LOCATION OPTIMIZATION OF COOLING DUCT THROUGH CFD ANALYSIS OF AIR FLOW AND TEMPERATURE DISTRIBUTION IN BUILDINGS FOR HUMAN COMFORT

POOJA GHODASARA, PRAGYAN JAIN

Abstract— Energy consumption for the heating and cooling of residential buildings accounts for nearly half of total use. Because the physical configuration of heating and cooling inlets and outlets is often determined from historical practice or convenience rather than optimal performance, it stands to reason that there is an opportunity to gain effectiveness of these systems by applying engineering principles to their design. In this study, a computational fluid dynamic analysis was performed to investigate the effect of the physical configuration of inlet and outlet vents on the temperature and flow patterns inside a room modeled for simplicity as a two dimensional enclosure. It was determined that for use cooling of a room, a low or floor located inlet vent coupled with an outlet that is positioned on the upper half of a wall yields the most desirable results in reaching, or nearly reaching, comfort conditions in the shortest amount of time. However, if either heating or cooling is expected to be the primary energy consumption, it may be advantageous to deviate from this configuration.

Keywords— CFD, Room Comfort Conditions.

I. INTRODUCTION

Although energy consumption per household decreased 31% from the year 1978 until 2011 in the India, the rise in the total number of homes results in an only slight decrease in total residential energy use per year. Heating and cooling costs account for a significant portion of this consumption; specifically, heating is identified as accounting for 8% of total residential energy consumption while air conditioning accounts for 41%. The consequence of this energy consumption pattern is that nearly half of the energy consumed in the residential sector is used by heating and cooling. It follows that with such a substantial portion of energy consumption in this area, any gain in efficiency or design of heating and cooling systems would result in a decrease in energy usage leading to reduced costs and, since most of the energy produced in the United States causes the production of greenhouse gases, reduced greenhouse emissions. Heating, ventilating and air conditioning (HVAC) systems are often designed by architects from experience, historical practice, and convenience. In order to reduce heating and cooling energy consumption by increasing the effectiveness of these systems, it would be advantageous to have a set of standardized design principles to guide the design of HVAC systems.

II. LITERATURE SURVEY

While many CFD studies have been conducted to determine flow patterns in buildings or to determine ideal control methods for HVAC systems, there were no studies found that investigated the effect of changing the vent locations in a living area. Also, no standards of vent configuration design were found. However, the following studies were useful in setting up proper CFD models for the purposes of this research.

A study by K. C. Chung was used extensively in the initial stages of this research because of the amount of data that was provided; Chung’s three-dimensional investigation into airflow in a partitioned environment provided a basis for a preliminary examination of potential models to be studied. Chung’s study included a detailed three-dimensional model that showed with velocities at various planes and temperature contours in those same planes. Additionally, details of the CFD model inputs were provided. However, in attempts to validate a model that was planned to be used in this current study, some vague model inputs and convergence problems did not allow a replication of the results. Accordingly, this preliminary CFD model validation attempt was dismissed.

The study by Sun et al. also provides a model of a CFD study of an indoor environment. A dynamic simulation was used to evaluate HVAC control systems based on a CFD model of a room, a mathematically modeled PID controller, and an actuation model. While the control model was beyond the scope of this current research, the CFD model was a helpful guide in gauging model inputs.

III. METHOD

A simple two-dimensional model of a room was analyzed using CFD-ACE+™, a commercial computational fluid dynamics software package, to examine airflow and temperature distribution under different circumstances. The model was set in summer conditions in which cooling was necessary and in winter conditions in which heating was
necessary. In both cases, the position of the inlet and outlet vents was parametrically changed yielding a large matrix of results. Additionally, a three-dimensional model was developed in much the same way as the two-dimensional model. With the same number of inlet and outlet locations as the two-dimensional case, a parametric study was planned; however, computing time and convergence problems prevented this study from being completed.

IV. TWO-DIMENSIONAL MODEL DESCRIPTION

Model Conditions
Each case of the simulation had been carried out in three dimensional model. The model replicates a standard class room in the institution. Transient analysis has been carried out for 1 hours with time step at the interval of 4 minutes. The model has been discretized with patch independent method. The convergent criteria for the model has been taken 1e-4. The physical model used to solve the flow regime is K-E model. Separate Source (user) has been defined as collective load in the room. For the three-dimensional model, steady state solutions were planned to be studied with transient analyses to follow. A description of the conditions and assumptions is given in the following sections.

Boundary Conditions
The hot conditions were meant to simulate a hot summer day. As such, a temperature of 300 K, or about 95 °F, was chosen as the outside air temperature. Additionally, the ground temperature was assumed to be adiabatic. The initial temperature inside the room was taken to be equal to the external ambient condition at 308 K. The inlet temperature of the model vents was set to be 2.3 K above the comfort condition at 298.5 K, or 77.6 °F.

5.1 Flow Module
The flow module determines the velocity and pressure fields by solving the two-dimensional momentum equations and the pressure correction equations, respectively. These equations are guided by the laws of conservation of mass and momentum, which lead to the use of the Navier-Stokes equations to iteratively resolve the flow solutions. The following sections describe the governing flow equations.

5.1.1 Mass Conservation Equation
The law of conservation of mass is applied to the model room which serves as the control volume; accordingly, the time rate of change of mass in the room must be balanced by the difference between the mass exiting and entering the room. This principle is described by the equation below.

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \]  (1)

where \( \rho \) is the air density and \( \vec{V} \) is the velocity vector. The first term on the left expresses the time rate of change of density while the second term describes the net mass flow through the control volume.

5.1.2 Momentum Conservation Equations
The law of conservation of momentum must also be applied; this states that the time rate of change of momentum of a fluid element is equal to the sum of...
the forces acting on the element. The equation below describes the two-dimensional x-component of this principle.

\[
\frac{\partial (p u)}{\partial t} + \nabla \cdot (p \vec{V} u) = \frac{\partial (-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + S_{Mx}
\]

where \( u \) is the fluid velocity in the x-direction, \( p \) is the pressure, \( \tau \) is the viscous stress, and \( S \) is the rate of increase in momentum caused by forces on the element. The left side of the equation is the rate of change of momentum in the x-direction while the left side describes the rate of change of total forces on the element in the x-direction. Similarly, the y-component is expressed below.

\[
\frac{\partial (p v)}{\partial t} + \nabla \cdot (p \vec{V} v) = \frac{\partial (-p + \tau_{yy})}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + + S_{My}
\]

where \( v \) is the fluid velocity in the y-direction.

### 5.1.3 Navier-Stokes Equations

To further develop the momentum equations given above, the variable viscous stresses for two-dimensional flows are defined below.

\[
\tau_{xx} = 2 \mu \frac{\partial u}{\partial x} - \frac{2}{3} \mu \left( \nabla \cdot \vec{V} \right)
\]

(4)

\[
\tau_{yy} = 2 \mu \frac{\partial v}{\partial y} - \frac{2}{3} \mu \left( \nabla \cdot \vec{V} \right)
\]

(5)

\[
\tau_{xy} = \tau_{yx} = \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)
\]

(6)

Combining equations 9 through 11 with momentum equations 7 and 8 results in the Navier-Stokes equations which are presented below.

\[
\frac{\partial (p u)}{\partial t} + \nabla \cdot (p \vec{V} u) = \frac{\partial (-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + + S_{Mx}
\]

(7)

\[
\frac{\partial (p v)}{\partial t} + \nabla \cdot (p \vec{V} v) = \frac{\partial (-p + \tau_{yy})}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + + S_{My}
\]

(8)

These above equations can be simplified and rewritten as the following equations.

\[
\frac{\partial (p u)}{\partial t} + \nabla \cdot (p \vec{V} u) = -\frac{\partial p}{\partial x} + \nabla \cdot (\mu \nabla u) + S_{Mx}
\]

(9)

\[
\frac{\partial (p v)}{\partial t} + \nabla \cdot (p \vec{V} v) = -\frac{\partial p}{\partial y} + \nabla \cdot (\mu \nabla v) + S_{My}
\]

(10)

### 5.1.4 Flow Properties

The following table shows the fluid properties that were held constant and used to calculate other model parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference Pressure</td>
<td>( P_{ref} )</td>
<td>101325 Pa</td>
</tr>
<tr>
<td>Universal Gas Constant</td>
<td>( R )</td>
<td>8.314472 J/mol K</td>
</tr>
<tr>
<td>Molecular Weight of Air</td>
<td>( MW )</td>
<td>29 kg/kmol</td>
</tr>
<tr>
<td>Dynamic Viscosity</td>
<td>( \mu )</td>
<td>1.846 ((10^{-5})) kg/m s</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>( C_p )</td>
<td>1007 J/kg K</td>
</tr>
<tr>
<td>Thermal Conductivity of Air</td>
<td>( k_{air} )</td>
<td>0.0263 W/m K</td>
</tr>
</tbody>
</table>

Using the above values, other variable model parameters were calculated within the CFD simulation. These calculations include the fluid density and the kinematic viscosity. The fluid density was calculated using the Ideal Gas Law. This equation is given below.

\[
\rho = \frac{(p + p_{eff}) MW}{R \cdot T}
\]

where \( \rho \) is density, \( p \) is pressure and \( T \) is the temperature of the air. Also, the kinematic viscosity was calculated with the following equation.

### VI. HEAT TRANSFER MODULE

The heat transfer module was used to calculate the temperature distribution within the model room as well as the interactions with the environment. These calculations were performed according to the law of conservation of energy. The equation that was solved in CFD-ACE+ was the total enthalpy equation, which is given below.

\[
\frac{\partial (h_{in} u_{in} + \nabla \cdot (h_{in} \vec{V} u_{in}))}{\partial t} + \nabla \cdot \left( h_{in} \left[ \frac{\partial u_{in}}{\partial x} \right] \right) \left( \frac{\partial \vec{V} u_{in}}{\partial x} \right) + \left( \frac{\partial \vec{V} u_{in}}{\partial x} \right) + S_{h}
\]

(13)

where \( h_{in} \) is the total enthalpy which is defined as:

\[
h_{in} = i + \frac{p}{\rho} + \frac{1}{2} (u^2 + v^2)
\]

(14)

where \( i \) is the internal energy, \( k_{eff} \) is the effective thermal conductivity, \( p \) is the static pressure, and \( \tau \) is the viscous stress tensor which was described in equations 9 through 11, above.

### 6.1. Thermal Boundaries

In addition to the above equations, it was necessary to define the conditions at the walls of the room to determine the thermal interaction with the ambient temperature outside of the model. The boundary consisted of three different elements: the ceiling, the walls, and the floor. While each of these sections had distinct thermal properties, a similar method of determining the effective heat transfer coefficients of each part was used. In general, an overall heat transfer coefficients was calculated using a thermal resistance model which took into account the conductive heat transfer within each layer of the boundary walls and the convective heat transfer to the ambient environment, if applicable. Overall, the heat transfer rate through the room’s boundaries can be expressed as follows:
\[ q = h_{\text{eff}} \cdot A \cdot (T_w - T_{\text{inf}}) \]  \hspace{1cm} (15)

where \( q \) is the heat transfer rate, \( h_{\text{eff}} \) is the overall heat transfer coefficient, \( A \) is the area of the boundary, \( T_w \) is the temperature on the inside of the boundary, and \( T_{\text{inf}} \) is the ambient temperature. Subsequently, the overall heat transfer coefficient, \( h_{\text{eff}} \), is defined as the reciprocal of the sum of the thermal resistances at the boundary, as shown in the equation below.

\[ h_{\text{eff}} = \frac{1}{R_{\text{total}}} \]  \hspace{1cm} (16)

where the \( R_{\text{total}} \) is the combined conductive and convective thermal resistances, \( R_c \) and \( R_h \), respectively. These properties are defined in the following equations.

\[ R_c = \frac{L}{k} \]  \hspace{1cm} (17)

where \( k \) is the material specific thermal conductivity and \( L \) is the thickness of the material layer.

\[ R_h = \frac{1}{h_{\text{inf}}} \]  \hspace{1cm} (18)

where \( h_{\text{inf}} \) is the convective heat transfer coefficient of air.

Detailed below are the assumptions, values, and calculations used for each of these three sections.

**VII. RESULTS**

The twenty five two-dimensional test cases were run under both hot and cold conditions. The data collected included the temperature data at the monitor point and the temperature distribution at all node points in the model room at the end of the 30 minute simulation. The normalized residuals of all computations were required to be at a 0.001 level. Under some cases, however, this level of convergence was not attained. If the residuals were below the acceptable level for a significant length of time, these simulations may be considered up to the point at which the solution began to diverge. Otherwise, if the solution only converged for a small number of time steps, these cases were excluded from the following discussion.

This flow pattern, which was dominated by natural convection, showed a lack of penetration of the heating or cooling air which in turn inhibited the incoming air from mixing with the room air. As a result, this general vent configuration was shown to give favorable results for cooling while showing poor results for cooling. An example of the natural convection flow and temperature pattern under cooling effect is shown in Figure XYZ. The incoming cool and relatively dense air almost immediately sinks to the floor where it only rises to the level of the outlet vent. Only the bottom portion of the room is cooled while the top stays close to the initial condition of 308 K. Additionally, while there is some degree of cooling that occurs near the floor, both the monitor point and the volume averaged temperature are above the comfort condition at over 305 K. Thus, since this option offers less than 3 K of cooling, it is clearly not an ideal configuration for cooling.
7.2 Temperature Data

Figure XYZ through Figure XYZ below show the transient temperature data at the monitor point for each configuration with all inlet options.

By examining this data set, some general trends can be discerned. While none of these configurations reach the desired comfort condition of 295.2 K, the data still provides insights into the patterns of temperature distribution. Looking at the cold condition, it can be seen that as the outlet is lowered from location one to location three.

Now, by examining the results of all configurations involving outlet 2, other generalizations about the results can be drawn. The following figures show the transient temperature data for at the monitor point for hot conditions.

This data set makes it possible to draw more generalizations about the results of the simulations. Furthermore, under the hot conditions, it can be noted that as the inlet location is moved from the top location one to the bottom location five, the cooling becomes more effective. Specifically, the best configuration, inlet 4, has a final monitor point temperature that is 13.71 K below the worst configuration and a volume averaged temperature that is 10.32 K below the worst configuration. A significant consequence of the preceding findings is that under both cold and hot conditions, better results are achieved with a lower inlet location.

Table 9 shows the final temperature reached at the monitor point in the transient test for each configuration under hot conditions while Table 10 gives the volume averaged temperature for the same time and conditions. Similarly, Table 11 and Table 12 provide the final monitor point temperature and volume averaged temperature at the end of the transient simulation under cold conditions. Because the comfort condition was never reached under cold conditions, there is only one table, Table 13, that shows the time required to reach the comfort condition.

Under hot conditions, a more negative difference in the tables above was considered to show more effective cooling. This is because the more negative the difference was, the lower the final temperature was below the comfort condition. Since the cooling system could simply be turned off before reaching temperatures below the comfort condition, a more negative difference also meant that the system would not have to operate as much to maintain thermal comfort. On the other hand, under cold conditions, the less negative the final temperature given in the tables was, the better the heating performance was considered to be. Because all of the cold conditions have the same heating input, a more negative final difference shows a smaller effect of heating on the

The tables have been highlighted to exhibit the configurations that performed the best; the green cells
indicate that the configuration resulted in a temperature that was better than the comfort condition while the yellow cells indicate that the configuration resulted in temperatures that were near the comfort condition.

CONCLUSIONS

While keeping either the inlet of the outlet location constant can help in determining trends in the results, it also helps to look at the complete data set. The transient monitor points were used to compare the amount of time required to attain the comfort condition. If the comfort condition was not attained, the data was still used to verify if the temperature had reached a steady state or if it was still in a transient mode. The most favorable result was considered to be one in which a steady state was reached at or near the comfort condition. Figure ABC below presents a matrix of monitor point temperature outputs for the duration of the transient tests.

As is indicated in the above figure, a higher outlet exhibits better cooling, when the position of dust is at the center the energy distribution is quite regular and uniform. Furthermore, with a high outlet, the cooling effectiveness is improved with a higher inlet. The cause of this trend can be determined from studying the flow patterns shown in Figure XYZ and Figure XYZ. As the air from the inlets flows down the left wall because of the natural convective forces and is then diverted to the right by the floor, it proceeds to stream over the outlet located on the floor. The flow then curls up the wall and is allowed to mix and fill the room. This prevents the flow from mixing with the room air and thus reduces the effectiveness of the heating or cooling.

On the other hand, when only considering the two outlets near the top, outlets one and two, outlet two, which is located on the side wall yields better results. This is caused because the air that curls up the right wall after being diverted from the bottom corner flows parallel to the outlet; a smaller fraction of this flow escapes before it is allowed to mix with the room air.

It is clear that the mode of operation, significantly affects the temperature distribution and flow patterns within the room. At the same time, there is no single configuration that reached the comfort condition under both modes of operation. Nevertheless, it was shown that a low inlet generally yields better performance for cooling and the configurations that showed the best results were obtained by the First Model. However, these results may not be best suited for every design scenario. Depending on energy usage patterns and geographical location or expected seasonal temperatures, one may desire to design a system such that it is optimized for either heating or cooling. If the environmental temperatures are typically cold and heating is the main energy use, the use of a low, wall oriented inlet with a low, wall oriented outlet would be ideal. Conversely, if the environment is characteristically warmer and cooling will be more often used an inlet that is centrally located on the height of a wall coupled with a high wall mounted outlet would be expected to yield the most desirable results.

With regard to the steady state investigation of the three-dimensional model, the data that was generated was not deemed viable because of the high residuals after evaluation. Accordingly, no reasonable conclusions can be drawn directly from these cases. However, it has been speculated that many of these model configurations do not actually have a steady state solution due to the effects of natural convection. Consequently, it would be necessary to run the simulations as transient in order to yield viable data.

RECOMMENDATIONS

While the original objective of this research was to use three-dimensional CFD analysis to determine a set of principles to guide the design of heating and cooling systems, many difficulties were encountered in devising, evaluating, and validating a three-dimensional model. As such, this leaves much room for future work in this area. Not only do problems such as computational limits and times complicate the matter of performing a three-dimensional investigation, but the sheer number of possible model configurations makes the development of a meaningful model more difficult as there are unlimited variations of actual physical room designs. Additionally, while a single model may be able to provide insights into the temperature and flow patterns within a room, changes as simple as an open doorway or any obstruction added within the room could dramatically affect the observed patterns within a physical building.

Areas of interest in future simulations would include changing of vent locations and changing of air injection methods. For example, a pulsed injection with a mean volumetric flow equal to the injection flow rate of a steady flow may offer more mixing and better results. Also, changing the number of inlets and outlets that are active in any given case may yield desirable results.
REFERENCES


★★★★