Turbulence characteristics of large factory building and its influence on airship transport process

Yuan Liu*, Hu Ye, Yongdong Zhang, Li Zhang, Min Li, and Taiqin Huang
Northwestern Institution of Nuclear Technology, Xi’an, China

*Corresponding author e-mail: zy2001336@buaa.edu.cn

Abstract. Aiming at the problem that the stability of airship’s posture is destroyed by turbulent flow when the airship leaves the large factory building, a Computational Fluid Dynamics (CFD) numerical simulation method is used to analyse the characteristics of large factory building disturbance under different inflow angles. The transport process of airship is simulated, and the equivalent force and moment of airship in variation with transport distance are acquired. The results show that the wind field changes along the transport path of the airship, which makes the equivalent force and moment acting on the airship body center fluctuate. In 60 degrees of the incoming flow, compared with other inflow angles, the turbulence at the exit of large factory building is severe, and the distance affected by the turbulent flow of large factory building is shorter, and the equivalent force and moment acting on the airship body center are larger in the process of airship transport. The simulation results provide an effective reference for the safety of airship transport process.

1. Introduction
Airship is a kind of aircraft which can fly by air buoyancy and propulsion system [1]. Airship can stay in the air for a long time with the low cost of manufacturing and flying, and it has good concealment performance and a large payload. Airship is widely used in communication relay, early warning detection, electronic countermeasures, intelligence investigation and other fields.

The capsule of airship, especially for the blimp, is usually made of flexible membrane material. Due to the small stiffness of airship, its shape is very sensitive to the change of incoming flow, thus assembling and inflating in the large factory building, and then transported to the outside for launching. The process of towing the airship from the large factory building to the outside is called transport. With the increasing requirements of airship flight and dwell time, the volume of modern airship tends to be large, and generally can reach 104 m³ orders of magnitude. The huge volume makes the airship vulnerable to the influence of the flow in all directions and the disturbance of large factory building, which makes it possible for the accident that the airship collides with the large factory building, thus it is of practical and important significance to study the transport process of airship. Computational Fluid Dynamics (CFD) is a technique for numerical simulation and analysis of physical phenomena such as fluid flow and heat transfer. In this paper, the numerical calculation model of large factory building and airship are established, and the fluid simulation software is used to simulate the large factory building turbulence and airship aerodynamic force, and then the influence of large factory building turbulence on airship transport process is analyzed.
In recent years, the research on airship aerodynamic performance is in-depth. Based on the potential flow theory, Fei Xie and Zhengyin Ye [2] used CFD numerical simulation to study the influence of different wave length, frequency and amplitude of airship skin on airship flow field characteristics. In order to reduce the drag coefficient of the airbag, Longtai Huang [3] analyzed the influence of the maximum section position, slenderness ratio and Reynolds number on the drag coefficient of the airbag by CFD method. The results show that the optimal design of the three is very necessary for the design of low resistance airbag and the search for the optimal shape. Pengjun Cao and Wan Jiang [4] used the response surface method and CFD numerical simulation to optimize the aerodynamic shape of a catamaran airship capsule with three main shape parameters: length width ratio, width thickness ratio and maximum section position. In order to reduce the aerodynamic drag and improve the maneuverability of airship, Weizhi Wang [5] used CFD method to find the optimal combination of hull, pod and tail. In order to describe the aerodynamic force of the airship in full state, B.Mueller [6] expressed the aero-dynamic force calculation formula of the rigid airship of which center of gravity and center of inertia do not coincide as a six degree of freedom equation, and considered the influence of the added mass and inertia, but did not consider the reciprocal of motion. Based on the wind tunnel test data of German Lotte-airship, Jinggang Miao [7] revised the existing theory, gave the six degree of freedom aerodynamic calculation model of general configuration airship, and identified the parameters of Lotte-airship aerodynamic calculation model by using the least square method. The identification results prove the applicability of the model. The above literatures mainly focus on the analysis of aerodynamic characteristics of airship in cruising or lingering conditions, but there is a lack of in-depth research on the aerodynamic resistance of airship in the process of transport.

2. Numerical calculation model
Due to the structural characteristics of the large factory building, it is easy to form turbulence around it. The CFD method can be used to establish the numerical model of the disturbance flow in a large factory building. The VOF model is used to solve the complex turbulence problem in the transport process of airship. The continuity equation and momentum equation of fluid are as follows [8]:

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) &= 0 \\
\frac{\partial \rho}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) &= -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + F_i
\end{align*}
\] (1)(2)

Where \( t \) is time and \( u_i \) is the velocity component. \( \rho \) is the density of the fluid and \( \mu \) indicates viscosity. \( \mu_t \) is turbulent viscosity and \( F_i \) is mass force.

The Reynolds number of the flow field studied in this paper is in the order of 106, so it can be judged that the flow is turbulent. The numerical calculation model is imported into the fluid analysis software for simulation, and the standard k-\( \varepsilon \) model is used for calculation according to the turbulence simulation method. The standard k-\( \varepsilon \) model is based on two transport equations to solve \( k \) and \( \varepsilon \). The coefficients are given by empirical formula. It is widely used and the amount of calculation is moderate, and has a lot of data accumulation. The convergence and calculation accuracy can meet the requirements of general engineering calculation. This paper adopts standard k-\( \varepsilon \) model as the turbulence model to simulate the transport process of airship. The transport equations of the turbulent kinetic energy \( k \) and dissipation rate \( \varepsilon \) are as follows [9]:

\[
\begin{align*}
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} &= \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon \\
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} &= \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + G_\varepsilon - \rho \varepsilon
\end{align*}
\] (3)
\[
\frac{\partial (\rho e)}{\partial t} + \frac{\partial (\rho e u_j)}{\partial x_j} = \frac{\partial}{\partial x_i} \left[ \mu \frac{\partial e}{\partial x_i} \right] + \frac{\partial}{\partial x_i} \left[ \frac{\partial p}{\partial x_i} \right] - \rho C_1 e \frac{\partial e}{\partial t} - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon}} \quad (4)
\]

Where \( G_k \) represents the generation term of turbulent kinetic energy \( k \) caused by the average velocity gradient, and \( C_1 \) and \( C_2 \) are empirical constants. \( \sigma_k \) is Prandtl number corresponding to turbulent flow energy \( k \), \( \sigma_\varepsilon \) represents the dissipation rate \( \varepsilon \) corresponding to Prandtl number. Where:

\[
\begin{align*}
\sigma_k &= 1.0, \sigma_\varepsilon = 1.2, C_\varepsilon = 1.9 \\
C_1 &= \max \left( 0.43, \frac{\eta}{\eta + 5} \right) \\
\eta &= \left( \frac{2E_\eta}{E_\varepsilon} \right)^{1/3} \frac{k}{\varepsilon} \\
E_\eta &= \frac{1}{2} \left( \frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_j} \right)
\end{align*}
\quad (5)
\]

3. Simulation of disturbance characteristics of large factory building

For complex flow problems, the numerical simulation method has the advantages of low costing, time consuming, and easy to obtain the data in the flow field. In this paper, the CFD software ANSYS FLUENT is used to simulate the disturbance flow of a large factory building, and the characteristics of the disturbance flow under different incoming flow conditions are analyzed.

3.1. Establishment of turbulence model for large factory building

A large factory building is established as a disturbance model, and the right-hand system as shown in Fig. 1 is established. The coordinate origin is set at the exit of the large factory building. The length, width and height of the large workshop are 266m, 140m and 117m respectively. According to the external dimension of large factory building, the dimension of calculation domain is set as 3000m × 2400m × 1800m, so that the flow can be fully developed [10].

ICEM CFD is used to mesh the computational domain. In order to improve the efficiency of mesh generation, unstructured tetrahedral mesh and prism mesh are used. The numerical calculation grid of large factory building surface is shown in Fig. 2.

3.2. Boundary condition setting

As for the setting of boundary conditions, the large factory building surface and ground are set as the wall boundary conditions. Wall boundary conditions are used to limit the fluid and solid regions. The flow surface of the external flow field is set as the velocity inlet, and the velocity inlet boundary
condition is used to define the flow velocity and the flow property related scalar of the flow inlet. The outlet of the outflow field is set as the outflow boundary condition. When the velocity and pressure at the exit are unknown before solving the flow problem, the exit boundary condition can be used. The surface and the side of the external flow field are set as symmetrical boundary conditions. When the flow in the field and the boundary shape have mirror symmetry, the symmetrical boundary condition can be set in the calculation.

Conditions set: The wind speed is constant at 4m/s, and the included angle between the wind speed and the longitudinal axis of large factory building is set as θ. When the wind speed is in the negative direction of z axis, θ is 0 degrees. θ rotates clockwise and every 30 degrees for a working condition. Since the model is symmetric about the yOz plane, the range of θ can be taken as [0, 180]. It can realize the simulation calculation under different incoming flow angles, As show in Fig. 3.

Figure 3. Schematic diagram of simulation conditions.

3.3. Simulation results and analysis
According to the established simulation model, the disturbance characteristics of large factory buildings with different inflow angles are obtained. The wind speed vectors under different inflow angles are shown in Fig. 4 to Fig. 10.

Figure 4. Wind speed vector diagram under 0 degrees inflow condition.  
Figure 5. Wind speed vector diagram under 30 degrees inflow condition.
Figure 6. Wind speed vector diagram under 60 degrees inflow condition.

Figure 7. Wind speed vector diagram under 90 degrees inflow condition.

Figure 8. Wind speed vector diagram under 120 degrees inflow condition.

Figure 9. Wind speed vector diagram under 150 degrees inflow condition.

Figure 10. Wind speed vector diagram under 180 degrees inflow condition.

The results show that under different flow angle conditions, there will be turbulence around and inside the large factory building. Under the condition of 30 degrees and 60 degrees inflow, the range of disturbance at the outlet of large factory building is small but the intensity of disturbance is large. The area of turbulence formation is mainly concentrated in the side and rear of the large factory building. Under the condition of 150 degrees and 180 degrees inflow, the range of disturbance at the outlet of
large factory building is large but the intensity of disturbance is low. The extreme value of wind speed appears at the condition of 120 degrees inflow, and the speed is 7.146m/s.

4. Aerodynamic simulation of airship transfer process
In order to further determine the influence of turbulence on airship transfer process, a three-dimensional simulation model of integrated airship and large factory building was established to simulate the aerodynamic force of airship transfer process under different incoming flow conditions.

4.1. Establishment of simulation model
The simulation models of the airship and the large factory building under different working conditions are established, and the right-hand system is established with the intersection of the plane of large factory building exit and the airship longitudinal axis as the coordinate origin, as shown in Fig. 11. Taking a test airship as the simulation model, the length of the test airship is 80m, the maximum diameter is 30m, and the X-shaped empennage structure is adopted. The height between the airship body center and the ground is 20m. The airship transports at a constant speed of 1m/s, and the distance between the foremost point of the airship and the origin of the coordinate system is taken as the transport distance.

![Figure 11. Schematic diagram of airship transfer model.](image)

According to the method in Section 3.1, topological computational domain is established, and the boundary conditions and calculation model are set. The turbulence during airship transport belongs to low-speed incompressible flow, and the Navier-Stokes equations for the Reynolds mean of incompressible flow and the k-ε turbulence models are used. The equation is discretized by finite volume method and the convection term is discretized by second order upwind scheme. The diffusion term adopts the central difference scheme, and the velocity pressure coupling method adopts SIMPLE algorithm.

4.2. Numerical calculation and result analysis
According to the established simulation model of airship transport process, the equivalent force and moment acting on airship body center along x, y and z axes during transport process are calculated, and the changes of resultant force and moment of airship in different incoming flow directions can be obtained, as shown in Fig. 12 to Fig. 18.
Figure 12. Equivalent force and moment curve at incoming flow of 0 degrees.

Figure 13. Equivalent force and moment curve at incoming flow of 30 degrees.

Figure 14. Equivalent force and moment curve at incoming flow of 60 degrees.

Figure 15. Equivalent force and moment curve at incoming flow of 90 degrees.
Figure 16. Equivalent force and moment curve at incoming flow of 120 degrees.

Figure 17. Equivalent force and moment curve at incoming flow of 150 degrees.

Figure 18. Equivalent force and moment curve at incoming flow of 180 degrees.

It can be seen from the simulation results in the process of airship transport that the equivalent force and equivalent moment acting on the airship body center fluctuate to a certain extent, and the variation of equivalent force and moment is different under different incoming flow angles, with the increase of transport distance. Under the condition of 60 degrees and 90 degrees inflow, the equivalent force and equivalent moment of airship fluctuate obviously within 80m distance, as the results of the disturbance of large factory building. When the airship leaves the disturbance range of the large factory building, the three-dimensional resultant force and moment tend to be stable. In the case of 0 degrees flow, the equivalent force in z-axis direction tends to 0, and the equivalent force and moment in other directions fluctuate obviously in the process of transport. Except for the case of 0 degrees flow, the equivalent moment on the z-axis is basically 0, and the equivalent force and moment in other directions are in the
state of fluctuation. The extremum of equivalent force and moment appear in the case of 60 degrees, the extremum of equivalent force is 83058 N and the extremum of equivalent moment is 618746 N·m.

Under the condition of 60 degrees inflow, the wind speed vector when the airship transfers 60m is shown in Fig. 19, and the wind load on the airship surface is shown in Fig. 20.

![Figure 19. Vector diagram of wind speed at 60m transport distance.](image1)

![Figure 20. Wind pressure diagram of airship surface at 60m transport distance.](image2)

The results show that a strong turbulence is formed on the airship surface and at the outlet of the large factory building. The wind speed near the windward tail of airship is larger than 10m/s. The turbulence around the airship results in the uneven distribution of wind load on the airship surface. The positive pressure at the front of the windward tail of the airship is large, and the extreme value is 55.99Pa. Negative pressure appears on the upwind side and bottom of the airship, and the extreme value of negative pressure is -72.89 Pa.

5. Conclusion

The conclusions of this paper are as follows:

1. The flow from different directions will form a disturbance around the large factory building. Under the condition of 30 degrees and 60 degrees inflow, the range of disturbance at the outlet of large factory building is small but the intensity of disturbance is large. Under the condition of 150 degrees and 180 degrees inflow, the range of disturbance at the outlet of large factory building is large but the intensity of disturbance is low.

2. In the process of airship transport, the wind load on the airship surface will change due to the disturbance of large factory building, and the equivalent force and moment acting on the airship body center will fluctuate to a certain extent. The variation of equivalent force and moment of airship is different under different incoming flow angles.

3. The wind speed around the airship changes dramatically due to the disturbance of large factory building, which results in the uneven distribution of wind load on the airship surface. The positive pressure mainly appears in the front of the windward tail of the airship, and the negative pressure mainly appears in the windward side and bottom of the airship.

References

[1] XF. Gao, DB. Duan, X. Guo. Envelope volume design and research on changing rule of net buoyancy for stratospheric airship. In: Long ML, Fu YF, editors. Proceedings of China aerostat Conference 2007. Beijing: Aviation Industry Press.; 2007. pp. 10–15.

[2] F. Fei and ZY. Ye, Influence of skin wave on airship resistance, Engineering mechanics. 2009, 26(01): pp. 250-256.

[3] LT.. Huang, Aerodynamic shape optimization of airship airbag based on CFD, Computer simulation. 2010, 27(09): pp. 44-47.
[4] JP. Cao, W. Jiang and H. Zhang, Aerodynamic shape optimization of a catamaran Airship Based on response surface method, Aviation computing technology. 2012, 42(01): pp. 39-42.
[5] WZ. Wang and XQ. Liu, Aerodynamic characteristics analysis of stratospheric airship configuration, Chinese spacecraft recovery & remote sensing. 2007, 28(03): pp. 55-61.
[6] B. Mueller. Development of an aerodynamic model and control law design for a high altitude airship. Aero-space Industries Association of America-2004-6279. Washington DC; 2004. pp. 1156-1159.
[7] G. Miao, X. Yang, JH. Zhou. Airship aerodynamic semi-empirical model and its parameter identification. In: Long ML, Fu YF, editors. Proceedings of China aerostat Conference 2007. Beijing: Aviation Industry Press.; 2004. p. 282–287.
[8] YJ. Qian, Fundamentals of fluid kinematics and dynamics, aerodynamics, Beijing University of Aeronautics and Astronautics Press., Beijing, 1999, pp. 84-87.
[9] Thorston L, Peter F, Andreas J. Summary of aerodynamic studies on the Lotte airship. In 4th International Airship Convention and Exhibition; 2002 July 28-31; Cambridge, England.
[10] PP. Mi, JH. Meng, MY. Lv, Aerodynamic performance and overall parameters analysis of floating hybrid airship. Journal of Beijing University of Aeronautics and Astronautics. 2015., 41(6): pp. 1108–1116.