1. INTRODUCTION

Convection is a branch of heat transfer that deals with a variety of engineering applications like heat exchangers, greenhouse effect generators, radiators, condensers, air conditioners, refrigerators, electronics, etc. [1, 2]. Numerous numerical and experimental studies were carried out by various researchers to investigate the free convection phenomenon across the horizontal and vertical cylinders [3–7]. However, methane gas is used in applications such as energy storage devices and reactors [8]. Methane is generally stored in liquid form. The applications of methane in different industries are in the manufacturing of organic chemicals, used as refrigerated liquid, to run the power engines and refrigerators, electronics, etc. [1, 2]. Methane gas is used as a fluid in a closed room by achieving a steady-state condition. Experiments were carried out using air as a fluid in a closed room by achieving a steady-state condition. Implicit scheme of finite difference method was adopted to numerically simulate the free convection phenomenon across vertical tube using LINUX based UBUNTU package. Numerical data were collected in the form of velocity, temperature profiles, boundary layer thickness, Nusselt number (Nu), Rayleigh’s number (Ra), and heat transfer coefficient. The results of the Nusselt number showed a good agreement with the previous studies. Results data of heat transfer coefficient indicate that there were some minor heat losses due to radiation of brass tube and curvature of the tube.

Keywords: Natural convection, vertical tube, Boundary layer, Finite difference method, heat transfer, Nusselt number, Grashof number.

Received: May 2021, Accepted: September 2021  
Correspondence to: Dr Jashanpreet Singh, Department of Engineering, Punjab State Aeronautical Engineering College, Patiala, Punjab, India  
E-mail: jashanpreet.singh@mrsptu.ac.in  
doi:10.5937/fme2104932S  
© Faculty of Mechanical Engineering, Belgrade. All rights reserved

Jashanpreet Singh  
Assistant Professor  
Department of Engineering  
Punjab State Aeronautical Engineering College, Patiala, Punjab, India

Chanpreet Singh  
Professor  
Department of Mechanical Engineering  
Punjabi University, Patiala, Punjab, India

Computational Analysis of Convective Heat Transfer Across a Vertical Tube

This paper deals with the numerical investigation of the convective mode of heat transfer across a vertical tube. Experiments were carried out using air as a fluid in a closed room by achieving a steady-state condition. Implicit scheme of finite difference method was adopted to numerically simulate the free convection phenomenon across vertical tube using LINUX based UBUNTU package. Numerical data were collected in the form of velocity, temperature profiles, boundary layer thickness, Nusselt number (Nu), Rayleigh’s number (Ra), and heat transfer coefficient. The results of the Nusselt number showed a good agreement with the previous studies. Results data of heat transfer coefficient indicate that there were some minor heat losses due to radiation of brass tube and curvature of the tube.
rion reaches a steady state condition after the passage of long time periods. Senapati et al. [16] numerically studied the natural convection from a cylinder with annular fins positioned in vertical direction using a FLUENT ANSYS code. They found that the Nu was increased with the increase in fin spacing-to-tube’s diameter (S/D) ratio and Ra.

From the literature, it can be said that few studies deal with the simulation of convective heat transfer using programming languages. Moreover, the flow of methane gas inside and outside can cause serious damages at room temperature. On the other hand, the practical experimentation using methane gas is practically very complex and nearly impossible in actual environmental conditions. So, there is a need to perform a numerical simulation study to investigate the natural convection phenomenon for methane. In this work, the LINUX-based UBUNTU package was used to code the free convection along a vertically hanged tube using a FORTRAN language. In this study, methane was taken as the fluid for numerical simulations. However, the experiments were performed for air by achieving a steady-state condition. The FORTRAN code was validated for the experiments of air then utilized for the methane gas inside and outside can cause serious damages at room temperature. On the other hand, the practical experimentation using methane gas is nearly impossible in actual environmental conditions. So, there is a need to perform a numerical simulation study to investigate the natural convection phenomenon for methane. In this work, the LINUX-based UBUNTU package was used to code the free convection along a vertically hanged tube using a FORTRAN language. In this study, methane was taken as the fluid for numerical simulations. However, the experiments were performed for air by achieving a steady-state condition. The FORTRAN code was validated for the experiments of air then utilized for the methane gas.

2. NUMERICAL MODELING

2.1 Mathematical model

Mathematical modeling was done in order to discretize the governing equations of conservation of continuity, energy, and momentum. The discretized equations are used to generate the code in FORTRAN 95 language while subjected to adequate boundary conditions. From this computer code, numerical data comes on the terminal as an output. Fig. 1 shows a two-dimensional coordinate system model (x, y) in the area across a cylindrical tube positioned in the vertical direction. Letters ‘Q’ and ‘u’ symbolize the flow of heat and velocity respectively. Numerical model was prepared for the cylindrical tube of length (l) = 0.504 m and diameter (d) = 0.038 m. The governing equations are written as:

\[
\begin{align*}
\frac{\partial v}{\partial y} + \frac{\partial u}{\partial x} &= 0 \quad (1) \\
u \frac{\partial u}{\partial x} + g\beta(T - T_d) - \nu \left( \frac{\partial^2 u}{\partial y^2} \right) - \nu \frac{\partial u}{\partial y} &= 0 \quad (2) \\
u \frac{\partial T}{\partial x} = \alpha \left( \frac{\partial^2 u}{\partial y^2} \right) - \nu \frac{\partial T}{\partial y} \quad (3)
\end{align*}
\]

where, Eq. (1), (2), and (3) are the governing continuity, momentum, and energy equations respectively. An implicit finite difference scheme was adopted to solve the governing equations. In this scheme, the Taylor series expansion was followed to solve the equations [17]. The approach used for solving the equations was combined in a manner i.e. combination of both Forward difference and Central difference algorithms. Central differontation was used for the partial derivatives of \( y \) and forward differontation was used for derivatives of \( x \). The discretized equations are written as:

- Discretised continuity equation:
  \[
  v_{i+1,j+1} = v_{i+1,j-1} - 2\frac{\Delta y}{\Delta x} \left[ u_{i+1,j+1} - u_{i+1,j-1} \right] \quad (4)
  \]

- Discretised momentum equation:
  \[
  u_{i,j} - \frac{\partial T_{i+1,j}}{\partial y} = -\frac{\nu}{\Delta y} \left( u_{i+1,j+1} - u_{i+1,j-1} \right) + \frac{\beta g \Delta x}{\Delta y} \left( T_{i+1,j} - T_d \right) \quad (5)
  \]

- Discretised energy equation:
  \[
  T_{i+1,j+1} = \frac{v_{i,j}}{\nu} \left( \frac{\partial^2 u}{\partial y^2} \right) \left[ T_{i+1,j+1} + T_{i+1,j-1} \right] + \frac{\Delta x}{\Delta y} \left[ \frac{\Delta T_{i+1,j}}{\Delta y} - \frac{\Delta T_{i+1,j}}{\Delta y} \right] \quad (6)
  \]

In the area across a cylindrical tube positioned in the vertical direction

\[
\begin{align*}
Q &= 0 \\
\frac{\partial Q}{\partial y} &= u \quad y \rightarrow \infty
\end{align*}
\]

Figure 1. Two-dimensional coordinate system model (x, y) in the area across a cylindrical tube positioned in the vertical direction.

Truncated or discretized Eq. (4) to (6) were used to generate a FORTRAN 95 code. Truncated forms of governing equations were imposed following boundary conditions:

\[
y = 0, T = T_s, u = 0, v = 0 \quad (7)
\]
In this equation, the $y = 0$ indicates the no-slip boundary condition that indicates the region of initiation of the boundary layer.

$$y \to \infty, T = T_o, u \to 0, v \neq 0$$  \hspace{1cm} (8)

In Eq. (8), value $y$ approaches to $\infty$ represent the free slip condition for the faraway region.

2.2 Computational domain

The computation domain was developed in the FORTRAN 95 programming language. The typical grid used in the present study is Fig. 2. Before simulations, a grid independence test was conducted for $m \times n$ sized grids of $1001 \times 51$, $1001 \times 101$, and $2001 \times 101$ cells. The balanced grid of size $m \times n$ was chosen for $1001 \times 101$ cells. The size of the grid was round-off to 0.5 (lengthwise) and 0.04 (widthwise) to avoid the complexity and to produce results according to the actual parameters. The points in 2D space across the tube in the $x$ and $y$-axis are represented by the symbol ‘$i$’ and ‘$j$’ respectively. The range of ‘$i$’ was taken as 1 to $n+1$ whereas 1 to $m+1$ for ‘$j$’. Computation model was prepared for the cylindrical tube of length ($l$) = 0.504 m and diameter ($d$) = 0.038 m. For the numerical simulations, a LINUX Ubuntu 2.6.321 machine was used. For the numerical simulations, a LINUX-based Ubuntu 2.6.321 operating system was used. The Gauss-Seidel iterations were used to solve the governing equations. The CPU time required to single run was noted as 10 to 15 sec.

3. EXPERIMENTS

Experiments were conducted by using the equipment shown in Fig. 3. The experiments were conducted in a closed room that was free from the influence of air. During the experiments, the temperature of the tube’s surface was measured at a steady-state condition on eight different locations where the RTD PT-100 type sensors were fixed. The experimental readings were taken after achieving the steady-state condition which took approximately 3 hours. The average temperature of tube surface ($T_s$) was measured from the experimental condition of air and taken as 310 K. The ambient conditions were maintained in a closed room during experiments and ambient temperature of the surrounding atmosphere ($T_a$) was measured as 294 K. By using the values $T_o$ and $T_s$, the film temperature was calculated as:

$$T_f = \frac{T_s + T_o}{2}$$  \hspace{1cm} (9)

From the above equation, $T_f$ was calculated as 302 K. At 302 K, the value of volumetric thermal expansion ($\beta$) was noted as $3.31 \times 10^{-3}$ K$^{-1}$. The fluid used for the experiments was natural air. However, the computational model was designed for methane (CH$_4$). Kinematic viscosity of air ($\nu$) was used as $1.7 \times 10^{-6}$ m$^2$/s, Prandtl number ($Pr$) = 0.7279, and thermal diffusivity ($\alpha$) = $2.4 \times 10^{-6}$ m$^2$/s. These values were put in the FORTRAN program to perform the numerical simulations.

Results obtained for the vertical tube were validated by using the equation of the vertical plate. According to this equation, a vertically positioned cylinder can be preserved as a vertically positioned plate if it satisfies the criterion [18]:

$$\frac{d}{l} \geq \frac{34}{\sqrt[4]{Ra}}$$  \hspace{1cm} (10)

By following this equation, the following condition satisfies i.e. $d/l = 0.0754$ which was found lesser than 0.291. Eq. (10) does not satisfy the analytical model. So, it implicates that there are some obvious heat losses due to the effect of curvature and radiation.

4. RESULTS AND DISCUSSION

In the previous section, the experimentation was done for the natural convection around the vertical tube using the air as a fluid. By using the experimentation data, further numerical simulations were done in order to obtain the numerical data. Numerical data was used to draw the results in the form of tables, charts, and graphs from the output of computer code. Numerical outputs were obtained in the form of velocity and temperature profiles which were used to obtain the hydrodynamic and thermal boundary layer thickness, respectively.
4.1 Velocity profiles

Velocity profiles and hydrodynamics boundary layer thickness of methane were studied for the different values of locations (x) on the vertical tube. Fig. 4 illustrates the velocity profiles for methane at different values of x. The boundary layer thickness is a function of kinematic viscosity and distance of a point from the leading edge. It is noticed that velocity (u) was observed minimum at x = 0.1 m however maximum at x = 0.5. As the vertical distance x (from bottom to top) increases, velocity (u) increases. Hence, the velocity curve at x = 0.5 m travels longer distance as compared to other profiles in the horizontal direction (y).

![Figure 4. Velocity profiles for methane in vertical direction at various locations along heated vertical tube](image)

4.2 Hydrodynamics boundary layer thickness

The distance up to which the velocity gradient effects are significant is the Hydrodynamic boundary layer thickness (δ_H). As the velocity (u) approaches zero hydrodynamic boundary layer ends. Hydrodynamic boundary layer thickness (δ_H) directly depends upon velocity gradient. The numerical data was generated from the output obtained from the computer code. Numerical data directly depends upon the initial parametric values of input and boundary-layer conditions. For the proof to be valid, the numerical results of the δ_H were compared with the analytical results calculated by using Von Karman’s method of solving free convective heat transfer from a vertical cylinder to adjacent fluid [1]. According to this method, the boundary layer thickness can be calculated by [1]:

\[
\frac{\delta_{anl}(H)}{x} = 3.93 \left( \frac{Pr + 0.952}{Gr_x} \right)^{1/4}
\]

where the analytical hydrodynamic boundary layer thickness was denoted by δ_{anl}(H) (in m). Symbol Pr, Gr_x, and x are the Prandtl number, analytical Grashof number, and vertical distance of the tube from the bottom to the top (x = 0.1, 0.2, 0.3, 0.4, and 0.5 m), respectively.
From these velocity profiles, the hydrodynamic boundary layer thickness was calculated. Fig. 6 shows the boundary layer thickness for methane at different values of vertical distance (x). In Fig. 6, the numerical boundary layer thickness (δ_b) at x = 0.1 m was observed as 0.0124 m which was found minimum. At x = 0.2, 0.3, and 0.4 the value of δ_b was observed 0.0158, 0.0172, and 0.0192, respectively. At x = 0.5 m, numerical boundary layer thickness (δ_b) was observed 0.0216 m which was found maximum. Numerical boundary layer thickness at different points of x is given in Table 1. From Table 1, the general observation indicates that the value of δ_b increases with the value of the Grashof number (Gr).

It was noticed that data between numerical hydrodynamic boundary layer thicknesses fall within the range of ±5% deviation with analytical boundary layer (+0.002 m).

### Table 1. Summary of numerical and analytical δ

<table>
<thead>
<tr>
<th>Parameter</th>
<th>0.1</th>
<th>0.2</th>
<th>0.3</th>
<th>0.4</th>
<th>0.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gr (×10^6)</td>
<td>1.7</td>
<td>13.6</td>
<td>45.9</td>
<td>108.8</td>
<td>212.2</td>
</tr>
<tr>
<td>δ_b (num)</td>
<td>0.0145</td>
<td>0.0172</td>
<td>0.0190</td>
<td>0.0204</td>
<td>0.0216</td>
</tr>
<tr>
<td>δ_b (anal)</td>
<td>0.0124</td>
<td>0.0158</td>
<td>0.0172</td>
<td>0.0192</td>
<td>0.0216</td>
</tr>
</tbody>
</table>

The distance up to which the temperature gradient effects are significant is termed as thermal boundary layer thickness (δ_T). As the readings of temperature approach ambient temperature, the thermal boundary layer ends.

### Figure 8. Comparison of temperature profiles of different fluids at location x = 0.5 m

Fig. 9 also shows the comparison between numerical and analytical boundary layer thickness. In Fig. 9, the numerical boundary layer thickness (δ_T(num)) was found minimum at x = 0.1 and observed as 0.0142 m. At x = 0.2, 0.3, and 0.4 m, it was observed as 0.0176, 0.0208, and 0.0236 m respectively. At x = 0.5 m, numerical boundary layer thickness (δ_T(num)) was observed maximum as 0.0260 m. For the proof of the results of thermal boundary layer thickness, we have an equation in which Prandtl number (Pr) is a connecting link between the velocity field and temperature, written as [17]:

$$\frac{\delta_T}{\delta_H} = \frac{1}{\sqrt{Pr}} \quad (12)$$

Comparison between numerical and analytical values of δ_T for methane at different points of x. is given in Table 2. Analytical δ_T observed at x = 0.1, 0.2, 0.3, 0.4, and 0.5 m was 0.0168, 0.0200, 0.0221, 0.0237, and 0.0251 m respectively. In Fig. 9, it is observed that data between numerical δ_T falls within in the range of ±5% deviation (+0.002 m) with analytical δ_T. Results of the numerical experimental thermal and hydrodynamics boundary layers were found consistent and in good agreement.

### Table 2. Comparisons between numerical and analytical thermal boundary layer thickness (δ_T)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>0.1</th>
<th>0.2</th>
<th>0.3</th>
<th>0.4</th>
<th>0.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>δ_T (num)</td>
<td>0.0142</td>
<td>0.0176</td>
<td>0.0208</td>
<td>0.0236</td>
<td>0.0260</td>
</tr>
<tr>
<td>δ_T (anal)</td>
<td>0.0168</td>
<td>0.0200</td>
<td>0.0221</td>
<td>0.0237</td>
<td>0.0251</td>
</tr>
</tbody>
</table>

It was also noticed that temperature falls early at x = 0.1 m. As the vertical distance (x) increases, a sudden decrease in the value of T drops. At point x = 0.5 m, the sudden fall of temperature was observed to be minimum. Hence, the temperature curve at x = 0.5 m travels more distance as compared to other profiles in the vertical direction at various locations along with the heated vertical cylinder.
horizontal direction (y). Thermal boundary layer thickness directly depends upon temperature gradient.

Figure 9. Comparison of numerical and analytical boundary layer thickness for methane

4.5 Nusselt number

From numerical data, various parameters related to fluid dynamics can be evaluated for instance Nusselt number, heat transfer coefficient (h), conduction heat (q), etc. By measuring the slope of temperature profiles for the various locations (x), the temperature gradient (dT/dy) was calculated. The temperature gradient dT/dy was calculated as 3353.50, 2430.62, 2059.75, 1854.53, and 1681.81 K/m respectively at x = 0.1, 0.2, 0.3, 0.4, and 0.5 m. Convective heat transfer coefficient (h_c) can be calculated using the empirical relationship written as:

\[ h_c = \frac{k dT}{dy} \left( \frac{T_s - T_a}{T_a} \right) \]  \hspace{1cm} (13)

The value of h_c was calculated by using the k = 0.03228 W/mK i.e. the value of conductive heat transfer coefficient, surface temperature, and ambient temperature. The value of h_c can be utilized to evaluate the local Nusselt number (Nu_x). Nu_x was calculated by using the heat transfer coefficient (h_c), which is written as [22]:

\[ Nu_x = \frac{h_c x}{k} \] \hspace{1cm} (14)

In the above equation, the value of k was taken as 0.03228 W/mK. Churchill and Chu [23] also proposed a correlation for the calculation of Nu based on the Pr and Ra, written as:

\[ Nu_x = 0.68 + \frac{0.67 Ra^{0.25} x^{4}}{Pr^{4/9}} \] \hspace{1cm} (15)

Present results of Ra and Pr satisfies the following condition: 0 < Ra_x < 10^6 and 0 < Pr < 10^6. In the present study, the value of Ra lies in the range 1.24×10^6 to 1.54×10^8 which follows the condition of laminar flow i.e. Ra ≤ 10^6.

In the popular literature, another correlation is available which was proposed by McAdams [25], given as:

\[ Nu_x = 0.59 \times Ra^{0.25} x \] \hspace{1cm} (16)

Figure 10. Analytical and numerical Nusselt number across the length of the vertical tube for the methane

The above equation is applicable to present results due to the agreement of condition: 10^6 < Ra_x < 10^8. By using the Eq. (14–16), the present numerical results of Nu were compared with the analytical results. Fig. 10 shows the analytical and numerical values of local Nu across the length of the vertical tube for the methane. It was observed that the present results showed a good agreement with Churchill and Chu [23] and McAdams [25]. A new model (namely Jashanpreet and Chanpreet Model) has been proposed based on present results, given as:

\[ Nu_x = 1.41 \times Ra^{0.19} \] \hspace{1cm} (17)

To test the above newly proposed model, the predicted data were plotted against the actual numerical results, and shows a good correlation coefficient (R^2) of 0.9989, as seen in Fig. 11.

4.6 Heat transfer coefficient

The value of convective h_c was evaluated as 6.76, 4.90, 4.15, 3.74, and 3.39 W/m^2K at x=0.1, 0.2, 0.3, 0.4, and 0.5 respectively. The direct average of h_c was evaluated as 4.59 W/m^2K. Fig. 12 indicates a variation in the values of numerically evaluated h_c with respect to constant values of analytical h_i. The average value of heat transfer coefficient along the overall length (h_i) was written as:

\[ h_i = \frac{4}{3} \bar{h}_c \] \hspace{1cm} (18)

From the above equation, the h_i comes out as 6.12 W/m^2K (approximately) without consideration of any curvature effect.

According to Newton’s law of cooling, the calculation of analytical heat transfer coefficient can be done using the following equation [26]:
\[ h_l = \frac{Q}{A(T - T_a)} \]  

(19)

From the above equation, the value of \( Q \) was calculated from the experimental current and voltage values and calculated as 6.759 W. The analytical value of \( h_l \) was calculated as 6.76 W/m²K (without any radiation loss). By comparison, the analytical and numerical values of \( h_l \) were found different. As mentioned in Sec. 3, the value of the average heat transfer coefficient doesn’t satisfy the equation of the plate, so there was a production of some radiation and curvature heat losses. To account for the effect of radiation loss, the following equation was used [27]:

\[ q_{\text{rad}} = \sigma \varepsilon (T_S^4 - T_0^4) \]  

(20)

Figure 11. Actual versus predicted results of the newly proposed model of Nusselt number for numerical convection around vertical tube

Figure 12. Variation of numerical heat transfer coefficient with respect to constant analytical heat transfer coefficient

In the above equation, the \( \varepsilon \) represents the emissivity of the brass tube that was taken as 0.1 and 0.2 for analysis purposes. To evaluate the effect of curvature loss, the curvature factor was calculated by using the following relationship [19]:

\[ F = 1.3 \left( \frac{d}{l} \right) \left( Gr_D = l \right)^{0.25} + 1 \]  

(21)

By using the above equation, the curvature factor \( (F) \) was calculated as 1.0265. Fig. 13 represents the heat transfer coefficient from the numerical and analytical results. Fig. 13 shows that by accounting for the curvature factor, the value of \( h_l \) changes to 5.09 W/m²K which is lower than the actual average value of numerical \( h_l \). However by taking the \( \varepsilon = 0.1 \) and 0.2, the value of \( h_l \) changes to 5.67 and 5.09 W/m²K, respectively. The value of \( h_l \) was found equal in the case of the effect of curvature and radiation loss \( (\varepsilon = 0.2) \).

Figure 13. Heat transfer coefficient from the numerical and analytical results for methane

5. CONCLUSION

This study was carried out to numerically study the convective heat transfer across a vertical tube. Implicit scheme of finite difference method (FDM) was adopted numerically simulate the free convection phenomenon across vertical tube using LINUX based UBUNTU package. A FORTRAN code was run to successfully obtain the velocity profiles and temperature profiles. Numerical data were collected for the velocity, temperature profiles, boundary layer thickness \( (\delta) \), Nusselt number \( (Nu) \), Rayleigh’s number \( (Ra) \), and heat transfer coefficient \( (h) \). The following conclusions are drawn on the basis of numerical simulation results:

(i) By comparison of temperature profiles, it can be concluded that the temperature and velocity gradient travel longer distance in the direction perpendicular to the vertical tube as compared to the air and water vapours.

(ii) From the velocity profiles, the effect of buoyancy was demonstrated. The \( Pr < 1 \) for methane indicates that the buoyant forces directly controlled the velocity profile. This effect was found to be lesser dominant in the case of air. However, the water vapours generated larger friction forces which tend to reduce the dominance of buoyant forces to some extent. Thus, the velocity profiles of the air and water travel a smaller distance than methane.

(iii) Results of numerical and experimental thermal and hydrodynamics boundary layers were found to be consistent and in good agreement.

(iv) In the present study, the value of \( Ra \) lies in the range 1.24×10⁸ to 1.54×10⁹ which follows the condition of laminar flow i.e. \( Ra \leq 10^6 \).

(v) The results of local Nusselt number \( (Nu) \) and boundary layer thickness showed a good agreement with the previous studies. The results of numerical
boundary layer thicknesses fall within the range of ±5% deviation with analytical boundary layer (±0.002 m).

(vi) Heat transfer coefficients found from the experimental and numerical study are also found to be in good agreement. Heat transfer coefficient (h) results indicate that there were some minor heat losses due to radiation of the brass tube and curvature of the tube.

ACKNOWLEDGMENT

The authors received no financial support for the research, authorship, and publication of this article.

REFERENCES


NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d$</td>
<td>Diameter of the tube [m]</td>
</tr>
<tr>
<td>$g$</td>
<td>Gravitational force [m$^2$/s$^2$]</td>
</tr>
<tr>
<td>$Gr$</td>
<td>Grashof number</td>
</tr>
<tr>
<td>$h$</td>
<td>Heat transfer coefficient [W/m$^2$.K]</td>
</tr>
<tr>
<td>$i$</td>
<td>Grid points in $x$ direction</td>
</tr>
<tr>
<td>$j$</td>
<td>Grid points in $y$ direction</td>
</tr>
<tr>
<td>$k$</td>
<td>Thermal conductivity [W/K.m]</td>
</tr>
<tr>
<td>$l$</td>
<td>Length of the tube [m]</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$Ra$</td>
<td>Rayleigh’s number</td>
</tr>
<tr>
<td>$Q$</td>
<td>Input heat [W]</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature [K]</td>
</tr>
<tr>
<td>$u$, $v$</td>
<td>Velocity components in Cartesian [m/s]</td>
</tr>
<tr>
<td>$x$</td>
<td>Locations on vertical tube [m]</td>
</tr>
</tbody>
</table>

Greek symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta$</td>
<td>Boundary layer thickness [m]</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity [m$^2$/s]</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Thermal diffusivity [m$^2$/s]</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Volumetric coefficient of thermal expansion [1/K]</td>
</tr>
</tbody>
</table>

Superscripts

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a$</td>
<td>Ambient</td>
</tr>
<tr>
<td>$anl$</td>
<td>Analytical</td>
</tr>
<tr>
<td>$avg$</td>
<td>Average</td>
</tr>
<tr>
<td>$f$</td>
<td>Film</td>
</tr>
<tr>
<td>$l$</td>
<td>Length</td>
</tr>
<tr>
<td>$max$</td>
<td>Maximum</td>
</tr>
<tr>
<td>$num$</td>
<td>Numerical</td>
</tr>
<tr>
<td>$s$</td>
<td>Surface</td>
</tr>
<tr>
<td>$x$</td>
<td>Location</td>
</tr>
</tbody>
</table>

РАЧУНСКА АНАЛИЗА КОНВЕКТИВНОГ ПРЕНОСА ТОПЛОТЕ ПРЕКО ВЕРТИКАЛНЕ ЦЕВИ

J. Синг, Ч. Синг

Рад приказује нумеричко истраживање конвективног преноса топлоте преко вертикалне цеви. За експеримент је коришћен ваздух као флуид у затвореном простору чиме је постигнут услов стационарног стања. Усвојена је имплицитна шема методе кончних резултата за нумеричку симулацију феномена слободне конвекције преко вертикалне цеви коришћењем Linuxa заснованог на Ubuntu. Нумерички подаци су се односили на брзину, температурне профиле, дебљину граничног слоја, Нуселтов број, Райлијев број и коефицијент преноса топлоте. Резултати добијени за Нуселтов број су се слагали са резултатима објављеним у ранијим истраживањима. Коефицијент преноса топлоте показује незнатне губитке топлоте што је узроковала месингана цев и закривљеност цеви.