NUMERICAL SIMULATIONS OF THE NATURAL VENTILATION DESIGN MIX OF ELEMENTARY SCHOOL CLASSROOMS

Hsin-hung Wu[1], Fu-Jen Wang[2], Che-ming Chiang[1] and Chi-ming Lai[3*]

[1] Department of Architecture, National Cheng-Kung University, Taiwan
[2] Department of Refrigeration and Air-Conditioning Engineering, National Chin-Yi University of Technology, Taiwan
[3*] Graduate Institute of Rural Planning, National Chung-Hsing University, Taiwan
250, Kuo-Kuang Road, Tai-chung City, 402, Taiwan, China
e-mail: lcm@dragon.nchu.edu.tw fax: +886-4-22878988

ABSTRACT

Most traditional elementary school classrooms here do not have air-conditioners and it often feel stuffy in classrooms located in the top floors. In addition, traditional classrooms widely have noise problem, so do the open educational classrooms. If the windows and doors are closed to reduce noise, it will result in the phenomenon of stuffiness, which largely prevents the external air entering the room. Therefore, it has become an urgent matter for building designers and educational administrators about how to solve the problem of ventilation as well as thermal comfort of elementary school classroom.

CFD (Computational Fluid Dynamics) simulation method was conducted to investigate how the 6 natural ventilation design mix (Double Roof, CRSO (Covered Ridge with Sidewall Opening), roof ridge opening and window opening) influence indoor thermal environment at summer season and low outdoor wind-speed. Moreover, this research discussed how to use these designs to improve the natural ventilation efficiency of classroom. The results demonstrated that when collocating with Double Roof, roof ridge opening and CRSO, both ventilation efficiency and thermal comfort of the room are superior to other cases, no matter the room adapt staggered or conventional window layout. The staggered window openings can increase the buoyancy ventilation effect, compared with conventional window openings, they can increase the air change rate about 1.21 (1/h), equivalently increase by 1.5 times more.

KEYWORDS

classroom, natural ventilation, CFD (Computational Fluid Dynamics), CRSO (Covered Ridge with Sidewall Opening), double roof

INTRODUCTION

Indoor air quality in Taiwan is poor due to congested living spaces, highly airtight buildings, poor air circulation and lack of ventilation. This problem has been receiving increasing attention recently. Furthermore, many typical indoor pollutants (oil-aerosol, water vapor, carbon monoxide, nitrogen oxides and volatile organic compounds) are found in Taiwanese buildings, owing to local cooking and showering customs. Properly inducing natural ventilation can significantly improve indoor air quality, while also decreasing reliance on air-conditioning, thus cutting energy consumption. This field of environmental design engineering requires further research. In February of 2003, the SARS disease spread rapidly in the Tropical Regions of Asia, which created an impact on the global society and economy. Human people began to reflect the content regarding building science. Since people spend much more time indoors than outdoors, the nature of architectural design should return to HUMAN application. Therefore, it is very important to make relative studies on indoor environment science for people who long-time stay indoors.

Most elementary school classrooms are not equipped with air-conditioners. Classrooms, located in the upper floors, are always stuffy. In addition, traditional classrooms widely have noise problem, so do the open educational classrooms. If the windows and doors are closed to reduce noise, it will result in the phenomenon of stuffiness, which largely prevents the external air entering the room. Therefore, it has become an urgent matter for building designers and educational administrators about how to solve the problem of ventilation as well as thermal comfort of elementary school classroom.

The objective of the present study is to provide insights into how the 6 natural ventilation design mix (Double Roof, CRSO (Covered Ridge with Sidewall Opening), roof ridge opening and window opening) influence indoor thermal environment at summer season and low outdoor wind-speed, considered for which little or no information is available. In doing so, numerical simulations via a finite volume method have been performed for the steady three-dimensional turbulent flow phenomena induced by outdoor wind and heat sources (residents' heat dissipation).

BACKGROUND

Natural ventilation can be defined as the air exchange caused by indoor and outdoor pressure gradient, which make indoor and outdoor air pass through opening, such as doors and windows. The main function of natural ventilation is to remove or dilute indoor contaminant and provide acceptable indoor air quality and heat transfer mechanism. Moreover, since fine natural ventilation can reduce the energy consumption of air-conditioner, it must be take into consideration about natural ventilation at the primary stage of the development of architectural design (CIBSE, 1997). The thermal mechanism of natural ventilation includes free convection and forced convection. The driving force of free convection comes from gravitational force, that is to say, the temperature difference between indoor air and outdoor air (Stack effect or buoyancy effect); while the driving force of forced convection comes from pressure and shearing stress, that is to say, external air pressure and mechanic fans (Etheridge and Sandberg, 1996). In the past, the content of natural ventilation related researches here in Taiwan mainly include: studying the influence of the position and size of windows upon natural ventilation, the relationship between ventilation route and air exchange efficiency, the influence of the location of furniture (for example, bed) upon the natural ventilation performance, the influence of widowing type upon the natural ventilation performance, and so on. The analyzed space includes bedroom and kitchen. Research methods mainly divide into full scale experiment and CFD numerical analysis to study the air flow field, contaminant concentration distribution, and ventilation (or air exchange) indexes, such as contaminant concentration, ACH (Air exchange rate per Hour), Age of air, AEE (Air Exchange Efficiency). The result of domestic researches showed that methods of enhancing natural ventilation mainly contain building skin design, raised floor ventilation, wind scoop and wind tower, and double roof design.

METHOD

Classroom under investigation

According to researches regarding the moduli of house and school by Building Research Institution, Ministry of the Interior, Taiwan, the most common construction of the elementary school classrooms are reinforced concrete, with classroom length of 9 meters and width of 7.2-7.8 meters. Thus, this study established investigated classroom with the size of 9 ×7.5 meters. According to the domestic Building Codes regarding classroom space, height of the overhang is about 3 meters. According to a survey on the window designs of elementary school classrooms, the size of each window is commonly 1.2m(height) × 1.2m(width), which can provide a ventilation opening of 1.2m(height) × 0.6m(width). The other dimensional parameters specified in this study are shown in Fig. 1.

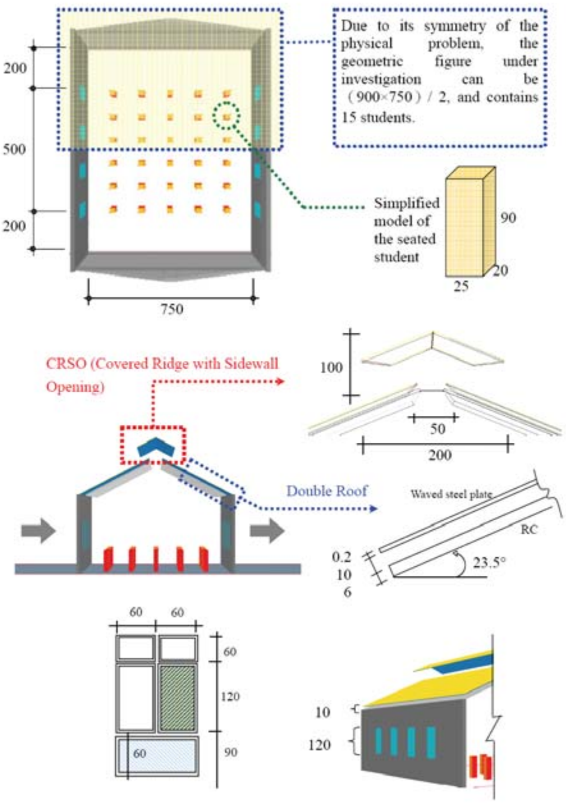
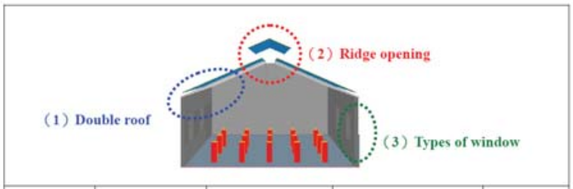


Fig.1 Schematic diagram of the investigated classroom (Unit: cm)

Physical Problem

CFD (Computational Fluid Dynamics) simulations were carried out to investigate how the 6 natural ventilation design mix (double roof, CRSO (Covered Ridge with Sidewall Opening), roof ridge opening and windows) influence indoor thermal environment at summer season with low outdoor wind velocity. Content of the 6 mix are illustrated in Table 1. Every condition is the same, except the controlled variables marked in Table 1: (1) Whether it has a double roof; (2) Whether the Ridge opening is opened; (3) types of window. Furthermore, the thermo-physical properties of the air are temperature independent, except for the density, for which the Boussinesq approximation is valid.

Table 1 Configurations for numerical simulation（No means not open/operate）



Mode	(1) Double roof	(2) Ridge opening	(3) (Types of) window	Illustrative
Basic Mode	No	No	No	
Mode 1	No	Yes	No	
Mode 2	Yes	No	No	
Mode 3	Yes	Yes	No	
Mode 4	Yes	Yes	Yes (parallel)	
Mode 5	Yes	Yes	Yes (staggered)	

Numerical Method

Numerical simulations of the physical problem under consideration have been performed via a finite volume method for solving the governing equations and boundary conditions mentioned above. This study applies the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm (Patankar, 1980) to solve those equations. The "two equation model" of turbulence, the k-epsilon model (Launder and Spalding, 1974), is adopted. To bridge the steep dependent variable gradients close to the solid surface, the "general wall function" is employed. The iteration calculation was continued until a prescribed relative convergence of 10^{-3} is satisfied for all field variables of this problem. A nearly 166×61×120 grid system is employed for the present calculations.

Reduced-scale Experiment

In this study, upper plate of the double roof is a wave-shaped steel plate, commonly used in the construction industry in Taiwan. Its waved shape and complex heat transfer coefficient makes it quite inconvenient when setting up CFD conditions, so first a pre-test has to be conducted. The wave-shaped steel plate is then considered to be a rectangular steel of 2 cm thickness in the CFD simulation process. Results of reduced-scale experiments and CFD simulations were compared repeatedly, with try and error method, to decide the heat transfer coefficient until the error is the slightest. In this way, the best heat transfer coefficient of wave-shaped steel plate was determined, and then specified into the following numerical simulation. The upper plate in the reduced-scale experiments was a wave-shaped steel plate, and lower plate to be reinforced concrete. Those combinations directly mimicked the practical double roof structure of school building. As for the size of the double roof, the length of air flow direction is (along Z axis in Fig.2) 100cm; the lateral length is 60cm (the direction perpendicular to X and Z axis in Fig.2); the width (along X axis in Fig.2) is 10cm. So, the aspect ratio of length and width is 10; aspect ratio of lateral and width is 6; the thermal and flow field can be regarded as a two dimensional (X-Z) phenomena.

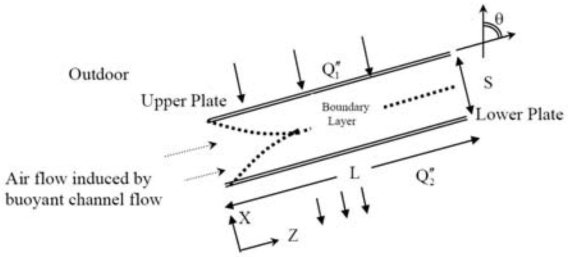
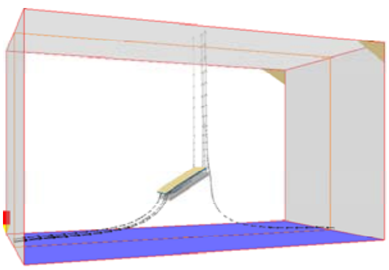


Fig.2 Physical model of the double roof prototype for pre-tests

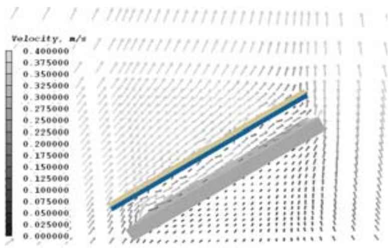
The slope of prototype in the reduced-scale experiments is 30°. To avoid influence of changeable solar radiation, tungsten halogen lamp without infrared reflection coating was used. The spectrum curve of the tungsten halogen lamp without infrared reflection coating shows that its electromagnetic wavelength, ranging from 0.38 to 3 μm, almost has the same shape as vertical solar radiation. Therefore, the 6 tungsten halogen lamps of 500W/2A/220V with no infrared reflection coating was used to simulate solar radiation, and by adjusting the interval space among lamps and their distance from upper plate, then the average solar radiation was about 600w/m^2, as shown in Fig. 3. TESTO 445 Multi-functional ventilation/air-conditioning detectors equipped with two sensing connectors were used to detect various environmental factors, such as wind velocity, temperature and humidity. The data were analyzed with an RS 232 transmitting line and the professional analyzing software ComSoft 3 (Testo 0554 0830) on the Windows platform.

Fig.3 Reduced-scale experiments

CFD simulations result shows that temperature of the upper plate (wave-shaped steel plate) increases after it receives solar radiation. The adjacent air under the upper plate is then heated and creates buoyant force, which makes the air, originally at 0m/s, in the channel flow up along the length-direction and flow out of the channel. The solar heat-gain, penetrating through upper plate, is taken out of the channel through the discharge of air in the channel, so as to reduce solar heat-gain which enters the indoor through lower plate. The result of CFD simulations were indicated in Fig.4 and Fig.5.



(a) Air trajectory



(b) Detailed airflow within the Double Roof

Fig.4 Results of flow field simulation of the pre-test

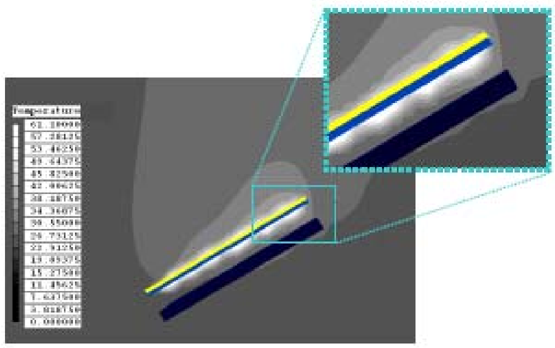


Fig.5 Results of temperature field simulation of the pre-test

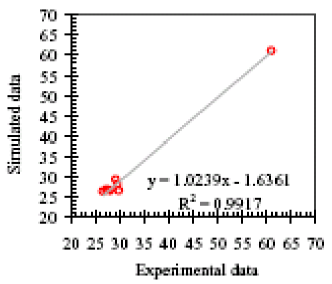


Fig. 6 Comparison of plate temperatures between experimental data and simulated data

When the wave-shaped steel plate was set in the following condition:

Steel at 27 deg C;
Energy source: Adiabatic;
Heat transfer coefficient: 500W/m k;
Thickness: 2cm;

as the relation showed in Fig.6, it is sure that the correlation coefficient between experiment and CFD simulation at each test temperature is 0.9958, and the estimated parameter P value is extremely less than 0.001, which is significant. Therefore, when using the above specifications to do further simulations, the established thermal conditions are reliable.

RESULTS AND DISCUSSION

Design mix's influences on natural ventilation performance are discussed latterly. This section only presented, through CFD simulations, the ventilation rate and average indoor temperature of each design mix as well as the visible result of temperature field and flow field in each design mix. Fig. 7 shows the CFD simulation results of Mode 4. Table 2 shows the analytic result of simulation of each design mix. The temperature field and flow field of each design mix have the following features:

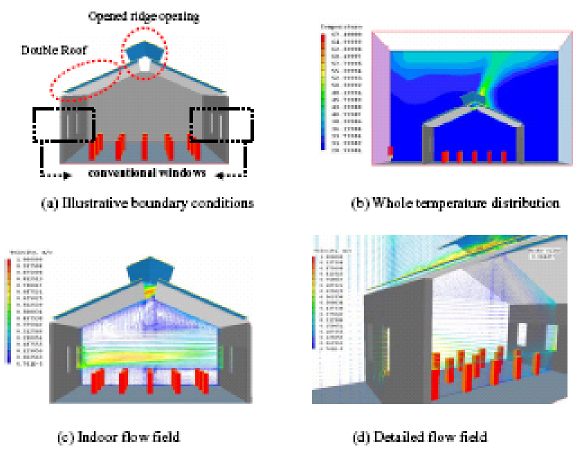


Fig.7 CFD simulation results of design mix Mode 4

Table 2 Analysis of the simulation results of each natural ventilation design mix

	illustration	ACH (h^{-1})	T (℃)	Performance	
Basic Mode	Single roof, Ridge opening closed, windows closed				
		0.03	33.4	Ventilation performance	●○○○○○
				Thermal comfort	●○○○○○
Mode 1	Single roof, Ridge opening opened, windows closed				
		0.05	31.8	Ventilation performance	●●○○○○
				Thermal comfort	●●●○○○
Mode 2	Double roof, Ridge opening closed , windows closed				
		0.68	32.4	Ventilation performance	●●●○○○
				Thermal comfort	●●○○○○
Mode 3	Double roof, Ridge opening opened , windows closed				
		0.78	30.9	Ventilation performance	●●●●○○
				Thermal comfort	●●●●○○
Mode 4	Double roof, Ridge opening opened , conventional window layouts				
		2.22	29.2	Ventilation performance	●●●●●○
				Thermal comfort	●●●●●●
Mode 5	Double roof, Ridge opening opened, high-low window layouts				
		3.43	29.4	Ventilation performance	●●●●●●
				Thermal comfort	●●●●●○

(1) Basic Mode

The flow field is seriously stuffy; outdoor air only can enter the room slightly through door's crack. Because there is no double roof to enhance the solar heat insulation and also no opened windows, solar heat-gain enters the room through RC roof, which make the room over-heated.

(2) Mode 1
The solar heat-gain through the RC roof and human heat dissipation will transfer into the indoor air and discharge through the ridge opening, due to the buoyancy effect. Thus, compared with the Basic Mode, ventilation performance and indoor thermal environment both improve in this design mix.

(3) Mode 2
After installing a double roof, indoor solar heat-gain through roof decreases. However, because the ridge is closed and the windows are closed, indoor heated air could not discharge successfully. As a result, the average indoor temperature decreases slightly, but ventilation performance is not adequate very well.

(4) Mode 3
After the upper plate and adjacent air in double roof channel are heated by solar radiation, there exists thermal buoyancy effect. Air flow in the channel, toward outside, then exists near CRSO. Along with the outdoor flow blowing through CRSO, the two flows (air out of the channel and outdoor wind blow) can induce indoor air discharge through CRSO opening. Thus the whole room ventilation quantity increases.

(5) Mode 4
Owing to the wind force and thermal buoyancy effect, a good indoor flow field can be achieved. The indoor air can discharge simultaneously through ridge opening and windows, it is obvious that outdoor air can blow into the room through the windows, and then a part of air flow can directly discharge from windows on the other side. Thus, the there is some short cut phenomenon, which can be seen clearly from this figure. The other part of air flow rise along the other-side wall to the ridge opening and is introduced out of the room by thermal buoyancy flow and outdoor air passing the CRSO.

(6) Mode 5
Owing to the wind force and thermal buoyancy effect, there is a good indoor flow field in Mode 5. Heated air can discharge simultaneously through ridge opening and high windows. Staggered open windows can make the thermal buoyancy ventilation performance better.

CONCLUSION

From the result of CFD simulations for 6 natural ventilation design mixes, the air exchange rate and the relative air exchange increasing of each design mix with summer low air velocity of 0.1m/s (thus, indoor flow field was influenced significantly by the thermal buoyancy effect) are shown as Fig. 8. The indoor average temperature and relative percentage of each design mix are shown as Fig. 9. It is shown that Basic Mode (single roof without CRSO) is the worst mode regarding natural ventilation and thermal environment. Designs such as double roof, CRSO, roof ridge opening and window opening can be incorporated to improve the indoor phenomenon of stuffiness and poor natural ventilation. Mode 3 (with closed windows) can be fabricated with staggered windows to become Mode 5, which can increase air exchange of 2.65 h-1 (equivalently to 4.4 times), and can decrease the average indoor temperature of 1.5℃ (about 5% of decrease). Mode 4 (with conventional windows) can be modified by staggered windows to become the design mix of Mode 5, which can increase air exchange of 1.21 h-1 (equivalently to 1.5 times), but there is no significant difference in the average indoor temperature (about 1%of slight increase).

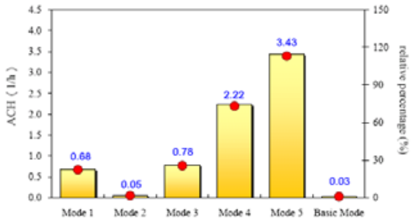


Fig. 8 Air exchange rates and the relative percentage of the 6 design mix

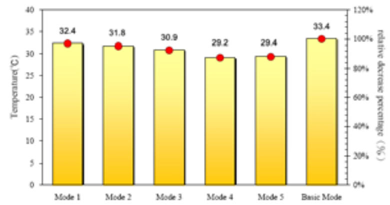


Fig. 9 Indoor average temperature and the relative percentage of the 6 design mix

ACKNOWLEDGEMENTS

Support from the National Science Council of ROC through grants NSC 95-2622-E-005-007-CC3, NSC 95-2221-E-426-014 and NSC 93-2211-E-426-003 in this study is gratefully acknowledged. My sincere gratitude also goes to Ms. Yu-Chun Huang, Master of Architecture at National Cheng-Kung University, Taiwan for her assistance throughout the numerical analysis.

REFERENCES

CIBSE. 1997. Natural Ventilation in Non-Domestic Buildings, CIBSE Applications Manual AM10. CIBSE Technical Publications Committee.

Etheridge, D and Sandberg, M. 1996. Building Ventilation: Theory and Measurement. John Wiley & Sons Inc., New York, pp. 5-6.

Patankar, S. V. 1980. Numerical Heat Transfer and Fluid Flow. Hemisphere Washington.