Int. J. Nonlinear Anal. Appl. 15 (2024) 8, 195–204
ISSN: 2008-6822 (electronic)
http://dx.doi.org/10.22075/ijnaa.2023.30424.4395

A modified zone model for investigating the airflow patterns in unified spaces with natural convection

Mostafa Rahimi, Farhad Sadegh Moghanlou, Khadijeh Samandari*

Department of Mechanical Engineering, University of Mohaghegh Ardabili, Ardabil, Iran

(Communicated by Seyed Hossein Siadati)

Abstract

Predicting the air flow pattern with proper accuracy and speed inside a building with natural ventilation is one of the important study topics of building design due to the economic importance of energy consumption. For this purpose, in this study, a chamber with certain dimensions and with natural connection has been investigated as a study sample to predict the air flow pattern and speed distribution with the help of regional models. Two air inlet and outlet valves were considered for natural air ventilation. After researching the previous models, the three-dimensional zone model coupled with heat transfer and air flow calculations (ZAER) was chosen as the basis for comparing the modeling results. In the following, after zoning, to improve the flow coefficient factor model, which is assumed to be a constant number, it was assumed to be variable and the results were recorded in different states. The behavior of the model has been checked in the flow coefficients between 0.3 and 1 in order to obtain the most optimal coefficient for the number of suitable cells. Comparing the results of the research with the results of other models and computational fluid dynamics (CFD) showed that the three-dimensional behavior of the flow shows better compatibility with the experimental researches.

Keywords: Regional model, Air flow, Natural convection, ZAER, Flow coefficient
2020 MSC: 37E35

1 Introduction

In many countries, the building sector accounts for 40-60% of the total energy consumption [8]. Considering the issues related to energy, it is necessary to develop suitable simulation tools for researching the movement of air flow and the thermal stratification of the environment in large spaces, especially for projects where energy costs are important. Building energy simulation software such as Energy Plus and DeST require functional computing modules and improvements for large spaces [10, 17].

The first zonal models (Lebrun in 1970, Howarth in 1980, Inard in 1988) were developed based on previous information of airflow patterns [13]. During several decades, three levels of modeling were developed to predict building energy consumption or temperature distribution and flow displacement in the building: 1) nodal method (single-zone and multi-zone model), 2) CFD computational fluid dynamics model, 3) zone model. Regional methods are based

*Corresponding author
Email addresses: rahimi@uma.ac.ir (Mostafa Rahimi), fmoghanlou@uma.ac.ir (Farhad Sadegh Moghanlou), samandarikh62@gmail.com (Khadijeh Samandari)

Received: February 2023 *Accepted:* May 2023

on dividing the room into a number of sub-areas and applying continuity and energy equations in each sub-area. This method is an intermediate approach between the single-node method (one-zone and multi-part) and the CFD model [14, 12]. Regional models ignore turbulence and conservation of momentum and instead consider the power law equation as a function between pressure and flow. Allard and Inard [1] extensively studied the previous regional models and examined the evolution of regional models. Recent developments and applications of regional models for indoor environments have been reviewed.

Macroscopic or multi-zone infiltration air flow models such as COMIS and CONTAM have been widely used to predict air flow in simple methods [5]. This method considers each room as a single area with a uniform temperature and calculates the flow between the areas. These models are suitable tools for designing ventilation systems for complex buildings; in such a way that they provide approximate shapes as input for thermal analysis tools. However, they are not able to evaluate the stratification of the indoor air and the air velocity distribution inside the buildings.

Figure 1 summarizes the evolution of regional models from 1970 to 2020. In 1970, the basis of the zone model for predicting thermal stratification in a room was presented. Earlier simple area models were gradually developed over 20 years. Since 2000, various regional models have been developed keeping the original advantage (balance between accuracy and efficiency) and overcoming the previous disadvantages of the models (by developing simplified N-S equations). For example, in 2012, the VEPZO (Velocity Propagating Zonal) model of the forces acting on the flow path was obtained in the form of an ordinary differential equation. The PLM (Power law model) method, as one of the old methods, has been developed by researchers such as Foucquier, Bouia, Dalicieux and Wurtz since 1989 [10].

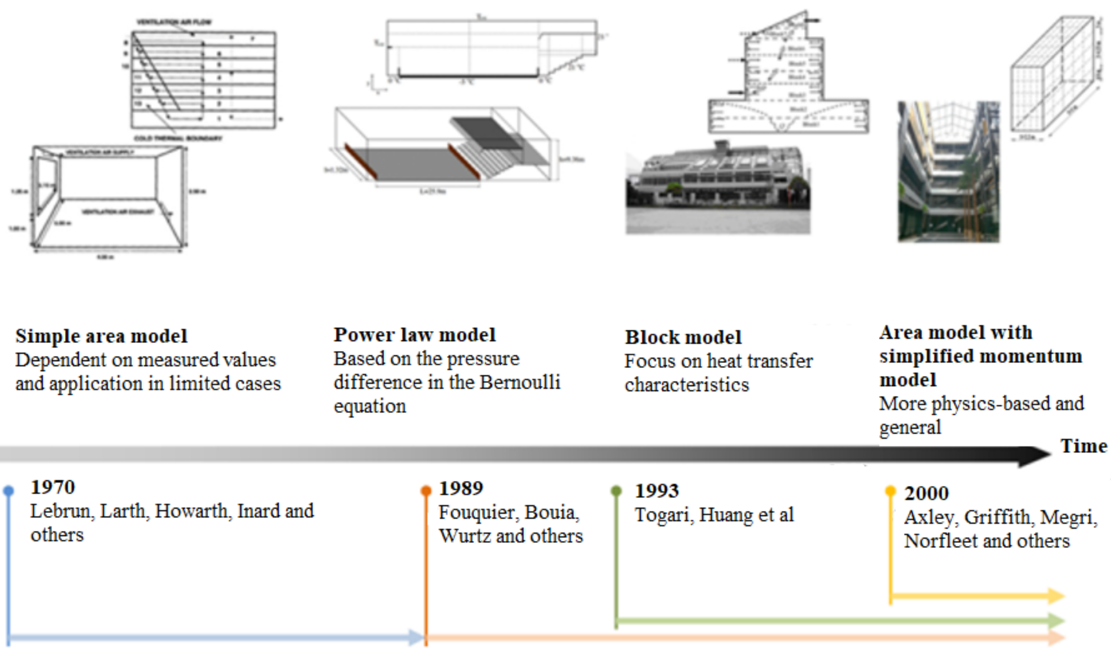

Figure 1: Time history of regional models in the last five decades [10]

Today's regional models include a unified space with a limited number of regions (cells or control volumes); Typically, less than a thousand cells are considered for 3D flow. The air inside each area is completely mixed and its mass and energy are assumed to be constant. These areas are connected with each other through the borders and through this the air flow and heat flux are transferred. In this type of regional models, calculating the air flow rate between the designated areas is the main problem in modeling [10]. One of the conventional methods of the regional model is the POMA method; that many researches have been done on improving the results of this method in regional modeling. For example, in one study, this model was rewritten and in the new POMA+ model based on the Navier-Stokes energy equation, the three A terms in the POMA energy equation were replaced. By using this new heat transfer expression in the POMA model, it is possible to improve the accuracy of the second order or higher depending on how the expression of the first derivative of temperature is described. It should be kept in mind that in this model, the flow is considered constant [14]. One of the conventional methods of the zone model is the POMA (Pressurized zone model with air diffuser) method; that many researches have been done on improving the results of this method in regional modeling. For example, in one study, this model was rewritten and in the new +POMA model, based on the Navier-Stokes energy equation, three terms $\rho V c_p \left(u\ \partial T_{ai/\partial x} + v\ \partial T_{ai/\partial y} + w\ \partial T_{ai/\partial z}\ \right)$ were replaced with convection heat transfer terms $\sum_{j=1}^{n} \dot{q_{ij}}$ in the POMA energy equation. By using this new heat transfer expression in the POMA model, it is possible to improve the accuracy of the second order or higher depending on how the expression of the first derivative of temperature is described. It should be kept in mind that in this model, the flow is considered constant [14].

In the first experimental research on the subject under investigation, measurements were made on a real-size experimental cell called Minibat CETHIL in 1998 by Castanet [16].

This test cell has parallel surfaces with a square base of 3.1 meters and a height of 2.5 meters. This cell is surrounded by a technical vacuum; which keeps the temperatures of the inner surface constant. One of the vertical faces of the cell (facing south) is made of glass and is exposed to weather changes; which makes the air temperature in the range of -10 to 40 degrees Celsius, with high stability (± 0.2 degrees Celsius). Next, in an experimental method, with the help of PIV particle image velocity measurement, they modeled the air flow pattern in the building space and compared it with the results of the zone model [11], which is used as a reference experimental work in this research.

In one of the latest researches, Yamasawa et al [18] presented the prediction of thermal and contaminant environment in a room with impinging jet ventilation system by zonal model. In another research, Hu et al. [6] investigated inter-zonal heat transfer in large space buildings based on similarity and presented a comparison of two classified air conditioning systems.

The zone model is based on dividing the room space or air volume inside the building into a limited number of separate parallel control volumes (zones) and in them, the air state variables (temperature, density and pressure) are considered constant. Next, the adjacent areas are connected by connections (borders) and through them, air flow and heat are transferred. In this model, the unknowns of the problem are air flow rate and air condition variables [7].

According to the stated research background, in this article, the development of a zone model for investigating flow patterns and temperature distribution in integrated spaces with natural convection was discussed. For this purpose, the model of the ZAER (Zonal Aerial model) method, which is a simplified method (Bouia 1993) [2] was expanded. This model uses the simplified form of conservation of momentum to determine the air mass flow rate and air temperature inside a building in calculating the inside pressure distribution.
In the first part of this article, the description of ZAER model was discussed. Its results were compared for air speed and temperature obtained from the Wurtz zone model, as well as experimental data and CFD analysis. Then, as the main innovation of this research, the effect of selecting the value of C_d discharge coefficient on heat transfer and air flow through large vents was measured and optimized by formulating the data. It was observed that by improving the fixed values of C_d used in previous models, the results of the model are improved.

2 Mathematical modeling method and introduction of equations

In the ZAER method, the desired interior space is divided into a number of cells. The speed of the air flow between two cells is created only by the pressure difference between the two centers of the cell and creates a flow with the discharge speed at the border of the cells, which crosses the border between them with the discharge coefficient C_d [4]. In this method, the mass flow rate at the boundary between two cells i and j for the vertical boundary (Figure 2) is as follows:

$$\dot{m}_{ij} = \sqrt{2\rho}\varepsilon_{ij} C_d S_{ij} \left|\Delta P_{ij}\right|^{\frac{1}{2}}, \tag{2.1}$$

where

$$\begin{cases} \Delta P_{ij} = P_j - P_i, \\ |\varepsilon_{ij}| = 1, \\ sign(\varepsilon_{ij}) = sign(\Delta P_{ij}). \end{cases}$$

$$\begin{cases} \rho = \rho_j, \quad if \quad \varepsilon_{ij} = 1, \\ \rho = \rho_i, \quad if \quad \varepsilon_{ij} = -1. \end{cases}$$

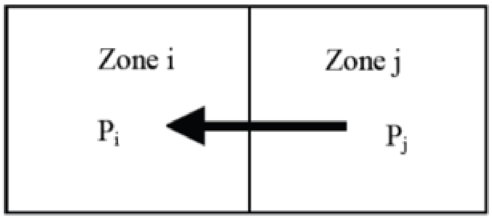

Figure 2: Passage of mass flow in the vertical boundary $P_j > P_i$)

Of course, equation (2.1) is derived from the relation of flow through an orifice:

$$Q = CA\sqrt{2\Delta P\rho}. \tag{2.2}$$

For the horizontal boundary (Figure 4), the equation for the mass flow from the cell boundaries is written as follows:

$$\dot{m}_{ij} = \sqrt{2\rho}\varepsilon_{ij} C_d S_{ij} \left| \Delta P_{ij} - \frac{1}{2}(\rho_i g h_i + \rho_j g h_j) \right|^{\frac{1}{2}}, \tag{2.3}$$
$$sign(\varepsilon_{ij}) = sign(\Delta P_{ij} - \frac{1}{2}(\rho_i g h_i + \rho_j g h_j)).$$

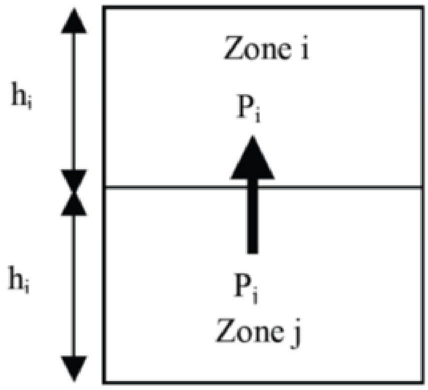

Figure 3: Passage of mass flow in the horizontal boundary (ε_{ij})

It should be noted that the zone model introduced has the following assumptions:

- The 3D grid used is Cartesian; in which the cells are placed in parallel planes.
- Temperature and pressure are constant in each cell.
- Analysis is done in steady state.
- The speed of current passing through the boundary of a cell is a constant value.
- Conduction of heat in the air is ignored and it is assumed that there is no internal heat source in the space.

With these assumptions, the mass and heat balances and the perfect gas law for each cell i of the room are expressed as follows:

$$\sum_j \dot{m}_{ij} = 0, \tag{2.4}$$

$$\begin{matrix} \sum_{j,\dot{m}_{ij}>0} \dot{m}_{ij} C_p T_j + \sum_{j,\dot{m}_{ij}<0} \dot{m}_{ij} C_p T_i + \\ \sum_w h_{c_i,w} S_{i,w}(T_w - T_i) = 0 \end{matrix}, \tag{2.5}$$

$$P_i - \rho_i \frac{RT_i}{M} = 0, \tag{2.6}$$

The system of equations formed by equations (2.4), (2.5) and (2.6) is non-linear and it is difficult to solve all the equations at the same time and it is not easily solved. To solve this problem, a numerical and iterative method in block form was chosen. The main advantage of this method is to reduce the number of equations and its implementation is numerically easier. Finally, the numerical solution is obtained according to the following steps.

Solving the nonlinear system, which is obtained by the mass balance equation (2.4), is performed by Broyden's method [16]. The power law used to represent the mass flow between cells is non-linear. To solve the problem, the power law becomes linear in the range 10^{-4} to 10^{-8}) Pa.

$$|\Delta P|^{\frac{1}{2}} = \begin{cases} -(\Delta P)^{\frac{1}{2}} & \Delta P \leq -\varepsilon \\ \varepsilon^{-\frac{1}{2}} |\Delta P| & -\varepsilon \leq \Delta P \leq \varepsilon \\ (\Delta P)^{\frac{1}{2}} & \varepsilon \leq \Delta P \end{cases} \tag{2.7}$$

After obtaining the pressure distribution, the mass flow rate between the cells is calculated. This mass flow rate will be the input of the thermal block. The temperature distribution can be obtained using the mentioned equations. The solution of the linear system obtained by the heat balance equations (2.5) is performed by the classical method of Gauss iteration [15].

Considering that the compressive term is applied in the Y direction, the Z direction equations will be the same as the X direction equations. The dimensions of the space are variable; However, in the Y direction, the compressive term $\rho_i g h_i$ is effective. At the inlet and outlet, velocity, mass flow rate, temperature and pressure are constant values. that's mean:

$$\dot{m}_{ij} = \sqrt{2\rho}\varepsilon_{ij} C_d A_i \left|\Delta P_{ij}\right|^n . \tag{2.8}$$

Here $n = 0.5$ and $A = S$ is the cell surface in Y direction. As a result, we reach the same equation (2.1) and the equations are the same in X and Z directions. The mass flow rate in the Z direction is:

$$\begin{aligned} &\dot{m}_{ij} = \sqrt{2\rho}\varepsilon_{ij} C_d S_{ij} \left|\Delta P_{ij}\right|^{\frac{1}{2}} \\ &\begin{cases} \Delta P_{ij} = P_j - P_i \\ |\varepsilon_{ij}| = 1 \\ sign(\varepsilon_{ij}) = sign(\Delta P_{ij}) \end{cases} , \quad \begin{cases} \rho = \rho_j & if \quad \varepsilon_{ij} = 1 \\ \rho = \rho_i & if \quad \varepsilon_{ij} = -1 \end{cases} \end{aligned} \tag{2.9}$$

After solving the equations in the Z direction, the perfect gas and energy balance equations for natural convection are modeled so that the temperature distribution in the desired space can be obtained. So, like the X direction, for the Z direction, we can also write:

$$\begin{aligned} &\textstyle\sum_{j,\dot{m}_{ij}>0} \dot{m}_{ij} C_p T_j + \sum_{j,\dot{m}_{ij}<0} \dot{m}_{ij} C_p T_i + \\ &\textstyle\sum_w h_{c_i,w} S_{i,w} (T_w - T_i) = 0 \end{aligned} \tag{2.10}$$

$$P_i - \rho_i \frac{RT_i}{M} = 0 \quad \rightarrow \quad P_i = \rho_i R_a T_i \tag{2.11}$$

where R_a is the air constant. The sum of inputs and outputs around a cell is solved using equations (2.9), (2.10) and (2.11) and mass flow rate and velocity distribution are obtained.

3 Simulation

A large indoor space was modeled by coding in MATLAB software environment. As shown in Figure **??**, an inlet opening and an outlet opening were considered as air flow ventilation. In order to validate the improvement made on the zone model, the experimental results of a recent research were used [9]. In the mentioned research, an experimental sample with a smaller size and conditions similar to the real sample has been designed and tested. We matched the model in terms of geometry and conditions in order to determine the accuracy of modeling by comparing the results.

The assumptions of experimental research simulation are as follows:

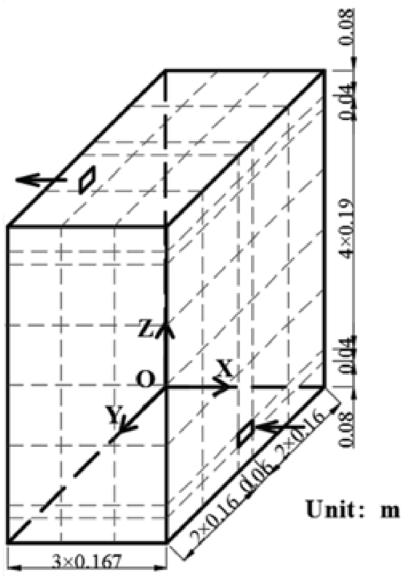

Figure 4: Zoning structure in simulation [9]

- A simple rectangular building (made of Plexiglas with a thickness of 10 mm) with dimensions of 500 mm (length) × 700 mm (width) × 1000 mm (height) with air fluid is available.
- Two entrance and exit openings on two opposite walls with dimensions of 60 mm × 40 mm are designed to simulate windows near the floor and ceiling surfaces and help ventilation.

The flow characteristics are specified as boundary conditions. The ventilation flow rate at the orifice is 0.001 kg/s for the simulated and experimental model. Inside the space, the maximum air temperature deviation is -0.16 degrees Celsius and the root mean square temperature error is 0.21 degrees Celsius.

In order to accelerate the convergence of the calculations, first a reasonable and non-zero flow velocity was estimated at the outlet. Then, the velocity at the inlet was calculated with the air mass balance in the space. In the following, the convergence criteria of temperature and pressure 10^{-6} and flow rate 10^{-4} were considered. Also, the flow velocity at the inlet was assumed to be 0.25 m/s, the ambient temperature was 22.4 degrees Celsius, and the ambient pressure was assumed to be 1 atmosphere. Finally, the space was divided into 120 areas (3 (length) × 5 (width) × 8 (height) =120). The results of the three-dimensional simulation in the present study are shown in Figure 5.

As can be seen in Figure 5, the incoming and outgoing currents affect the cells close to it, and the farther a cell is from the incoming or outgoing current, the weaker the current is. For example, in cells close to the $x = 0$ axis, there is almost zero current (compared to other cells). Also, it can be seen that the flow on the side walls is almost zero, which shows a good prediction of the flow state. For a better investigation, two-dimensional slices of the simulated volume are needed. In order to better investigate, a section of Figure 5 was selected at the center of the room and the place where the flow enters and exits and was compared with the results of the research done by Lu et al. [9].

As can be seen in Figure 6, the behavior of the flow near the walls does not fully match the reality. In the following, the results of the previous research are shown in Figure 7 in the form of two-dimensional cuts.

As can be seen in Figure 7, the flow behavior near the walls is in better agreement with the existing reality. For example, at height $Z < 0.5$ and length $X < 0.1667$, the differences in different levels of the flow are well defined and the vortexes created by the flow can be seen. In the following, the ambient temperature changes according to the height from the floor of the test room are as follows:

As can be seen from Figures 8 and 9, the pattern of temperature changes in terms of height obtained in this research has a similar behaviour to what was mentioned in the research and reached from approximately 25°C to 40°C. The difference in the graphs (which is mainly due to the linearity and existing fractures) is due to the difference in the number of considered meshes and numerical solution iterations, and both graphs follow the same overall behaviour.

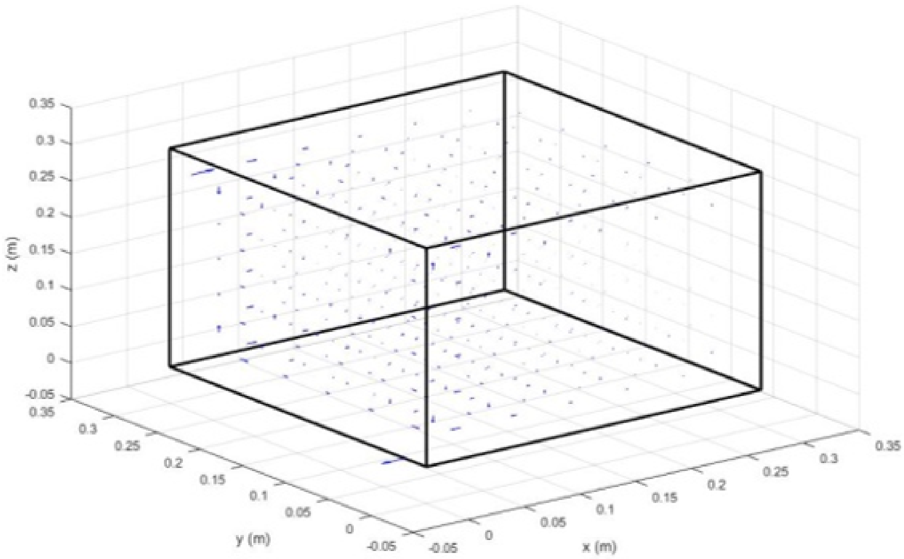

Figure 5: A view of the three-dimensional simulation of the flow with real values in the proposed model

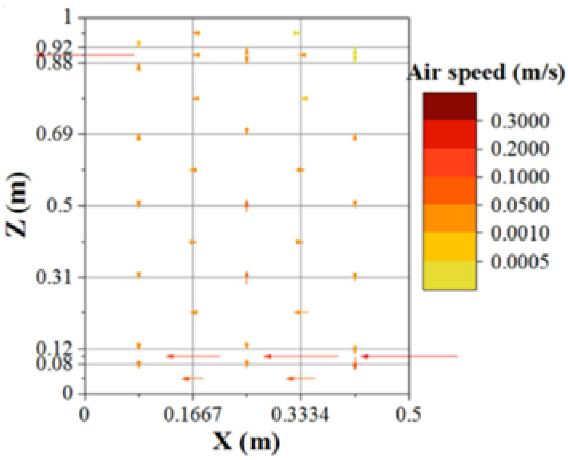

Figure 6: The flow pattern in a two-dimensional cut in the research of Lu et al. [9]

3.1 Investigating the effect of flow coefficient

In the following, the flow coefficient parameter is discussed and the behaviour of the model in the flow coefficients between 0.2 and 1 has been checked in order to obtain the most optimal coefficient for the number of suitable cells. In this regard, Figure 10 shows a section of the room space in the direction of the intersection of the inlet and outlet of the flow for two different values of the flow coefficient and it is enlarged to show its important parts.

According to the results obtained from Figure 10, increasing the flow rate for the inlet and outlet sections has resulted in opposite results. This means that at the entrance, with the increase of the flow coefficient, the flow rate has also increased; While this phenomenon has been a decrease in output. Practically, a higher flow coefficient leads to a higher flow rate in cells further away from the inlet and outlet. With the obtained results, the importance of this coefficient in flow modeling is shown and it corrects the statement that: "increasing the flow coefficient necessarily increases the flow rate".

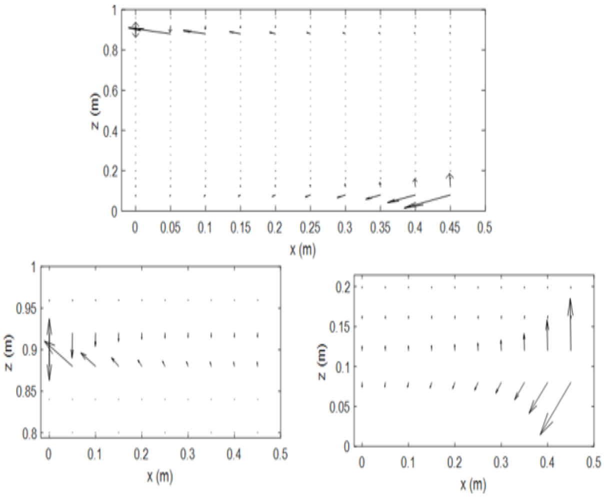

Figure 7: The flow pattern in a two-dimensional cut with the method introduced in this the present research

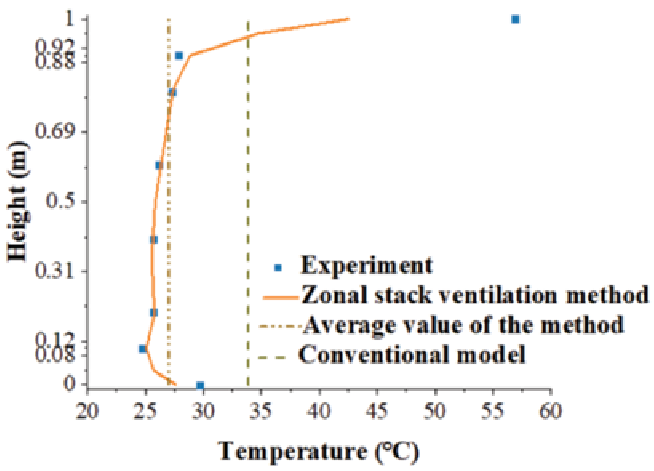

Figure 8: The pattern of temperature changes according to the height in [9]

4 Discussion and conclusion

The study of convection in an indoor place with natural convection is important, and large internal vents play an important role in heat transfer between rooms and even in indoor air quality and thermal comfort. ZAER three-dimensional zone model has been developed for this purpose. Comparing the results between the ZAER model, the Wurtz zone model and experimental measurements for the studied sample, which has a natural connection, showed that the ZAER model is a suitable tool for evaluating the layering of temperature and air flow between and inside the rooms; While, the time required for its calculations does not exceed five minutes. Further, the investigations determined that the distribution of pressure and speed of air flow from large internal valves depends on the selection of C_d coefficients. This research showed that it is possible to significantly improve the results of air speed and temperature by choosing variable and non-constant flow coefficients for each of the cells. With the above interpretations, the improved model in this research can be introduced as an optimized zone model.

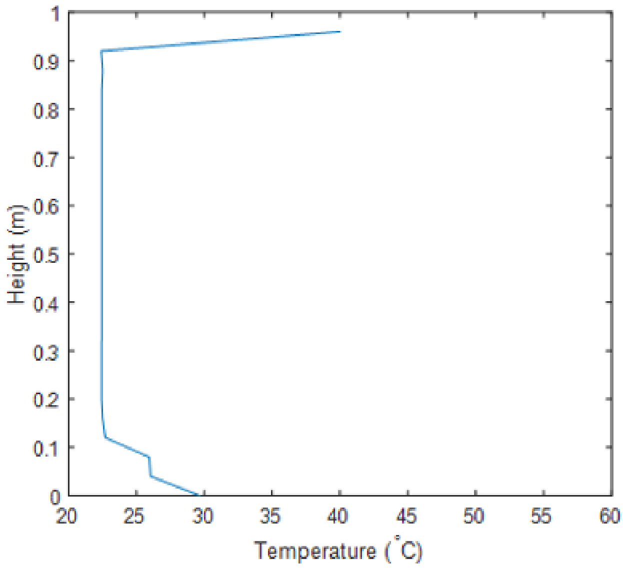

Figure 9: The pattern of temperature changes according to height in the present study

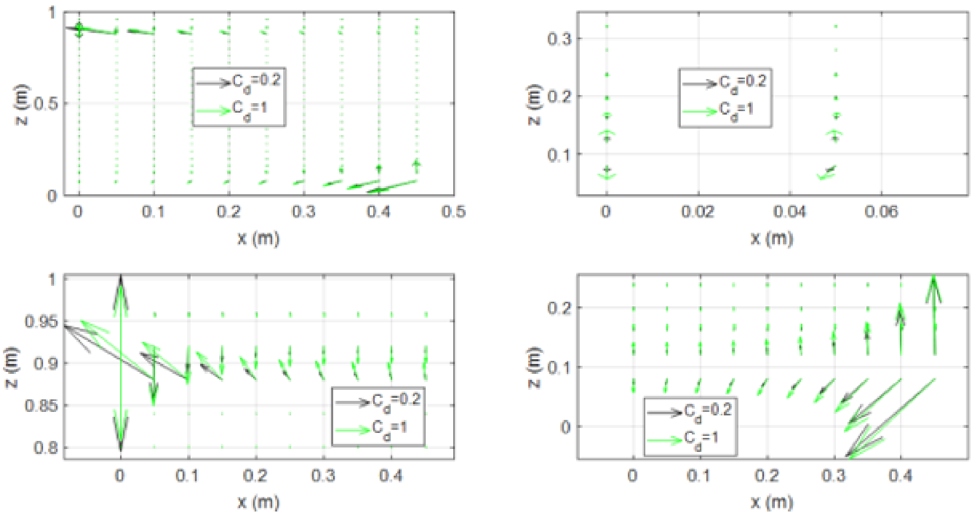

Figure 10: Comparison of flow pattern $C_d = 1$ and $C_d = 0.2$

5 Symbols

i	Density (kg/m3)
C_d	Flow coefficient
T_i	Temperature in cell i (℃)
V	speed (m/s)
X	length (m)
S	area (m^2)
Y	width (m)
P	Pressure (Pa)
Z	height (m)

References

[1] F. Allard and C. Inard, *Natural and mixed convection in rooms: Prediction of thermal stratification and heat transfer by zonal models*, Proc. ASHRAE Conf. Room Air Convect. Ventil. Effect., Tokyo, Japan, 1992, pp. 335–342.

[2] Y. Boukhris, L. Gharbi, and N. Ghrab-Morcos, *Simulating air flow, with a zonal model, for natural convection*, Int. J. Ventil. **7** (2008), no. 3, 207–219.

[3] C.G. Broyden, *A class of methods for solving nonlinear simultaneous equations*, Math. Comput. **19** (1965), no. 92, 577–593.

[4] L. Gharbi, N. Ghrab-Morcos, and J.J. Roux, *ZAER: A zonal model for heat transfer and air flow in unconditioned buildings.- An experimental validation*, Int. J. Ventil. **3** (2004), no. 1, 11–20.

[5] F. Haghighat and A.C. Megri, *A comprehensive validation of two airflow models-COMIS and CONTAM*, Indoor air **6** (1996), no. 4, 278–288.

[6] H. Hu, H. Wang, Z. Zou, and J. Zhu, *Investigation of comparison of two stratified air-conditioning systems*, Energy Build. **254** (2022), 111602.

[7] B. Lan, Z.J. Yu, P. Zhou, and G. Huang, *Optimal zoning for building zonal model of large-scale indoor space*, Build. Envir. **225** (2022), 109669.

[8] X. Li, W. Wu, and C.W.F. Yu, *Energy demand for hot water supply for indoor environments: Problems and perspectives*, Indoor Built Envir. **24** (2015), no. 1, 5–10.

[9] Y. Lu, J. Dong, Z. Wang, Y. Wang, Q. Wu, L. Wang, and J. Liu, *Evaluation of stack ventilation in a large space using zonal simulation and a reduced-scale model experiment with particle image velocimetry*, J. Build. Engin. **34** (2021), 101958.

[10] Y. Lu, J. Dong, and J. Liu, *Zonal modelling for thermal and energy performance of large space buildings: A review*, Renew. Sustain. Energy Rev. **133** (2020), 110241.

[11] Y. Lu, Z. Wang, J. Liu, and J. Dong, *Zoning strategy of zonal modeling for thermally stratified larg spaces*, Build. Simul. **14** (2021), 1395–1406.

[12] A.C. Megri, *Building load and energy simulation programs and the design process*, Int. J. Ventil. **6** (2007), 177–192.

[13] A.C. Megri and F. Haghighat, *Zonal modeling for simulating indoor environment of buildings: Review, recent developments, and applications*, Hvac & R Res. **13** (2007), no. 6, 887–905.

[14] A.C. Megri and Y. Yu, *New calibrated zonal model (POMA) for temperature and airflow predictions*, Build. Envir. **94** (2015), 109–121.

[15] G. Meurant, *Computer Solution of Large Linear Systems*, Elsevier, 1999.

[16] R. Teodosiu, D. David, C. Teodosiu, G. Rusaouën, V. Ilie, and C.-K. Nguyen, *Experimental and numerical investigation of mechanically ventilated rooms*, Energy Procedia **112** (2017), 252–260.

[17] US Department of Energy, *Energy Plus™ version 9.3.0 documentation –engineering reference*, US Department of Energy, 2020.

[18] H. Yamasawa, T. Kobayashi, T. Yamanaka, N. Choi, M. Cehlin, and A. Ameen, *Prediction of thermal and contaminant environment in a room with impinging jet ventilation system by zonal model*, Build. Envir. **221** (2022), 109298.