International Journal of Structural Engineering and Analysis
Vol. 1: Issue 1
www.journalspub.com

Novel Approach to Improve Stack Effect in High Rise Buildings

Kevin Jones*, Helen Santhi, C.P. Karthikeyan
School of Mechanical and Building Sciences, VIT Chennai, India

Abstract
This paper aims at improving stack effect in high rise buildings using an innovative approach. Parameters influencing stack effect in high rise buildings are identified based on standard theory and practice. A new draft number for estimating stack effect in high rise buildings is derived using Buckingham-Π theorem. Airflow data of a university high rise building is considered in estimating stack effect. The draft number developed is enclosed in ANSYS Fluent for estimating stack effect in buildings. A 15 storey hostel block of the university is taken into consideration for the study. Simple experiments and equations from Chapter 16 of the 2009 ASHRAE Handbook of Fundamentals are used for validating the draft number formulation. The case considered in this study, had a narrow space in the interior of the building on account of which sun rays could not heat the air inside the building. For most time of the day, the air remained at a very low temperature reducing the upward movement of air due to natural draft. The air movement was found to be influenced by wind flow direction and angle with respect to the building.

Keywords: *Stack effect, computational fluid dynamics, Buckingham-n theorem, finite volume method, high rise buildings, ANSYS Fluent*

**__Author for Correspondence__: Email ID: kevin.jonesj2014@vit.ac.in*

INTRODUCTION

High Rise Building

The term high rise building doesn't have internationally agreed definitions. In most countries, the most prominent tall buildings are referred to as high rise buildings. In Britain and some European countries, tower blocks are referred to as high rise buildings [1]. Depending on the distance between floors, it has been referred to about seven to ten stories. In feet measurements, this has been set variously between 75–100 feet. Electricity charges and comfort levels are key factors of focus in a high rise building. The airflow in and around the building decides the comfort level of occupants inside a high rise building[2].

The airflow in and around a building will affect the comfort level of occupants with respect to temperature, the transfer rate of heat in and out of the building and also plays a vital role in determining whether a good quality air exists inside the building or not.

Stack Effect

The phenomenon that causes hot combustion gases to rise in a chimney (or) chimney stack is called stack effect. The house or building that we live and work inside is considered as a giant chimney. The stack effect is often considered proportional to the height of the building. With respect to temperature, colder the air temperature, higher the stack effect. Cold air tends to move down on account of higher density and lighter hot air tends to move up[3]. Stack system is a remarkable energy efficient system. It was observed that the active stack provides substantial increase in the internal air velocities[4]. Designing for natural ventilation in high rise buildings is complicated by the interaction between the flow rates and the indoor air temperatures. A good design should account for all important parameters that influence both the wind induced and stack induced pressures[5].

Factors Affecting Stack Effect

There are two types of physical phenomena which induce natural airflow in the buildings- one by wind effect and the other by stack effect. They can occur simultaneously but the effect of one of them generally predominates over the other. If there is wind, the wind effect is usually more important than the stack effect but the interaction between these two phenomena can be more complicated[6]. During a simulation study by Maatouk *et al*., four parameters have been considered. Some of the parameters affecting stack effect are air-tightness of the building, temperature of the stairwell, wind speed velocity and the wind direction.

The total infiltration increases with the increase in wind speed and the air-tightness[7]. Nyuk *et al*., in their CFD simulations considered five design factors, which include stack size, stack location, door operations, and external wind effect and stack fan operations[8]. Another research, considers the inside and outside temperatures, effective cross sectional areas and height of the building.

Computed ventilation rates under various fixed conditions depend mainly on the opening cross section, but also strongly on the location of the openings and their geometry[9].The factor that triggers stack effect and its magnitude, i.e., pressure gradient between the outdoor environment and indoor spaces, is determined by the height of the building and temperature differences between the outdoor environment and indoor spaces[10]. Another research says that the stack effect can create significant pressure differences and stairwells, shafts, elevators, interior partitions, floors, and fire separations can have significant effect in stack effect[11]. Building shape, building envelope design, internal room design, local landscaping and site location are the important factors that influence the natural air movement in and around the building. Building arrangement (shape, spacing and orientation) floor planning (partition, window size and location etc.) and local weather conditions (wind speed and direction) are the major factors that contribute to a good building design[12].

It was also taken into account that the velocity increase along with the increase in stack size. Studies have shown that the increase in air velocities, especially at higher temperature enhances the thermal comfort condition. To improve the airflow in a building a draft can be provided to serve the purpose. A draft is a hollow space provided in the building which is open at top. A temperature difference between the outside and inside air will create a "natural draft" forcing air to flow through the building.

Temperature highly influences the direction of flow. The air movement in the hollow space of a building depends on the inside air temperature, outside air temperature, inside air density and outside air density. For cold air to flow into the building, two conditions are necessary- inside air temperature should be higher than outside air temperature and inside air density should be less than outside air density.

Field measurements and the simulation prediction values were compared and the reliability of the simulation program was studied by Taesub *et al*. Stack effect can be calculated from the following relation[12].

$$\Delta Ps = Ps(o) - Ps(i) = (\rho o - \rho i)\, g\, (h_{NPL} - h) = \rho o \left(\frac{Ti - To}{Ti}\right) g\, (h_{NPL} - h) \qquad \text{Eq. (1)}$$

A section exists where the air pressures outside and inside the building become the same, which is called a neutral pressure level (NPL). The pressure difference Ps is the difference between outside pressure Ps(o) and inside pressure Ps(i) for any random height h. Stack effect can be calculated from the following relation. Also, the rate at which air flows depends on several factors which include the inside and outside air temperatures, the area of the openings, and the height difference between the top and bottom openings. The draft or draught flow rate induced by the stack effect can be calculated with equation.

$$Q = \alpha A \sqrt{\frac{2\, g\, \Delta P}{\rho}} = \alpha A \sqrt{2\, g\, h\, \frac{Ti - To}{Ti}} \qquad \text{Eq. (2)}$$

Research by Mathews *et al*. helps to estimate the pressure difference due to stack effect. The simulations were carried out with all the doors, windows and ventilations closed[5].

$$\Delta P = 3460 \left(\frac{1}{To} - \frac{1}{Ti}\right) h \qquad \text{Eq. (3)}$$

They also suggest that mechanical ventilation is required for better draft. For single-sided ventilation with a single large opening the flow rate caused by buoyancy can be expressed through the following equation[7].

$$Qv = \frac{1}{3} C_d A_{eff} \sqrt{\left|\frac{(Ti - To) \,.\, g \,.\, he}{Ti}\right|} \qquad \text{Eq. (4)}$$

A_{eff} is the effective cross-sectional opening area of the single opening (m^2) and h_e effective opening height of the free area of the opening (m). Analytical equations for air flow rates are very consistent with simple steady state airflow network simulations. Therefore, in the architectural design phase of natural ventilation, potentials of different opening configurations may be estimated by the above mentioned equations.

METHODOLOGY

Ventilation by Wind and Stack with Open Area

For the model, the ventilation air flow rate is a function of wind speed and thermal stack effect, along with the area of the opening being modeled. This object can be used alone or in combination with Zone Ventilation: Design Flow Rate objects. This model is intended for simplified ventilation calculations as opposed to the more detailed ventilation investigations that can be performed with the Airflow Network model. A "Wind and Stack with Open Area" model was used and the air flow rate can be controlled by a multiplier fraction schedule applied to the user-defined opening area and through the specification of minimum, maximum and delta temperatures. A single constant value or a variable which varies with time is considered as temperature. The ventilation wind rate is derived from the equation given by Eq. (37) in Chapter 16 of the 2009 ASHRAE Handbook of Fundamentals[13]

$$Q_w = C_w A F_{schedule} V \qquad \text{Eq. (5)}$$

where, Q_w is volumetric air flow rate in m^3/s, A is the opening area in m^2, $F_{schedule}$ is the open area fraction and it is dimensionless, C_w is the opening effectiveness which is also dimensionless and V is the velocity of wind in m/s.

If the user specifies "auto-calculate" for the opening effectiveness input field, the opening effectiveness is calculated for each simulation time step based on the angle between the actual wind direction and the effective angle (a user-defined input) using the following equation:

$$C_w = 0.55 - \frac{(Effective\ angle - Wind\ direction)}{180} * 0.25 \qquad \text{Eq. (6)}$$

The difference (Effective Angle – Wind Direction) should be between 0 and 180 degrees. If the difference is greater than 180, the difference is reset to be –180 degrees. This equation is a linear interpolation using the values recommended by 2009 ASHRAE Handbook of Fundamentals (page 16.13): 0.5 to 0.6 for perpendicular winds and 0.25 to 0.35 for diagonal winds. The equation used for calculating the ventilation rate due to stack effect is given below.

$$Q_s = C_D A_{opening} F_{schedule} \sqrt{2g\Delta H_{NPL}((T_{zone} - T_{odb})/T_{zone}} \quad \text{Eq. (7)}$$

where, Q_s is the Volumetric air flow rate due to stack effect(m^3/s), C_D is the discharge coefficient for opening (dimensionless), $A_{opening}$ is the opening area (m^2), $F_{schedule}$ is the open area fraction (user-defined schedule value, dimensionless), ΔH_{NPL} is the height from midpoint of lower opening to the neutral pressure level (m). T_{zone} is the zone air dry-bulb temperature (K) and T_{odb} is the local outdoor air dry-bulb temperature (K).The discharge coefficient for an opening can be calculated using the following equation, which is given as:

$$C_D = 0.40 + 0.0045(T_{zone} - T_{odb}) \quad \text{Eq. (8)}$$

The total ventilation rate calculated by this model is the quadrature sum of the wind and stack air flow components:

$$Ventilation_{Wind/stack} = \sqrt{Q_s^2 + Q_w^2} \quad \text{Eq. (9)}$$

Draft Number

Key factors influencing airflow inside a high rise building are listed as follows. Length, breadth and height of building, density of air inside the building, pressure and velocity of air flow around the building.

Using these parameters, a dimensional analysis is done using Buckingham-Π method. The draft number "N" as a function of these parameters is given by Eq. (10).

$N = \phi\ (L, B, H, V, \rho, P)$

Where N is the draft number, L is the length of building, B is the breadth of building, H is the height of building, V is the velocity of air flow, ρ is the density of air, P is the pressure difference.

n = Number of parameters = 7
k = Number of primary dimensions = 3
(In this case, M, L and T)
Number of P terms = n – k = 7 – 3 = 4

$\phi\ (P_1, P_2, P_3 \text{ and } P_4) = 0$

Since L, B, H has same dimensions they can't be in the same P group.

$P_1 = L\ V^a\ \rho^b\ P^c$
$P_2 = B\ V^a\ \rho^b\ P^c$
$P_3 = H\ V^a\ \rho^b\ P^c$ and
$P_4 = N\ V^a\ \rho^b\ P^c$

In terms of primary dimensions,
$M^0L^0T^0 = (L)\ (LT^{-1})^a\ (ML^{-3})^b\ (MLT^{-2})^c$

Equating the powers we get;
$b + c = 0;\ a + c - 3b + 1 = 0;\ a + 2c = 0$

Solving these equations, we get;
$a=1;\ b= 0.5;\ c= -0.5$

Substituting these values in the term P, we get;
$P_1 = L*V*\sqrt{\rho}/P$

Similarly, $P_2 = B * V * \sqrt{\rho} / P$
$P_3 = H * V * \sqrt{\rho} / P$
$P_4 = N * V * \sqrt{\rho} / P$

Equating these Π terms we get the final equation as–

$$N = (L * B * H * V^2 * \rho)/P \quad \text{Eq. (10)}$$

Computational Fluid Dynamics (CFD)

Design optimization studies in residential and high rise buildings have been extensively carried out to optimize energy and comfort levels. A design strategy for optimizing natural ventilation in high rise buildings in China using CFD modeling is reported by Zhou *et al.*[14]. Building orientation and spacing were adjusted, wind paths were introduced, and alternations in the position of windows were done.

The optimized design reduced the age of air to less than 6 minutes in more than 90% of rooms which was only 30 minutes in 50% rooms in conventional design. Combined methodology of wind tunnel experiments and CFD simulations were used to study the potential of using active stack to enhance natural ventilation in residential apartments[15].

The active stack strategy was considered with five design factors. The design factors considered include stack size, stack location, door operations (opened or closed), wind tunnel operations (on or off) and extract fan operations (on or off). Using active stack strategy, the researchers could significantly increase the average air velocity within the residential unit by 47%.

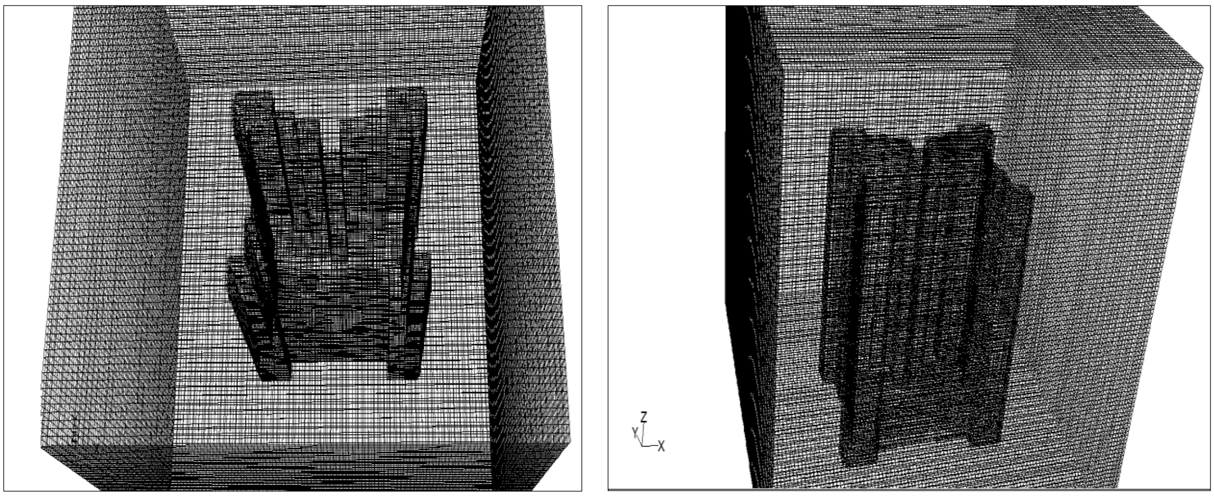
These studies ensure the use of active stack strategies in residential/high rise buildings for balancing energy consumption and comfort level. This study is different in its approach by using dimensional analysis coupled with CFD for predicting stack effect in interiors of high rise buildings.

Airflow modeling based on computational fluid dynamics solves the fundamental conservation equation for mass, momentum and energy in the form of Navier-Stokes equation, the general form of which is–

$$\frac{\partial(\rho\emptyset)}{\partial t} + div(\rho V\emptyset - diff_{\emptyset} grad\emptyset) = S_{\emptyset} \quad \text{Eq. (11)}$$

The first term of Eq. (11) represents the transient term of the dependent variable *Φ*. The diffusion and convection terms of *Φ* are grouped together in the second. The right hand side of the governing equation is often balanced by a source term.

The Navier-Stokes equations and the scalar equations for turbulence are solved by superimposing a grid of cells that describe the physical building geometry. The building geometry was discretized into a uniform grid of hexahedral cells.



(a) (b)

Fig. 1*: Meshed Model of the High Rise Building (a) Plan View and (b) Elevation.*

The dimensions of the building in meters are Length (40), Width (30) and Height (60). 3D-pressure based implicit unsteady solver was used to solve the basic governing equations (mass, momentum and energy). Standard *k*-*ε* model was used to simulate the turbulence around the building. Where, *k* represents turbulent kinetic energy (m^2/s^2) and *ε* represents the dissipation rate (m^2/s^3). SIMPLE (Semi-Implicit Method for Pressure Linked Equation) was used as the pressure velocity coupling scheme. Velocity of air entering the domain was taken as 60 m/s.

RESULTS AND DISCUSSION

Initially, pressure and temperature contours were plotted to understand the stack effect. The velocity readings were compared to that of the experiments conducted using a vane anemometer.

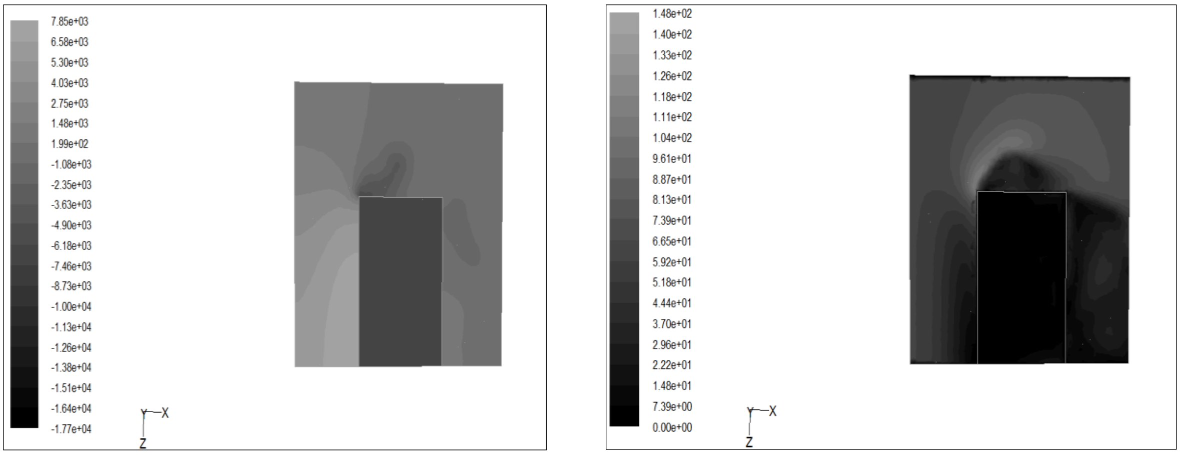


Fig. 2: *Pressure and Velocity Contours inside the Building Parallel to Wind Flow Direction.*

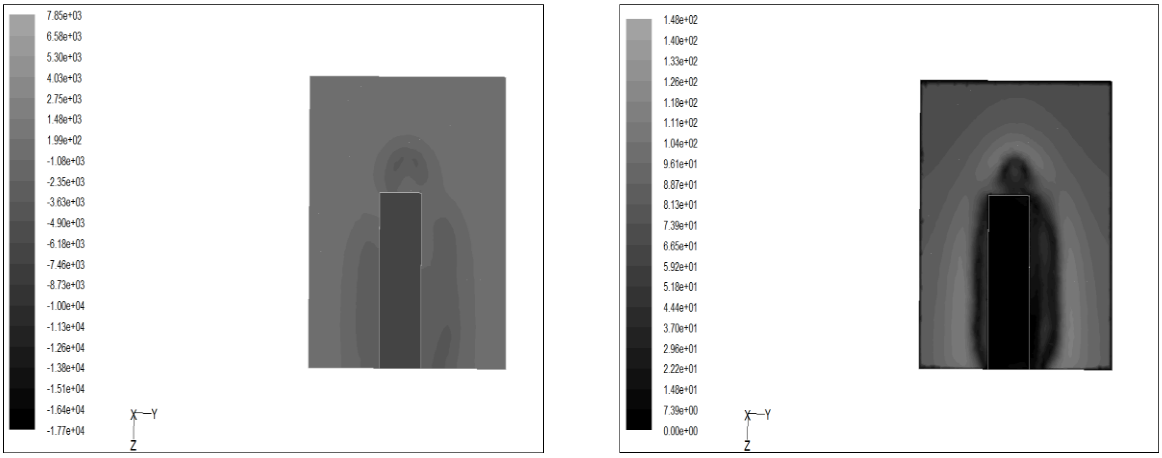


Fig. 3*: Pressure and Velocity Contours inside the Building Perpendicular to Wind Flow.*

Vertical lines were created in the building interior space (hollow space) and points were created on them. The points of measurements were made floor wise (1–15) to check the velocity and pressure in the vertical direction as shown in Figure 4.

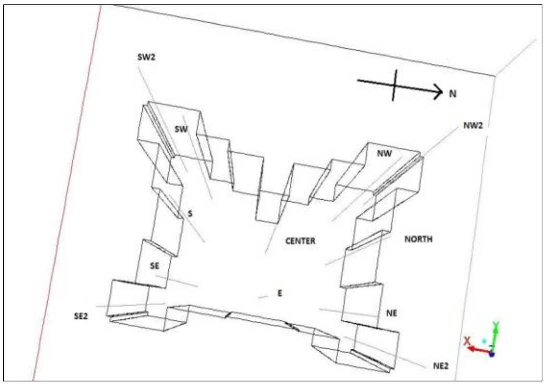


Fig. 4: *Points of Measurements for Velocity and Pressure along the Vertical Direction.*

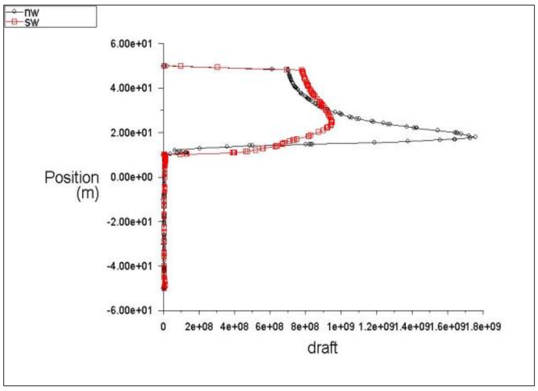


Fig. 5: *Draft Vs Position in North West and South West Direction.*

The simulation domain varies from –50 to +50 m (i.e., a total of 100 m). The height of the lines used for measurement is 100 m. The building measures 60 m high from the ground level.

Figure 5 shows the variation of draft using the number developed along two different lines of measurement (NW and SW represented in Figure 4). The trends obtained from the simulation predict high air movement above the building. The airflow at the top of the building enters at a height of 1.00 e+01 in the y-axis (60 m from the ground level); after which significant reduction in draft values are noticed on the x-axis.

The reductions in values signify poor draft in hollow space of the building for the reference lines mentioned. Same trend is observed for other lines of measurements and the values correspond to near zero values as shown in Figures 6, 7 and 8. Figure 6 shows the same trend observed along three lines of measurement: center, north and south.

The intensity of draft is less when compared to the previous results in Figure 5. The draft movement is significantly higher on one side of the hollow interior space namely NW and SW as depicted in Figure 5.

The reason being the air flow direction is from south to north in this study as shown in Figure 4. The movement is highly influenced by the air inlet direction and angle with respect to the building hollow space.

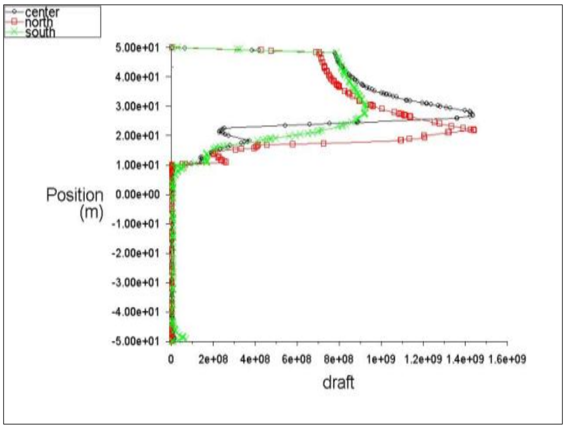


Fig. 6: *Draft Vs Position in North, Center and South.*

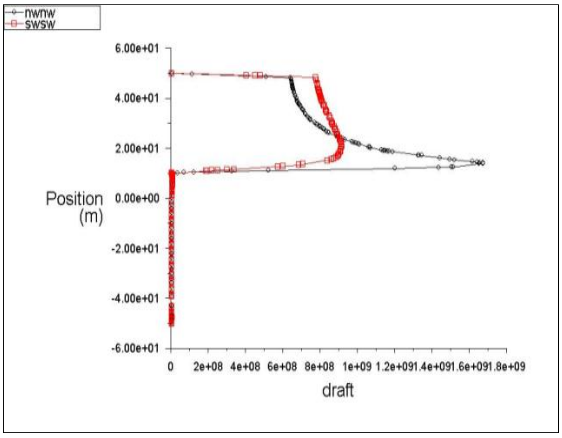


Fig. 7: *Draft Vs Position in Extreme North West and South West Direction.*

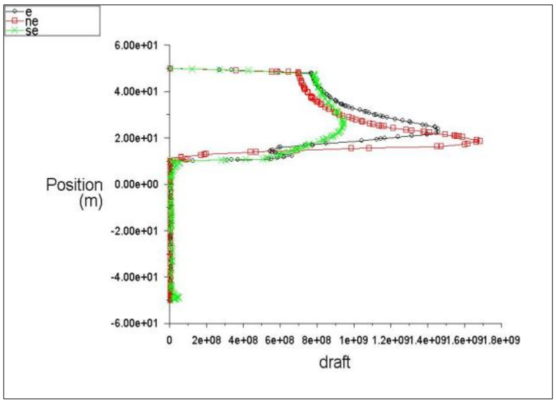


***Fig. 8**: Draft Vs Position in North East, East and South East Direction.*

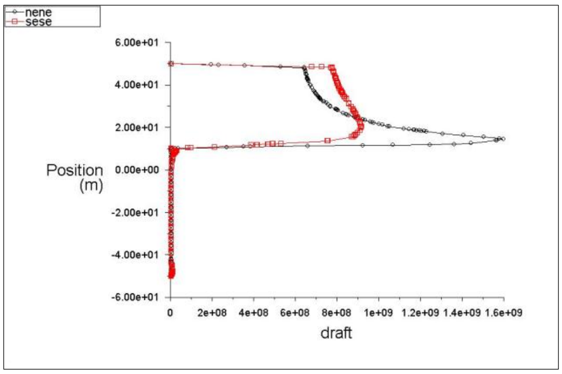


***Fig. 9:** Draft Vs Position in Extreme North East and South East Direction.*

Figures 8 and 9 shows the same trend where air movement is found to be higher along the length (NE, SE, SW, and NW) of the building when compared to width of the building. The width of the building correspond to directions north, center and south which are perpendicular to the direction of the wind. The dimensions of the length and width play a major role on the draft movement. In this study, the length (40 m) of the building is much longer than width (30 m). There is also a difference in velocity/air movement values along the two extreme opposite lengths of the building hollow space. This variation can be accounted for the angle of the building space with respect to the inlet air.

The variation of air movement along the height of the hollow space is much less when compared to the length which signifies poor draft. Combining both horizontal and vertical air movement, draft estimated in the hollow space of the building is poor. The results of the CFD simulation comply well with anemometer measurements.

CONCLUSIONS

The case considered in this study, had a narrow space in the interior of the building on account of which sun rays could not heat the air inside the building. For most time of the day, inside air remained at a very low temperature reducing the upward air movement due to natural draft. The CFD results predict high air velocity on top of the building and near-zero velocity in the open area. Aspect ratio is found to be a significant contributing factor to a better stack effect on the interiors of high rise buildings. The numerical results showed significant deviations with the equations for stack effect prescribed by ASHRAE standards. The reasons for the deviations can be contributed to the fact that the study used an incompressible approach with several assumptions.

To summarize the study in brief, a new draft number is developed to estimate stack effect in high rise buildings. Comparison of results from the CFD simulation has good agreement with experiments, though deviations were noticed in comparison with ASHRAE standards. The direction of airflow and angle with respect to the building are the parameters to be included in the draft number in future research.

REFERENCES

1. Challinger D. *From the ground up: security for tall buildings*. CRISP Report. Alexandria, VA: ASIS Foundation Research Council; 2008: 4.
2. Knoke M .E. *High-rise structures: life safety and security considerations*. In: Protection of Assets Manual. Alexandria, VA: ASIS International; 2006.

3. Joseph W. Lstiburek. How do buildings stack up and the hollow building. ASHARE *Journal Building Insights*. BSI-002.
4. Priyadarsini R., Cheong K.W., Wong N.H. Enhancement of natural ventilation in high-rise residential buildings using stack system. *Energy and Buildings*. 2004; 36: 61–71p.
5. Mathews E. H., Rousseau P.G. A new integrated design tool for naturally ventilated buildings part 1: ventilation model. *Building and Environment.* 1994; 29(4): 461–71p.
6. Gładyszewska-Fiedoruk K., Gajewski A. Effect of wind on stack ventilation performance. *Energy and Buildings*. 2012; 51: 242–7p.
7. Khoukhia M., Yoshinoa H., Liub J. The effect of the wind speed velocity on the stack pressure in medium-rise buildings in cold region of China. *Building and Environment*. 2007; 42: 1081–8p.
8. Wong N.H., Heryanto S. The study of active stack effect to enhance natural ventilation using wind tunnel and computational fluid dynamics (CFD) simulations. *Energy and Buildings*. 2004; 36: 668–78p.
9. Schulze T., Eicker U. Controlled natural ventilation for energy efficient buildings. *Energy and Buildings*. 2013; 56: 221–32p.
10. Lee J., Song D., Park D. A study on the development and application of the E/V shaft cooling system to reduce stack effect in high-rise buildings. *Building and Environment* 2010; 45: 311–9p.
11. Khoukhia M., Al-Maqbali A. Stack Pressure and Airflow Movement in High and Medium Rise buildings. *Energy Procedia*. 2011; 6: 422–31p.
12. Lim T., Cho J., Kim B.S. Predictions and measurements of the stack effect on indoor airborne virus transmission in a high-rise hospital building. *Building and Environment*. 2011; 46: 2413–24p.
13. *ASHRAE Handbook – Fundamentals*. Chapter 16-Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers Inc. 2009.
14. Zhou C., Chen Z.Q., Pei J. Design optimization and field demonstration of natural ventilation for high rise buildings. *Energy and Buildings*. 2014; 82: 457–65p.
15. Jo J.H., Lim J.H., Song S.Y., *et al*. Characteristics of pressure distribution and solution to the problems caused by stack effect in high-rise residential buildings. *Building and Environment*. 2007; 42(1): 263–77p.