Green Buildings and Materials
Volume 1 Issue 1 No. 1
Published Online July 10 2022

Theory, technology and application of wind simulation in green building

Bo Su, Jiaqi Wu, Tianxiang Wang, Guojun Yu
Faculty of Civil Engineering and Mechanics, Jiangsu University, P.R.China,

Abstract Green building has been the trend of social development at present. The quality of wind environment acts as an important aspect of green building. Based on the theory of computation fluid dynamics (CFD) and using the method of program compiling and numerical solutions, the paper achieve to perform simulation of indoor and outdoor wind environment of buildings, provide technical support for indoor air duct design, buildings' distribution and optimization, and the reasonable planning of urban function zones. Moreover, wind environment simulation technology can supply surface wind pressure data, which is superior to filed measurement and wind tunnel test in saving human resources and materials.

Keywords: Green building, wind field simulation, CFD, theory application

Introduction

In terms of building environment, the quality of the wind environment, is a significant aspect to assess the green building. Good building ventilation environment can offer a more comfortable living environment, can be more conducive for energy conservation and emission reduction, and can promote the development of smart city. Therefore, the study of wind environment is a crucial source to be considered in urban planning.

The research methods of wind environment and wind load mainly include wind tunnel tests, field measurement and numerical simulation of computational fluid dynamics (CFD) [2]. Through making the scale model of an actual building, wind tunnel test can simulate actual wind environment and building structure. Besides, experimental instrument, which is arranged in the model surfaces and surrounding, is used to determine the actual structural wind effects. However, the price of wind tunnel test is rather expensive and model making requests high requirements. Field measurement method is the most direct means of wind environment research. However, field test has also been restricted by plenty of conditions. For instance, the field measurement cannot be carried out before the building construction; it can simply be applied to wind loads and structural response measurements in existing building surface; it still needs a large number of human resources, materials and time.

With the rapid development of the level of science and technology, in addition to the traditional study method like the wind tunnel experiments as well as field research methods, the researchers gradually utilize computational fluid dynamics (CFD) technology based on the principles of aerodynamics. This technology calls numerical simulation technique as well. CFD has been developed especially for the simulation of the flow behavior, and has proven its use in the investigation and optimization of many processes. The objective of CFD-Simulation is to identify complex flow problems in the construction as well as in existing systems and to help optimizing the processes [3]. Significant development and improvement has taken place in computational fluid dynamics (CFD) and its application in engineering, science and environment in the last 30–40 years [4]. And as a result, due to the development of computers, researchers began to employ computational fluid dynamics (CFD) to study various of areas in a building including the redistribution of snow [5], the indoor climate [6] and building ventilation. At present, the numerical simulation method has become a new effective way to predict the wind load and wind environment of buildings. It not only can effectively save human resources and materials, but also can be convenient to achieve the simulation of the wind environment of buildings.

1 Computational numerical simulation method

Numerical simulation of wind environment mainly includes node method, mathematical model method and computational fluid dynamics method [7]. The computational fluid dynamics (CFD) method has been widely used in engineering problems because of its advantages of high-speed, convenience, accuracy, effectivity, low cost and others. Gradually, this method has become the effective means to deal with engineering problems and has been widely recognized.

1.1 Integrated modeling and grid division

Computational fluid dynamics method plays an extremely important role in building wind environment evaluation, the found of building geometry model, the division of calculation regional grid and other aspects. The geometric shape of the building would affect the quality of the local grid. In order to avoid the quality degradation of the local mesh in several individual buildings caused by the irregular geometry, it is necessary to be simplified for the building geometry. In addition, the choice of calculation area is also quite crucial. Owing to the large domain of wind field's action, the calculation area should be large to some extent. However an excessive increase of the computational domain would greatly increase the computational cost. Hence, choosing a reasonable calculation area would reduce the amount of computation.

1.2 Boundary conditions and governing equation

When utilizing CFD technology to simulate the indoor wind environment, it is an important link to determine the reasonable boundary conditions in order to guarantee the accurate simulation computation outcome.

For the flow boundary conditions, the wind velocity in the flow direction is uniformly distributed. And the flow velocity increases along the height of the building according to the gradient. The wind velocity is set up at different heights, according to the theory of the atmospheric boundary layer. The calculation formula of the height and wind velocity is as follows [8]:

$$V_h = V_0 \left(\frac{h}{h_0}\right)^n \tag{1-1}$$

The full development of turbulence needs to be considered when air flows on the flow surface of the building. The flow boundary conditions are set up according to the free exits.

Based on the mass conservation, momentum conservation and energy conservation law of fluid flow, the CFD method establishes mathematical governing equations. The general form of the turbulence model is as follows [9]:

$$\frac{\partial(\rho\emptyset)}{\partial t} + div(\rho\vec{U}\emptyset) = div(\Gamma_\phi grad\phi) + S \tag{1-2}$$

in this type can be velocity, turbulent kinetic energy, turbulent dissipation rate and temperature etc..

1.3 Convergence criteria

According to the continuity equation and the momentum equation, the residual of the termination criteria in CFD numerical simulation algebraic equation is below 1.0E-4. While according to the energy equation, the residual is below1.0E-7[10]. The convergence curve and the value of the observation point are shown in the following figure.

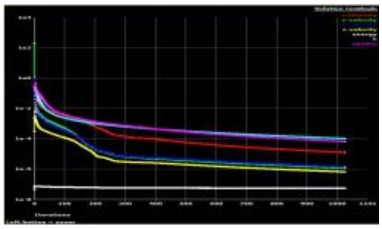


Fig. 1 convergence curves calculated by CFD

2 Case 1: CFD Numerical simulation of wind environment around buildings

Taking a residential district as an example, indoor and outdoor wind environment is simulated, including summer, winter, and transitional season outdoor wind speed and wind pressure, to get the indoor wind speed and air age map. Figure 2.1 and 2.2 are respectively for building model of the residential district and the computational domain mesh generation.

Fig. 2.1 Building Model

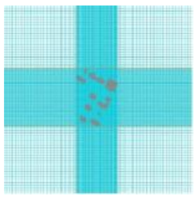


Fig. 2.2 Computational Domain Mesh Generation

Figure 2-3 and 2-4 are respectively for the regional model and the indoor computational mesh of the typical floor in building 1.

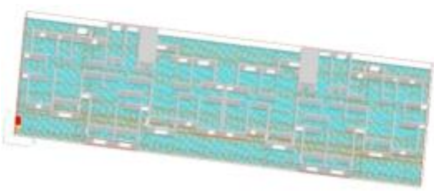


Fig. 2.3 Regional Model of Typical Floor in Building 1

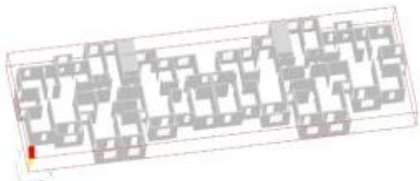


Fig. 2.4 Computational Mesh of Indoor Typical Floor in Building 1

Taking the summer outdoor wind environment of the residential district and the indoor air environment of the typical floor as an example, the wind speed and wind pressure map of summer indoor and outdoor are obtained by computer simulation in Figure 2.5 to 2.9 respectively.

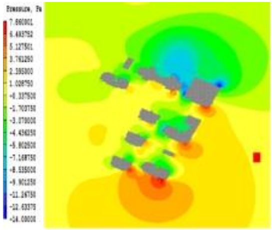


Fig. 2.5 Summer Outdoor Wind Pressure

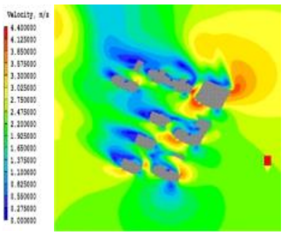


Fig. 2.6 Summer Outdoor Wind Speed

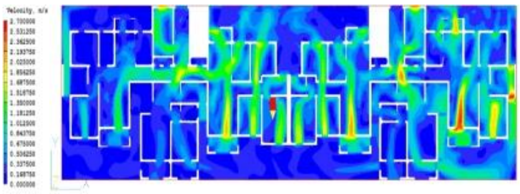


Fig. 2.7 Summer Indoor Wind Speed of Typical Floor

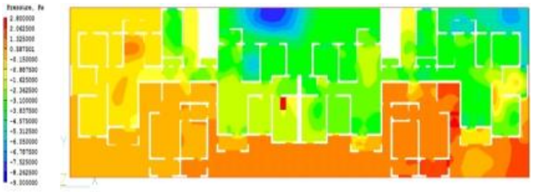


Fig. 2.8 Summer Indoor Wind Pressure of Typical Floor

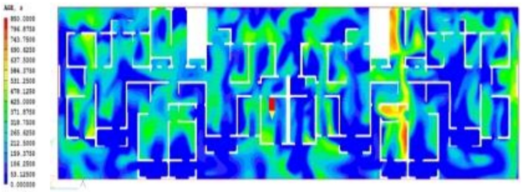


Fig. 2.9 Summer Indoor Air Age of Typical Floor

From Figure 2-5 and Figure 2-6 shown above, it can be seen that computer simulation technology can simulate the buildings' surrounding wind pressure, wind speed and other parameters, thus to provide effective technical support for rational layout of buildings in the design stage. The wind speed, wind pressure and air age in the building can be quantitatively seen from the Figure 2.7-2.9. These results can provide an effective numerical reference for the evaluation of green building.

3 Case 2: CFD Numerical simulation of wind loads on buildings

The basic parameters such as wind load and wind load can be determined by wind tunnel test in the early stage of long-span structure design. And in recent years, numerical simulation method has been applied gradually, and has become an effective method of studying building wind environment. It can carry out wind vibration response analysis on the structure even when in the lack of corresponding experimental data.

In this project, the wind load on the surface of a long-span roof is obtained by numerical simulation. The structure diagram is shown in Figure 3-1; computational domain and mesh generation are shown in Figure 3.2, 3.3.

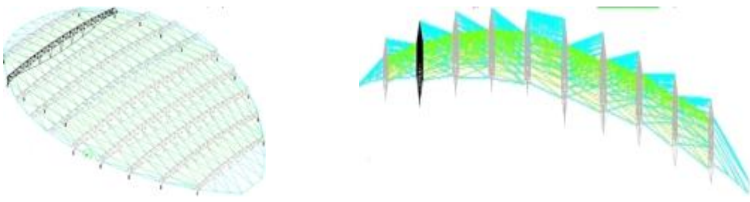


Fig. 3.1 Structure Diagram

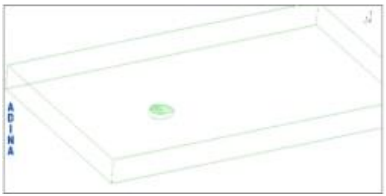


Fig. 3.2 Computational Domain

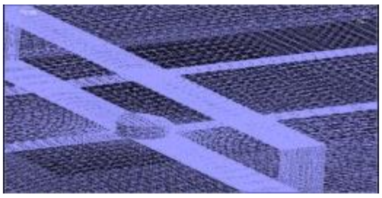


Fig. 3.3 Mesh Generation

3.1 Turbulence model and boundary conditions

The boundary conditions of the simulation and the selection of the turbulence model are shown in the following table.

Parameter condition	Content selection
turbulence model	RNG$\kappa - \varepsilon$ turbulence model
Inlet boundary conditions	Uniform flow using 15m/s
Exit boundary conditions	Fully developed flow boundary conditions
Ground boundary condition	Solid wall, Wall function is used to simulate the near wall flow.
Both sides and above of the computation domain boundary conditions	Solid wall slip condition
Building surface	Solid wall slip condition

3.2 Analysis of wind pressure on the roof surface

Equation of mean wind pressure coefficient [11]:

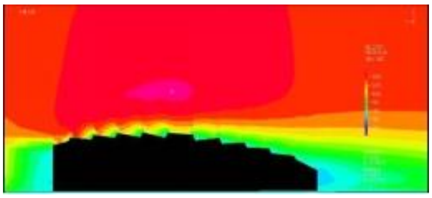
$$C_p = \frac{p}{\frac{1}{2}\rho u_0^2} \qquad (3\text{-}1)$$

In the Equation, p-Surface absolute pressure, ρ-Density of air, ρ=1.29kg/m^3, u_0-Inflow wind speed

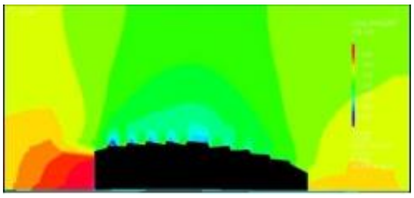
3.3 Computer simulation of surface wind pressure

CFD numerical simulation method is used to analyze the variation of wind pressure coefficient on the surface of long-span structure, thus to realize the application of CFD numerical simulation technology in the static wind load of the actual structure.

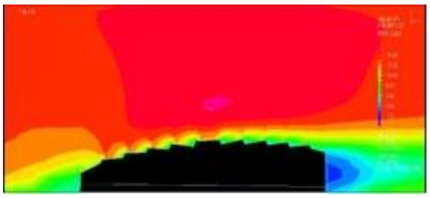
Cross-section wind speed and wind pressure scatter diagram at 0º-180º wind direction were given in Figure 3-4. It can be clearly seen from this figure that the wind speed and wind pressure at each corner of a long-span roof and eaves:



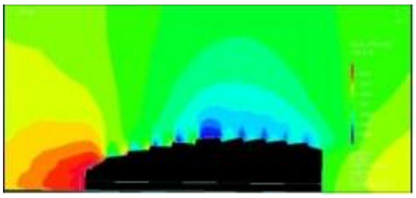
(a) Wind Speed（Wind Direction 0º）



(b) Wind Speed（Wind Direction 0º）



(c) Wind Pressure（Wind Direction 180º）



(d) Wind Pressure（Wind Direction 180º）

Fig.3-4 Wind Direction at 0º-180º, Cross-section Wind Speed and Wind Pressure Scatter Diagram

4 Conclusions

In this paper, the wind field simulation technology is used to simulate the wind field environment of buildings. The evaluation of indoor and outdoor wind environment quality and the optimization of the structural system or the component can be available based on simulation result, to promote the construction of green building [12].

This project uses the innovative computer simulation algorithm to make the program, and the dynamic equations of the airflow around the building are solved by numerical method. The technique can be implemented to calculate the physical parameters of the wind field of buildings, which provides a new means and method for the calculation of wind load, and also serves as a tool for the green building. According to the wind pressure, wind speed and the wind age map from the building wind field simulation, Indoor ventilation channel shall be reasonably arranged when designing a single building, architectural complex layout be rationally planned and functional region be rationally divided when overall arranging the city., so as to improve the indoor outside natural ventilation effect

Acknowledgement

The Project Sponsored by Jiangsu Planned Projects for Postdoctoral Research Funds (No.1301048C); National Natural Science Foundation of China (No.51508237); Natural Science Foundation of Jiangsu Province (No.BK20140560).

Reference

[1]Shuzo Murakami, editor in chief, translated by Zhu Qingyu. CFD and architectural environment design. Beijing: China Building Industry Press, 2007
[2] Embarek Belhadj, Stephen Chilton, William Nimmo, Hubert Roth, Mohamed Pourkashanian. Numerical simulation and experimental validation of the hydrodynamics in a 350 kW bubbling fluidized bed combustor[J]. Int J Energy Environ Eng, 2016, (7): 27-35
[3] Lei Peng, Peter V. Nielsen, Xiaoxue Wang, Sasan, Sadrizadeh, Li Liu, Yuguo Li. Possible user-dependent CFD predictions of transitional flow in building ventilation[J]. Building and Environment, 2016, (99): 130-141
[4] Xuanyi, Zhou, Luyang, Kang, Ming, Gu, Liwei, Qiu, Jinhai, Hu. Numerical simulation and wind tunnel test for redistribution of snow on a flat roof[J]. Journal of Wind Engineering and Industrial Aerodynamics, 2016, (153): 92-105
[5] Daniel Risberg, Mikael Risberg, Lars Westerlund. CFD modelling of radiators in buildings with user-defined wall functions[J].Applied Thermal Engineering,2016,(94):266-273
[6] Qin Yanyao, Dai Huizi, Wu Sirui, Liu Jun. Study on the natural ventilation design of ultrahigh-rise building with the numerical simulation technology [J]. Chongqing architecture, 2015, 02:8-10.
[7] Zhang Xiangting, editor in chief. Structural wind engineering – theory, standard, practice. Beijing: China Building Industry Press, 2006
[8] Tominaga Y, Akabayashi S, Kitahara T, et al. Air flow around isolated gable-roof buildings with different roof pitches: Wind tunnel experiments and CFD simulations [J]. Building and Environment, 2015, 84: 204-213.
[9] Shang Tao, Qian Yi. Wind environment simulation and evaluation of the residential district Wuhan: A case study of Chagang district in Wuhan University [J]. Huazhong Architecture, 2013, (01): 48-51
[10] An K, Fung J C H, Yim S H L. Sensitivity of inflow boundary conditions on downstream wind and turbulence profiles through building obstacles using a CFD approach[J]. Journal of Wind Engineering and Industrial Aerodynamics, 2013, 115: 137-149.
[11] Yang Li, editor in chief. Green building design - building wind environment. Shanghai: Tongji University press, 2014