surroundings. Buoyancy creates a significant effect in movement of fluid flow which contributes in transverse velocity that has a large effect on the heat transfer process [4].

In early days, Sakiadis [5] performed a numerical study to investigate the effect of skin friction on boundary layer thickness over a horizontal plate of finite length. They used integral method to numerically solve the governing equations by applying the appropriate boundary conditions. They reported that the integral method was effective in solving the turbulent boundary layer on a moving continuous flat surface. Peters [6] performed a numerical study to investigate the laminar hydrogen-
oxygen diffusion flame on a flat plate. They investigated the non-equilibrium and nearly-equilibrium structure at various positions in the boundary layer. They used 15 elementary reactions between 8 species and investigated the effect of variable properties including thermo diffusion. Results showed that no chemical reactions occurred at larger distances from the starting edge of plate due to the low temperatures. Janssen and Armfield [7] investigated the boundary layer stability of steady state cavity convection flow. They applied artificial perturbation to generate the start-up waves and analyzed the stability character of the boundary layers in fully developed flow. During the experiments, they kept the top and bottom cavities as insulated. They found that heated and cooled walls exhibit travelling wave instabilities in the thermal boundary layers that form on the walls. They also found that these waves were observed at start-up for the water ($Pr=7.5$) at $Ra=6\times10^8$. Additionally, the thermal boundary layers in the fully developed flow have approximately the same stability character as the start-up flow. In 21st century, CFD has received an extensive attention amongst the community of researchers/investigators to solve the engineering problems numerically by using computers. Mahgoub [8] performed a numerical study to investigate the heat transfer due to forced convection across a flat plate in a porous media. They found that the local heat transfer coefficient starts decreasing as the axial distance increases on the flat plate.

Various researchers have done a lot of work on boundary layer analysis. The various methods and techniques to solve the boundary layer are Approximate Generalized Method [9], Newton-Raphson method [9], Homotopy analysis method [10], Crank-Nicolson method [11], Integral method [12], etc. From literature survey, it is found that these methods are very tedious and complex. Moreover, these methods are difficult to unproductive to obtain approximate solutions of fluid flow problems. On the other hand, Finite difference method and Gauss-Seidel iteration are very simple techniques to analyze laminar boundary layer for flow over a flat plate. In this context, present study was carried out to analyze the velocity and temperature profiles of boundary layer over a flat plate using finite difference method using Gauss-Seidel iteration. Air and water were considered as the flowing fluids.

2. Mathematical modelling

2.1. Mathematical equations
Three different governing equations were used to solve the forced convection around flat plate. These equations are explained below [1]:

Continuity Equation:
\[
\frac{du}{dx} + \frac{dv}{dy} = 0
\]  
(1)

Steady-state Navier-Stokes Equation:
\[
u \frac{du}{dx} + v \frac{du}{dy} = \nu \left[ \frac{d^2u}{dx^2} + \frac{d^2u}{dy^2} \right]
\]  
(2)

Energy equation:
\[
u \frac{dT}{dx} + v \frac{dT}{dy} = \alpha \left[ \frac{d^2T}{dy^2} \right]
\]  
(3)

2.2. Finite difference method
Discretization of governing equation was done by using explicit scheme of Finite difference method. Discretization of Navier-Stokes equation was done to find x-component of velocity ($u$) whereas the discretization of the continuity equation was done to find the velocity ($v$). However, the discretization the energy equation of thermal boundary layer was done to find out the value of temperature. Discretization of governing equations is as follows [13, 14]:

Continuity equation:

\[ v_{i+1,j+1} = v_{i+1,j-1} - 2 \frac{\Delta x}{\Delta x} [u_{i+1,j} + u_{i,j}] \]  \hspace{1cm} (4)

Navier-Stokes equation:

\[ u_{i+1,j} = u_{i,j} - \frac{\Delta x \cdot v_{i,j}}{2 \Delta y \cdot u_{i,j}} [u_{i,j+1} - u_{i,j-1}] + \frac{v \cdot \Delta x}{\Delta y^2} [u_{i,j+1} + u_{i,j-1} - 2u_{i,j}] \]  \hspace{1cm} (5)

Energy equation:

\[ T_{i+1,j} = T_{i,j} - \frac{\Delta x \cdot v_{i,j}}{2 \Delta y \cdot u_{i,j}} [T_{i,j+1} - T_{i,j-1}] + \frac{v \cdot \Delta x}{\Delta y^2} [T_{i,j+1} + T_{i,j-1} - 2T_{i,j}] \]  \hspace{1cm} (6)

3. Numerical model & boundary conditions

3.1. Physical model of problem

The physical model of problem consists of a horizontal flat plate placed on a smooth surface, schematically illustrated as Figure 1. The length \((alen)\) of plate was taken as 0.5 m. The thickness \((t)\) of plate was taken as 0.1 m.

\[ u = 3 \text{ m/s} \]

\[ t = 0.1 \text{ m} \]

Length of plate \((alen) = 0.5 \text{ m}\)

\[ \text{Figure 1. Physical model of problem.} \]

3.2. Solution domain

The solution domain was divided into \(m\) and \(n\) number of cells. Number of grid points in \(x\) and \(y\) direction were designated as \(i\) and \(j\) respectively. The schematic diagram of solution domain is shown in Fig 2. The spacing between grid points along \(x\)-direction was solved by using the following correlation:

\[ delx = \frac{alen}{m-1} \]  \hspace{1cm} (7)

Where, the symbol \(alen\) represents the length of plate on which fluid flows, and \(m\) is the value of number of grid points along \(x\)-direction \((i=1, 2, 3, \ldots, m+1)\) whose value was taken as 1001. However, the spacing between grid points along \(y\)-direction was solved by using the following correlation:

\[ delx = \frac{t}{n-1} \]  \hspace{1cm} (8)

Where, \(t\) is the thickness of plate = 0.1 m, and \(n\) is the number of grid points along \(y\) direction \((j=1, 2, 3, \ldots, n+1)\) whose value was taken as 51.
3.3. Boundary conditions
Various boundary conditions to solve momentum equation, continuity equation and thermal energy equation are defined. Four different boundary conditions (namely no slip, free slip, inflow, and outflow conditions) were applied to solve the discretized governing equations. The boundary conditions are written as:

- **No Slip boundary conditions**: The no-slip boundary condition for viscous fluids states that the fluid achieves zero velocity (i.e. hydrodynamic condition) and temperature gradient (i.e. thermal condition) at a solid boundary. In present study, the solid boundary was the surface of horizontal flat plate. So, the no slip boundary conditions were applied as follows:

  \[ u(i,1) = v(i,1) = 0 \quad \text{for} \quad 1 < i < m \]  
  \[ u(i,1) = v(i,1) = 0, \quad \text{and} \quad T(i,1) = T_w \quad \text{for} \quad 1 < i < m \]  

- **Inflow boundary conditions**: Inflow boundary conditions were applied for the inlet of fluid (in Figure 2, at the left hand side of mesh i.e. \( i = 1 \)). Hydrodynamic as well as thermal boundary conditions were applied as follows:

  \[ u(1,j) = u_o, \quad v(1,j) = 0 \quad \text{for} \quad 2 \leq j \leq n \]  
  \[ u(1,j) = u_o, \quad v(1,j) = 0, \quad T(1,j) = T_o \quad \text{for} \quad 2 \leq j \leq n \]  

- **Outflow boundary condition**: These conditions were applied at the outlet of flow while the fluid passes the plate i.e. \( i = m \). In this case, velocity remains constant i.e. change in velocity is zero. Outflow boundary condition is written as:

  \[ \frac{du}{dx} = 0, \quad u_i = u_{i-1} \]  

**Figure 2.** Discrete grid points in the mesh.
Or
\[ u(i, j) = u(i-1, j), \quad v(i, j) = v(i-1, j) \]  (14)

- **Free slip boundary conditions:** Free slip boundary conditions were applied at top of boundary layer i.e., \( j = n \). Free slip boundary conditions are:
  Hydrodynamic,
  \[ u(1, n) = u_o, \quad v(1, n) = 0 \]  (15)
  Thermal,
  \[ u(1, n) = u_o, \quad v(1, n) = 0, \quad T(1, n) = T_o \]  (16)

4. **Numerical simulations**

In present work, the numerical simulations were performed by developing a numerical code in FORTRAN language. For performing the numerical simulations, the governing equations were solved by using the Gauss-Seidel iterations for a fine grid spacing \((m \times n)\). The grid size in the \( x \)-axis is about 1\% of total length of flat plate and 0.1\% of transverse dimension in the \( y \)-axis. Numeric iterations were run using a LINUX-based Ubuntu 2.6.321 operating system on Intel(R) core™ i5 machine having 2.67 GHz processing unit and 4 GB RAM. Average time taken by CPU to run a single iteration was 6-7 s. The properties of the air and water applied in numerical code are specified in Table 1. Surface temperature of plate \( (T_s) \) was taken as 300 K and temperature of fluid \( (T_o) \) was taken as 323 K. Grid taken for air/water have 1001×101\((m \times n)\) nodes.

| S. No. | Fluid   | Kinematic viscosity (\(v\)) | Inflow velocity (\(u\)) |
|--------|---------|-----------------------------|-------------------------|
| 1      | Air     | \(17 \times 10^{-6} \text{ m}^2/\text{s}\) | 3 m/s                   |
| 2      | Water   | \(0.66 \times 10^{-6} \text{ m}^2/\text{s}\) | 3 m/s                   |

5. **Results and Discussion**

Numerical models were recently made in FORTRAN programming language were run to numerically predict the boundary layer thickness, velocity and temperature profiles. The detailed results are as follows:

5.1. **Velocity profiles**

In Figure 3(a), as the value of \( x \) increases as the velocity \((u)\) decreases. It was seen that there is an initial sudden rise in velocity \((u)\) at \( x = 0.1 \) m. This initial sudden rise reduces as value of \( x \) increases. At \( x = 0.5 \) m, this rise becomes minimum. In Figure 3(b), the effect of buoyancy force can be clearly observed. Velocity curve moves sharply above 3 m/s under the effect of buoyancy force. Moreover, the buoyancy force acts from downward position so at lower distance \((x = 0.1)\), this effect was found maximum. As distance \( x \) increases, this effect rapidly decreases.

In case of water, characteristic of velocity curve remains same as air. However, the velocity curve of water travels a large vertical distance \((\gamma)\) as compared to air (In Figure 4). It was also observed that the initial sudden rise of velocity (up to 3 m/s) curve of water was higher than fluid air. Figure 5(b) shows the comparison of velocity profiles while buoyancy force is acting on the fluids. It was observed that characteristic curves moves more sharply and an initial sudden rise exists. Similar observation was made by Chang [15]. In case of air, characteristic curve has an initial sudden rise but it is not as sharper as in case of water.
5.2. Hydrodynamic boundary layer (δ)

Validation of results is carried out by comparing the numerical model’s results with the analytical results. The Blausius theory was applied to find the analytical results of boundary layer thickness. The Blausius equation for the laminar boundary layers over a flat plate is written as [16]:

$$\delta \approx 4.91 \sqrt{\frac{Lx}{u}} = 4.91 \frac{Lx}{\sqrt{Re_x}}$$  \hspace{1cm} (17)

Where, $Re_x$ is the Reynolds Number at distance $x$ on flat plate, $\delta$ is the overall thickness (or height) of the boundary layer (m), $\nu$ is the kinematic viscosity, $\nu=\mu/\rho$ (m²/s), $x$ is the distance downstream from the start of the boundary layer (m), $\rho$ is the density of the fluid (kg/m³), $u_0$ is the free stream velocity, and $\mu$ is the dynamic viscosity (Pa-s or kg/m-s).

The comparison between present numerical simulation results and Blausius model [16] is shown in Figure 6. By using the properties of air, the hydrodynamic boundary layer thickness from Blausius equation was found as 0.0035 m for air at $x = 0.1$ m. Generally, the hydrodynamic boundary layer thickness ($\delta$) is defined normal to the wall, and the point where the free stream velocity ($u$) is essentially that of the free stream is customarily defined as the point where:

$$u(y) = 0.99u$$  \hspace{1cm} (18)

Above equation indicates that the velocity of fluid equal to 99% of velocity at a particular distance ($x$).
In present study, the value of initial velocity is equal to 3 m/s so, the 99\% of value $u$ is calculated as 2.97 m/s. From Figure 6(a), at velocity 2.97 m/s value of $y$ is approximately equal to 0.0032 m which is equal to the boundary layer thickness as calculated from Blausius equation [16] i.e. 0.0035 m. It also confirms the validity of model. By comparison of results, it could be said that the present numerical model holds a good agreement with the analytical model and lies within the standard deviation of ±6.5\%.

Furthermore, the boundary layer thickness of water was found lesser as compared to water. The boundary layer thickness depends directly upon kinematic viscosity of fluid and distance of point from leading edge. The kinematic viscosity of water is lesser than air therefore, the boundary layer thickness of water was found lesser than that of air. Moreover, a stationary water particle near the stationary surface has lesser retarding effect as compared to the particles away from the surface [5]. Due to this, the effect of stationary surface in water stream forms up to a smaller distance than that of air stream.

5.3. Reynolds’ number (Re)

Reynolds’s number is the defined by the ratio of inertia to viscous force which is given as [17]:

$$Re_x = \frac{\rho u L}{\mu}$$  \hspace{1cm} (19)
Symbol $Re_x$ represents the value of Reynolds number at distance $x$ on flat plate which was calculated as $\approx 17647$ for air and $\approx 455000$ for water at $x = 0.1$ m. From the value of $Re$, it was concluded that the boundary layer thickness remains laminar for the fluid air whereas turbulent for water. Similarly, the value of $\delta$ and $Re_x$ at $x = 0.2, 0.3, 0.4, 0.5$ for air and water is shown in Figure 7.

Figure 7. Reynolds’s number of (a) air and (b) water at vertical distance above flat plate.

5.4. Thermal boundary layer ($\delta_T$)
Thermal boundary layer can be calculated by using the temperature profiles. Prandtl number ($Pr$) is related to the velocity and temperature field, given by [18, 19]:

$$\frac{\delta_T}{\delta} = \frac{1}{\sqrt{Pr}}$$

(20)

Figure 8. Numerically found thermal boundary layer thickness for air and water at $x = 0.1$.

From Figure 7, the temperature was evaluated as 322.46 K at 99% of velocity ($u$). Corresponding to this temperature, the boundary layer thickness ($\delta$) was evaluated as 0.0037 m. By putting these values of $\delta$ and $\delta_T$ in Eq. 20, the value of $Pr$ was calculated as 0.844 (approx.) which lies in the same range of $Pr$ (0.7 to 0.8) for air. Similarly for water, value of $\delta$ and $\delta_T$ was evaluated as 0.00069 and 0.0004 respectively. Putting these values in Eq. 20, the value of $Pr$ was found as 4 (approx.) for water. So, the present simulation results hold a good deal with the literature also.
5.5. Heat transfer coefficient

By using the results of boundary layer thickness and dimensionless numbers, various other parameters like shear stress, viscous force etc. can be evaluated. The slopes were plotted on the velocity and temperature profiles in order to evaluate the velocity and temperature gradients. From Figure 8, the value of gradient $du/dy$ at a particular point was calculated by putting value of coefficient of friction using the shear stress formula, written as:

$$\tau = -\mu \frac{du}{dy} \quad (21)$$

In the Figure 8(a), change in velocity ($du$) and vertical distance ($dy$) was 2.75 m/s and 2 m respectively at a point A. Therefore, velocity gradient $du/dy$ was evaluated as 1.375 s$^{-1}$. Here, the value of coefficient of viscosity ($\mu$) is 690 kg/hr-m for air). The value of shear stress ($\tau$) was calculated as 948 N/m$^2$ (approx.). Further, the shear force can be calculated by putting value of shear stress ($\tau$) and area of plate on which shear stress occur ($A = 0.25$ m$^2$), by using the following empirical correlation:

$$F = \tau A \quad (22)$$

The value of shear force ($F$) was evaluated as 237 N.

![Figure 9](image)

**Figure 9.** Variation of x-component of (a) velocity and (b) temperature, with the vertical distance above flat plate.

Similarly by using Figure 8(b), heat transfer (conductive and convective) was calculated by using temperature gradient $dT/dy$ from graph of thermal boundary layer from graph [20]:

$$q = -k \frac{dT}{dy} \quad (23)$$

Where, $k$ is the conductive heat transfer. At a point A, the change in temperature ($dT$) was 21 K (321-300), and change in vertical distance is ($dy$) was 0.0004 m. So therefore temperature gradient $dT/dy = 52500$ K/m. By putting value of heat transfer coefficient ($k = 0.0268$ W/m-K for air) and temperature gradient $dT/dy$ in Eq. 23, heat transfer ($q$) was calculated as 1407 kJ. The convective heat transfer ($h$) was also calculated, by using following correlation [1]:

$$h = \frac{k}{T_o - T_i} \left( \frac{dT}{dy} \right)_A \quad (24)$$
In above equation, the value $T_s = 300 \text{ K}$ (temperature at surface of plate), $T_o = 323 \text{ K}$ (temperature of fluid), $k = 0.0268 \text{ W/m-K}$ (heat transfer coefficient for air). The value of convective heat transfer ($h$) was calculated as $\approx 60 \text{ W/m}^2\text{K}$.

6. Conclusions

In this paper, a numerical code was developed in FORTRAN programming language to analyse the forced convection across flat plate. Following conclusions are drawn on the basis of present work:

- Buoyancy force acts from downward position so it was observed maximum at lower distance of plate in $x$-axis. However, buoyancy effect reduces as the distance of across $x$-axis increases.
- Velocity profile curves moves more sharply and an initial sudden rise exists. In case of air, characteristic curve has an initial sudden rise but it is not as sharper as in case of water.
- The velocity curve of water travels a large vertical distance (in $y$-axis) as compared to $x$-axis. There was an initial sudden rise of velocity (up to 3 m/s) curve of water was higher than fluid air.
- From the value of $Re$, it was concluded that the boundary layer thickness remains laminar for the fluid air whereas turbulent for water.
- Boundary layer thickness of water was found thinner as compared to fluid water.

7. References

[1] Singh J and Singh C 2017 Proceed. IMech. Part E J. Process Mech. Eng. 231 405.
[2] Singh J Singh C and Kumar S 2017 Advance. Intelligent Sys. Comput. 546 331.
[3] Singh J Kumar S Singh JP Kumar P and Mohapatra SK 2019 Particul. Sci. Technol. 37 275.
[4] Som SK 2008 Introduction to heat transfer (New Delhi: PHI learning Pvt. Ltd.).
[5] Sakiadis BC 1961 American Inst. Chem. Eng. 7 221.
[6] Peters N 1976 Int. J. Heat Mass Trans. 19 385.
[7] Janssen R and Armfield S 1996 Int. J. Heat Fluid Flow 17 547.
[8] Mahgoub SE 2013. Ain Shams Eng. J. 4 605.
[9] Kaw AK Kalu EK and Nguyen D 2011 Numerical Methods with Applications (Florida: University of South Florida).
[10] Liao SJ 1999 Int. J. Non-Linear Mech. 34 759.
[11] Crank J and Nicolson P 1947 Math. Proceed. Cambridge Phil. Society 43 50.
[12] Singh RK Sahu KB and Mishra TD 2013. Int. J. Eng. Res. App. 3 766.
[13] Totala NB Shimpi MV Shete NL and Bhopate VS 2013 Int. J. Eng. Sci. 3 27.
[14] Subramanyam D Chandrasekhar M and Lokanadham R 2013 Int. J. Mech. Eng. Technol. 4 54.
[15] Chang CL 2006 J. Phys. D: Appl. Phys. 39 1132.
[16] Blasius H 1908 J. Appl. Math. Phys. 56 1.
[17] Lindstedt M and Karvinen R 2012 J. Phys.: Conf. Ser. 395 012059.
[18] Prandtl L 1904 Int. Conf. of Mathematicians (Heidelberg: Germany) vol 3 (Verhandlinger) p 484.
[19] Ishak A 2009 J. Phys. A: Math. Theor. 42 195501.
[20] Mitra A 2017 IOP Conf. Ser.: Mater. Sci. Eng. 225 012096.