Numerical Modeling of Buoyancy-driven Natural Ventilation in a Simple Three Storey Atrium Building

Shafqat Hussain and Patrick H. Oosthuizen

Department of Mechanical and Materials Engineering, Queen’s University, Kingston, ON, Canada K7L 3N6

Email: oosthuiz@me.queensu.ca

ABSTRACT
A simple three storey atrium building was designed to investigate numerically the development of buoyancy-driven natural ventilation airflows and temperature distributions induced by solar radiation and heat sources present on each floor of the building. The ventilated space is warmed by heat gains from the solar radiation and other heat sources, as a result buoyant plumes are generated that entrain the surrounding air and transport warm air upwards, forming a warm, stratified layer in each storey and a column of warm air in the atrium that transports the ventilation flow. The airflow patterns, temperature distributions and ventilation flow rates predicted using Computational Fluid Dynamics (CFD) techniques are presented in this paper. The CFD simulations of the conditions in the building were carried out by solving the Reynolds Averaged Navier-Stokes (RANS) equations using SST k-ω turbulence model along with the DTRM radiation model. The resultant steady state governing equations were solved using the commercial CFD solver FLUENT®. The ventilation flow rates, interface heights, layer reduced gravities were non-dimensionalized and were compared with the design curves developed by others using analytical models for a simple geometry atrium building. It was found that CFD predictions agreed favorably with the general trends described by the analytical models indicating the capability of CFD modeling for predicting the buoyancy-driven displacement natural ventilation in atria buildings.

1. INTRODUCTION
Over recent decades natural ventilation has been widely recognized as one way towards achieving low-energy building design. Conventional ventilation systems based on mechanical components consume electric power. Due to energy crisis worldwide, the need to reduce energy consumption in buildings has developed interest to explore energy-efficient natural ventilation strategies. Wind field is thought to be the main source to provide driving force for the natural ventilation but in the absence of an external wind field, the alternative is to explore the use of stack pressure as the natural force to drive a ventilation flow. Stack pressure is the pressure difference between the interior and exterior of the building which can be created naturally by density differences caused by warm air generated inside the building. The pressure difference between inside and outside of the building drives ventilation flow through a building by means of thermal stack effects.

The architectural features that can enhance the stack effect, for example, tall solar chimneys, light wells or atria are sometimes employed. These structures potentially increase the height of the column of warm air inside the building and as a result increase the stack driving force which draws cooler air from the exterior – near the bottom or sides of the building – and vent it out after it has warmed from convective heat exchange with the internal space. The air is normally vented at the top of the building. Advanced stack-ventilated atria buildings have the potential to consume much less energy for space conditioning than typical mechanically ventilated buildings. The proper design of the natural ventilation system must be based on the detailed understanding of airflows within enclosed spaces governed by pressure differences due to wind and buoyancy forces. At the design stage, CFD modeling techniques can be utilized to investigate the possible ventilation flow rates, temperature distribution and thermal stratification within the ventilated space.

Ji et al. [1-2] studied the effect of ventilated and unventilated atriums connected to multi-storied spaces on the flow characteristics (thermal
stratification and airflow rates) by using numerical techniques. Cook and Lomas [3] showed comparison between analytical, physical and CFD modeling of natural convection flow for a single space with a localized point heat source and openings connected to the ambient environment. Chen [4] noted that simulation results using the Renormalisation Group (RNG) k-ε turbulence model agreed favorably with the analytical model and experimental measurements. Holford and Hunt [5] studied the fluid dynamics of a single ventilated space connected to a tall atrium and developed simple analytical models and validated using small-scale salt bath experiments. It was shown that at the design stage these techniques can be useful for understanding the flow characteristics, including the likely ventilation flow rate, fresh air distribution and temperature distributions. To observe the applicability of the analytical techniques suggested by Holford and Hunt [5], a simple three-storied atrium building was designed in the present work to investigate the use of buoyancy-driven natural ventilation.

2. BUILDING DESCRIPTION

Using the design approach developed by Holford and Hunt [5] for an atrium design, a simple three-storey atrium building shown in Fig.1 was designed to investigate the development of buoyancy-driven ventilation airflows.

Fig. 1 Geometry used in the CFD simulations

The building was assumed to be located in the region of Montreal, with the atrium on the southwest façade of the building-35 degrees west of south. In the same location, we studied numerically an atrium space in the Engineering Building of the Concordia University using CFD modeling approach and CFD model used was validated by comparing the numerical predictions against the experimental data available i.e, see [6-10]. Heat sources were assumed to be concentrated in the centre of each floor. The atrium exhaust is located on the highest point in the atrium and the storey inlets are located at the side walls of the rooms (see Fig.1) which were sized on each floor using the design curves developed by Holford and Hunt [5]. The effective atrium outlet opening area was selected equivalent to the inlet opening areas for all the floors.

3. NUMERICAL SOLUTION PROCEDURES

3.1 CFD Model

The validated CFD model was used in the present numerical study of a simple three-storied atrium building (see Fig.1) using a commercial software GAMBIT/FLUENT. In the model the key dimensions used are the atrium height (M=16m) and storey height (H=4m). Ambient temperature was set to \( T_{\text{amb}} = 25°C \). In practice the airflow inside the connected spaces is mixed by conduction, convection and radiation heat transfer effects. In this work, conduction effects were only considered for the glazing façade wall while all the other walls were assumed to be adiabatic to meet the assumptions of the mathematical models developed by Holford and Hunt [5]. When the first-order fluid parameters are of main concern (e.g mean temperature and flow rate) rather than turbulent fluctuation details, two equation eddy-viscosity turbulence models are generally thought suitable for modeling indoor flows.

After the numerical modeling and validation of the indoor flows in atria of two existing atria buildings in our previous studies [6-10] it was noted that the SST k-ω turbulence model with a DTRM radiation model, would be more suitable for the present CFD simulations. Pressure coupling was treated using the SIMPLE algorithm. The second-order upwind scheme was used to discretize the momentum, turbulent kinetic energy, dissipation rate and energy conservation equations. The body force weighted scheme was used to discretize pressure-velocity coupling. These were solved in a segregated manner. Convergence was considered to have been reached when the enthalpy residual was less than 0.1% and the flow variables residuals varied by 1% over the last 100 iterations. In all the simulations, the buoyancy flux \( B \) equal to \( 22.63 \times 10^3 \text{m}^3\text{s}^{-3} \) (equivalent to a heat source of 823W in the centre of each floor, 2x2m) and \( 14.57 \times 10^3 \text{m}^3\text{s}^{-3} \) (equivalent to a heat source of 530W in the centre of the atrium floor, 2x2m) was used. The under-relaxation factors were adjusted to achieve convergence. The mixed thermal boundary conditions were used for the façade glass surface.

The turbulence parameters such as the hydraulic diameter and the turbulent intensity were specified at the inlet using the relations hydraulic diameter =
2L/W/L+W and turbulence intensity = 0.16 Re\(^{-1/8}\). The optical properties of the glazing (semi-transparent) used in the previous studies with solar transmittance of 36% and absorptivity of 17.5% were used. The modeling of the glazing was simplified as a single glazing with effective thermal conductivity of 0.0626 W/m\(^2\)-K and total overall thickness of 24 mm. The radiation exchange between the facade and the sky was also taken into account. The sky temperature was calculated to be 14.1°C using the Mills [11] correlation, \(T_{\text{sky}} = [\varepsilon_{\text{sky}} T_{\text{out}}]^{1/4}\) where the emissivity of the sky (\(\varepsilon_{\text{sky}}\)) for the daytime was calculated to be 0.82 using the relation, \(\varepsilon_{\text{sky}} = 0.727 + 0.0060 T_{\text{out}}\) with an ambient temperature (\(T_{\text{out}}\)) of 25°C. All other boundaries of the domain, except the ventilation openings and the heat source, were modeled as no-slip wall boundaries with zero heat flux. A constant relative pressure of 0 Pa was imposed across the room inlets and the atrium outlet.

### 3.2 Radiation Model

To account for radiation, radiation intensity transport equations (RTEs) are solved. Local absorption by fluid and at boundaries links RTEs with energy equation. DTRM radiation model available in FLUENT™ was found suitable for the present studies. The main assumption followed in the DTRM model is that radiation leaving a surface element in a specific range of solid angles can be approximated by a single ray. It uses a ray-tracing algorithm to integrate radiant intensity along each ray and is relatively a simple model, and increase accuracy by increasing number of rays while applies to a wide range of optical thicknesses. A solar calculator is also available in FLUENT™ to calculate the beam direction and irradiation. The solar calculator was used to find the sun’s location in the sky with the given inputs of time, date and global location.

### 3.3 Mesh Independence Test

Three mesh densities were investigated: Mesh1 (415k cells), Mesh2 (8120k cells) and Mesh3 (1235k cells). In Mesh2 and Mesh3 more cells were located where more velocity and temperature gradients were expected e.g. near walls, ventilation openings and the area potentially occupied by the thermal plume in order to capture the variation in airflow in these areas. The numerical results shown in this section are based on the conditions for achieving the same ventilation flow rate each storey. Using three mesh densities the numerical results indicating the volume flow rates at each floor of the left-hand side of the building are shown in Table-1. From the results it is seen that they are very close to each other with a difference less than 1%. Considering both accuracy and computational time, it was decided that Mesh2 (800k cells) is fine enough to accurately predict volume flow rate, and temperature distributions in the building.

### Table-1 Volume flow rates at three floors using three mesh densities

<table>
<thead>
<tr>
<th>Floors</th>
<th>Volume Flow Rate (m(^3)/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh1</td>
<td>0.42</td>
</tr>
<tr>
<td>Mesh2</td>
<td>0.42</td>
</tr>
<tr>
<td>Mesh3</td>
<td>0.42</td>
</tr>
</tbody>
</table>

### 4. Results

The numerical results obtained from a series of CFD simulations are presented in two sections: Section 4.1 presents the results where the total effective opening area \(A_{\text{pt}}\) on each storey is the same. In section 4.2 the results where the effective opening areas were sized to have the same ventilation flow rate for each storey are presented.

#### 4.1 Simulations Using the Same Effective Opening Area \(A_{\text{pt}}\) on each Storey

Table-2 shows the quantitative results of the volume flow rates (m\(^3\)/s) of the buoyancy-driven ventilation with the variation of total effective opening area \(A_{\text{pt}}\) in the left-hand side rooms of the building.

<table>
<thead>
<tr>
<th>Total effective opening area (A(_r)/H(^2))</th>
<th>Volume flow rate (m(^3)/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Left side</td>
</tr>
<tr>
<td></td>
<td>Ground floor</td>
</tr>
<tr>
<td>0.0087</td>
<td>0.27</td>
</tr>
<tr>
<td>0.017</td>
<td>0.44</td>
</tr>
<tr>
<td>0.026</td>
<td>0.59</td>
</tr>
<tr>
<td>0.035</td>
<td>0.75</td>
</tr>
<tr>
<td>0.044</td>
<td>0.85</td>
</tr>
</tbody>
</table>

It was noticed that with the increase of the total effective opening area \(A_{\text{pt}}\), the ventilation flow rate increases until it reaches maximum for a certain value of the total effective opening area \(A_{\text{pt}}\). In order to confirm the accuracy of the CFD predictions, the quantitative numerical results obtained for the interface height, volume flow rate...
and reduced gravity were compared in non-dimensional form with the design curves developed analytically by Holford and Hunt [5].

Fig. 2(a, b, c) shows the comparison between the analytical and present CFD model predictions for the non-dimensional interface height (a), volume flow rate (b) and reduced gravity (c) with the same total effective opening area $A_t$ on each floor of the building. It was found that the CFD predictions of interface height, volume flow rate and reduced gravity agreed favorably with the general trends described by design curves drawn developed by Holford and Hunt [5]. It was observed that the lower storey has higher interface height and volume flow rate than that of the upper storey, while the temperature of the stratified layer in the upper storey is higher than that of the lower storey as was also reported in the analytical results by Holford and Hunt [5]. From the numerical results obtained, it was noted that by increasing the total effective opening area for a floor causes a rise in the interface height and volume flow rate, and a decrease in the reduced gravity of the stratified layer for that storey. The same observation was noted in the analytical study [5]. In order to obtain the same ventilation flow rate on each storey, the total effective opening area for each storey needs to be different. The higher storey requires larger opening area than the lower storey to compensate for the smaller stack effect.

### 4.2 Openings Sized to Give the Same Ventilation Flow Rate on Each Storey

In order to have the same ventilation flow rate on each storey, inlets openings were sized using the different total effective opening area ($A_t/H^2$) for each storey. Table-3 shows the results of volume flow rates ($m^3/s$) of the buoyancy-driven ventilation in the left side rooms of the building with different total effective opening area $A_t$ on each floor. It is seen that for each storey the ventilation flow rates are almost equal. The velocity and temperature contours inside the building to have same ventilation rate on each storey are shown in Fig.3. Interfaces separating the warm upper layer from the cool ambient layer below are clearly visible. As the heat source strengths are the same on each floor, the air temperatures of the upper layers on each floor should be same. This was predicted successfully by the CFD model (see Fig.3).
Table-3 Volume flow rates (m$^3$/s) of the buoyancy-driven ventilation with different total effective opening area $A_{t}$ on each floor of the left side rooms of the building.

<table>
<thead>
<tr>
<th>Floors</th>
<th>Total effective opening area($A_t/H^2$)</th>
<th>Volume flow rate (m$^3$/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ground floor</td>
<td>0.0170</td>
<td>0.42</td>
</tr>
<tr>
<td>First floor</td>
<td>0.0235</td>
<td>0.42</td>
</tr>
<tr>
<td>Second floor</td>
<td>0.0380</td>
<td>0.40</td>
</tr>
</tbody>
</table>

Fig. 3 CFD predictions of velocity (a) and temperature contours (b) in the building with the same ventilation flow rate on each storey.

The quantitative results of the CFD simulations in the non-dimensional form were compared with the design curve [5] and are shown in Fig. 4. A close agreement was found between the analytical and CFD predictions.

Fig. 4 Comparison of the analytical [5] and the CFD predictions for the non-dimensional interface height (a), volume flow rate (b) and reduced gravity (c) simulations where the openings have been sized to give the same ventilation flow rate on each storey.
4.3 Conclusions

In the work presented, the use of buoyancy-driven natural ventilation in a simple three-storied building was investigated numerically. The building was designed based on the concepts of the analytical models developed by Holford and Hunt [5]. The steady-state CFD simulations of buoyancy-driven natural ventilation air flows and temperature distributions in the building were undertaken using SST-k-\omega turbulence model with a DTRM radiation model. The numerical results obtained were compared with the design curves developed by Holford and Hunt and [6]. A favorable agreement was achieved between the analytical models calculations and CFD predictions of the key parameters, i.e. the interface height, non-dimensional volume flow rates and reduced gravities (buoyancy force) which successfully demonstrate the ability of a commercial CFD code to accurately predict three-dimensional buoyancy-driven displacement ventilation flows in a simple multi-storey spaces connected to a common atrium.

However, for the predictions of interface height quantitative discrepancies were observed. This is thought to be due to the assumption in the analytical model that the buoyant layer remains homogenous and hydrostatic for all interface heights which was not the case in the CFD simulation as the interface approached the ceiling. Mesh density may also contribute to the inaccurate interface prediction for large ventilation openings. When applying the CFD techniques to real buildings it is expected that methods applied would give accurate results for bulk air flow parameters such as stratification depth, buoyancy of the upper layer and ventilation flow rate provided that the effective opening areas are accurately represented.

ACKNOWLEDGEMENTS

This work was supported by natural Sciences and Engineering Research Council Canada (NSERC)

REFERENCES


[9.] S. Hussain, P.H. Oosthuizen, Validation of numerical modeling of conditions in an atrium space with a hybrid ventilation system, Building and Environment, 52(2012) 152-161.
