BEYOND ARROWS: CFD MODELING OF A NEW, NATURALLY VENTILATED, DOUBLE SKIN FACADE CONFIGURATION IN A CHICAGO HIGH-RISE OFFICE BUILDING

Mona Azarbajjani\textsuperscript{1}, Jim Anderson\textsuperscript{2}
\textsuperscript{1}Center for Integrated Building Design Research; School of Architecture; University of North Carolina at Charlotte
\textsuperscript{2}School of Architecture, University of Illinois at Urbana Champaign (UIUC)

ABSTRACT

There is a great deal of interest in using double skin facade (DSF) strategies in new and retrofitted buildings, as they provide many possibilities for energy conservation, and at the same time create better thermal comfort. For hundreds of years, architects have tended to rely on intuitive guesses to design naturally ventilated buildings without detailed analyses. The lack of numerical airflow information that demonstrates the complexity and challenges in the domain of designing large naturally ventilated buildings is addressed in the literature reviews.

In this paper, the energy performance of a high-rise office building equipped with conventional insulated glazing will be calculated by EnergyPlus and compared to a new DSF configuration. The DSF solution is innovative because it combines two common typologies: shaft and corridor type. In this research, CFD software (computational fluid dynamics) Fluent was used to evaluate various thermal comfort parameters for the new configurations. It helped to determine the possibility of improved indoor thermal comfort in a humid climate through natural ventilation strategies for a high-rise office building. The results proved that the new configuration had a major impact on enhancing natural ventilation and as a result, a reduction in energy usage.

Fluent was used to study the airflow and temperature distribution in the occupied spaces, evaluating different possibilities of exploiting natural ventilation for different outside conditions.

For natural ventilation analyses, simulation programs measured the airflow across an opening by determining the pressure differential that exists across that opening. One component is the pressure inside the building, which mostly depends on temperature. Another component is the wind pressure on the opening.

In this study two driving forces—wind and stack effect, i.e., buoyancy forces—are investigated to study the possibility of providing building.

DOUBLE SKIN FACADE

The concept of a DSF is not new and dates back to many years ago in central Europe when houses utilized box-type windows to increase thermal insulation (Oesterle, 2000). The DSF is an architectural phenomenon driven by the aesthetic desire for an all-glass facade and the practical desire to have natural ventilation for improved indoor air quality. Until recently the use of DSFs had become more popular in many European high-rise buildings.

A number of studies, research, and simulation programs have been done on incorporating natural ventilation in buildings and DSFs in thermal performance. Most have been carried out for solar chimneys—one way to increment natural ventilation and to improve indoor air quality—and Trombe walls prior to DSFs. Most designers found out that natural ventilation is possible in summer, even in multistory buildings (Wong, 2006). The potential of using a DSF for natural building ventilation in climates other than Europe has not, however, been fully studied.

In this study, wind-driven ventilation improved with stack effect in the novel DSF configuration and will be tested to see if it can maintain adequate comfort during summer and spring. The first step would be to study the ambience that will be used as CFD boundary conditions. Initial studies of the macroclimate were carried out.
through Ecotect, which allowed for efficient visualization of the local climatic conditions.

CLIMATE
The simulations were performed using the Ecotect-weather tool with EnergyPlus meteorological data measured at Midway airport. The building under the study assumed to be located in town, so it is necessary to allow for the effects of the city terrain. Figure 1 shows the ambient temperature during the year, which typically are either in the comfort zone or below the comfort zone.

![Figure 1. Ambient dry bulb temperature for Chicago.](image)

Since the office building is internally load dominated, it needed to be cooled most of the time during these periods. Therefore, this study is going to test whether employing natural ventilation in a generic open office building can achieve thermal comfort.

METHODOLOGY
The final goal of the research is to look into the possibility of natural ventilation in a high-rise office building in a humid continental climate (Chicago) using a new DSF configuration. To achieve that goal, intuitive diagrams of natural ventilation are drawn (Figure 2).

The office module equipped with the new configuration is constructed in 3-D Gambit Fluent with geometrical dimensions of 27x7 m, and 3.5 m ceiling height. The external screen model has openings on panes with 6- mm thick glass. In the new configuration, the DSF has a ventilated-shaft, 1.5x1.5 m depth, and 7-story height, with two openings on the low and high levels of the chimney. This study introduced a shaft to improve the natural ventilation’s stack effect to extract heat from offices and improve airflow rates required to reach thermal comfort level within the interior office space. In addition, an energy performance of this model was compared with the two typical DSF types of and an average office building single skin facade system as a base case.

CONFIGURATION
The new configuration takes advantage of strategies such as ventilation driven by different combinations of wind and external stack. The most distinguishing feature is its cooling stack towering over the building’s south side. This configuration combined both shaft and corridor types through the building’s facade. The cooling stacks allow for further ventilation on hot, stagnant summer days so the building can remain cool within reasonable comfort levels.

![Figure 2. Intuitive diagrams: Natural ventilation and DSF (section of the new configuration).](image)
Figure 3. Vertical cross section of the new DSF configuration.

The effectiveness of ventilation driven by thermal buoyancy, or stack effect, is determined by the inlet air temperature, height between the inlet and outlet openings, and the size of these openings. Figure 3 shows the section of office modules and configuration of inlet and exhaust openings.

AIRFLOW MODEL DESCRIPTION

The first stage of the internal CFD modeling is to construct the office module with geometrical dimensions of 27x7 m, 3.5 m ceiling height, and 1.5 m cavity corridor in front of the offices. The single skin external model facade has openings on panes with 6-mm thick glass. The DSF has one opening (inlet) at the outer pane and two openings (air inlet and exhaust) at the inner pane. The shaft is duplicated every 7 m along the facade and connected to exhaust air coming from offices. The model is constructed in 3-D in Gambit, as shown in Figure 4, and boundary conditions illustrated in Figure 5. To verify the final design for the ventilated facade, Fluent was implemented.

Figure 4. 3-dimensional grid model of the new DSF configuration constructed in Gambit.

Figure 5. Boundary conditions of the CFD model.

The boundary conditions for all stages of the modeling are:
- Simulations run for 2 periods of time morning (10 a.m.) and afternoon (2 p.m.)

Average hourly Dry bulb temperature C

<table>
<thead>
<tr>
<th>Month/Time</th>
<th>May</th>
<th>June</th>
<th>Oct.</th>
</tr>
</thead>
<tbody>
<tr>
<td>10:01-11:00</td>
<td>22</td>
<td>25</td>
<td>18</td>
</tr>
<tr>
<td>14:00-15:00</td>
<td>25</td>
<td>27</td>
<td>20</td>
</tr>
</tbody>
</table>
Average hourly relative humidity percentage:

<table>
<thead>
<tr>
<th>Month/Time</th>
<th>May</th>
<th>June</th>
<th>October</th>
</tr>
</thead>
<tbody>
<tr>
<td>10:01-11:00</td>
<td>77</td>
<td>79</td>
<td>81</td>
</tr>
<tr>
<td>14:00-15:00</td>
<td>56</td>
<td>57</td>
<td>58</td>
</tr>
</tbody>
</table>

Daily average wind velocity m/s:

<table>
<thead>
<tr>
<th>Month</th>
<th>May</th>
<th>June</th>
<th>October</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average m/s</td>
<td>5.6</td>
<td>4.9</td>
<td>5.2</td>
</tr>
</tbody>
</table>

- Wind direction perpendicular to the wall system
- DSF opening size for inner pane 0.3 m
- Cavity Depth 1.5 m
- DSF opening size for outlet 0.3 m

<table>
<thead>
<tr>
<th>Domain material</th>
<th>Air outdoor temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference pressure</td>
<td>100,000 Pa</td>
</tr>
<tr>
<td>Gravitational acceleration</td>
<td>-9.81 in Y direction</td>
</tr>
<tr>
<td>Heat source external</td>
<td>11.43 W/m²</td>
</tr>
<tr>
<td>Heat source internal</td>
<td>7.93 W/m²</td>
</tr>
<tr>
<td>Velocity inlet</td>
<td>5 m/s</td>
</tr>
<tr>
<td>Buoyancy model</td>
<td>Boussinesq</td>
</tr>
<tr>
<td>Buoyancy reference temp</td>
<td>Outdoor temperature</td>
</tr>
</tbody>
</table>

*Table 1. Boundary conditions*

**ASSUMPTIONS**

The following assumptions were made for the preliminary feasibility analysis.

The effects of site condition on wind, such as adjacent buildings, walls, and vegetation were not considered in the study, yet they would greatly affect the airflow through a building.

Winds are perpendicular to the surface and exert maximum pressure. However, in the site under study, there is also a fairly large range of wind directions, resulting in better indoor ventilation as they generate greater turbulence effect.

For comfort ventilation, openings should be at a level close to the floor, and the outlet vent in order to collect hot air should be located near the ceiling. Generally the inlet and outlet size should be the same.

The simulations are performed under a steady state condition using a k-epsilon turbulent model. Simulated wind speeds are used to model expected wind velocities at the levels under study with corresponding ambient temperatures and relative humidity shown in the tables above.

**Wind effects on a high rise building**

In assessing the wind effects on buildings, it is important to consider the wind’s characteristic nature. Wind is turbulent, and this means speeds vary with height. Vertical profiles of mean wind speed for boundary layers are approximated by taking the wind speed to be proportional to the height raised to some power—a power law variation (Davenport, 1965). The simple expression, which is used extensively has the form:

\[ V_h = 0.35 V_{net} h^{0.25} \]

Where \( V_h \) is the local wind speed at height, \( h \), and \( V_{net} \) is the meteorological wind speed. Based on this formula, wind has been calculated for specific times that CFD analysis was performed.

The wind speed from the meteorological data corresponds to the wind speed at 10 m height in open country. Since the building is located in an urban environment, the appropriate wind profile was assumed to be urban (0.35 and 0.25 are the constants which depends on the train in the vicinity of the buildings).

**Wind pressures**

The distribution of wind pressure around a building depends very closely upon the local variation in wind velocity that the building produces. In accordance with the elementary pressure-velocity relationship, the pressure distribution is represented by a dimensionless pressure coefficient \( C_p \):

\[ P_w = C_p \frac{\rho V_r^2}{2} \]

Where,

\[ P_w = \text{wind pressure, Pa} \]
\[ \rho = \text{air density, kg/m}^3 \]
\[ V_r = \text{wind speed at specified height, m/s} \]

This formula has been used to calculate boundary conditions for the inlet and outlet pressure in Fluent.

**CFD**

Ventilation rates are calculated by solving a network consisting of nodes connected by flow elements that correspond to openings between spaces, and between spaces and the outside. Buoyancy driven flows are predicted using space temperature calculated with 1000 iterations between airflow and thermal calculations. The flow is produced by a combination of wind and stack effects in the shaft.

Through analysis, it was discovered that thermal buoyancy in the shafts was great enough to induce a stack effect between the cavity and external space, resulting in warmed return air extracted from the space from the top of the shaft façade. An additional modification was to incorporate a light shelf into the lower side of the takeout slot, which would serve to direct flow into the shaft and thus the heat extracted from the room would not be mixed with the intake air.
DEMONSTRATION OF SIMULATION RESULTS
The base case with the new DSF configuration has been generated in CFD and airflow patterns; temperature profiles within the DSF have been illustrated for certain times of the year. Based on those data, the level of thermal comfort within the space will be determined. To study the thermal comfort, the specific temperature and velocity in the internal space at a certain height (1.2 m) has been analyzed. The air velocity through the cavity is due to buoyancy and wind forces, and is quite high. Figure 6 shows the model of air velocity in the building. The velocity in this model ranges from 7 m/s inlet to 2.3 m/s outlet through the chimney. The inflow from the external screen is 1 m/s and the internal screen inflow velocity is 0.45 m/s while the exhaust airflow is 0.46 m/s on average. The exhaust air velocity to chimney on average is 1.3 m/s. The greatest velocity is near the inlet to the chimney and exhaust from the stack. As shown in the figure below, velocities are increased relative to the one-story-high cavity due to higher buoyancy.

Figure 6. Velocity vectors colored by velocity magnitude (m/s).

Figure 8. Velocity vectors in the exhaust shaft.

Figure 9. Velocity vectors in the room.

Figure 10. Velocity gradients at the level of 1.2 m from the floor on the second floor.

Figure 7. Velocity vectors in the shaft.

Figure 9 shows an air-movement trend from laminar close to wall boundaries to turbulent in the room’s center. Airflow inside the rooms is less than 1 m/s and more than 0.1 m/s.

Figure 10 shows the section of airflow at the horizontal opening. Velocities are greatest near the opening, when the air is forced through a smaller area. In the back of the room, the air velocity is high as the air moves toward the exhaust to get out from the stack. As illustrated, the air velocity in the chimney is higher close to the back wall and it is generally laminar when only driven by buoyancy without wind effects.
Figure 11 shows the cavity air temperature. The temperature goes from 18 °C at the bottom opening to about 19.5 °C at the top opening in the cavity. The stack air temperature increases towards the top of the chimney in a fairly linear progression, as shown qualitatively in Figure 11 below. All of these figures depict a cavity model with 20 °C outdoor air temperature and 480 watts/m²-incident solar energy obtained from weather data illustrated earlier in this chapter. The interior air temperature is slightly higher than the lower half of the chimney on an average of 1 °C per floor.

![Temperature profile](image)

**Figure 11. Temperature profile.**

**RESULTS**

The results for a south-facing new type of DSF with external wind velocity of 5 m/s and air humidity of 50%, respectively, are tabulated in Table 2. Results as shown in Figure 12 indicate that the DSF air gap size of 0.3 m gives a comfort result for particular conditions in a natural ventilated building.

The lower floor of the office space would generate the lowest operative temperature due to the stack effect provided by the DSF configuration. This has enhanced the natural ventilation strategy to provide better internal thermal comfort conditions for the office spaces.

<table>
<thead>
<tr>
<th>Floor Level</th>
<th>Temp. C</th>
<th>Air Velocity</th>
<th>Radia at Temp</th>
<th>RH %</th>
<th>PMV</th>
<th>OT</th>
</tr>
</thead>
<tbody>
<tr>
<td>May</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>20</td>
<td>0.58</td>
<td>22.93</td>
<td>40.7</td>
<td>0.18</td>
<td>20.2</td>
</tr>
<tr>
<td>5</td>
<td>22.4</td>
<td>0.16</td>
<td>22.53</td>
<td>47.3</td>
<td>0.29</td>
<td>20.3</td>
</tr>
<tr>
<td>7</td>
<td>24.8</td>
<td>0.11</td>
<td>33.64</td>
<td>45.6</td>
<td>0.37</td>
<td>20.5</td>
</tr>
<tr>
<td>June</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>23.7</td>
<td>0.50</td>
<td>27</td>
<td>51.1</td>
<td>0.5</td>
<td>20</td>
</tr>
<tr>
<td>5</td>
<td>24</td>
<td>0.19</td>
<td>37.50</td>
<td>50.7</td>
<td>0.66</td>
<td>21.3</td>
</tr>
<tr>
<td>7</td>
<td>24.6</td>
<td>0.15</td>
<td>27.19</td>
<td>56.6</td>
<td>0.76</td>
<td>21.1</td>
</tr>
<tr>
<td>Oct</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>22</td>
<td>0.57</td>
<td>23.3</td>
<td>36.4</td>
<td>-0.32</td>
<td>22</td>
</tr>
<tr>
<td>5</td>
<td>22.3</td>
<td>0.20</td>
<td>23.4</td>
<td>42.4</td>
<td>0.11</td>
<td>22.8</td>
</tr>
<tr>
<td>7</td>
<td>22.8</td>
<td>0.19</td>
<td>24.6</td>
<td>45.4</td>
<td>0.23</td>
<td>22.0</td>
</tr>
</tbody>
</table>

There is an internal temperature difference of 1 °C for the DSF mid-floor which may be due to slower internal air velocity generated. The south-facing DSF configuration has produced an 82 percent acceptability limit for the 0.3 m opening for external temperature 23 °C, according to the Thermal Environmental Conditions for Human Occupancy from ANSI/ASHRAE Standard 55-2004.

Figures 7 through 12 illustrate detailed indoor temperature, velocity magnitude, velocity vector, and base on a calculated PMV index. From this index, it can be seen that indoor thermal comfort is non-uniform and the area with higher air velocity provides better indoor thermal comfort than the stagnant spaces.

**Figure 12. Thermal comfort evaluation.**

**ENERGY COMPARISON**

The annual energy usage per square feet area of the new DSF type, which is a combination of two typical DSF types, has been tabulated and illustrated below. The energy intensity of the new type compared with shaft and corridor types as well as an average office building in Chicago as a base case model is also shown.

![Energy comparison](image)

**Figure 13. Simulation results for different alternatives.**

It was discovered that the heating energy intensity was reduced by 50 percent in the new type from the base
case, and there was a 28 percent reduction in cooling energy intensity.

In total, compared with the base case (an average office building in Chicago), the corridor type reduced energy usage by 12 percent, shaft by 11 percent and the new type by 29 percent, respectively.

<table>
<thead>
<tr>
<th>CASE</th>
<th>Energy Intensity Kbtu/SF yr</th>
<th>Heating Consumption Kbtu/SF yr</th>
<th>Heating Reduction %</th>
<th>Cooling Consumption Kbtu/SF yr</th>
<th>Cooling Reduction %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference building</td>
<td>86.87977</td>
<td>45.5</td>
<td>16.8</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Corridor type</td>
<td>75.86</td>
<td>34.5</td>
<td>24</td>
<td>15.6</td>
<td>7</td>
</tr>
<tr>
<td>Shaft type</td>
<td>76.6095</td>
<td>33.8</td>
<td>25</td>
<td>16.0</td>
<td>5</td>
</tr>
<tr>
<td>New Configuration</td>
<td>61.07612</td>
<td>22.7</td>
<td>50</td>
<td>12.0</td>
<td>28</td>
</tr>
</tbody>
</table>

Table 3. Energy usage comparisons for different alternatives.

CONCLUSION

A proposed DSF configuration was studied to evaluate thermal comfort of occupants in high-rise buildings. The office spaces were assumed to be open usages. With the completion of the simulation stages, numerous runs had been carried out with such variables as various ambient temperatures, different external air velocities, different periods during the day, etc., in order to find out if the DSF could provide sufficient comfort while reducing energy usage. The CFD results appear to confirm the design's effectiveness in two ways. First, the airflow follows the ceiling and exits through the chimney openings, which was enhanced by buoyancy effects. The second outcome of the airflow modeling was to understand the actual airflow behavior in this design. Accurately solving for the airflow rate and patterns within the cavity requires the use of CFD. The main driving forces in the cavity are buoyancy and wind pressure. This research has shown that this new DSF configuration has the possibility of providing acceptable internal thermal comfort through natural ventilation strategy in a humid climate. These findings will be of utmost important in determining whether a DSF is a real possibility in incorporating natural ventilation and reducing energy usage in both heating and cooling in a humid climate.

REFERENCES


