SIMULATION OF DISCRETE GUST ACCORDING TO AIRSHIP SAFETY STANDARDS

Xuan Tung Ta[1], Manh Hieu Vu[1,*], Duc Quyen Vu[1], Nam Quy Nguyen[2]

[1]*Faculty of Mechanical Engineering, Le Quy Don Technical University, Hanoi, Vietnam*
[2]*Faculty of Special Equipment, Le Quy Don Technical University, Hanoi, Vietnam*

Abstract

One of the factors that greatly affects the flight safety of airships is weather conditions and in particular, discrete gust with a speed equivalent to the speed of the airship. This article presented a method of numerical simulation of discrete gust according to flight safety standards. The discrete gust is considered from the bottom up with velocity in two forms: rectangular velocity distribution and variable velocity as a function of cosin. Research shows that in order for the velocity distribution to be most similar to the flight safety standards, it is recommended to choose a calculated area of size $25L \times 10L \times 2.5L$. This simulation method can be used to study other discrete gust according to flight safety standards in the actual operation of airships.

Keywords: Airship; discrete gust; numerical method.

1. Introduction

When operating, the airship must fully meet the requirements of flight safety standards, flight modes as well as possible situations must be carefully and accurately considered and calculated in advance to ensure absolute safety during operation. To do so, the airship must meet the flight safety standards presented in documents [1, 2]. In order to know if an airship meets those safety standards, manufactured airship must be tested. This makes the process of designing and manufacturing airships lengthy and costly. Currently, this problem can be solved by numerical simulation method. However, in our country there are no published studies on this method. There is a lot of research on discrete gust in the world. In the past, gust was studied experimentally [3, 4]. In recent years, with the development of computational methods, gust has been simulated numerically [5-8]. The effect of gust on airplanes has been studied extensively, but there is less research on the effect of gust on airship. The authors continue a series of studies on airships presented in [9, 10] by building a method to numerically simulate of discrete gust acting on the airship. In this study, simulation of discrete gust is carried out to get the closest to the flight safety standard in both 2D and 3D cases.

The airship is assumed to be subjected to the loads resulting from encounters with the following atmospheric gusts in level flight:

Discrete gust of $U_m = 7.62\,\text{m/s}$ while flying at speed V_H.

Discrete gust of $U_m = 10.67\,\text{m/s}$ while flying at speed V_B.

Gust shapes and intensities are defined as follows: $U = \dfrac{U_m}{2}\left(1 - \cos\dfrac{\pi X}{H}\right)$

where U_m is gust velocity, X is penetration distance, $0 \le X \le 2H$, H is gust gradient length $\dfrac{L}{4} \le H \le 244$ (m), L is length of airship, V_H is maximum level flight speed at sea level, V_B is design speed for maximum gust intensity.

In this study, we will clarify two issues:

- Simulation discrete gust according to flight safety standards in case no airship has passed.

- Select the optimal flight altitude of the airship in order to optimize computer resources and for the velocity distribution to be most similar to the flight safety standards.

When simulation the discrete gust, we make the following assumptions to simplify the problem:

- The discrete gust does not change the direction and magnitude of the velocity of the airship which means that the motion of the airship is always a uniform linear motion;

- The flow of the discrete gust and the flow near the airship is turbulent.

The problem is solved numerically, using the program ANSYS 15 (lisence number 00632255). We consider the discrete gust from the bottom up with velocity in two forms: constant velocity (rectangular velocity distribution) and variable velocity as a function of cos, where gust velocity $U_m = 10.67\,\text{m/s}$. The velocity description function is written in C^{++} for the rectangular velocity distribution and is included in Ansys Fluent as a UDF function.

2. The system of aerodynamic equations

To determine the parameters of the air flow, it is necessary to solve the system of equations describing the movement of the air flow around the airship. The equations include:

- Continuity equation:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \tag{1}$$

- Motion equation:

$$\frac{\partial \rho \overline{u_i}}{\partial t} + \frac{\partial \rho \overline{u_i u_j}}{\partial x_j} = -\frac{\partial \overline{p}}{\partial x_i} + \mu \frac{\partial^2 \overline{u_i}}{\partial x_j \partial x_i} - \frac{\partial \rho \overline{u_i' u_j'}}{\partial x_j} \tag{2}$$

The component $(-\rho \overline{u_i' u_j'})$ in this equation is the Reynolds stress tensor and is denoted by τ_{ij}.

- Equation for scalar quantities (Commonly written for scalar quantities: kinetic energy, velocities of dissipation, specific velocities of dissipation):

$$\rho \frac{\partial \overline{\phi}}{\partial t} + \rho \overline{u}_j \frac{\partial \overline{\phi}}{\partial x_j} = A - B \frac{\partial}{\partial x_i}\left[(\mu + C)\frac{\partial \overline{\phi}}{\partial x_j} \right] + D \tag{3}$$

The symbols in equation are listed in Table 1.

Table 1. Scalar quantities

Scalar quantities	$\overline{\phi}$	A	B	C
Kinetic energy	k	$\tau_{ij}\dfrac{\partial \overline{u_i}}{\partial x_j}$	$\beta^* \rho k \omega$	$\dfrac{\mu_t}{\sigma_k}$
Velocities of dissipation	ε	$c_{\varepsilon 1}\dfrac{\varepsilon}{k}\tau_{ij}\dfrac{\partial \overline{u_i}}{\partial x_j}$	$c_{\varepsilon 2}\rho\dfrac{\varepsilon^2}{k}$	$\dfrac{\mu_t}{\sigma_\varepsilon}$
Specific velocities of dissipation	ω	$\alpha\dfrac{\omega}{k}\tau_{ij}\dfrac{\partial \overline{u_i}}{\partial x_j}$	$\beta\rho\omega^2$	$\mu_t \sigma_\omega$

In this article, the SST $k-\omega$ model is used, which is a combination of two models $k-\varepsilon$ and $k-\omega$. The values of the constants in the model are taken from [11].

- Equation of state:

$$p = \rho R T \tag{4}$$

Here, we only consider when the airship is in steady motion, so the equations in the system are independent of time.

The system of equations is a closed differential equation system and to solve this system needs to be discretized to return to the system of algebraic equations. All these steps were done with using Ansys Fluent.

3. Calculation results

3.1. Simulation of discrete gust when no airship is passing

When simulating the discrete gust, we must ensure that the velocity distribution in the discrete gust is as close to the velocity distribution according to the flight safety standards as possible, and that the outside of the discrete gust is static atmosphere. However, when simulation discrete gust in finite space, there are some notes: The discrete gust will expand in space in the direction of the flow, the boundary conditions affecting the extent of the discrete gust and the velocity outside the discrete gust in that finite space. Therefore, for this problem, determining the size of the calculated area to ensure the above requirements plays a very important role. In order to determine the size of the calculated area and the necessary boundary conditions, we simulate the discrete gust in 2D form. With the 2D problem, we simulate with two cases:

- The calculated area size is 10L × 5L:
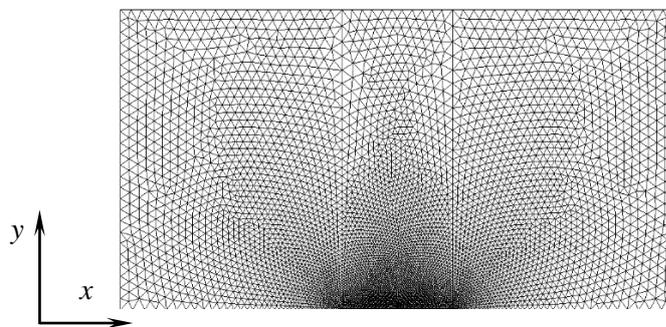
Fig. 1. 2D calculated area is 10L × 5L, rectangular velocity distribution.

This calculated area is divided into 11559 triangular grid cells. The cells are smoothed where the discrete gust enters to simulate the discrete gust more accurately (bold part in Fig. 1). The discrete gust section with width 2L is set with velocity inlet boundary condition with $V_x = 0$ and $V_y = 10.67\,\text{m/s},$ the remaining edges are set with velocity inlet condition with $V_x = 0$ and $V_y = 0\,\text{m/s}$ (Fig. 2). Select such a grid to use the dynamic grid method to calculate for future research. The grids will be rebuilt over time, and the grid criteria will determine the size of the grid cells. The simulation results obtained the pressure field and the velocity field (Fig. 3). We see that around the discrete gust two vortices appear, which distort the velocity distribution in the spatial region around the discrete gust. Inside the discrete gust, the velocity is not uniform.

The discrete gust is deviated from the lower boundary of the calculated area and changes the velocity profile at the discrete gust cross-section.

As a result, the cross-sectional velocity distribution at a position farther from the discrete gust lower boundary is relatively different from the original rectangular velocity distribution. One of the reasons for this is that the calculated area size is not large enough. Another cause of vortices is the type of boundary conditions placed where V_x, V_y values are on the boundaries of the calculated area. Thus, in order to eliminate unwanted vortices and have a velocity distribution close to the flight safety standards inside and outside of discrete gust, we will increase the size of the calculated area and change the boundary conditions accordingly. The calculated area will increase to the size of 25L × 10L (this is the optimized size studied specifically in [12] to optimize computational resources, calculation time and ensure reliable results), and the values V_x, V_y on the boundary as well as the type of boundary condition are changed.
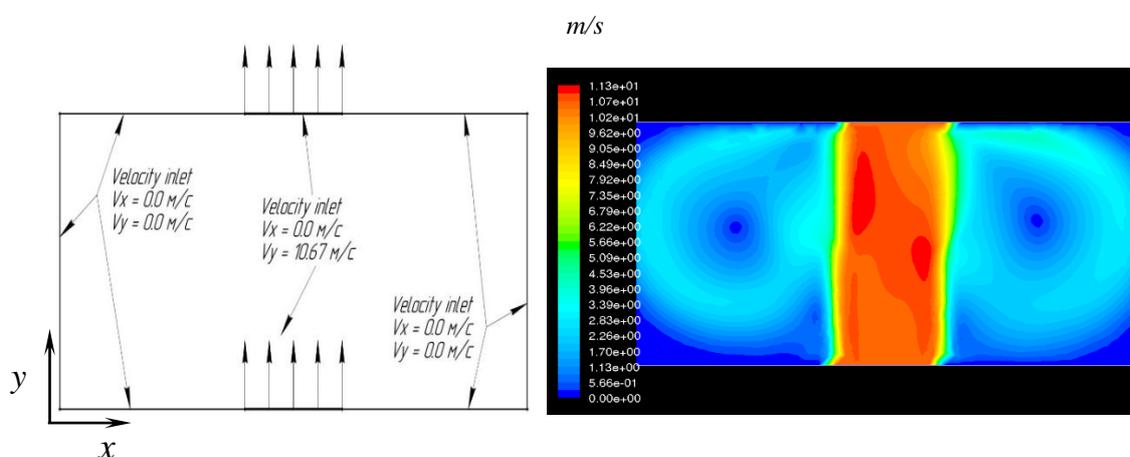
Fig. 2. Boundary conditions.   *Fig. 3. Velocity field when calculated area is 10L × 5L.*

- The calculated area size is 25L × 10L:

The 25L × 10L calculated area will have 14847 triangular grid cells (Fig. 4), and the grid cells in the face where the discrete gust in will be smoothed and the number of grids increased to ensure the calculation results. On the remaining boundary of the calculated area, we will set outflow condition and at the lower and upper surface where there is noise flow, set the velocity inlet condition, the rest of the upper surface $V_x = 0$ and $V_y = 0.4\,\text{m/s}$ (Fig. 5). The velocity field is shown in Fig. 6. From this it can be

observed that with this calculated area size and boundary condition, the discrete gust keeps the velocity distribution close to the velocity distribution in the original cross section (including even at sections that are quite far from the original one) and do not form vortices in the surrounding space.
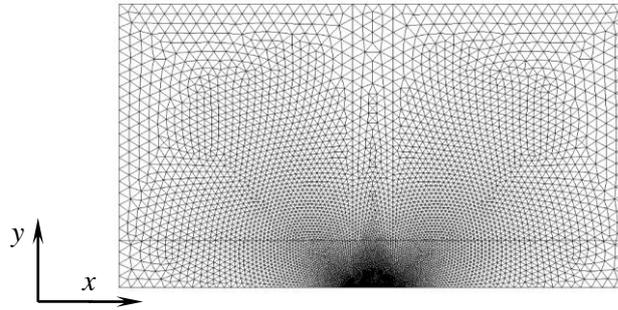

Fig. 4. 2D calculated area size is 25L×10L, rectangular velocity distribution.

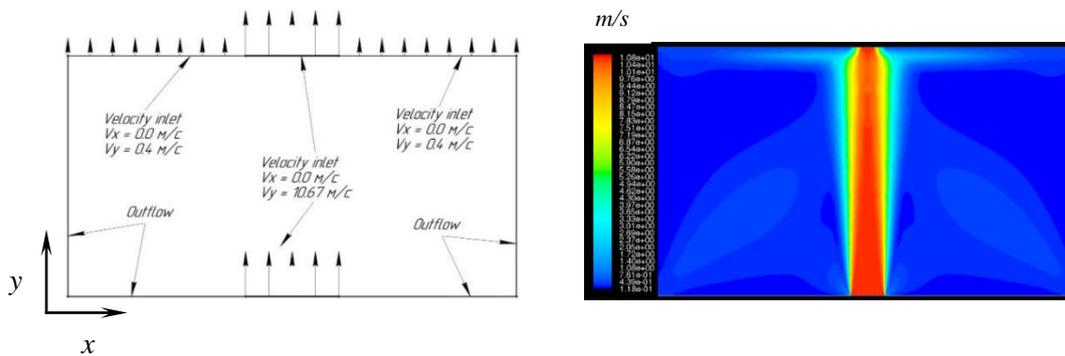*Fig. 5. Boundary conditions.*   *Fig. 6. Velocity field when calculated area is 25L × 10L.*

We obtained the velocity distribution along the trajectory of the airship (at a distance $h = 2.5L$) that most closely to the flight safety standard. Here are some results obtained with V_x and V_y at different distances h: $h = 2.5L; 5L; 7.5L$.

From V_y's graphs in Fig. 7, it can be seen that the discrete gust retains its shape very well even at locations far from the original cross-section. From Fig. 8 we see that the velocity modulus $|V_x| \leq 0.42 \, \text{m/s}$ and its effect on the flow surrounds the airship flow is small and can be ignored. At the distance $h = 2.5L$, the longitudinal velocity value outside the discrete gust is the smallest, so in the later calculations we choose $h = 2.5L$.

In the flight safety standard, there is a case of velocity distribution according to the cos function, the research results obtained with the discrete gust in this case are in Fig. 9 and Fig. 10.

24

For this case we see that the core of the discrete gust fades away from the first cross-section, and that the discrete gust tends to widen at the upper edge of the calculated area.
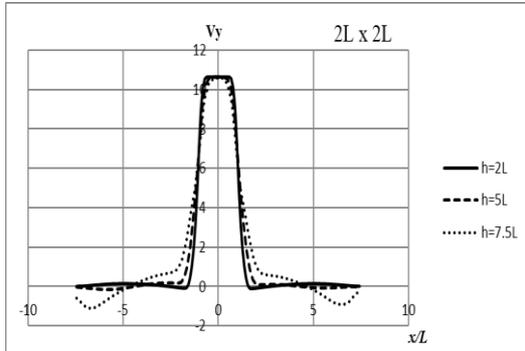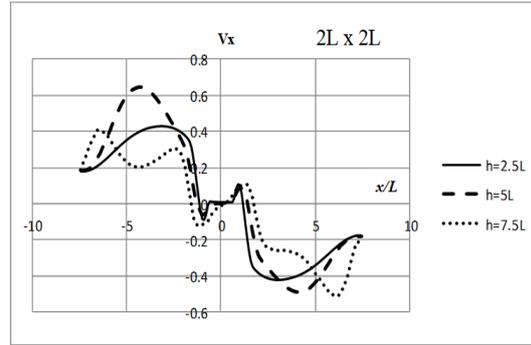

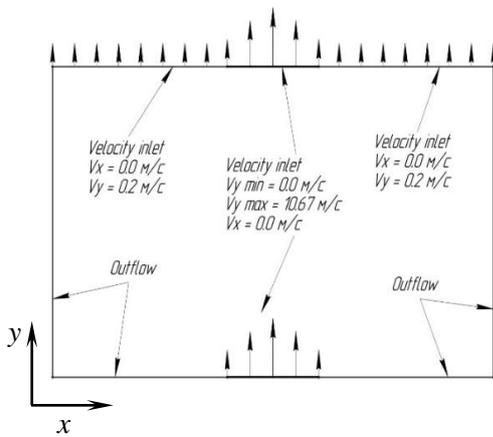*Fig. 7. V_y when b = 2L.*

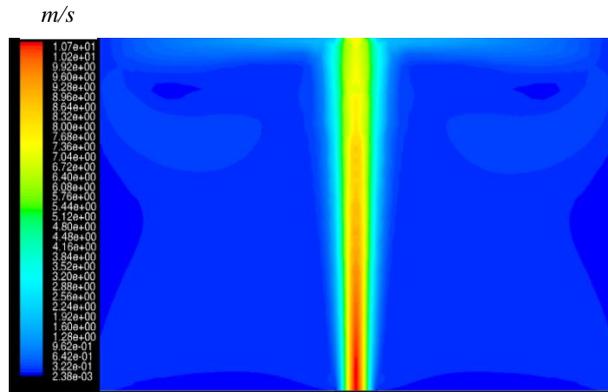*Fig. 8. V_x when b = 2L.*

Fig. 9. Boundary conditions.

Fig. 10. Velocity field when calculated area is 25L × 10L.

- 3D calculated area: The calculated area has a rectangular box shape (Fig. 11) with the size of 25L × 10L × 5L, the number of calculated cells is 1.09 million cells and is concentrated mostly in the discrete gust area to ensure the accuracy of the results. For the 3D calculated area, the boundary conditions are similar to the 2D simulation case (Fig. 12 and Fig. 13) to ensure the requirements of the problem, in addition, the V_z component in the part out of the discrete gust has zero value.

The following are some simulation results with V_x, V_y and V_z (Fig. 14 - 16) for the case of rectangular velocity distribution, and for the case of cosine velocity distribution, which are presented in detail in [3].
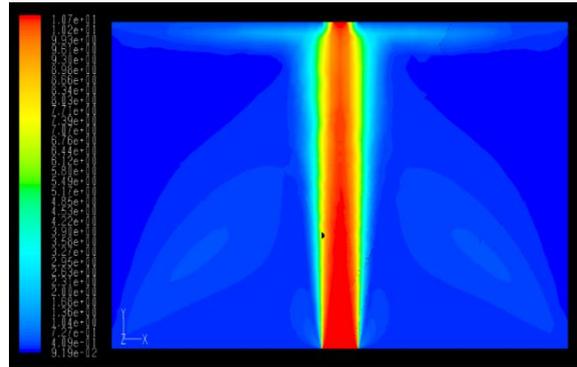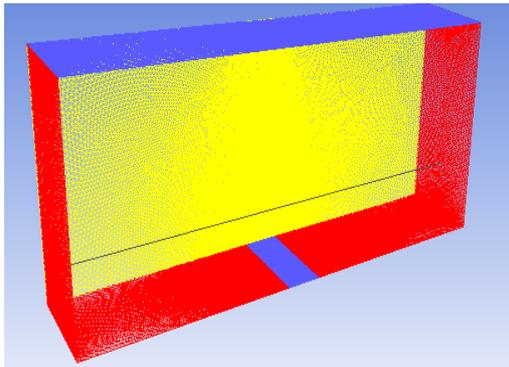
Fig. 11. Calculated area 3D: 25L × 10L × 5L

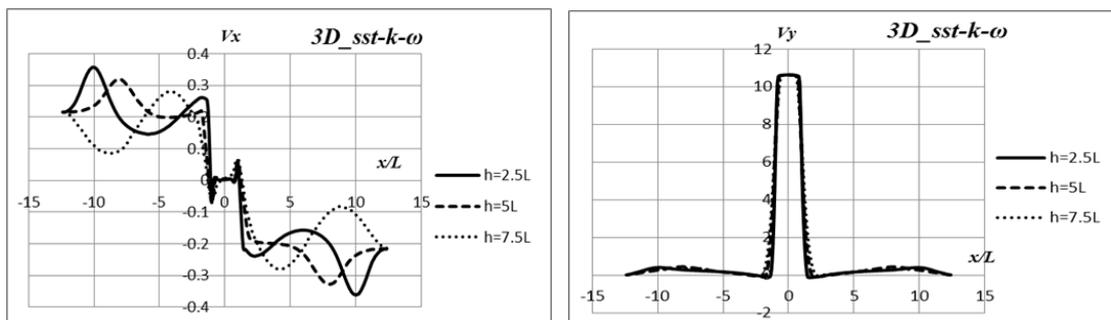*Fig. 12. Velocity field at the surface Oxy.*

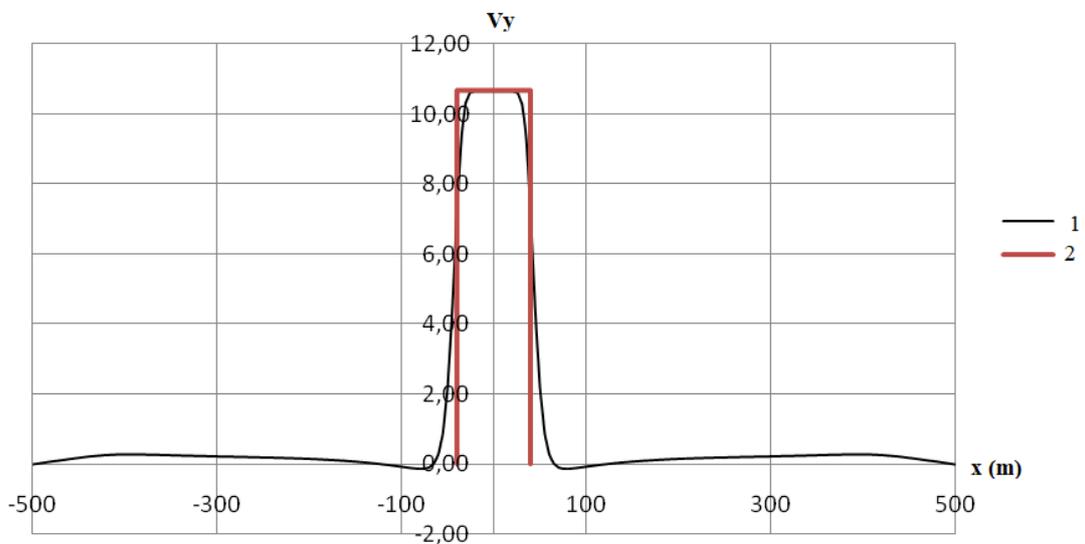*Fig. 13. V_x and V_y velocities in 3D simulation.*

Fig. 14. Velocity distribution V_y when the airship's altitude h = 2.5L:
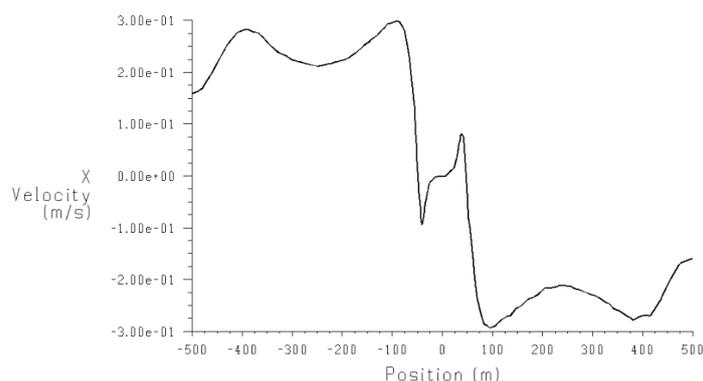*1 - Simulation results; 2 - According to flight safety standards.*

Fig. 15. Velocity distribution V_x at the airship's altitude h = 2.5L.

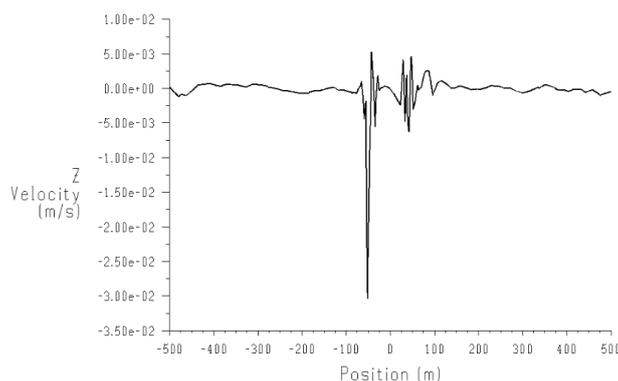*Fig. 16. Velocity distribution V_z when the airship's altitude h = 2.5L.*

It is seen that for $h = 2.5L$ and such simulation we have velocity components $V_x \leq 0.3$ m/s, $V_z \leq 0.03$ m/s, the most influential change is V_x but only accounts for about 2.8% of V_y's value, so the influence of components V_x and V_z can be ignored. The airship can therefore be considered to be moving in the region outside the discrete gust in still air conditions.

3.2. Select the flight altitude of the airship

This is an important issue when studying the interaction between the airship and the discrete gust, it determines the accuracy of the results of future research. When choosing the altitude of the airship, we must choose so that the trajectory of the airship is parallel to the lower boundary of the calculated area and the velocity at the core of the discrete gust must be distributed according to the flight safety standards while the region velocity outside the discrete gust is zero or small enough to not affect the calculation results. From the above results, we choose $h = 2.5L$ as the altitude of the airship when studying the interaction between the airship and the discrete gust.

4. Conclusion

The study presented a method to simulate discrete gust according to airship safety standards in both 2D and 3D cases. The discrete gust is considered from the bottom up with velocity in two forms: rectangular velocity distribution and variable velocity as a function of cos;

In order for the velocity distribution to be most similar to the flight safety standards, and to optimize computer resources, it is recommended to choose a calculated area of size 25L × 10L × 2.5L.

References

[1] *Критерии летной годности для дирижаблей*. Авиарегистр Межгосударственного Авиационного Комитета от 9 октября, 1996 г.

[2] *Airship Design Criteria*, FAA P-8110-2, 1992.

[3] J. Jr. Gilman and R. M. Bennett, "A wind-tunnel technique for measuring frequency-response functions for gust load analyses", *J Aircraft*, Vol. 3, pp. 535-540, 1966.

[4] M. Kobayakawa and H. Maeda, "An experiment on the gust response of a transport airplane by free-flight model", *J Aircraft*, Vol. 15, pp. 540-541, 1978. DOI: 10.2514/3.58403

[5] Z. Wu, Y. Cao, and M. Ismail, "Gust loads on aircraft", *The Aeronautical Journal*, Vol. 123, Iss. 1266, pp. 1216-1274, August 2019. DOI: 10.1017/aer.2019.48

[6] G. Yang and S. Obayashi, "Numerical analyses of discrete gust response for an aircraft", *J Aircraft*, Vol. 41, pp. 1353-1359, 2004. DOI: 10.2514/1.2531

[7] R. Singh and J. D. Baeder, *Generalized moving gust response using CFD with application to airfoil-vortex interaction*, AIAA Paper 1997.

[8] R. E. Bartels, *Development, verification and use of gust modeling in the NASA computational fluid dynamics code FUN3D*, 2012.

[9] Та Суан Тунг, "*Численное моделирование обтекания и расчет аэродинамических характеристик дирижабля, перемещающегося через атмосферные течения струйного типа*", Диссертация, 2018.

[10] Xuan Tung Ta, Manh Hieu Vu, Van Chien Vu, "The effectiveness of the new-generation airship's tail-wing schemes and its effect on the aerodynamic characteristics of the airship", *Journal of Science and Technique*, Vol. 17, No. 01, pp. 17-24, 2022. DOI: 10.56651/lqdtu.jst.v17.n01.298

[11] И. А. Белов, С. А. Исаев, *Моделирование турбулентных течений: Учебное пособие*. СПб.: Балт. гос. техн. ун-т, 2001, 143 с.

[12] В. Т. Грумондз, Н. В. Семенчиков, О. В. Яковлевский, "Аэромеханика дирижабля", *Наука*, pp. 41-50, 2017.

MÔ PHỎNG DÒNG NHIỄU ĐỘNG KHÔNG KHÍ
THEO TIÊU CHUẨN AN TOÀN BAY CỦA KHÍ CẦU

Tạ Xuân Tùng[a], Vũ Mạnh Hiếu[a], Vũ Đức Quyến[a], Nguyễn Nam Qúy[b]

[a]*Khoa Cơ khí, Trường Đại học Kỹ thuật Lê Quý Đôn*
[b]*Khoa Thiết bị đặc biệt, Trường Đại học Kỹ thuật Lê Quý Đôn*

Tóm tắt: Một trong những yếu tố ảnh hưởng lớn tới an toàn bay của khí cầu là điều kiện thời tiết bất lợi và cụ thể là các dòng gió nhiễu có vận tốc xấp xỉ tốc độ chuyển động của khí cầu. Bài báo này trình bày phương pháp mô phỏng số dòng gió nhiễu theo tiêu chuẩn an toàn bay. Dòng gió nhiễu được xem xét thổi từ dưới lên ở hai dạng: phân bố dạng hình chữ nhật và phân bố theo hàm cosin. Nghiên cứu chỉ ra rằng, để phân bố vận tốc giống với tiêu chuẩn an toàn bay nhất cần chọn vùng tính toán có kích thước 25L × 10L × 2,5L. Phương pháp mô phỏng này có thể sử dụng để nghiên cứu các dòng gió nhiễu khác theo tiêu chuẩn an toàn bay trong thực tế khai thác và vận hành các loại khí cầu.

***Từ khóa:** Khí cầu; dòng gió nhiễu; phương pháp số.*

❑