NUMERICAL SIMULATION OF BUOYANCY AFFECTED TURBULENT AIR FLOW IN A ROOM

A.Nouri-Borujerdi, A. Fathi-Gishnegani
School of Mechanical Engineering, Sharif University of Technology
Azadi Avenue, Tehran, Iran
Email: anouri@sharif.edu

ABSTRACT
In this paper a three-dimensional steady state incompressible turbulent air flow is considered in a large single room. The buoyancy affected turbulent air flow is simulated by solving governing equations numerically. The turbulence modeling includes both $k-\varepsilon$ and zero-equation models and their results are compared to the experimental data.

The paper reviews several aspects such as displacement of radiator system performance, temperature and flow field distribution and comfort. The results show that the best temperature distribution and comfort obtain when radiator is installed under the window and its height be equal to or greater than that of the window.

Keywords: Natural convection, large cavity, turbulent flow, three-dimensional flow, numerical method

INTRODUCTION
In modern building, it is important to resolve the relationship between geometric room parameters and air flow patterns produced by heating or cooling systems. These parameters have major impacts on the air flow patterns and performance of the heating or cooling systems in the room. The flow that forms in this case is usually incompressible and often turbulent due to the velocity levels and dimensions involved.

Two approaches of computational simulations are available for the study of indoor air temperature distribution. The first approach is the computational fluid dynamics (CFD) method and the second is simplified flow simulation method such as zonal model. The zonal model approach that has been developed in past two decades gives fast approximation, but it is unable to provide the required detailed information.

The CFD method solves numerically a set of partial differential equations for the conservation of mass, momentum and energy. The solution provides the field distributions of air temperature and velocity. Because turbulence models are approximated models they need to be validated by experimental data before being used as a design tool. Some researchers have made efforts to measured airflows in the real rooms or used small-scale model to represent a full scale room [1]. However, the experimental methods are much more expensive than using the CFD technique and lack the flexibility to simulate various boundary conditions such as a complex diffuser. With the CFD techniques, whole field distribution can be obtained from the solutions, while in most experiments the measurements can only be carried out in a few locations of the room.

Related studies on air movement with buoyancy affected flow in the room, also, take either experimental or numerical approaches. Ball and Bergman [2] used the Chebyshave collection technique to calculate the enclosure with differentially heated side walls. They calculated enclosure with a range of Rayleigh number up to $10^8$.

Peeters et al. [3] presented a numerical simulation of a natural convection in a square cavity. Low Raynoldes number model were used to study flows with Rayleigh number up to $10^{14}$. They found that the position of laminar to turbulence transition in a vertical boundary layer highly depends on the turbulence model used and the same Rayleigh number and grid spacing could produce multiple solutions.

Cheesewright et al. [4] measured natural convection in a cavity with a Rayleigh number of $5 \times 10^9$. Air velocity, temperature, and turbulence quantities were measured. They presented a method to correct the influence of the heat lost due to imperfect insulation. The nonsymmetrical boundary layer on hot and cold wall and relaminarisation at the bottom of hot wall boundary were found. This experiment was numerically simulated by Chen et al. [5].

Olsen et al. [1] measured natural convection in a full scale room as well as in a small scale model with differentially heated walls. The aspect ratio of a room was $H/L = 0.3$. This is a case very close to a real room. They found turbulent boundary
layers on vertical walls and recirculation flow in core region. This feature was not found in previous studies on natural convection in the enclosure. This is a very challenging problem that should be studied.

Schwenke [6] conducted a series of experiments on a ventilated room with a heated wall. He investigated the effect of Archimedes number on mixed convection in a ventilated room. Baly et al. [7] also investigated a ventilated room with heated flow. They measured mean velocity and temperature in the model room. They found that the direction of eddy in the room center is depending on Froude number of the cavity.

In this study we aim at prediction of velocity and temperature distribution for buoyancy affected turbulent air flow in a room such as natural convection. The experimental data will be used to verify accuracy of the numerical results.

GOVERNING EQUATIONS

Often airflow calculations use the Buossinesq approximation. The approximation takes constant air density in the momentum equation except in the buoyancy term. The indoor airflows are usually turbulent. With an eddy-viscosity model, the indoor airflows can be described by the following time-averaged Navier-Stokes equations for the steady state conservation of mass, momentum, and energy.

\[
\frac{\partial}{\partial x_i} \left( \rho V_i \right) = 0
\]

\[
\frac{\partial \rho V_j}{\partial x_j} = -\frac{\partial P}{\partial x_j} + \rho g \beta (T - T_{ref}) + \mu \left( \frac{\partial V_i}{\partial x_i} \right) + S
\]

where \( \mu \) is eddy viscosity and defined as follows.

If \( y^+ \leq 30 \)

The eddy viscosity based on Prandtl mixing length hypothesis [8], and Van Driest [9] damping function to include wall damping effects for zero equation model is:

\[
\mu_i = C_v \rho k^{1/2} l
\]

where \( C_v = 0.5478 \). K is turbulent kinetic energy; \( I \) is turbulence intensity and can be expressed through mean flow velocity as:

\[
I = \frac{\sqrt{\frac{2k}{\mu}}}{\sqrt{u^2 + v^2}}
\]

\[
\mu_i = 0.5478 \rho l \frac{\sqrt{u^2 + v^2}}{\sqrt{2}}
\]

The average turbulence intensity for indoor air flow is assumed to be 10%. This value is a good estimate for the occupied zone in mixing ventilation, yet it can be far from reality near or within the jet region.

Chen and Xu [10] used assumption of uniform turbulence intensity and derived an algebraic equation to express eddy viscosity as a function of local mean velocity, \( V \), and a length scale, \( l \), a distance to the nearest surface of enclosure.

\[
\mu_i = 0.03874 \rho l \sqrt{u^2 + v^2}
\]

CALCULATION PROCEDURE

The room considered is divided to small control volumes or cells, (Fig. 1). The differential equations are integrated over each control volumes to obtain discretization equations of finite difference form.

\[
a P \phi_P = a_W \phi_W + a_E \phi_E + a_S \phi_S + a_N \phi_N + a_T \phi_T + a_B \phi_B + b
\]

\[
b = S_c \Delta x \Delta y \Delta z
\]

where \( a_i \)s are the neighbor coefficients representing the convection and diffusion flux at the cell boundary surfaces. The subscripts W, E, S, N, B and T represent the neighbor grids of P, as show in Fig. 1. \( S_c \) And \( S_p \) are linearization coefficients of the source term, S in Eq. (3), [11].
The QUICK differencing scheme is used to determine the expression of the neighbor coefficient [12]. The SIMPLE algorithm with collocated grid used for the solution of the finite difference governing partial differential equations to the control volume approach. 

Boundary conditions required for governing equations are as:

On the wall:

\[
\begin{bmatrix}
    u \\
    v \\
    w
\end{bmatrix} = \begin{bmatrix}
    0 \\
    0 \\
    0
\end{bmatrix}, \quad \frac{\partial T}{\partial n} = 0
\]

where \(n\) is a local normal direction to the wall.

On the radiator and window:

\[T_{rad} = T_a, \quad T_{win} = T_c\]

**RESULTS AND DISCUSSION**

A CFD program with a zero equation turbulence model has been developed. The program provides a source code that can be easily changed to study the model performance and numerical algorithm. In this section, the first Phase focuses on the solution of air velocity and temperature based on the different following circumstances.

**Natural convection with differentially heated walls:** The physical model selected for investigation is a two dimensional rectangular enclosure with 2.5 m height and 7.9 m length used by Olsen et al.[1]. The two opposite vertical wall are at 35.3 °C and 19.9 °C. The ceiling and floor are assumed to be insulated.

Figs. 2a shows velocity vector field based on the two-layer zero equation model. The recirculation flow on the ceiling and floor appears in the computational results. The results show two large eddies at the upper-right and the lower-left corners. Olsen argued that this is because the horizontal mass flow on the ceiling is too great to entrain entirely into the cold wall boundary layer. After turning down the outer portion of the flow goes upward and forms the eddy and reverses flow. The two-layer zero equation model can consider relaminarization and low Reynolds effects near the walls. Fig. 2b shows velocity vector field based on the standard k-\(\varepsilon\) turbulence model. The results show no recirculation. It is probably due to the fact that the standard k-\(\varepsilon\) model cannot handle low-Reynolds number effects. Fig. 2c illustrates the experimental work of Olsen et al. [1]. Turbulence was found near the vertical walls. Two horizontal flow loops on the ceiling and floor were also observed. The flow pattern in the experiment shows two large eddies at the upper-right and the lower left corners. This suggests that a buoyancy force in the corner region is important.
The comparison of Figs. 2a and 2b shows that the two-layer zero equation model predicts better than the standard \( k-\varepsilon \) model.

Fig. 3 shows temperature distribution along the x-axis on the floor of the room. The x and y-axis are dimensionalized by the room length (L=7.9 m) and difference temperature between the hot and cold wall respectively. The results of the present work show a good agreement with the experimental data.

A similar plot, but in the vertical direction of the room is depicted in Fig. 4. In comparison with the Olsen's data, a good agreement is found in the core but not near the wall regions. This may be because of the imperfect insulation in the experimental work caused heat losses, gain to the ceiling or from the floor.
Natural convection in a room including a window and a radiator: Fig. 5 depicts the application of the CFD code with two-layer zero equation model for prediction of room airflow including one window and one radiator as a heat sink and heat source respectively. The room dimensions are 5 m long, 4 m width and 3 m height. The window dimensions are 2.5 m wide, 1 m height window with uniform temperature ($T_{\text{win}} = 17 \, ^\circ C$). The corresponding heat source is a uniform temperature radiator ($T_{\text{rad}} = 67 \, ^\circ C$) with 1 m width, and 1 m height.

In this section the combine fluid flow and heat transfer is simulated in a three dimensional room for various radiator dimensions and displacement. Rayleigh number which is just in the transition regime ($Ra=1.8 \times 10^7$) is based on the average height of window and radiator and their temperature difference.

Figs. 6 and 7 illustrate air flow pattern and temperature contours in z-y plane at the middle of the x-axis. As shown In Fig. 7, the temperature variation across the room is not uniform and varies from $T_{\text{win}}$ to $T_{\text{rad}}$.

In the second case the window and radiator are assumed to be installed on two neighbor walls, Figs. 8, 9. These figures are plotted on y-z and y-x plane at $x/L=0.5$ and at $z/W=0.5$ respectively. There is a strong upward warm jet near radiator and one downward cold jet near the window. Therefore, the cold air penetrates to the room.

Figs. 10 and 11 illustrate Temperature contours on the y-z plane at $x/L=0.5$ and y-x plane at $z/W= 0.5$ respectively. The Results show that a non-uniform temperature distribution across the room and some cold regions on the floor of room.

In the third case the radiator and window are assumed to be installed on the same wall. Figs. 12-13 are plotted on the y-z plane at $x/L=0.5$. There is a strong upward warm jet near the radiator and one downward cold jet near the window. Hence, the cold and hot jet is mixed and the cold air couldn’t penetrate to the core region of the room. There is, also, a suitable uniform air distribution in Fig. 13. This case suggests that the widow and radiator at the same wall is the best selection for winter heating.
Fig. 6 Air room flow pattern with one widow and one radiator on two opposite wall at $x/L = 0.5$

Hwind=1m
Hrad=1m

Fig. 7 Air room temperature contours with one widow and one radiator on two opposite wall at $x/L = 0.5$

Fig. 8 Air room flow pattern with one widow and one radiator on two neighbor walls at $x/L = 0.5$

Fig. 9 Air room flow pattern with one widow and one radiator on two neighbor walls at $z/W = 0.5$

Fig. 10- Air room temperature contours with one widow and one radiator on two neighbor walls at $x/L = 0.5$

Fig.11- Air room temperature contours with one widow and one radiator on two neighbor walls at $z/W = 0.5$
Figs. 14 and 15 illustrate two cases in which the radiator heights are different with the same window height. As shown in the figures, the strong air flow direction changes from upwards to downwards when the height of radiator changes from 1.25 to 0.75 m respectively. In addition, the air temperature contours next to the respective radiators are 25 and 20°C respectively. Comparison of Figs. 14 and 15 also show that the upward warm jet next to the larger radiator is stronger than the downward cold jet next to the window.

Finally, Fig. 16 illustrates that the window and radiator heat flux versus room temperature. It is clear that the two lines have the same trend, because in the steady state case the heat loss from the radiator should be identical to the heat gain by the window. The results indicate that increasing the room temperature causes heat loss increases from the window. On the other hand, one degree centigrade increment of the room temperature results 6% increase in heat transfer from the window.
CONCLUSION

This paper proposes a two-layer zero equation model for prediction of room air flow pattern and temperature. The model is derived from the Navier-Stokes equations using the concept of eddy-viscosity. The main difference between this method and the conventional CFD approach with k-ε model is that the former model does not solve transport equations for turbulent quantities. The results demonstrate the capability of the two-layer zero equation model by applying it to predict the airflow with buoyancy affected flow in a room such as natural convection. This model can predict reasonably good indoor airflow pattern and air temperature distribution. Since the two-layer zero equation model does not solve transport equations for turbulence, the computer memory needed is much smaller, and the convergence speed is 10 times faster than that of the k-ε model.

The best temperature distribution and comfort conditions obtain when radiator is installed under the window and its height be equal to and greater than that of the window. Distance between radiator and window has not major effect on temperature distribution in the room, but increasing the distance between window and radiator results in decrease heat loss from the window. Finally, one degree centigrade increment of room temperature results 6% increase in heat loss from the Window.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>b</td>
<td>constant</td>
</tr>
<tr>
<td>C_p</td>
<td>Specific heat</td>
</tr>
<tr>
<td>F</td>
<td>flux</td>
</tr>
<tr>
<td>g</td>
<td>gravity acceleration</td>
</tr>
<tr>
<td>H</td>
<td>height</td>
</tr>
<tr>
<td>I</td>
<td>turbulence intensity</td>
</tr>
<tr>
<td>k</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>K</td>
<td>turbulence kinetic energy</td>
</tr>
<tr>
<td>l</td>
<td>characteristic length</td>
</tr>
<tr>
<td>L</td>
<td>length</td>
</tr>
<tr>
<td>P</td>
<td>pressure</td>
</tr>
<tr>
<td>P_ref</td>
<td>pressure at return</td>
</tr>
<tr>
<td>Ra</td>
<td>Rayleigh number g β(T_h - T_c)H^3 / αν</td>
</tr>
<tr>
<td>S</td>
<td>source term</td>
</tr>
<tr>
<td>T</td>
<td>temperature</td>
</tr>
<tr>
<td>u</td>
<td>x-component velocity</td>
</tr>
<tr>
<td>v</td>
<td>y-component velocity</td>
</tr>
<tr>
<td>V</td>
<td>Velocity vector</td>
</tr>
<tr>
<td>w</td>
<td>z-component velocity</td>
</tr>
<tr>
<td>W</td>
<td>width</td>
</tr>
<tr>
<td>x</td>
<td>x-coordinate</td>
</tr>
<tr>
<td>y</td>
<td>y-coordinate</td>
</tr>
<tr>
<td>z</td>
<td>z-coordinate</td>
</tr>
</tbody>
</table>

Greek letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>α</td>
<td>Thermal diffusivity</td>
</tr>
<tr>
<td>β</td>
<td>expansion coefficient</td>
</tr>
<tr>
<td>μ</td>
<td>viscosity</td>
</tr>
<tr>
<td>ρ</td>
<td>density</td>
</tr>
<tr>
<td>φ</td>
<td>fluid property</td>
</tr>
<tr>
<td>ν</td>
<td>Kinetic viscosity</td>
</tr>
</tbody>
</table>

Subscript

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>Cold</td>
</tr>
<tr>
<td>h</td>
<td>Hot</td>
</tr>
<tr>
<td>in</td>
<td>inlet</td>
</tr>
<tr>
<td>m</td>
<td>mixing</td>
</tr>
<tr>
<td>out</td>
<td>outlet</td>
</tr>
<tr>
<td>ref</td>
<td>reference</td>
</tr>
<tr>
<td>t</td>
<td>turbulent</td>
</tr>
</tbody>
</table>
Superscript
+ dimensionless
- average

REFERENCES