International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 10 Issue 12 ǁ December 2022 ǁ PP. 151-157

Thermal comfort evaluation of a theater with natural ventilation by CFD simulation

Zhongwei Huang, Siyi Hou, Yang Liu*
School of Architectural and Surveying & Mapping Engineering, Jiangxi University of Science and Technology, Ganzhou, China

Abstract
To investigate the flow field features of natural ventilation in the theater, a three-dimensional simulation model of the theater was established. The distribution of the velocity field and inside temperature is simulated using the k-epsilon turbulence model. Based on the numerical simulation results, the natural ventilation characteristics of the theater and its influence on indoor thermal comfort and building energy consumption are analyzed. The findings demonstrate that indoor warm air will create thermal stratification in the upper portion of the building area and dead zones at the corner of the structure, leading to poor thermal comfort under natural ventilation conditions. As a result, the theater's seating arrangement for patrons should be as close to the thermal stratification interface as possible.

Keywords:*numerical simulation,natural ventilation, thermal comfort, building energy consumption.*

Date of Submission: 01-12-2022 Date of acceptance: 10-12-2022

I. INTRODUCTION

Indoor air quality (IAQ) has received increasing attention in recent years. Humans spend roughly 80% of their time indoors[1]. The quality of the indoor environment significantly and directly affects public health. During a performance, both the actors and the audience must spend a lot of time in the theater. Sadrizadeh et al. [2] confirmed that improving indoor environmental quality can increase the productivity and concentration of indoor occupant.On the contrary, prolonged exposure to an unpleasant interior environment increases the risk of building syndrome, lassitude, and cognitive deterioration in people[3]. In addition,With the increasing attention to thermal comfort and indoor air quality, many researchers begin to carry out more in-depth and detailed research [4,5]. In comparison to the comfort zone under an air-conditioned facility, the acceptable comfort zone for humans in natural ventilation has a larger temperature range and can save more energy[6]. As a result, it is critical to study the effects of natural ventilation on air quality and interior thermal comfort.

Given the current worldwide trend of reducing energy consumption and carbon dioxide emissions, many architects and building designers are turning to natural ventilation or mixed-mode technology[7,8]. Due to the unexpected nature of natural ventilation systems, much attention must be given during the design process to guarantee that the ventilation strategy will work successfully under predictable climatic and occupancy conditions, and in particular that acceptable air change rates will be reached[9]. In mechanically driven systems, the prediction of these parameters is much easier because the designer can know with a high degree of accuracy the volumetric flow rates produced by the various components that make up the system [10]. However, when designing natural ventilation strategies, the only way to predict flow velocity is through physical or mathematical models [11]. In this paper, the mature commercial software ANSYS-FLUENT is used to calculate the fluid dynamics to realize the simulation of natural convection, and the temperature distribution and air flow characteristics in different areas are obtained, which provides reference for improving the thermal environment quality of the theater.

II. Numerical methodology

Computational fluid dynamics (CFD) has been playing a key role in the analysis and research of natural ventilation flow field. With the rapid development of computer performance, numerical simulation method is widely used in the measurement of indoor environment.Most CFD simulations are performed using the Reynolds-averaged Navier-Stokes equations, which are used to calculate the point dependence of the fluctuating velocities[12]. The momentum equation and the continuity equation are the governing equations in this analysis since they follow conservation laws. The finite volume method is used to obtain the computational grid, generate the computational points, determine the object's boundary conditions, establish the discrete equation, repeat the test until the solution value converges, and finally display and output the calculation results, as shown inFig. 1.

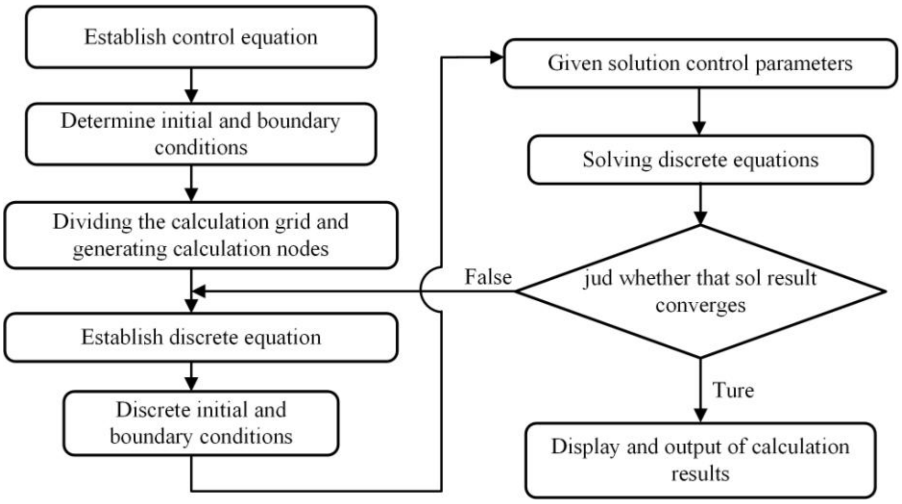


Fig. 1Flowchart for CFD calculation

2.1 Geometrical considerations

To acquire the interior temperature and flow field distribution as similar to the real building size is possible, we built a theater in Spaceclaim with size specifications of 15m×12m×4.6 m, with the theater separated into two parts: stage and auditorium. Fig. 2 depicts the geometric model's exterior contour, internal structure, and Cartesian coordinate system orientation. we can see the dimensions of doors and windows, the width of auditorium steps, etc in Table 1. Mesh generation and simulation were performed in Ansys Fluent R 18.0. The operating boundary conditions are set to velocity inlet and pressure outlet.The wall is defined as a no-slip boundary condition.

Table 1: Geometric dimension of model.

Serial No.	Part name	XDirectionlength inm	YDirectionlength inm	ZDirectionlength inm
1	Theater	15	12	4.6
2	Front door	0.1	2	1
3	Back door	1	2	0
4	Step	0.8	0.18	4.6

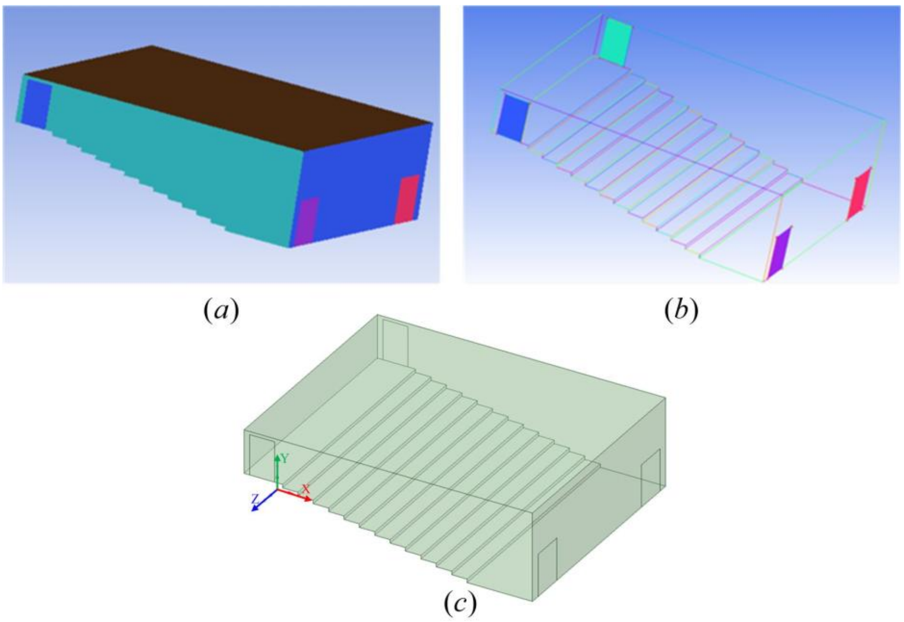
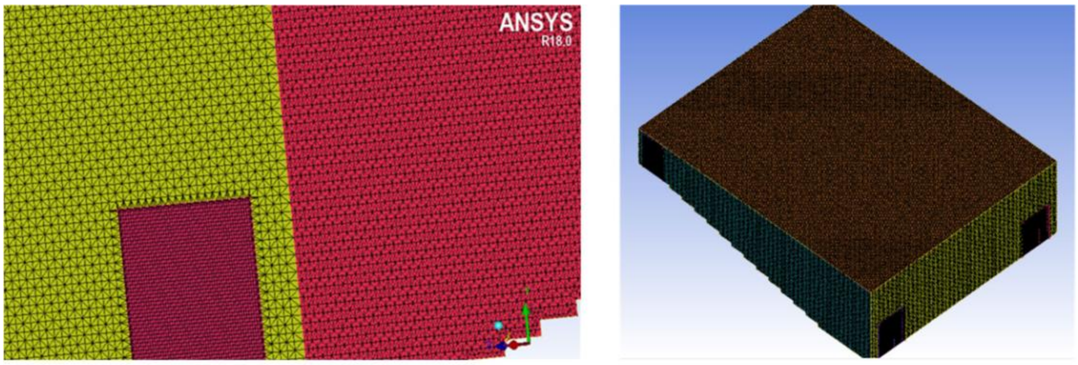


Fig. 2Theatre model. (*a*) is exterior contour, (*b*) is internal structure, (*c*) is coordinate orientation

2.2 Meshing

For meshing the theater ANSYS Workbench is used, and the solver preference is selected as Fluent. The theater model is relatively regular and is completely carried out by using structured grids. Local refinement treatment is performed during the division process around regions with strong flow changes (such as windows and doors), as shown in Fig. 3(*a*), to capture the possible boundary layer flow phenomena. The grid of the entire theater area is illustrated in Fig. 3 (*b*), and the number of grid nodes in the calculation domain exceeds 1.53 million. Because the study focuses mostly on interior flow characteristics, the present number of grid nodes match the calculation criteria.



(*a*) (*b*)

Fig. 3mesh generation result. (a) display details of local mesh refinement , (b) display the overall situation of the grid

2.3Mathematical Model

2.3.1 governing equations

For incompressible flow, the governing equations describing the flow and heat transfer can be expressed by the following equations[13]:

Continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

Incompressible flow equation:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = f_i - \frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(v \frac{\partial u_i}{\partial x_j} \right) \tag{2}$$

The Reynolds time average method is mainly the ensemble average of the physical parameters in the flow field, expressed as the sum of the time mean and the pulsation value:

$$f = \overline{f} + f' \tag{3}$$

The parameters in the flow field are expressed as the representation of equation (3) above, and then brought into the flow control equation system represented by (1) and (2), and the average Reynolds time equation for the average motion of the incompressible fluid is simplified by simplifying the time-averaging characteristics:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{4}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = \overline{f}_i - \frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + v \frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} + \frac{1}{\rho}\frac{\partial(-\rho \overline{u'_i u'_j})}{\partial x_j} \tag{5}$$

Based on the preceding equation, Reynolds stress varies with pulse-value velocity and is nonlinear. The number of control equations remains at four after ensemble averaging, but the unknown quantity is raised to five owing to Reynolds stress. To solve the system of equations numerically, the Reynolds stress must be expressed in a known number and a new equation must be added to the system of equations to make it closed and solvable.

2.3.2 Turbulence model

To tackle the problem of closure of the time-homogenized turbulent flow control equation, a specific study of the newly added unknown Reynolds stress factor is required. Because there is no exact theory to address the turbulence problem and only certain approaches can be utilized to approximate the assumption of modifying Reynolds stress, the newly added Reynolds stress term is a reflection of turbulent pulsation features. The vortex viscosity theory developed by Boussinnesq is the most typical of the various approximation modulation approaches[14]. The phrase is as follows:

Boussinnesq approximation:

$$-\rho\overline{u_i^{'}u_j^{'}} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t\frac{\partial u_i}{\partial x_j}\right) \tag{6}$$

In equation (6), the eddy viscosity coefficient is a new unknown quantity. One way to solve this problem is to use the time-averaged parameters of turbulent flow to express the eddy viscosity coefficient, so as to introduce a new expression and equation, namely the so-called turbulence model.The RNG two-equation turbulence model is better than the other turbulence models in predicting the flow with strong swirl and curved wall.

2.4 Numerical Calculation

2.4.1 Initial Condition

The simulation of theater natural ventilation in winter can be summarized as a three-dimensional steady heat conduction problem of incompressible gas. The air distribution flow is considered as turbulent flow and is solved by k-ε Realizable two-equation model.In order to simplify the simulation calculation, the following assumptions are made for the physical model: 1) normal temperature, low speed, incompressible fluid flow; 2) isobaric flow in accordance with the equation of state of gas; 3) Boussinesq approximate; 4) The influence of outdoor temperature, humidity and wind direction on indoor air flow is not considered.SIMPLE algorithm is used to solve the coupled pressure and velocity equations. The pressure term of the governing equations is discretized using a second-order upwind scheme.In order to ensure the convergence and stability of the solution of nonlinear equations, the convergence residuals of all physical quantities are set to 10^{-4}.

2.4.2 Boundary condition

In the calculations of this study, the following boundary conditions were used: Boundary condition of inlet is given velocity inlet, boundary type is set as Velocity Inlet, velocity is 1.5m/s, turbulence intensity *I*=5%, turbulent kinetic energy dissipation rate is 10, inlet temperature is set as 0℃;Outlet boundary condition: according to the setting principle of CFD boundary condition, when velocity inlet condition is given, pressure outlet boundary condition shall be given to facilitate the convergence of numerical calculation. In this calculation, the inlet is regarded as the outlet, Stactic Pressure, relative pressure is given as 1000Pa, turbulence intensity *I*=5%, and turbulent kinetic energy dissipation rate is 10.Solid wall boundary condition: all the walls are set as smooth non-slip solid wall boundary condition; The heat flux of the wall is set at 5000w/m^2.Setting of fluid and solid physical characteristics is shown in Table 2.

Table 2 physical parameter

Serial No.	Parameter name	*I*(A)
1	Density of wall material(kg/m^3)	1000
2	Specific heat capacity of wall material (J/kg·K)	970
3	Heat conductivity of wall material [W/(m·℃)]	1.7
4	Density of glass material (kg/m^3)	2500
5	Specific heat capacity of glass material (J/kg·K)	840
6	Heat conductivity of glass material [W/(m·℃)]	0.96
7	Density of wood material (kg/m^3)	730
8	Specific heat capacity of wood material (J/kg·K)	2310
9	Heat conductivity of wood material [W/(m·℃)]	0.147
10	Glass emissivity	Internal 0.96, external 0.96
11	Concrete emissivity	Internal 0.7,external 0.6

III. Simulation results and discussion

3.1 Numerical Calculation

The geometric model of the theater is imported into the CFD calculation application, where the numerical computation is finished when the grid creation is finished in the pre-processor. A fully implicit multi-grid linked iterative method is employed in this study. The parameter residual iteration results are displayed in Fig. 4when the residual value of each parameter is less than the error limit.

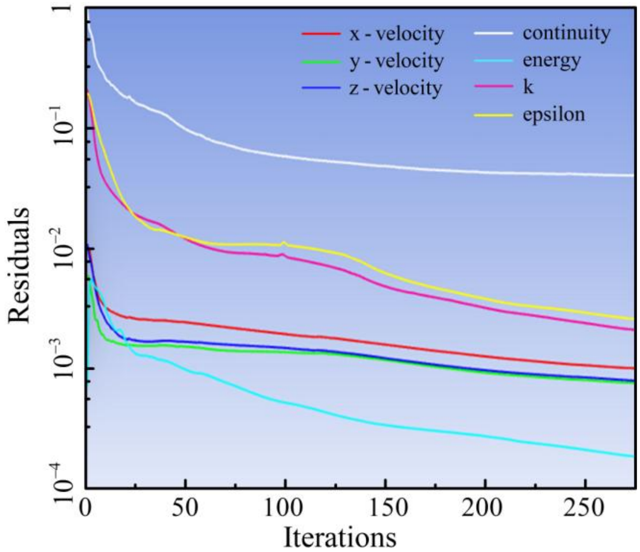


Fig. 4 The residual iterative computation reaches a point of convergence

3.2 velocity field analysis

The flow field of the entire theater was selected for study, as shown in Fig. 5 (*a*), and the streamlines were obtained. There is a big volume of cold air pouring in at the two inlets, and the speed reduces dramatically after entering the vast space, and there is a massive vortex at the lowest level of the theater, and only a little percentage of the air goes out of the outlet. Fig. 5 (*b*)depicts a velocity vector diagram for the whole fluid domain, indicating the amount and direction of the velocity, with the general velocity revolving clockwise but with minor regions of turbulence at the theater's corners and heating.Figure 5.3 depicts the velocity pattern on the segment at 0.5m height. The wind speed throughout the entire theater is around 0.16m/s, but at the open window, the speed clearly increases to approximately 0.7m/s, and the influence domain of this flow extends to the middle of the theater due to the viscous action of air, resulting in many vortex forms there. However, the wind speed in the centre of the theater is often lower than in the surrounding region, and the comfort level is definitely higher.

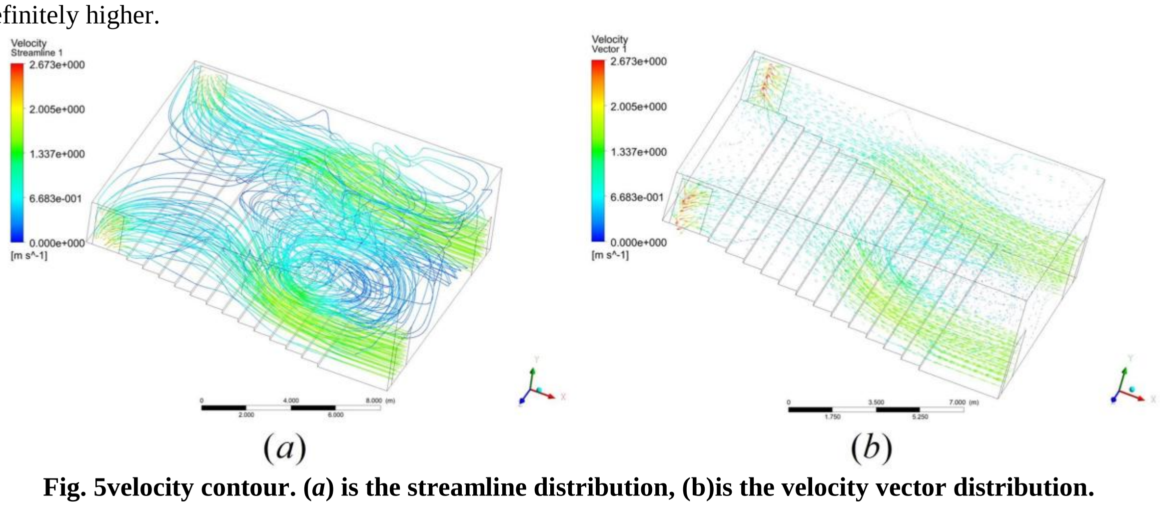


Fig. 5velocity contour. (*a*) is the streamline distribution, (b)is the velocity vector distribution.

Fig. 6 show the velocity distribution at different heights. As the heater heats the air, causing thermal expansion and contraction, the air density near the heater decreases and rises, resulting in higher velocity here.It is also observed that the maximum velocity of air flow decreases from 1.02m/s to 0.73m/s with the increase of height, that is to say, the air flow at the top of the theater is weak. From the longitudinal section, this arrangement faces the problem that the ventilation in the middle of the theater is not smooth enough, and the natural wind does not spread to the whole room well.

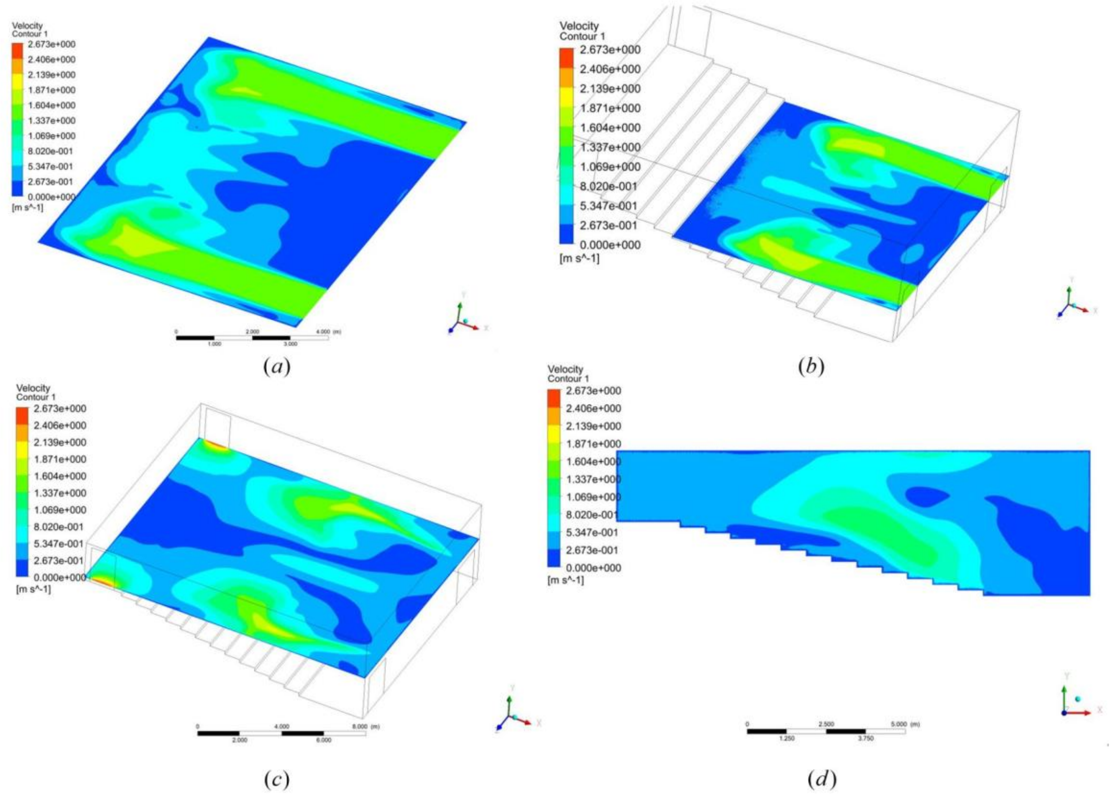


Fig. 6Sectional velocity field at different positions.

3.3temperature field analysis

Analyzing the temperature distribution at various theater heights, it becomes clear that the temperature at the theater's entrance is roughly 10°C lower than that in the rear row's high ladder position. Fig. 7 (*a*) shows the diffusion and circulation of high temperature gases throughout the theater due to natural convective heat transfer. The theater design is reasonable since it can be shown that the temperature distribution is roughly even, at around 21°C, and that it is roughly the same at the 1 m, 1.5 m, and 2.5 m sections.It can be clearly observed from Fig. 7 (*b*) that the temperature of the lower part of the whole theater is obviously lower than that of the upper part, about 5℃ lower, and the temperature on the side of the theater with heating is also several degrees higher than that on the side without heating. The occupant’s seats can be arranged according to this structure.

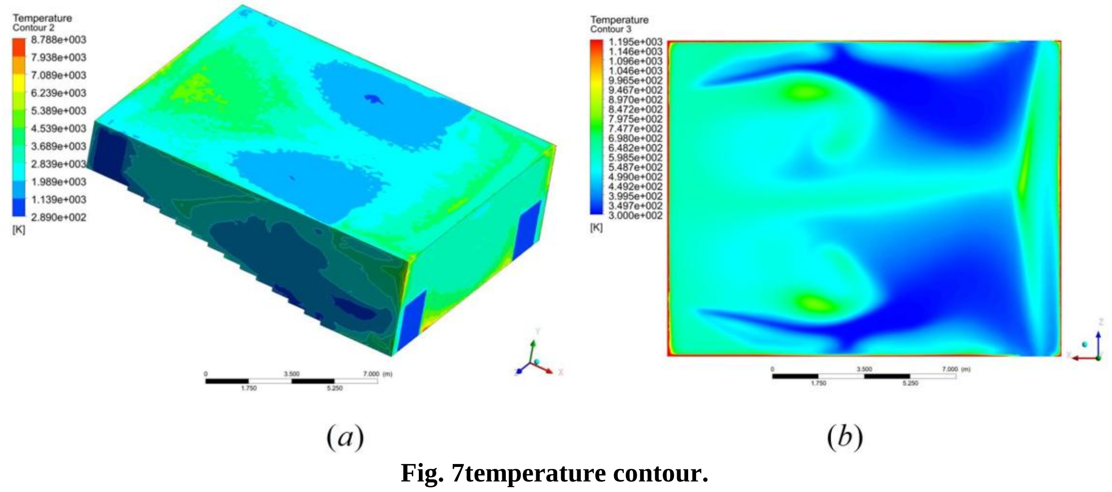


Fig. 7temperature contour.

IV. CONCLUSION

Due to the high frequency of use in business places such as theaters, the air distribution of the auditorium is suitable to adopt low air supply mode to ensure that the majority of the space can maintain good indoor environment and thermal comfort, so as to meet the physical health and inner satisfaction of indoor occupants.

(1) The distribution of indoor temperature field is directly related to the wind speed.
(2) Natural ventilation promotes the diffusion of air with varying temperatures while also lowering the temperature in the area around the door.
(3) Although the theater's interior can often be kept at a comfortable temperature, insufficient ventilation at the back may cause lower oxygen levels and audience tiredness.
(4) The temperature around theater is slightly lower than that in the middle, and the temperature near the ground is slightly lower than that in the upper part of the theater, resulting in uneven spatial distribution of temperature.

REFERENCES

[1]. Chen C, Zhao B (2011) Review of relationship between indoor and outdoor particles: I/O ratio, infiltration factor and penetration factor. Atmospheric Environment 45(2): 275-288.
[2]. Sadrizadeh S, Yao R, Yuan F et al (2022) Indoor air quality and health in schools: A critical review for developing the roadmap for the future school environment. Journal of Building Engineering 57: 104908.
[3]. Sakiyama NRM, Carlo JC, Frick J et al (2020) Perspectives of naturally ventilated buildings: A review. Renewable and Sustainable Energy Reviews 130: 109933.
[4]. Bhagat RK, Davies Wykes MS, Dalziel SB et al (2020) Effects of ventilation on the indoor spread of COVID-19. J Fluid Mech 903.
[5]. Ahmed T, Kumar P, Mottet L (2021) Natural ventilation in warm climates: The challenges of thermal comfort, heatwave resilience and indoor air quality. Renewable and Sustainable Energy Reviews 138: 110669.
[6]. Jia L-R, Han J, Chen X et al (2021) Interaction between Thermal Comfort, Indoor Air Quality and Ventilation Energy Consumption of Educational Buildings: A Comprehensive Review. Buildings 11(12): 591.
[7]. Raji B, Tenpierik MJ, Bokel R et al (2020) Natural summer ventilation strategies for energy-saving in high-rise buildings: a case study in the Netherlands. Int J Vent 19(1): 25-48.
[8]. Bamdad K, Matour S, Izadyar N et al (2022) Impact of climate change on energy saving potentials of natural ventilation and ceiling fans in mixed-mode buildings. Build Environ 209: 108662.
[9]. Chen C-M, Lin Y-P, Chung S-C et al (2022) Effects of the Design Parameters of Ridge Vents on Induced Buoyancy-Driven Ventilation. Buildings 12(2): 112.
[10]. Ward J, Noel C (2022) Basic Modes of Mechanical Ventilation. Emergency Medicine Clinics of North America 40(3): 473-488.
[11]. Baeumle R, Hunt GR (2018) Capturing the needs of architects: a survey of their current information requirements for natural ventilation design. Int J Vent 17(2): 120-147.
[12]. Maulik R, Sharma H, Patel S et al (2021) A turbulent eddy-viscosity surrogate modeling framework for Reynolds-averaged Navier-Stokes simulations. Computers & Fluids 227: 104777.
[13]. Motamedi H, Shirzadi M, Tominaga Y et al (2022) CFD modeling of airborne pathogen transmission of COVID-19 in confined spaces under different ventilation strategies. Sustainable Cities and Society 76: 103397.
[14]. Gray DD, Giorgini A (1976) The validity of the boussinesq approximation for liquids and gases. International Journal of Heat and Mass Transfer 19(5): 545-551.