

Research Article

A Study on Computational Analysis for Natural Convection in Tall Building – A Macroscopic Approach

Rupa Rajkumar ¹, Vasanthi Padmanabhan ², Nagaraj Meenaskshi Sundaram ³
and Umamaheswari Kandasamy ⁴

¹Misrimal Navajee Munoth Jain School of Architecture, Chennai, India

²Department of Civil Engineering, B.S. Abdur Rahman Crescent Institute of Science & Technology, Chennai, India

³Institute of Agriculture Engineering, Saveetha School of Engineering, Saveetha Institute of Medical and Technical Sciences, Chennai, India

⁴Kebridehar University, Kebridehar, Ethiopia

Correspondence should be addressed to Vasanthi Padmanabhan; svraj कुमार@ymail.com
and Umamaheswari Kandasamy; umasashwini@gmail.com

Received 22 February 2022; Revised 16 March 2022; Accepted 21 March 2022; Published 25 April 2022

Academic Editor: Samson Jerold Samuel Chelladurai

Copyright © 2022 Rupa Rajkumar et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

The development of convective heat transfer in buildings are found to be due to the movement of air with respect to the turbulence in air to the temperature. The heat transferred is observed in 3 levels - the roof exposed floor, the seventh floor and the lower floor of the 14 storey naturally ventilated tall building. This paper discusses the eulerian approach which is a macroscopic approach to understand the temperature in the building since the density stratification occurs in the indoor volume of the building. The volume averaged data is considered to be in eulerian approach. The CFD analysis is used to artificially create the environment with the different temperatures and velocities. Heat transfer happens due to convection, hence the boundary conditions - initial outdoor temperature has been kept constant at 30°C and 23°C. The change in the outdoor temperature at the air velocities 10 m/sec at 12 noon were observed as 29.37°C, 29.49°C, 29.89°C at constant temperature 30 with respect to 1st floor, 7th floor and roof. Finally, it is noted that, as the air velocity increases, the outdoor temperature also increases. Conjugate heat transfer analysis is followed in CFD analysis to include the resistance of the exterior concrete wall to transfer the temperature to the control volume. K-omega model is used to resolve the turbulence and to analyse the interior domain with Boussinesq hypothesis in the momentum equation to include the effect of the density difference in heat transfer. It is observed the velocity helps to take the temperature from the exterior domain to the interior part of the building. Hence the results clearly indicate that when the outdoor temperature is higher and the outdoor velocity is also higher, the temperature which enters the indoor environment is also high.

1. Introduction

Emphasizing sustainable living in the built environment has been the need of the hour in many countries which are facing global warming. Natural ventilation and passive methods to cool buildings and to minimize energy consumption has been the primarily goals of sustainable development. Migration of people to the urban areas due to rapid increase in jobs has resulted in the expansion of cities. This recent trend

has paved the way for new developments and buildings in the urban sprawl. Increase in various high-rise residential buildings coming up in peri-urban areas due to further urbanization and land scarcity in the urban areas have been resulted. Various changes in the environmental parameters such as air temperature, air velocity, wind direction, and subsequently the micro-climate is also noticed. Particularly in the residential sectors, this substantial change in the micro-climate has an impact on the thermal comfort with

an increase in the use of energy consumption [1–13] throughout the world.

India has a flexible climate divided into 5 climatic zones. Indian codes standards for thermal comfort [14] for air-conditioned buildings for all climatic zones are (23°C -26°C) in summer and (21°C to 23°C) in winter. 45% of the Indian cities are warm and humid [15] according to the Bureau of Energy Efficiency (BEE), India. The weather seasons in India has been divided as [16] winter (December, January, and February), Pre-monsoon/summer (March, April, and May), South-west monsoon/summer monsoon (June, July and August) and Post-monsoon/Northeast monsoon – September, October, and November according to Indian Meteorological Department (IMD).

Heat gain control method, microclimate, natural ventilation, induced ventilation, forced ventilation, evaporative cooling, night sky radiative cooling, phase changing material based time lag cooling and earth cooling are the techniques used to cool the building using low energy. The cooling potential was assessed for 5 Indian cities – Chennai, Bangalore, New Delhi, Jaipur, and Jammu & Kashmir through evaporative cooling, nocturnal radiative cooling and phase change material (PCM) based free cooling system [17]. Trees and semi-outdoor spaces are to be placed near open able envelopes in buildings oriented towards the prevailing wind direction [18]. Being site-specific the building designers and architects could evolve interesting energy-efficient concepts, selection of appropriate building materials and incorporating suitable passive cooling techniques such as minimize heat gain, dissipate internal heat and modulate the heat in buildings to reduce energy consumptions [19].

Wall to window ratio, window to floor ratio, building position, and orientation are the elements that are dependable for effective natural ventilation. Further architectural elements such as louvered windows, vertical shading devices, shape and form of aperture and use of vernacular materials can produce significant results [20]. Hybrid ventilation systems can be provided to reduce the feeling of inadequate natural ventilation, when the outdoor temperatures are very high in naturally ventilated high rise buildings during summer [21]. In the pre-design stage detailed CFD simulation with outdoor air velocity and indoor air velocity are essential for building design strategies for natural ventilation [22]. In practice CFD models when compared to other models were mainly used to examine the natural ventilation, indoor air quality and stratified ventilation in buildings [23]. In buildings with large window openings, when both the outdoor air quality and the thermal environment are tolerable mechanical heating or cooling is not required, however [24]. The building ventilation improved when the space between two buildings were five times the width of the building [25].

Computational Fluid Dynamics (CFD) models have numerous advantages over other methods as it provides specific data at every point on the computational domain, full scale simulations can be made, it can be used in parametric studies for accurate results [26–32]. To assess the airflow in a naturally ventilated building, the position of the inlet and outlet opening on the façade is important along with the wall porosity [33].

Strait Sen et al. [34] have analysed the impact of placing curved obstacles on natural convection in enclosures with differentially heated side walls is analyzed. The lattice Boltzmann method (LBM) was used. A standard natural convection problem with left wall at high temperature, right wall at low temperature, and top and bottom adiabatic walls was considered for validation purpose. Goodarzi et al. [35] stated that various numerical methods were developed to simulate natural convection internal transfer of heat. Based on lattice Boltzmann method was used to develop FORTAN code. The same analysis was done with commercially available CFD for verification purpose.

From the literature, it was found that, no adequate work was carried out related to natural convection heat transfer multi storey building. In this research work, a detailed study is carried out to analyse on computational analysis for natural convection in tall building. The temperature maximum and minimum were kept constant, inlet velocities were varied from 1 m/s, 2 m/s, 5 m/s, 8 m/s, 10 m/s. The pressure outlet was kept static with zero due to the atmospheric pressure.

2. Numerical Analysis

In fluid dynamics the movement of a group of particles, as in our environmental variables such as temperature, relative humidity and air velocity which are in fluid state are expressed in Lagrangian method or the Eulerian method. In the Lagrangian method, the coordinates of the moving particles- temperature, relative humidity and velocity are presented as functions of time. At a particular arbitrary time t_0 , the coordinates are (x_0, y_0, z_0) , identified and followed through the fluid. Hence the particles are denoted by a set of equation of the form.

Usually in fluid dynamics the lagrangian approach is not much used as the equations are large and unmanageable. Whereas in the Eulerian approach the velocities at various points and are expressed as functions of time as shown in Equation (1).

$$u = f_1(x, y, z, t), v = f_2(x, y, z, t) w = f_3(x, y, z, t) \quad (1)$$

where u , v , and w are the components of velocity in x , y , and z directions, respectively. The velocity change in the vicinity of a point in the x direction is given by Equation (2).

$$du = \frac{\partial u}{\partial t} dt + \frac{\partial u}{\partial x} dx + \frac{\partial u}{\partial y} dy + \frac{\partial u}{\partial z} dz \quad (2)$$

The small distances moved by a particle in time dt can be expressed in Equation (3).

$$dx = udt, \quad dy = vdt, \quad dz = wdt \quad (3)$$

Thus, dividing Eq. (1) by dt and using Eq. (2) we get the *substantial derivative* of the velocity u .

To obtain precise and accurate solutions at minimal cost in fluid flow, one of three basic methods: differential, experimental, and control volume can be utilized. In our CFD model control volume approach is used as it is simple yet



FIGURE 1: Plan showing Block-A, Block-B, Block-c, Block-D with swimming pool in the center.



FIGURE 2: (a) Map of Tamil Nadu (b) Location map of the building considered in the present study Alaka Palazzo.

gives accurate results. Newton’s second law for a mass m exposed to a net force $F \rightarrow$ is expressed in Equation (4)

$$\sum \vec{F} = m\vec{a} = m \frac{dV}{dt} = \frac{d}{dt} (m\vec{V}) \quad (4)$$

here $mV \rightarrow$ is the linear momentum of the system. It denotes both the velocity and density may change from point to point; Thus Newton’s second law can be given in Equation (5).

$$\sum \vec{F} = \frac{d}{dt} \int_{sys} \rho \vec{V} dV \quad (5)$$

Where $\delta m = \rho dV$ is the mass of a differential volume element dV , and $\rho V dV$ is its momentum.

Grid independence study is performed for block - A to eliminate/reduce the influence of the number of grids/grid



FIGURE 3: Showing the isometric view and image of the apartment.

size on the computational results. Dirichelet boundary condition for free stream air is used with 30°C and 23°C. Mixed boundary condition was included for penetration of wall.

The simulations model is validated with wind tunnel experiments (33) with PIV by Karava et al. [28]. The impact

TABLE 1: Carpet area of the apartment.

| S.no | Type of apartment | Area in Sq. m |
|------|-------------------|---------------|
| 1. | Single bedroom | 62.5 |
| 2. | Double bedroom | 124 |
| 3. | Three bedroom | 161 |
| 4. | Four bedroom | 255 |



FIGURE 4: Data Logger.



FIGURE 5: Data Logger recording.

of a wide range of computational parameters is investigated, including the size of the computational domain, the resolution of the computational grid, the inlet turbulent kinetic energy profile of the 4 atmospheric boundary layer, the turbulence model, the order of the discretisation schemes and the iterative convergence criteria.

The Reynolds number has to be calculated and compared with the turbulence level. The Reynolds number should be more than the turbulence level. Reynolds number is a dimensionless number. It is the ratio of inertial forces to viscous forces. It is used to find whether the fluid is in laminar or turbulent flow. The Reynolds number, N_{Re} , is defined as given in Equation (7).

$$N_{Re} = \frac{\rho v d}{\mu} \quad (6)$$



FIGURE 6: Eurolab digital thermometer.



FIGURE 7: HTC™ vane Anemometer (Model: AVM – 07).



FIGURE 8: Reading taken with Hotwire Anemometer.

where.

$$\begin{aligned} \rho &= \text{density} \\ v &= \text{velocity} \\ d &= \text{diameter} \\ \mu &= \text{viscosity} \end{aligned} \quad (7)$$

K-omega model is used to resolve the turbulence level and to analyse the interior domain with Bossiness hypothesis in the momentum equation to include the effect of the density difference in heat transfer. The K-omega model is a low Re model used for flows with low Reynolds number.

TABLE 2: Instruments used to Measure Temperature, Relative Humidity, Air Velocity and Globe temperature on site.

| S.no | Description | Quantity | Parameter measured | Range | Accuracy |
|------|---|----------|--------------------|----------------|------------|
| 1 | Probe thermometer with black painted table tennis ball 40 mm diameter | 4 Nos | Globe temperature | (-60 to 155°C) | ±0.5°C |
| 2 | Hot wire anemometer | 4 Nos | Air velocity | 0.01-50 m/sec | ±0.1 m/sec |
| 3 | Relative humidity data loggers | 4 Nos | Humidity | 0 to 100% | ±5% |
| 4 | Temperature data loggers | 4 Nos | Air temperature | -40 to 70°C | ±0.2°C |

It is commonly known as the scale of turbulence. It may be used to resolve the viscous sub layer and where the boundary layer is comparatively thick.

3. Methodology

A computational analysis for natural convection was carried out in one of the blocks of a 14 storied residential naturally ventilated tall building. The tall building consists of 4 blocks namely block A, B, C, D as shown in Figure 1. It is located in Chennai, India. Natural heat transfer in the tall building was examined to study the movement of air and its respective indoor temperature at different altitudes to subsequently understand the indoor comfort conditions. The data collection process was initially carried out with a field investigation in a lagrangian approach followed by a eulerian approach with a CFD solvent.

This paper discusses the eulerian approach to understand the transfer of heat in the indoor environment with respect to height in the 3 different levels – lower floor level, mid floor level and roof exposed floor level of the 14 storied apartment.

3.1. Description of the City and the Building. India has a coastline of about 7500 km., of which about 5400 km. belong to the peninsular India and the rest to the Andaman, Nicobar and Lakshadweep islands. Tamil Nadu is one of the major states with a coastal length of 1076 km [35] as shown in Figure 2(a). Chennai, is the metropolitan capital city of Tamilnadu. It lies at 13°04' N latitude and 80°17' E longitude. Chennai has the second longest urban coastline in the world. The city is placed at 6.9 m above the sea level. It has an estimated population of 9 million spread across 426 km² of land. According to Koppen climate classification, a major part of Tamil Nadu lies under the tropical savanna climate. It experiences hot temperatures for most of the months in the year. The city receives rainfall due to two trade winds, the southwest monsoon which flows towards the northern hemisphere and northeast monsoon which flows in winter. The summer seasons are the months from March to May and the winter seasons are the few months from November to January. It is during the summer months, the temperature rises from 30°C to 40°C and in the winter the temperature roves between 20°C to 30°C. The location of the building considered in the present work is shown in Figure 2(b).

The naturally ventilated high-rise apartment, located in Chennai is a gated community comprising of 207 apartments in four blocks from the mezzanine floor to the 14th

TABLE 3: Physical Variables Measured on Site.

| Four important physical variables in thermal comfort theory | |
|---|-------|
| Ait temperature (ta) | °C |
| Radiant temperature (Tr) | °C |
| Air velocity (v) | m/sec |
| Relative humidity (H) | % |

floor as shown in Figure 3. Double basements and the ground floor housed the car parking, two-wheeler parking, all services pertaining to the building services, and the maintenance department. The apartment also has a swimming pool which is surrounded by the four blocks. The apartment located in the Chennai suburban area, about 23 km from the coastline, experiences a hot and humid climate. The summer is hot with a higher range of air temperature and moderate to high humidity while the winter has lower to comfort air temperature with a high humidity level.

The building is oriented in such a way the longer sides face the north-south direction. Block – A and Block – D are housed in the eastern and western side, respectively, while Block – B and Block – C housed in a row adjacent to Blocks A and Block – D, respectively, as shown in Figures 1 and 3. The 207 apartments have different types of houses with single bedroom, double bedroom, three-bedroom, and four-bedroom with the carpet area as shown in Table 1. All houses in the apartment had entry through the corridor and had semi-outdoor spaces like balcony.

3.2. Data Collection

3.2.1. Lagrangian Approach. For the study purpose in the lagrangian approach, the 14 storied apartment was categorized and examined in 3 levels – the lower level consisting of the mezzanine floor at +4 m above the ground level, the mid-level comprising of the 6th, 7th and 8th floor and the roof exposed floor ie the 14th floor. The properties of air - temperature, velocity and relative humidity were recorded in the lower level, mid-level, and the roof exposed floors and examined.

To begin the investigation data loggers as shown in Figures 4 and 5 were placed in the living room of the A-block apartments in the 3 levels- the lower floor, the mid floor and the roof exposed. The data loggers recorded the temperature and humidity continuously for every 10 minutes for a period of 2 months and 15 days in the two seasons – winter 2017 and summer 2018. Eurolab digital thermometer with its temperature sensor at the centre of a

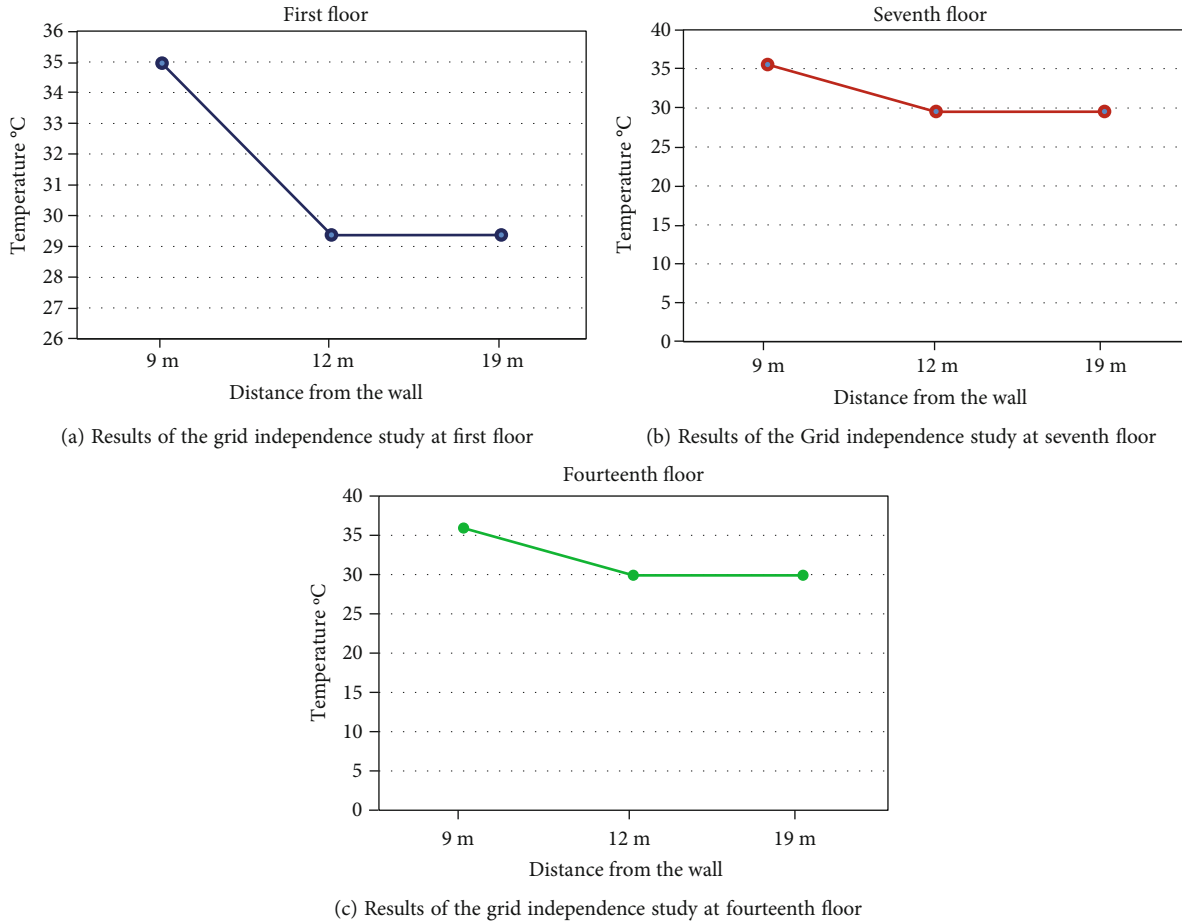


FIGURE 9: (a-c). Results of the grid independence study.

black painted table tennis ball 40 mm dia as shown in Figure 6 were used to measure the Globe temperature. Air velocity were measured using HTC™ vane Anemometer (Model: AVM – 07) shown in Figure 7. The variables temperature, relative humidity and air velocity were recorded for 7 consecutive days in peak winter of December 2017 and peak summer 2018.

To move on, in the lagrangian approach the spot data of the properties of the indoor environment – temperature, relative humidity and velocity were collected at an interval of one hour in random fashion at a fixed height of 1.1 m above the floor in the 3 levels during the day using hotwire anemometer as shown in Figure 8. The study was conducted for 7 consecutive days in the peak winter of December 2018 and peak summer in May 2019. Hence for better understanding, the experimental readings were recorded for four seasons during the two consecutive years. The indoor environmental variables were measured using the instruments shown in Tables 2 and 3 indicates the physical variables measured on site. The results of the lagrangian approach is generic as the spot data collected at a height of 1.1 m which is the lower part of the room is cooler than the upper part of the room due to the buoyancy force. In order to understand the physics of the space better, the indoor volumetric data was examined in its equilibrium state in an eulerian approach using a CFD solvent STARCCM+.

TABLE 4: Wall Properties.

| Details | Values |
|--------------------------------|---------------------------|
| Thermal conductivity | 0.53 W/mK |
| Density | 2240 kg/m ³ |
| Specific heat | 750 J/kg-k |
| Properties of air | |
| Dynamic viscosity | 1.85508E-5 pa-s |
| Density | 1.18415 kg/m ³ |
| Specific heat | 1003.62 J/kg-K |
| Thermal conductivity | 0.0260305 W/m-K |
| Prandtl number | 0.9 |
| Thermal expansion co-efficient | 0.0033/K |

The outdoor climatic variables temperature, relative humidity and air velocity used in the CFD analysis were assumed from the data's collected from the Indian Metrological Department.

3.2.2. Eulerian Approach. The eulerian approach gives a much realistic average volumetric data. It helps to understand the physics of the space in equilibrium state as the colder air in the lower part of the room and the hotter air in the upper part of room cannot be considered separately

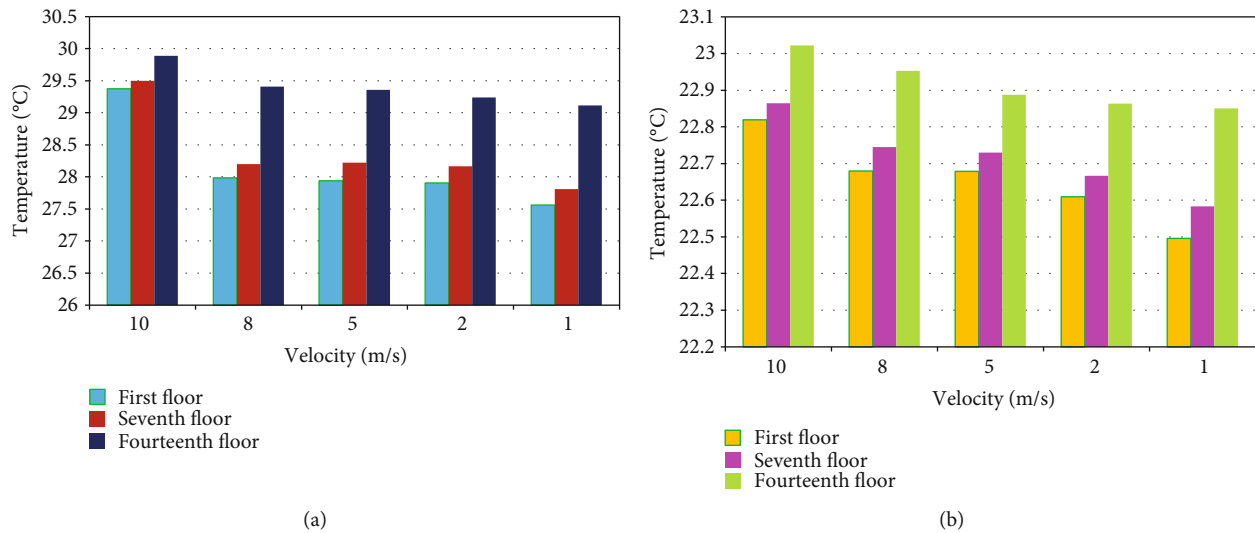


FIGURE 10: (a) Shows variations in the Air Temperature at 3 different heights during a typical day in summer – Experimental data. (b) Shows variations in the Air Temperature at 3 different heights during a typical day in winter – Experimental data.

as in the case of lagrangian approach. The CFD model was created for block- A and 5 different inlet air velocities were used in the model from a maximum of 10 m/s to 1 m/s to study the natural convection process in the 3 different levels. As the indoor environment depends on heat transfer due to both convection and conduction, conjugate heat transfer method is used in the CFD solvent.

To study the convection process, the following assumptions were adopted in the CFD model.

- (1) Windows are fully opened
- (2) The wind direction was assumed from north to south (- y to + y) based on the 30 years wind climatic data collected from the Indian Metrological Department

The Grid independence study as shown in Figure 9 was performed for block – A to form the geometry. To reduce the influence of the number of grid size on the computational results the mesh was created. Dirichelet boundary condition for free stream air is used with maximum outdoor temperature of 30°C and a minimum outdoor temperature of 23°C. Mixed boundary condition was included for penetration of wall. Since the exterior façade wall was made of concrete, the wall properties and air properties are shown in Table 4 and these values are used for the development of the CFD model. Discretization is the spacing between each point in the space.

The outdoor temperature was kept constant and the inlet air velocity was varied because the process of convection happens only when the air velocity is varied. The heat transfer analysis was studied for the lower floors namely the 1st and 2nd floors, 7th floor being the mid floor level and the 14th floor being the roof exposed floor using the CFD solvent. The boundary conditions - initial outdoor temperature has been kept constant at 30°C on the higher side and 23°C on the lower side and the initial outdoor velocity is varied from 1 m/s, 2 m/s, 5 m/s, 8 m/s and 10 m/s at 12 noon when

the outdoor temperature is maximum. As the inlet air velocity varies the rate of heat transfer will also vary subsequently bringing changes in the indoor environment.

In the CFD analysis wind pressure is an important boundary condition while calculating the natural ventilation. Flow patterns and turbulence due to wind pressure on the exterior facades of the building are proportional to the atmospheric pressure outside the building. Wind pressure on the façade is a dimensionless number usually called the pressure coefficients (C_p). The CFD simulation is validated with the wind tunnel test. The boundary layer thickness was determined from the Reynolds number. Wall whiteness is a dimensionless distance from the wall to the first cell. Wall whiteness should be less than 5. In this paper the wall whiteness is assumed as 3. The mesh was done accordingly; however for accurate results the problem was done based on the skin friction thickness and wall whiteness.

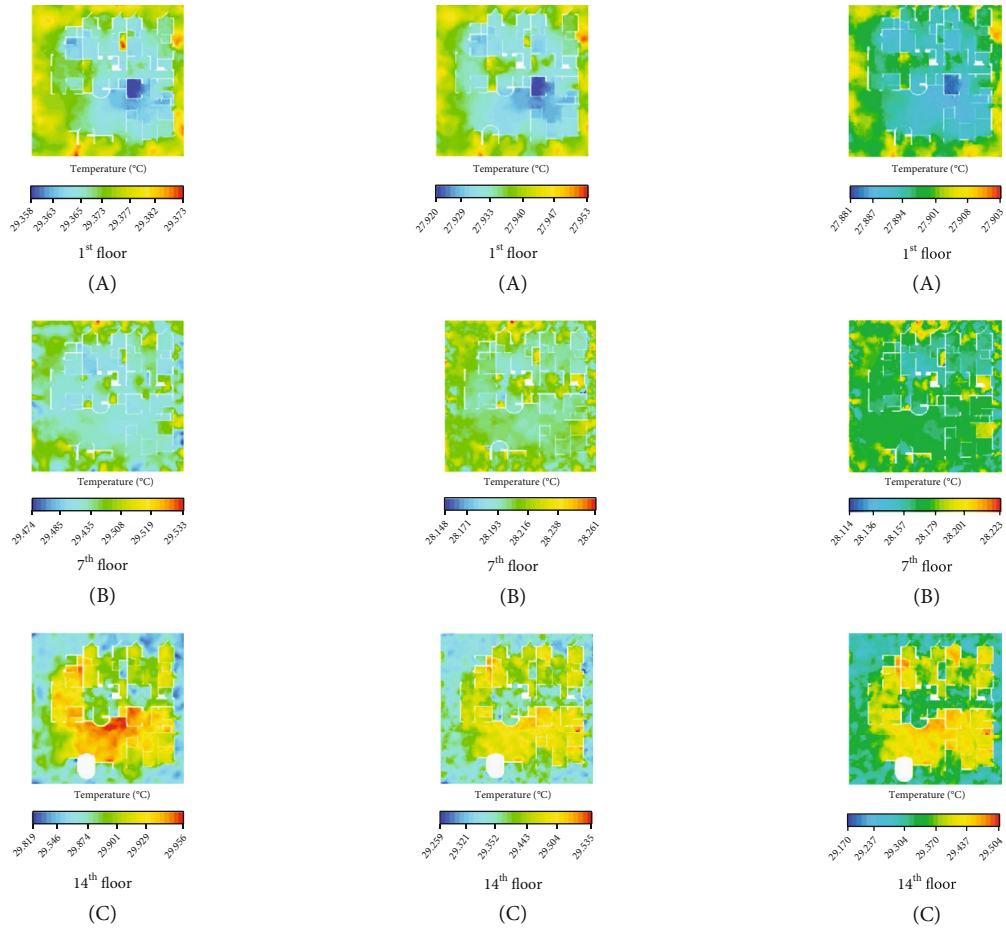
Heat transfer through conduction in the building happens within the exterior concrete wall material. Hence the wall material concrete is assumed and the rate of conduction in the lower floor, mid floor and roof exposed floor are examined and the results are discussed.

4. Results and Discussion

The results indicate an analysis of a numerical simulation of few computational parameters of a generic isolated building with wind induced ventilation.

4.1. Limitations of the Study

- (i) The CFD simulation was performed only for Block – A of the isolated building due to expensive computational cost. The building geometry and the position and size of the openings were not investigated
- (ii) The CFD simulation was carried out for a specific aerodynamic roughness length and its related

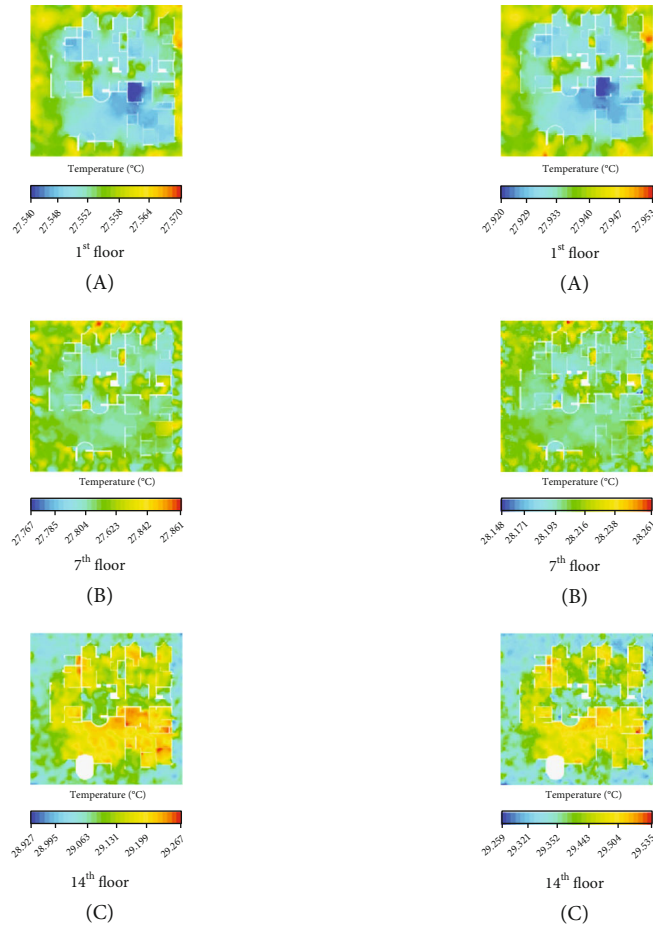


(a) Plans showing the heat transfer process in the 3 levels at temperature 30°C with air velocity 10 m/s

(b) Plans showing the heat transfer process in the 3 levels at temperature 30°C with air velocity 8 m/s

(c) Plans showing the heat transfer process in the 3 levels at temperature 30°C with air velocity 5 m/s

FIGURE 11: Continued.



(d) Plans showing the heat transfer process in the 3 levels at temperature 30°C with air velocity 2 m/s
 (e) Plans showing the heat transfer process in the 3 levels at temperature 30°C with air velocity 1 m/s

FIGURE 11: (a-e). Plan of the heat transfer process plots at 30°C with varied air velocities: 10 m/s, 8 m/s, 5 m/s, 2 m/s and 1 m/s.

incident atmospheric boundary layer profiles of mean wind speed, turbulence dissipation rate and turbulent kinetic energy

- (iii) The study primarily concentrated on the mean air speed
- (iv) In the computational domain the upstream and downstream length 3H and 15H, respectively, were fixed, with the building height H, while the cross-section (width and height) was varied

According to the macroscopic study both the inlet air velocity and the thermal resistance play a major role in setting the equilibrium temperature of the indoor space. The thermal resistance of the exterior wall of the building decreases the temperature from the external environment. Figure 10(a) and Figure 10(b) shows the temperature plots with outdoor temperature at 30°C and 23°C, respectively. The X- axis represents the inlet air velocities namely 10 m/s, 8 m/s, 5 m/s, 2 m/s and 1 m/s. The Y- axis represents the corresponding indoor temperature due to the heat transfer process. The maximum outdoor air velocity 10m/s was assumed from the 30 year wind data collected from the Indian Metrological Department

(IMD). The minimum outdoor velocity 2 m/s and 1 m/s which are nil air or no air condition were assumed to check the heat transfer during these conditions.

According to fluid dynamics heat transfer happens only when fluids flow. The boundary layer thickness was calculated as shown in Table 4 for all the 5 velocities assumed with a series of formulas and the Y value was determined. The model was simulated each time using the Y values with respect to the inlet air velocity and temperature plots showing natural convective heat transfer was developed as shown in Figure 11(a)–11(e), for the maximum and minimum temperatures at 12 noon condition.

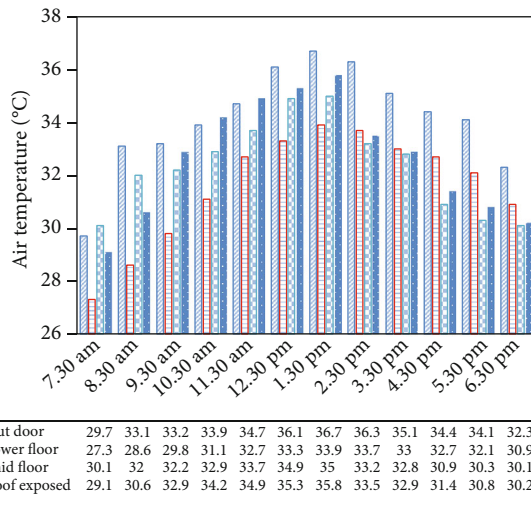
Table 5: shows the detailed calculation of the boundary layer thickness y using the velocities 10 m/s, 8 m/s, 5 m/s, 2 m/s and 1 m/s calculated.

Re- Reynolds Number: cf – Coefficient of Friction: τw – $V_r = \sqrt{\tau w / \rho}$; Y –Boundary Layer Thickness.

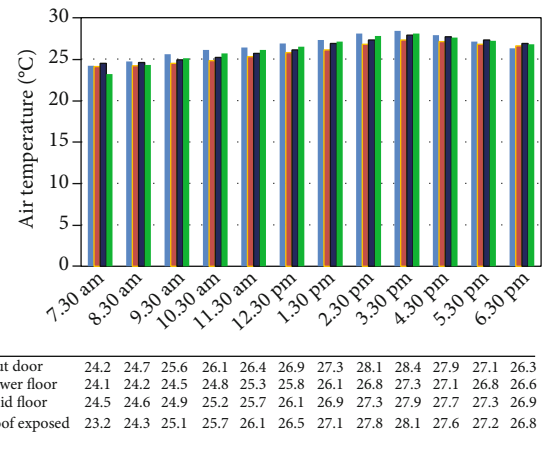
It is observed from Figure 10(a) and Figure 10(b) that the velocity plays a major role in setting the equilibrium temperature but thermal resistance plays a stronger role too. Hence as the velocity increases, it also increases the temperature of control volume. The thermal resistance decreases the temperature from the external environment. However it

TABLE 5: shows the detailed calculation of the boundary layer thickness by using the velocities 10 m/s, 8 m/s, 5 m/s, 2 m/s and 1 m/s.

| S.no | Air velocity m/s | $Re = \rho v l / \mu$ | $cf = 0.058 \times Re^{-0.2}$ | $\tau_w = \frac{1}{2} cf \times v^2 \rho$ | $Vr = \sqrt{\tau_w / \rho}$ | $Y = y^+ \times \mu / v \tau \times \rho$ |
|------|------------------|-----------------------|-------------------------------|---|-----------------------------|---|
| 1. | 10 | 1.8×10^7 | 2.031×10^{-3} | 0.1127 | 0.3187 | 1.58×10^{-4} |
| 2. | 8 | 1.5×10^7 | 2.124×10^{-3} | 0.1058 | 0.3685 | 1.37×10^{-4} |
| 3. | 5 | 0.9×10^7 | 2.333×10^{-3} | 0.0323 | 0.1705 | 2.96×10^{-4} |
| 4. | 2 | 0.37×10^7 | 2.802×10^{-3} | 0.0062 | 0.0747 | 6.77×10^{-4} |
| 5. | 1 | 0.18×10^7 | 3.219×10^{-3} | 0.0017 | 0.0391 | 12.98×10^{-4} |



(a)



(b)

FIGURE 12: (a) Shows variations in the Air Temperature at three different heights during a typical day in summer – Experimental data. (b) Shows variations in the Air Temperature at three different heights during a typical day in winter – Experimental data.

TABLE 6: Indoor Temperature variations at 30°C and 23°C for various air velocities.

| S.no | Inlet air velocity (m/s) | Outdoor temperature | First floor temperature (°C) | | Seventh floor temperature (°C) | | Roof exposed floor temperature (°C) | |
|------|--------------------------|-----------------------------------|------------------------------|-------|--------------------------------|-------|-------------------------------------|-------|
| | | | 30°C | 23°C | 30°C | 23°C | 30°C | 23°C |
| | | | 1. | 10 | 29.37 | 22.82 | 29.49 | 22.86 |
| 2. | 8 | Corresponding indoor temperatures | 27.98 | 22.68 | 28.20 | 22.75 | 29.40 | 22.95 |
| 3. | 5 | | 27.94 | 22.68 | 28.22 | 22.73 | 29.35 | 22.89 |
| 4. | 2 | | 27.90 | 22.61 | 28.16 | 22.67 | 29.24 | 22.86 |
| 5. | 1 | | 27.56 | 22.50 | 27.81 | 22.58 | 29.11 | 22.85 |

has been observed in both the cases CFD model and the experimental data as shown in Figure 10(a), Figure 10(b) and Figure 12(a), Figure 12(b), respectively, at 12 noon condition, the temperatures are always higher in the roof exposed floor when compared with the other floors.

It is also observed from Figure 10(a) and Figure 10(b) that the temperature plot shows a decrease in temperature with respect to height because of resistance offered due to the movement of air temperature and the resistance offered by conduction = L/kA and resistance offered by convection = $1/hA$. The difference between the lower temperature 23°C and higher Temperature 30°C is very feeble because the velocity plays the major role in forced convection.

When comparing the temperature plots we can understand that the concrete gets more heated than the inner air. Hence the concrete wall temperature should be lowered through some construction techniques or use of low conductivity wall material. However if we increase the ventilation in roof exposed floor by providing comparatively larger openings, the velocity will decrease with an increase in pressure which will impact the temperature directly.

Table 6 shows the difference between the minimum temperature 23°C and maximum temperature 30°C is very feeble because of velocity due to convection. The velocity plays the major role in forced convection. It can see from Figure 10(a) and Figure 10(b) the lower floor temperature

is observed lowest when compared to roof exposed floor. This is due to environmental impacting capability; the roof exposed floor has more volume of air which can dominate the temperature.

It is inferred from the heat transfer plans as shown in Figure 11 series that places where the inner air is in contact with the external environment the temperature is higher. The leading edge has the higher temperature along with the spreading rate when compared to the trailing edge. It is also observed from Figure 11 series in the first floor plan, those places where there is direct contact with the external environment, it has a free high temperature due to free steam air. Similarly, in places where the air is in contact especially through windows, it is observed the room temperature is lesser when compared to the inlet air temperature. The dominance of the environment is more here because of the prevailing inlet air velocity.

5. Conclusion

- (i) The study of heat transfer in buildings is necessary as it helps to identify the appropriate building fabric, window openings and construction technique to obtain the desired indoor thermal conditions for energy conservation and sustainable living
- (ii) A CFD simulation model provide a comprehensive analytical model in the initial stage of the design to determine a strategic plan for the natural ventilation potential in the different floors for energy conservation and low energy consumption
- (iii) Although CFD model is complex and time consuming it can predict accurate wind pressure around the building to examine the thermal behaviour of the internal spaces in the building. Due to the movement of air with respect to the turbulence in air natural convective heat transfer in buildings is developed. Hence as the velocity increases
- (iv) It is observed the temperature in the control volume also increases. The thermal resistance decreases the temperature from the external environment
- (v) From the eulerian approach in this paper, it is understood that the velocity in forced convection acts as a catalyst to take the temperature from the exterior domain to the interior part of the building
- (vi) The density stratification occurs in the indoor volume of the building. Hence when the outdoor temperature is higher and the outdoor velocity is also higher the temperature which enters the indoor environment is also higher
- (vii) Velocity plays a major role in setting the equilibrium temperature the thermal resistance also plays a stronger role too. Hence with limitations in wall opening sizes like windows, doors etc and to reduce the indoor thermal environment, exterior façade can be of suitable low conductivity materials

or alternate construction techniques such as cavity wall can be tested for saving energy with less use of active cooling systems for sustainable living. This is one the directions for future study

- (viii) The indoor temperature is increased as the inlet air velocity increases. The minimum of 22.50°C temperature was observed at air velocity 1 m/s at 1st floor at outdoor temperature 23°C. The maximum of indoor temperature of 29.89°C was observed at air velocity 10 m/s at roof at outdoor temperature 29.9°C

Data Availability

The data used to support the findings of this study are included within the article.

Conflicts of Interest

The authors declare that they have no conflicts of interest regarding the publication of this paper.

References

- [1] M. A. Humphreys, "Field studies of thermal comfort compared and applied," *Building Services Engineer*, vol. 44, pp. 5–27, 1976.
- [2] R. J. Dedear and A. Auliciems, "Validation of the predicted mean vote model of thermal comfort in six Australian field studies," *ASHRAE Transactions*, vol. 91, no. 2, pp. 452–468, 1985.
- [3] G. E. Schiller, E. Arens, F. Bauman et al., "A field study of thermal environment in office buildings," *ASHRAE Transactions*, vol. 94, no. 2, pp. 280–308, 1988.
- [4] G. E. Schiller, "A comparison of measured and predicted comfort in office buildings," *ASHRAE Transactions*, vol. 96, no. 1, pp. 609–622, 1990.
- [5] C. Benton, F. Bauman, and M. Fountain, "A field measurement system for the study of thermal comfort," *ASHRAE Transactions*, vol. 96, no. 1, pp. 623–633, 1990.
- [6] R. J. Dedear and M. E. Fountain, "Field experiments on occupant comfort and office thermal environments in a hot-humid climate," *ASHRAE Transactions*, vol. 100, no. 2, pp. 457–475, 1994.
- [7] W. C. Howell and P. A. Kennedy, "Field validation of the Fanger thermal comfort model," *Human Factors: The Journal of the Human Factors and Ergonomics Society*, vol. 21, no. 2, pp. 229–339, 1979.
- [8] W. C. Howell and C. S. Stramler, "The contribution of psychological variables to the prediction of thermal comfort judgments in real world settings," *ASHRAE Transactions*, vol. 87, no. 1, pp. 609–621, 1981.
- [9] D. S. Fishman and S. L. Pimbert, "Survey of the subjective responses to the thermal environment in offices," in *Indoor Climate*, P. O. Fanger and O. Valbjorn, Eds., pp. 677–698, Danish Building Research Institute, Copenhagen, 1979.
- [10] J. F. Busch, "Thermal responses to the THAI office environment," *ASHRAE Transactions*, vol. 96, no. 1, pp. 859–872, 1990.

- [11] Y. Xia, R. Zhao, and Y. Jiang, "Thermal comfort in naturally ventilated houses in Beijing," *HV & AC*, vol. 29, no. 2, pp. 1–5, 1999.
- [12] K. Cena and R. J. Dedear, "Field study of occupant comfort and office thermal environments in a hot, arid climate," *ASHRAE Transactions*, vol. 105, no. 2, pp. 204–217, 1999.
- [13] A. A. De Paula Xavier and L. Roberto, "Indices of thermal comfort developed from field survey in Brazil," *ASHRAE Transactions*, vol. 106, pp. 45–58, 2000.
- [14] BIS, "National Building Code," *Bureau of Indian Standards*, 2005.
- [15] *Energy Conservation Building Code User Guide*, Bureau of Energy Efficiency, India, 2009.
- [16] S. D. Attri and A. Tyagi, *A report on 'climatic profile of India*, India Meteorological Department, 2010.
- [17] K. Panchabikesan, K. Vellaisamy, and V. Ramalingam, "Passive cooling potential in buildings under various climatic conditions in India," *Renewable and Sustainable Energy Reviews*, vol. 78, pp. 1236–1252, 2017.
- [18] "effects of building microclimate on the thermal environment of traditional Japanese houses during hot-humid summer," *Buildings*, vol. 9, no. 1, p. 22, 2019.
- [19] N. B. Geetha and R. Velraj, "Passive cooling methods for energy efficient buildings with and without thermal energy storage," *Energy Education Science and Technology Part A: Energy Science and Research*, vol. 29, no. 2, pp. 913–946, 2012.
- [20] A. Aflaki, N. Mahyuddin, Z. al-Cheikh Mahmoud, and M. R. Baharum, "A review on natural ventilation applications through building façade components and ventilation openings in tropical climates," *Energy and Buildings*, vol. 101, pp. 153–162, 2015.
- [21] B. Raji, M. J. Tenpierik, R. Bokel, and A. van den Dobbels-teen, "Natural summer ventilation strategies for energy-saving in high-rise buildings: a case study in the Netherlands," *International Journal of Ventilation*, vol. 19, no. 1, pp. 25–48, 2020.
- [22] R. Ramponi and B. Blocken, "CFD simulation of cross-ventilation for a generic isolated building: impact of computational parameters," *Building and Environment*, vol. 53, pp. 34–48, 2012.
- [23] Q. Chen, "Ventilation performance prediction for buildings: a method overview and recent applications," *Building and Environment*, vol. 44, no. 4, pp. 848–858, 2009.
- [24] J. Seifert, Y. Li, J. Axley, and M. Rosler, "Calculation of wind-driven cross ventilation in buildings with large openings," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 94, no. 12, pp. 925–947, 2006.
- [25] J. O. P. Cheung and C. H. Liu, "CFD simulations of natural ventilation behaviour in high-rise buildings in regular and staggered arrangements at various spacings," *Energy and Buildings*, vol. 43, no. 5, pp. 1149–1158, 2011.
- [26] T. Van Hooff and B. Blocken, "Coupled urban wind flow and indoor natural ventilation modelling on a high-resolution grid: A case study for the Amsterdam ArenA stadium," *Environmental Modelling and Software*, vol. 25, no. 1, pp. 51–65, 2010.
- [27] S. Kato, S. Murakami, A. Mochida, S. Akabayashi, and Y. Tominaga, "Velocity-pressure field of cross ventilation with open windows analyzed by wind tunnel and numerical simulation," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 44, no. 1-3, pp. 2575–2586, 1992.
- [28] Y. Jiang, D. Alexander, H. Jenkins, R. Arthur, and Q. Chen, "Natural ventilation in buildings: measurement in a wind tunnel and numerical simulation with large-eddy simulation," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 91, no. 3, pp. 331–353, 2003.
- [29] P. Heiselberg, Y. Li, A. Andersen, M. Bjerre, and Z. Chen, "Experimental and CFD evidence of multiple solutions in a naturally ventilated building," *Indoor Air*, vol. 14, no. 1, pp. 43–54, 2004.
- [30] D. W. Etheridge and M. Sandberg, *Building Ventilation: Theory and Measurement*, John Wiley & Sons, Chichester, New York, 1996.
- [31] G. Evola and V. Popov, "Computational analysis of wind driven natural ventilation in buildings," *Energy and Buildings*, vol. 38, no. 5, pp. 491–501, 2006.
- [32] B. Blocken, T. van Hooff, L. Aanen, and B. Bronsema, "Computational analysis of the performance of a venturi-shaped roof for natural ventilation: Venturi-effect versus wind-blocking effect," *Computers and Fluids*, vol. 48, no. 1, pp. 202–213, 2011.
- [33] P. Karava, T. Stathopoulos, and A. K. Athienitis, "Airflow assessment in cross-ventilated buildings with operable façade elements," *Building and Environment*, vol. 46, no. 1, pp. 266–279, 2011.
- [34] S. Sen, H. Tarun, D. Arumuga Perumal, and A. K. Yadav, "A Numerical Approach for Natural Convection With Curved Obstacles in an Enclosure Using Lattice Boltzmann Method," *ASME Open Journal of Engineering*, vol. 1, article 011004, 2022.
- [35] M. Goodarzi, M. R. Safaei, A. Karimipour et al., "Comparison of the Finite Volume and Lattice Boltzmann Methods for Solving Natural Convection Heat Transfer Problems inside Cavities and Enclosures," vol. 2014, Article ID 762184, pp. 1–15.