NUMERICAL APPROACH TO A MODULARIZED DATA COMPUTER ROOM USING A FRESH AIR-COOLING SYSTEM

by

Jing MA[a], Yin LIU[b*], and Ren-Bo GUAN[c]

[a] Henan University of Technology, Zhengzhou, Henan, China
[b] Zhongyuan University of Technology, Zhengzhou, Henan, China
[c] SIPPR Engineering Group Co., Ltd, Zhengzhou, Henan, China

Original scientific paper
https://doi.org/10.2298/TSCI1904421M

A fresh air-cooling plan is proposed to deal with high energy consumption in a data center and continuous improvement of temperature resistance performance in an IT equipment. The modularized data center, which is designed to adopt outdoor fresh air-cooling method for reducing temperature, can greatly reduce the energy consumption in data center. Numerical approach to study on the distribution of flow field, temperature field and velocity field is given using the CFD software. The results show that the flow field distribution is uniform, and the temperature distribution meets the requirements of the data center system. The numerical results can be used for optimization of the modularized computer room.

Key words: *data center, fresh air cooling, modularized, CFD*

Introduction

With the popularizing of the internet and the continuous innovation of computer technology, informatization has been continuously improved and widely used in all vital fields of our life, study, work and the entire social production. As the carrier of informatization, the data center has attracted more and more attention. During recent years, due to the accelerating development of the *Internet of Things*, cloud computing and big data industry, China's data center industry has entered a new stage of large-scale planning and construction, and has shown a trend of development towards large-scale, centralized, environmental-friendly, and rational deployment features.

It was estimated that the energy consumption of building parts accounted for 40% of the primary energy in the USA. The heating, HVAC systems approximately accounted for about 65% of the total energy consumption for the buildings in China [1, 2]. The rapid development of the data center industry has brought with a sharp increase of data center energy consumption. Research data shows that, in 2011, China's data center power consumption reached nearly 50 billion kWh, and by 2015 China's data center power consumption reached nearly 100 billion kWh [3]. One of the key reasons for the excessive energy consumption in the data center and the excessive value of power usage effectiveness (PUE) [4] is that the energy consumption of its air-conditioning system is too high. According to statistics, the energy consumption of air-conditioning systems accounts for about 40% of the total energy consumption of data centers in China. In some certain data centers, this rate even approaches or

* Corresponding author, e-mail: hvacr@126.com

exceeds 50% [5]. Among the above, the energy consumption of the refrigeration units is the main composing part of the total power consumption of the air-conditioning systems, which is about 60. And the power consumption rate of the water pump and the cooling tower is about 20%. The energy consumption of the refrigeration system in the air-conditioning system of the data center accounts for about 80%. Therefore, reducing the energy consumption of the air-conditioning system can reduce the PUE value of the data center and achieve the goal of effective energy saving in the computer room. The temperature reduction and energy saving of the data center can be realized by using the natural cooling source of the brand fresh outdoor air to replace the high-energy-consumption refrigeration system in the data center. This is an important way of saving energy for the data centers [6].

Brand fresh air-cooling data center

With the continuous development of electronic component technology, the temperature resistance performance of servers and switches in the data center has been continuously improved. For the performance of the latest IT equipment, ASHRAE has made adjustments for the design data center air-conditioner parameters in 2011 White Paper of Data Center Air-conditioner Technology (Thermal Guidelines for Data Processing Environments – Expanded Data Center Classes and Usage Guidance, 2011), which adjusted the upper limit of the upmost temperature to be 45 °C. With the continuous development of IT technology, the performance of IT equipment has been continuously improved as well. Companies as Intel and Dell have launched data servers featured in good performance of temperature resistance and stability, which could even work in environment of 70~80 °C. All the above development has well prepared for the energy-saving air conditioner design for the data centers. The outdoor temperatures in summer of major cities in China are as shown in tab. 1. All temperatures are below 37 °C, which makes it possible to use fresh air-cooling technology in data centers [7].

Table 1. Summer outdoor temperature in major cities of China

City	Dry-bulb temperature of ventilation for summer [°C]	Dry-bulb temperature of air conditioning for summer [°C]	Relative humidity [%]
Beijing	29.9	33.6	26.3
Shanghai	30.8	34.6	28.2
Chongqing	32.4	36.3	27.3
Urumqi	27.4	33.4	18.3
Wuhan	32.0	35.3	28.4
Guangzhou	31.9	34.2	27.8
Haikou	32.2	35.1	28.1
Nanjing	30.6	34.8	28.1

A new type of fresh air-cooling modularized computer room is designed, which uses the induced draft fan to introduce the outdoor cold air into the large space where the modularized computer room is located in. The distributed air-flow circulation temperature control system is adopted to distribute the cold air evenly to all modules. The cold air could enter the module by passing through the two air-inlet fans equipped for each computer room module. Temperature reduction could be realized by the low-suction type ventilation under three elements of air-inlet fan, server chassis exhaust fan, and the exhaust fan. The hot air heated by the heat exchange with the server would be sent by the exhaust fan to the vent duct and then be exhausted to outdoor environment.

Model establishment of computer room

A 2.7 m × 2 m × 2.8 m computer room module is built up according to the actual parameters of the modularized data center to bring the outdoor air into the data center as a cold source. Three racks for the server installation (1 m × 0.6 m × 2.4 m) are arranged in the middle of the module. Each rack is ready for installing 8 sets of servers; each set of servers includes 10 servers, and each set of servers is equipped with 5 exhaust fans with air volume of 30 cfm/fan, the total heat of servers in each rack is about 7000 W. Two air blowers are arranged at 2.2 m on the counter-side of the model, while two exhaust fans shall be set at the top of the other side. The diameter of both the blower and the exhaust fan shall be 0.2 m. The server is arranged in a hollow flow form. The side facing to the blower is set with is an orifice plate having an opening ratio of 0.35 while installing 120 exhaust fans on the other side with diameter of 0.04 m. In order to fully utilize the outdoor cold air sent into the module, the gaps between the server and the two side walls and the top are completely blocked to prevent the cold air from flowing downstream, so that the cold air sent into the room could be made flow through the server to improve the overall utilization rate of the cold air.

This simulation uses the MIT zero-equation model for indoor air-flow simulation calculations for ventilation and air-conditioning rooms. The MIT zero-equation model was proposed in 1998 by Li *et al.* [8]. It is a new turbulence model based on the results of direct numerical simulation of indoor natural air convection and mixed convection [9]. For the Rayleigh number ranging from 2.6 × 1010 to 3.0 × 1010 for a non-isothermal flow in the room, the eddy viscosity coefficient is proportional to the fluid density, local velocity and the closest distance from the wall surface [10], and the proportional coefficient is obtained by fitting of the results of direct numerical simulation. The MIT zero-equation model is mainly used for numerical simulation of non-isothermal and mixed convection airflow in air-conditioning ventilation rooms. Compared with the turbulent equation model, the indoor MIT zero-equation model is used to simulate the indoor air-conditioning ventilation environment. For the mixed convection of natural air convection and forced convection, it can obtain more accurate results than those by the equation turbulence model, and can more accurately simulate the flow pattern of the mixing convections [11]. At the same time, the zero-equation turbulence model uses only algebraic equations instead of differential equation to relate the turbulent viscous coefficient to the time-averaged value, so its calculation is also more time-saving.

The fresh air inlet and server exhaust side of the module are taken as the inlet boundary. The air inlet side of the server and the air outlet flowing to the return air path are taken as the outlet boundary. According to the annual meteorological parameters of Yangzhou city, the outdoor dry-bulb temperature of 20~25 °C has the longest time of distribution. The intermediate temperature of the above 23 °C is taken as the inlet air temperature of the module room. The ambient temperature of the module is the same as the supply air temperature, so it is set to be 23 °C. The air volume of the two inlet fans is 2800 cfm. The exhaust air volume of each exhaust fan on the exhaust side of the server is 30 cfm; the exhaust air volume of the two exhaust fans is 2800 cfm.

The quality of the mesh plays an important role in the accuracy of the simulation results, so that it needs to make a reasonable meshing for the established model. The hexahedral mesh of CFD software is used for free meshing and meets the following conditions:

– The maximum size of the grid unit in the computer room is 1/20 of the corresponding size in the X, Y, and Z directions.
– For the place where the temperature and speed changes greatly, such as the air outlet, the server exhaust fan and the air outlet, the grid density should be increased.

This model has a total of 209,826 meshes, 221,098 nodes, and the grid quality is shown in fig. 1.

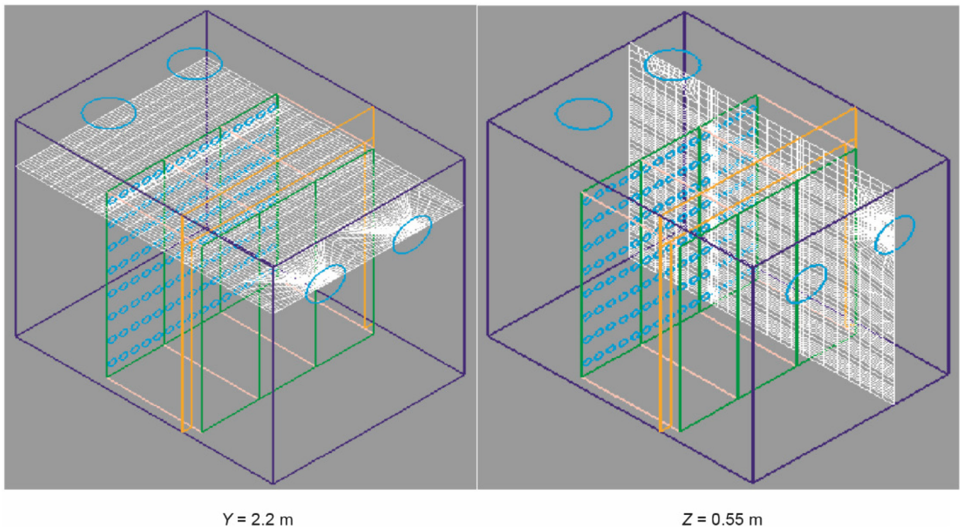

Figure 1. Mesh distribution (*for color image see journal web site*)

Analysis of simulation results

The distribution of the air-flow field of different co-ordinate axes directions in the computer room module is shown in fig. 2. The outdoor fresh air is sent into the module from

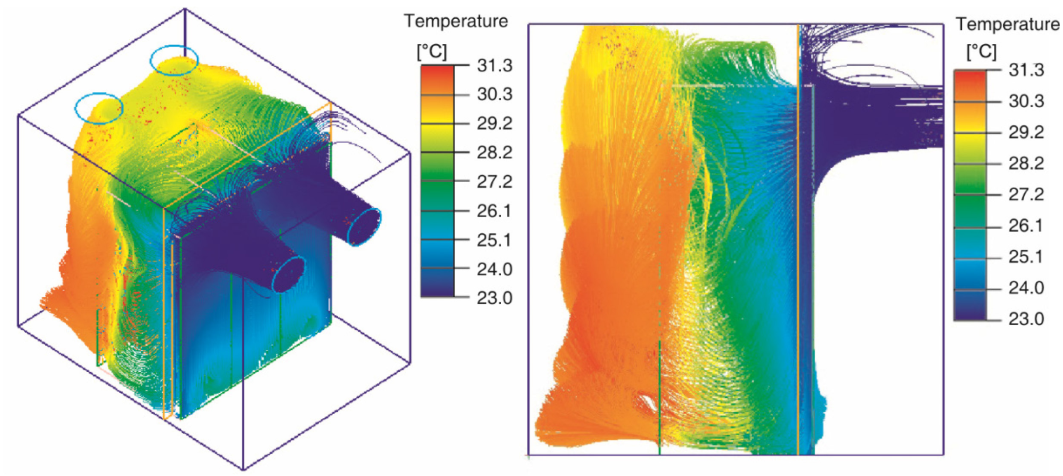

Figure 2. Flow fields distribution in the module (*for color image see journal web site*)

the opposite side of the server by the two air outlets horizontally. The air-flows into the server through the inlet baffle with the small orifices of the server, taking away the heat generated by the operation of the IT equipment, and discharging it from the server under the act of the exhaust fan. The hot air is then expelled from the module by the negative pressure entrainment of the two exhaust fans at the top of the module, into the vent duct and finally sent out of the room. The air inlet is located in the place that right faces to the upper position of the server. A part of the cold air directly enters the server, while another part of the cold air-flows down the baffle and enters the server through the openings of other height positions. At the same time, the entrainment effect of the module air outlet is obvious, and forms two vortices at the lowering position of the air outlet to enhance the air-flow disturbance on the exhaust side of the module, so that the hot air can be quickly discharged out of the module, maintaining a good cooling effect in the entire module.

For the temperature field distribution in the module, it is analyzed by taking the horizontal cross-sections at four heights in the Y-axis direction: $Y = 0.15$ m (lower cross-section of the server), 1.2 m (cross-section at intermediate height of the server), 2.2 m (cross-section right face to the air inlet), 2.6 m (cross-section at the middle part that the server top to the module top). The temperature field distribution is as shown in in fig. 3.

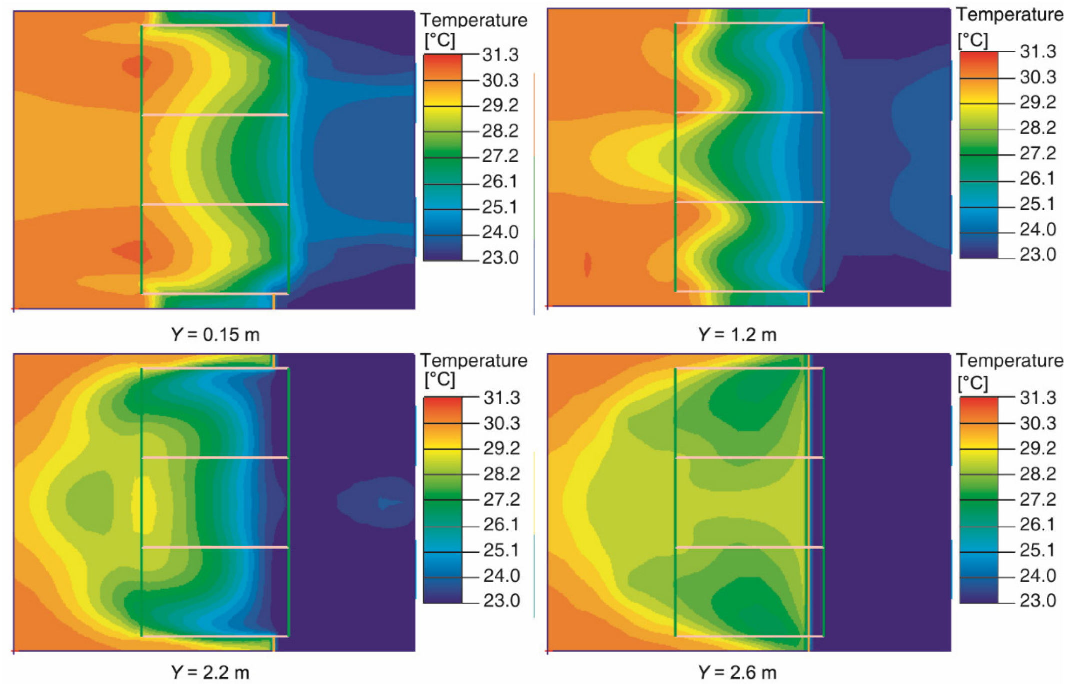

Figure 3. Temperature field distribution for cross-section inside the module
(*for color image see journal web site*)

In fig. 3, the temperature fields of the four cross-sections are symmetrically distributed along the vertical section of $Z = 1$ m. The stratification of the cross-section temperature of $Y = 0.15$ m, and 1.2 m is obvious. The temperature inside the module rises from the air inlet side, the inside of the server, to the exhaust side one by one, among which, the temperature of the air supply side floats at 24.0 °C, the temperature on both sides is low, and the temperature in the middle area is slightly higher. The temperature field inside the server gradually rises from about 25.0 °C to about 30.5 °C. The temperature field at the exhaust side is maintained at about 30.5 °C. Due to the horizontal direction of the cold air at the air inlet, the temperatures at Y = 2.2 m cross-section and Y = 2.6 m are lower than those at Y = 0.15 m and Y = 1.2 m cross-sections. The cold air concentrates on the side of the air inlet, which maintains a temperature of 23.0 °C. The temperatures at the server or above the server are below 30.0 °C. The maximum temperature in the whole module is 31.3 °C, and the temperature gradient is evenly distributed.

Figure 4 is a velocity vector diagram of the Z = 0.55 m and 1.0 m cross-sections in the module. For the process that Z = 0.55 m cross-section passes through the vertical section of the air inlet and air outlet, there are three places where the air-flow speed is comparatively larger, *i. e.*, the air inlet, the air outlet and the server exhaust fan. The speeds at the air inlet and air outlet are relatively large, reaching 10.6 m/s, and the speed of the server exhaust fan is relatively larger than the rest of the area, about 2.8 m/s. The Z = 1 m cross-section is the central symmetrical section in the module. Except for the air gathering at the server exhaust fan and the air whose velocity is high, the velocity vector lines in other areas is more smoothly distributed. After passing through the air inlet, the cold air forms a vortex in the upper part of the air inlet side of the module. The cold air is effectively disturbed and enters the server. A vortex is also formed in the area directly under the middle of the two air outlets. The hot air after exchange can be discharged from the module in time by the action of the vortex.

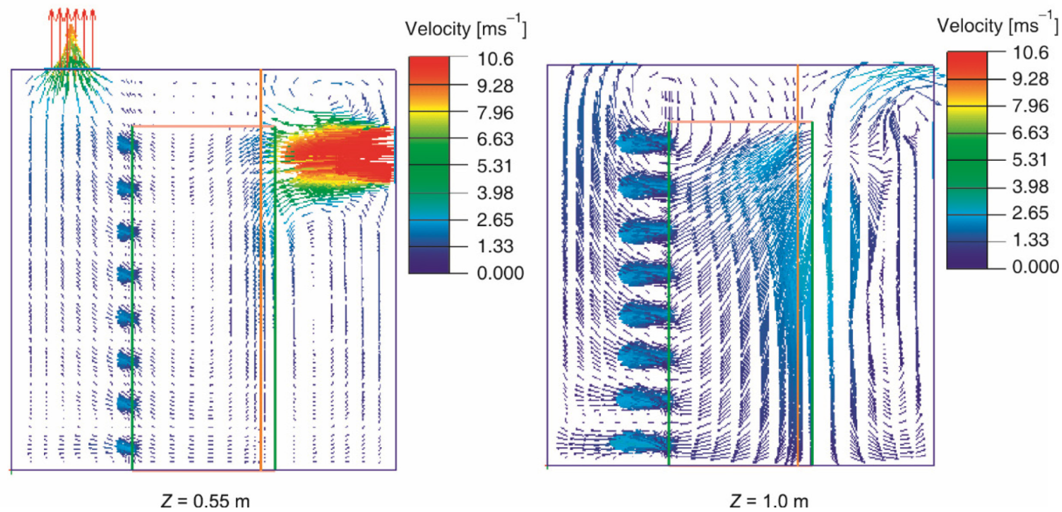

Figure 4. Velocity vector diagram at cross-sections inside the module
(*for color image see journal web site*)

Figure 5 is a horizontal cross-sectional velocity field distribution of four heights in the Y-axis direction of the module: Y = 0.15 m, 1.2 m, 2.2 m, and 2.6 m, respectively. It can be seen that the maximal velocity of the velocity field in the cross-section of Y = 0.15 m in the module is concentrated on the exhaust fan position of the server, and the speed of other regions is smaller and does not change much. The high flow rate at the exhaust fan position speeds up the air-flow inside the server and effectively prevents the local temperature from

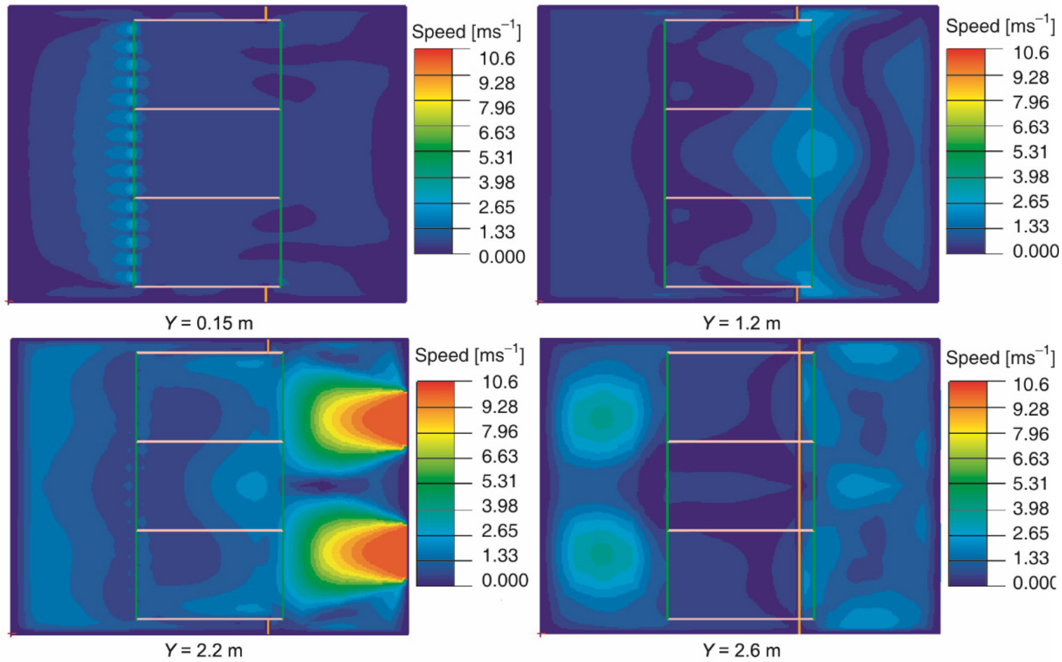

Figure 5. Velocity vector diagram at cross-sections inside the module
(*for color image see journal web site*)

being too high. In the velocity field distribution at the cross-section of Y = 1.2 m in the module, the velocity at the inlet orifice plate is larger than that of other regions. There are three positions having high velocity, *i. e.*, the middle part of air inlet side of the server, and the both sides. The highest speed is 2.5 m/s. At the cross-section of Y = 2.2 m that faces to the air inlet, due to the horizontal jet of the blower, the speed of the cold air at the air inlet is the highest, reaching 10.6 m/s. The air inlet speed of the module is relatively large, with comparatively strong spoiler phenomenon. The velocity inside the server and by the exhaust side is distributed in gradient pattern. Except for the larger flow velocity at the air inlet and its surrounding areas, the velocity of most of the entire section fluctuates between 0.5 m/s and 2.5 m/s. In the velocity field of the intermediate height Y = 2.6 m section between the top of the server and the top of the module, a higher velocity occurs in the area directly below the two air outlets. Due to the turbulence on the air supply side, the velocity distribution has a certain fluctuation. The velocity distribution at the server area is in uniform pattern with small velocity under 1 m/s.

Conclusion

The new air-cooling system can effectively realize the temperature control for the modularized computer room by setting distinct hot and cold areas to sufficiently use the outdoor natural coolant. The analysis results show that the air flow rate in the modularized computer room is reasonable, and the flow field distribution is uniform. The temperatures at the server area and above the server are both controlled under 30.0 °C. The maximal temperature in the whole module is 31.3 °C, and the temperature is distributed in uniform gradient pattern, which ensures the stable work of the data servers.

Acknowledgment

This research was supported by Key Project of Science and Technology Research of Henan Provincial Department of Education (13B480301).

References

[1] Liu, Z., *et al.*, Design of High-Performance Water-in-Glass Evacuated Tube Solar Water Heaters by a High-Throughput Screening Based on Machine Learning: A Combined Modeling and Experimental Study, *Solar Energy*, *142* (2017), Jan., pp. 61-67
[2] Liu, Z., *et al.*, Feasibility and Performance Study of the Hybrid Ground-Source Heat Pump System for One Office Building in Chinese Heating Dominated Areas, *Renewable Energy, 101* (2017), Feb., pp. 1131-1140
[3] Sun, W. C., Huang, Y., Data Center Energy Consumption Model Analysis and Research (in Chinese), *Shanghai Energy Conservation*, (2014), 1, pp. 29-32
[4] Christian, B., *The Green Grid Data Center Power Efficiency Metrics: PUE and DCiE*, The Green Grid, Portland, Ore., USA, 2007
[5] Qian, X. D., Li Z., Energy Saving Research on Air Conditioning System in Data Centers, *Journal of HV&AC, 42* (2012), 3, pp. 91-96
[6] Liu, Y., *et al.*, Research of Data Center Fresh Air Ventilation Cooling System, *Lecture Notes in Electrical Engineering, 262* (2014), Jan., pp. 299-306
[7] Liu, Y., Zhang, K., The Green Data Center Fresh Air Cooling System Design, *Proceedings,* 18th National Academic Conference on Heating Ventilation Air-Conditioning and Refrigeration, Yantai, China, 2012
[8] Li, C., *et al.*, A New Zero-Equation Turbulence Model for Micro-Scale Climate Simulation, *Building and Environment, 47* (2012), 1, pp. 243-255
[9] Zhan, Z. S., *et al.*, *Turbulence Theory and Simulation*, Tsinghua University Press, Bejing, 2017
[10] Su, Y. X., Application of Zero-Equation Model in Outdoor Air Flow Around Building Simulation, Ph. D. thesis, Tsinghua University, Beijing, 2010
[11] Ng, K. C., *et al.*, On the Effect of Turbulent Intensity towards the Accuracy of the Zero-Equation Turbulence Model for Indoor Airflow Application, *Building and Environment, 46* (2011), 1, pp. 82-88

Paper submitted: May 4, 2018
Paper revised: June 26, 2018
Paper accepted: June 26, 2018

© 2019 Society of Thermal Engineers of Serbia.
Published by the Vinča Institute of Nuclear Sciences, Belgrade, Serbia.
This is an open access article distributed under the CC BY-NC-ND 4.0 terms and conditions.