PAPER • OPEN ACCESS

Research on several key Factors of Numerical Wind Tunnel in the Simulation

To cite this article: Zhuangnan Zhang *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **490** 032010

IOP Conf. Series: Materials Science and Engineering **490** (2019) 032010 doi:10.1088/1757-899X/490/3/032010

Research on several key Factors of Numerical Wind Tunnel in the Simulation

Zhuangnan Zhang, Xinyu Li* and Chungang Wang

School of Civil Engineering, Shenyang Jianzhu University, Shenyang, China

*Corresponding author e-mail: 393480672@qq.com

Abstract. With the improvement of computer performance and the gradual maturity of the computational methods, numerical wind tunnel in the simulation based on computational fluid dynamics has been developed rapidly and turned into one of the hottest investigated topics in computational wind engineering. Simulation method of turbulence, inflow turbulence generator method and the size of the grid scale are mainly included in the influence factors of the wind-induced response around the building in the computational domain. The above influence was analysed and summarized. The results show that the numerical wind tunnel in the simulation has made some good progress, but there are still a lot of contents that need to be further studied.

1. Introduction

Numerical wind tunnel in the simulation has been researched as an emerging method in recent years for analyzing the effects of wind load on the flow field and building surfaces. Numerical wind tunnel in the simulation is considered to be a promising development direction in structural wind engineering [1]. The computational fluid dynamics (CFD) is used by numerical wind tunnel in the simulation to simulate the changes of the wind field around the structure by using a computer and solve the wind load on the surface of the structure [2]. Numerical wind tunnel in the simulation can be divided into two categories, one is steady-state numerical wind tunnel in the simulation, and the other is unsteady numerical wind tunnel in the simulation. Compared with the obtained instantaneous calculation results by the steady-state Numerical Wind Tunnel in the Simulation, the time variable data of the wind load effect can be generated, the cloud atlas of wind pressure and the dynamic picture of streamline with time can be obtained by the unsteady numerical wind tunnel technology. The research on numerical wind tunnel in the simulation in China was started late. Currently, the wind resistance performance of large-span and high-rise buildings is still at the stage of research and experiment. The design and optimization of numerical wind tunnel based on CFD have broad prospects. However, there were very few cases where CFD has been applied to aerodynamic design of numerical wind tunnels combining the engineering application so far, and are limited to localized auxiliary designs [3]. The performance of numerical wind tunnel in the simulation is mostly based on CAARC model [4]. Numerical Wind Tunnel in the Simulation not only saves test costs, shortens test cycles, but also provides very detailed information compared to wind tunnel tests. At present, the fundamental purpose of research on numerical wind tunnel in the Simulation is to ensure that more accurate and controllable analysis of computing results can be obtained. Simulation methods and algorithms are needed to optimize by us for improving the accuracy of numerical wind tunnel in the simulation.

2. Simulation methods of turbulence

2.1. Simulation Method of turbulence Development

The turbulence numerical simulation methods commonly used in numerical wind tunnel in the simulation are mainly divided into the following three types: direct numerical simulation (DNS), Reynolds average navier-stokes equations (RANS), and large eddy simulation (LES). The earliest DNS can be traced back to 1972, until now, this method has been widely researched. However, DNS is still faced with the characteristics of many computational data and long computation time. LES is widely used by a large number of scholars at present, LES is proposed by Smagorinsky in 1963. Because people have been optimistic about the direct numerical simulation of turbulence in previous years, the study of LES was once overlooked. However, as people realize the arduousness of DNS, and LES is promoted with the advent of parallel computing platforms. LES has been noticed by the researcher of the structural wind engineering.

Satisfactory results is obtained by used the unsteady RANS [5] to calculate appendages and small separation flows, however, Large-scale separation flows cannot be simulated accurately by using this turbulence models. Large eddy simulation has been developed due to the deficiency of RANS. The theory of LES is N-S equation and continuity equation.

$$\frac{\partial \rho \bar{u}_i}{\partial t} + \frac{\partial(\rho \bar{u}_i \bar{u}_j)}{x_j} = -\frac{\partial \bar{\rho}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \bar{u}_i}{\partial x_j}\right) - \frac{\partial \rho(\overline{u_i u_j} - \bar{u}_i \bar{u}_j)}{\partial x_j} \tag{1}$$

$$\frac{\partial \rho \bar{u}_i}{\partial x_i} = 0 \tag{2}$$

1. N-S equation of Large Eddy Simulation
2. Continuity equation of Large Eddy Simulation

2.2. Simulation method of turbulence comparison

In order to overcome the insufficiency of the RANS method to simulate the separation flow and effectively improve the calculation efficiency of the LES method, in 2006, Xiao Zhixiang of Tsinghua University studied the separation flow using the hybrid method of RANS and LES. Good results were achieved [6]. Through the simulation of the separation flow, this method has a strong ability to simulate the separation flow, and the flow of the surface of the building can be simulated very well.

To research the advantages and disadvantages of these Simulation methods of turbulence, in 2012, Lu Chunling used the above three simulation methods of turbulence to obtain the corresponding pressure power spectrum of cavity and compare the data. The results showed that the results of large eddy simulation are very close to data of direct numerical simulation, with only slight differences at high frequencies. The results of the non-stationary Reynolds average navier-stokes equations at the low-frequency part can coincide with the direct numerical simulation, but there is a large error in the high-frequency part [7]. Large eddy simulations require much less time and space than direct numerical simulation, and more turbulent information can be obtained relative to Reynolds average navier-stokes equations.

In 2016, Yang Lvlei conducted numerical wind tunnel in the simulation and wind tunnel tests on the test tower of the KONE in Kunshan City, Jiangsu Province [8]. The results show that the calculated wind field statistics and power spectral density by large eddy simulation are consistent with the theoretical formula and meet the engineering accuracy requirements by comparison, the calculated wind coefficient of the layer by large eddy simulation is basically consistent with the results of wind tunnel test. These show that the distribution of wind pressure on the building surface can accurately be predicted by large eddy simulation.

In recent years, The LES has been widely used. Cao Zhengzhi used LES to simulate the twin-tower building in 2016 [9], he proved that the downwind and across-wind loads on the building surface can be well simulated by LES. But the numerical wind tunnel test based on LES of complex buildings and

IOP Conf. Series: Materials Science and Engineering **490** (2019) 032010 doi:10.1088/1757-899X/490/3/032010

the application of the results to the wind resistance design of the actual project remains to be further studied.

3. Inflow turbulence generator method

Another important aspect of the numerical wind tunnel in the simulation is the turbulent boundary condition of the fluid at the entrance. The fluid at the entrance is needed to meet certain turbulent conditions, power spectrum characteristics and also have a certain turbulent structure. In the simulation of the RANS model, it can only be converted into turbulent kinetic energy (k) and dissipation rate of turbulence (ε), so the turbulent structure cannot be simulated. In the LES, the method of artificial simulation is used, after many years of development, there are two main methods for generating inflow turbulence. The one is to calculate the turbulent pulsation wind field of entrance directly in an additional computational domain as a boundary condition and then to simulate the computational domain. However, the inflow turbulence generator by this method is difficult to satisfy the power spectrum. The other method of inflow turbulence generator by generating random data was proposed by Hoshiya in 1972.

From now on, the DSRFG method for generating inflow turbulence has attracted the attention of everyone. The DSRFG method is currently approved by some researchers. This method was proposed by Huang Shenghong in 2010[10]. Huang Shenghong used the DSRFG method and compared it with other simulation methods to verify the following advantages of the DSRFG method:

1) The DSRFG method should strictly ensure that the inflow turbulence meets the continuity condition;

2) Based on strict theoretical derivation, the generated fluctuating velocity field satisfies the specified spectrum density function;

3) Spatial correlation of inflow turbulence can be adjusted by adjusting correlation scale factor;

4) The turbulence generation process at each coordinate point is independent to each other and is suitable for parallel computation.

The DSRFG method is a new method of inflow turbulence generator based on large eddy simulation. The turbulent wind field by the DSRFG method accords with the Von Karman spectrum. Because the Von Karman spectrum has been confirmed that it is in good agreement with the wind spectrum of the actual atmospheric boundary layer wind field. Figure 1is a picture of turbulent wind velocity at the entrance by the DSRFG method.

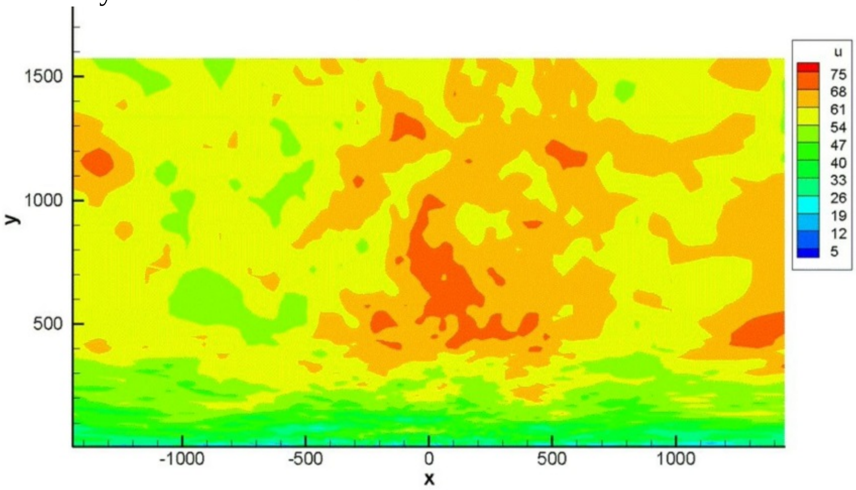

Figure l. Generation of turbulent wind velocity at the entrance.

4. Analysis of grid

4.1. Appropriate meshing

The selection of appropriate meshing in computational domain will have a significant impact on the accuracy of numerical wind tunnel in the simulation. In general, structured grid or unstructured grids are used in simulation. Because the most of actual buildings are complex, structured grid is generated difficultly, and at the same time, because of the large computational domain, structured grids take up a lot of grid resources and increase computing time.

For FLUENT software platform, the number of interpolating points of tetrahedral mesh is 4, while that of hexahedron mesh is 6. Therefore, the solution performance of the hexahedral mesh is significantly better than that of the tetrahedral mesh, but the generation of the hexahedral mesh is spent more power and time. Zeng Kai has carried out a study on the grid influence factors affecting the computational wind engineering [11]. The program of mixing factor of 0.75 was used in the simulation. It is suggested that tetrahedron or hexahedron grids should be selected according to different software platforms, and a mixed scheme of second order or second order should be adopted about Interpolation of convective kinematics scheme, so as to ensure the accuracy of calculation.

4.2. Encryption of grid

In the numerical simulation, although the accuracy of the calculation can be greatly improved by reducing the mesh scale in the entire model, but the time of calculation undoubtedly is increased. The astringency of the calculation and the accuracy of the calculation result are affected directly by the factors of the grid, so the grids of the separated flow district around bluff bodies and the circulation region are encrypted sufficiently, the performance of simulation is improved.

5. Conclusion

In this paper, simulation method of turbulence, inflow turbulence generator method and the size of the grid scale which are main factors of influencing the accuracy of the calculation results are summarized. At present, the accuracy and credibility of numerical wind tunnel in the simulation have not been yet reached the ideal standard. With the improvement of computer performance and the development of numerical computational methods, the calculation speed of numerical wind tunnel in the simulation and the accuracy of result will be greatly improved. Because the expenditure can be saved and the rich data be provided for computational domain in every points on the building's surface. In the future, numerical wind tunnel in the simulation is expected to become a powerful method of the wind-induced response analysis of super high-rise buildings and long-span structures. Therefore, further research and development of numerical wind tunnel in the simulation is very meaningful.

Acknowledgments

This work described in this paper was supported by China Northeast Architectural Design & Research Institute, their supports are gratefully acknowledged.

References

[1] Xiangting Zhang. Structural wind engineering [M]. Beijing: China Architecture & Building Press,2006(6).(in Chinese)
[2] Dan Lu, Chengming Li, Guojian Wang. CFD unsteady state numerical simulation of wind-induced vibration of the Shanghai Jinmao building[J].China Civil Engineering journal, 2008(8).(in Chinese)
[3] Yi Dai, Zuogang Chen, Ning Ma, et al. Numerical simulation of flow field inside the low-speed wind tunnel [J]. Acta Aerodynamica Sinica, 2014(4). (in Chinese)
[4] Mou Ben, Jiehe Bao, Xuezhao Dong, et al. Numerical simulation of the effects of buildng dimensional variation on wind pressure distribution [J].Engineering Applications of Computational Fluid Mechanics, 2017(1).
[5] Xiaochun Chen, Yuan Wang. The numerical simulation of separated flow around bluff bodies [J]. Acta Aerodynamica Sinica,2007(12).(in Chinese)
[6] Zhixiang Xiao, Haixin Chen, Qibing Li, Song Fu. Simulation of separation flow with RANS/LES hybrid methods [J]. Acta Aerodynamica Sinica,2006(6) .(in Chinese)
[7] Chunling Lu. Numerical wind tunnel simulation of wind effect on super high-rise buildings and long-span roof structures[R].Changsha: Hunan University, 2012.(in Chinese)

[8] Lvlei Yang, Yixiang Wang, Minfeng Gong. Wind-induced response analysis of super high-rise building based on large eddy simulation[J]. Building Structurel,2016(10).(in Chinese)
[9] Zhengzhi Xiao, Yanqi Li, Chengwei Huang, et al. Study of wind load characteristics of twin-tower building based on large eddy simulation[J]. Industrial Construction,2016(5). (in Chinese)
[10] Huang S H, Li Q S, Wu J R. A general inflow turbulence generator for large eddy simulation. Journal of wind engineering and industrial aerodynamics, 2010, 98: 600-617.
[11] Kai Zeng, Congjun Wang, Bencai Hang, et al. Suggestion and analysis of several key factors in computational wind engineering [J]. Acta Aerodynamica Sinica, 2007(12). (in Chinese)