Research Status on Inflow turbulence generation method with Large Eddy Simulation of CFD numerical wind tunnel

Zhuangnan Zhang¹, Xiaochun Bao*
School of Civil Engineering, Shenyang Jianzhu University, Shenyang, China

*Corresponding author e-mail: 2397241854@qq.com

Abstract. With the rapid development of computer science and the deepening of theoretical research, numerical wind tunnel simulation technology has been more and more attention as one of the important research methods in structural wind engineering. A representative method of numerical wind tunnel simulation technology: large eddy simulation method was introduced. One of the key technology of it called Inflow turbulence generation method was also analyzed. Meanwhile, three inflow turbulence generation methods: the Precursor Simulation Methods, the Synthesis Methods, and the DSRFG Methods were compared. It can provide reference for the further research on numerical wind tunnel simulation.

1. Introduction
A representative method of numerical wind tunnel simulation technology: large eddy simulation method was introduced. One of the key technology of it called Inflow turbulence generation method was also analyzed. Meanwhile, three inflow turbulence generation methods: the Precursor Simulation Methods, the Synthesis Methods, and the DSRFG Methods were compared. It can provide reference for the further research on numerical wind tunnel simulation.

2. Research Status on structural wind engineering
Currently, there are mainly four research methods in the field of structural wind engineering: field measurement, theoretical analysis, wind tunnel testing and numerical wind tunnel simulation on the base of CFD method.[1]

The numerical wind tunnel simulation has its advantage when compared with the other methods because the field measurement can only analyze the building after it was built, the theoretical analysis cannot solve complex flow problems and the wind tunnel testing is costly. And with the continuous improvement of the computer level and the development of the model theory, wind tunnel simulation technology will have broader development prospects in structural wind engineering and computational fluid dynamic in the future.

3. Common numerical simulation methods for turbulence
Currently, there are mainly three kinds of numerical simulation methods for turbulence commonly used in computational fluid dynamics: DNS, RANS, LES. Those three methods represent three different resolutions of turbulence.[2]

In these three methods, the Direct Numerical Simulation is precise but it need huge computer resources. Although the Reynolds average method can achieve high Reynolds number simulation, it has poor calculation accuracy. Based on these, in 1963, the American meteorologist Smagorinsky proposed the turbulent simulation method of large eddy simulation for the first time.[3]
4. Large Eddy Simulation (LES)

The central idea of this method is to divide the turbulent vortex shedding into more than two dimensions: the large-scale vortices mesh and the size of the spatial filter mesh. The large-scale vortices contain most of the energy, and their motion conditions are greatly influenced by individual flow conditions.

For small-scale vortices, its main “effect” in turbulent movement is energy consumption. The effect of its movement on large-scale vortices is simulated by establishing a sub-grid scale model. This method is between the two methods mentioned above. It can save the computer resources, better reflect the nature of turbulent motion, and improve the accuracy of simulation.

4.1 Research Status on Large eddy simulation of inlet pulsation wind

In recent years, with the advancement of theoretical research, the LES has become a very useful tool in the field of structural wind engineering. In the process of large eddy simulation, one of the most important parts is the method of generating the inlet turbulent wind field.

In addition to satisfying the turbulent control equation: Navier-Stokes equation, it need to meet the target wind speed spectrum meanwhile,[4]. That is, it is closer to the atmospheric boundary conditions and satisfies the spatial randomness. Also in the actual simulation process, it is necessary to facilitate the adjustment of various parameters such as the turbulence intensity and the scale of the lattice model.

4.2 Inflow turbulence generation methods of Large eddy simulation

At present, in the field of structural wind engineering, there are mainly three LES inflow turbulence generation methods, which are the Precursor Simulation Methods and the Synthesis Methods, and the Discretizing and Synthesizing Random Flow Generation Methods which was obtained based on the second method.

4.2.1 Precursor Simulation Methods

The Precursor Simulation Methods is to add an additional calculation domain before the determined calculation domain. The turbulent wind field of the flow is calculated in this additional area and used as the input turbulence parameter in the main calculation field.

This method comes from the wind tunnel test. In the scale model, the cusp tip, rough element and so on are simulated to simulate the ground roughness, and turbulence conditions similar to atmospheric boundary conditions are generated.

As is shown in the Dig 1, the calculation domain B is the main calculation domain. Before the calculation, an additional simulation calculation domain A is established before it, and the incoming turbulent wind field of the region A is directly calculated. At the same time, in the area A, the M plane vertical to the velocity orientation of the wind speed at the entrance is arranged, the turbulence intensity and time parameters of the plane are recorded, and these parameters are stored in the database. Then, the data in the database is extracted and applied to the entrance plane N of the main simulation area B to simulate the entrance turbulence condition of the area B.
The advantages of this method are: it can simulate the turbulent wind field under realistic atmospheric boundary conditions, satisfy the Navier-Stokes equation, etc. However, this method requires a lot of time in the process of generating the wind field in the additional area, and the storage requirement for the computer is also very high. In 2012, Zhu Weiliang, Yang Qingshan [5] and others adopted the cycle inflow boundary to study the applicability of the large eddy simulation entrance. The numerical results obtained can be used for the B wind resistance research institute in China.

4.2.2 Synthesis Methods
The synthesis method is a series of time series. The time series is characterized by satisfying the Navier-Stokes equations and the required turbulence parameters. This time series is then added to the computation domain ingress node.

This method can be divided into two methods. The first synthesis method was first proposed by Hoshiya (1972), and then improved by Kondo et al. (1997). In this method, the inverse Fourier transform of the target wind speed spectrum is performed and summed to obtain the fluctuating wind speed time course. The pulsation speed is showed by the following function(1)[6]:

$$\chi_j(t) = \sum_{N=1}^{N} \sum_{p=1}^{N} \left[ \alpha_{lp}(\omega_n) \cos \{ \omega_n t + \phi_{lp}(\omega_n) \} + b_{lp}(\omega_n) \sin \{ \omega_n t + \phi_{lp}(\omega_n) \} \right]$$

The advantage of this method is that the target spectral density, velocity and variance can be defined in the turbulence generation procedure. The disadvantage is that it cannot satisfy the continuity condition of the generated flow field.

The second synthesis method was proposed by Kraichanan (1970) [7]. At the same time, the generated isotropic wind field is modified by changing the scale size and orthogonal transformation to transform into an anisotropic wind field that satisfies the target pulsation characteristics.

The advantages of this method are: (1) It is possible to generate a wind field with no divergence. (2) It can meet the anisotropy of spatial scale and length scale.

However, it can be seen from the Dig 2 that, compared with the wind speed power spectrum of the atmospheric boundary layer, the wind speed power spectrum of the flow field generated by this method is well fitted in the large-scale energetic region but in the energy dissipation region and the inertial The area has dropped sharply, and the difference is very large.

Dig 2. The display of three turbulence subdomains at high-Reynolds-numbers.

4.2.3 Discretizing and Synthesizing Random Flow Generation Methods
In 2010, Huang Shenghong et al. proposed a new turbulence generation method based on the random flow generation method put forward by Smirnov. [8] That is DSRFG Methods.[9] And through numerical calculations, the advantages of this method satisfying the above-mentioned second method
are verified. The above second method can only generate fluctuating wind fields that satisfy the Gaussian spectrum. While the Discretizing and Synthesizing Random Flow Generation Methods can synthesize anisotropic turbulence that satisfies spatial correlation and non-diffusion conditions for any target wind speed spectrum.

The main features of this method are:

1. It is possible to generate a wind field with no divergence.
2. It can meet the anisotropy of spatial scale and length scale.
3. The spectral density function developed by the wind speed power spectrum generated by this method.
4. The turbulence generation parameters at the entrance are independent of each other and can be calculated in parallel, saving time.
5. With dynamic characteristics, suitable for engineering applications.

Based on those advantages, this method has been widely used at present.

5. Conclusion

This paper introduces three methods for generating inlet turbulence in the numerical wind tunnel simulation method represented by large eddy simulation in structural wind engineering. The advantages and disadvantages of the three methods are compared and analyzed. China started late in the field of structural wind engineering, but in recent years, with the advancement of computer science and the deepening of theoretical research, rapid progress has been made in all aspects. Inlet turbulence method is one of the key technologies in numerical wind tunnel simulation. Further development and research are necessary.

Acknowledgments

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

References

[1] Li Chao. Study on CFD simulation of turbulence wind field near ground[D]. Harbin Institute of Technology, 2010. (in Chinese)

[2] Lu Chunling. Numerical Wind Tunnel Simulations of Wind Effects on Super High-rise Buildings and Long-span Roof Structures[D]. Hunan University, 2012. (in Chinese)

[3] Smagorinsky J. General circulation experiments with the primitive equations. Mon. Weath. Rev., 1963, 91(3): 99-164.

[4] S.H. Huang, Q.S. Li, J.R. Wu. A general inflow turbulence generator for large eddy simulation[J]. Journal of Wind Engineering and Industrial, 2010, 98: 600-617.

[5] Zhu Weiliang. Research on the turbulent inflow for large eddy simulation of fluctuating wind loads[D]. Beijing Jiaotong University, 2011. (in Chinese)

[6] Shen Lian, Han Yan, et al. Research on Generating Method of Fluctuating Wind Field of LES Based on WAWS [J]. Journal of Hunan University, 2015, 42(11): 64-71. (in Chinese)

[7] Kraichnan, R.H. Diffusion by a random velocity field[J]. Physics of Fluids, 1970, 13(1): 22-31.

[8] Zhu Weiliang, Yang Qingshan. Research on the applicability of LES simulation pulse entrance [J]. Acta Aerodynamica Sinica, 2011, 29(05): 124-128. (in Chinese)

[9] B.W. Yan, Q.S. Li. Inflow turbulence generation methods with large eddy simulation for wind effects on tall buildings[J]. Computers & Fluids, 2015, 116: 158-175.