JAST, Vol. 14, No. 1, pp

Scientific-Research Article

Investigation of fin changes effects on aerodynamic performance of airship

Sahar Noori [1]*, Mohammad Saleh Afshar [2] , Nima Karimi [3]

1- Amirkabir University of Technology, School of Aerospace Engineering

2- Iranian Space Research Center, Space Transportation Research Institute

3-Aerospace Research Institute

* Tehran, Hafez St.

Email: S_noori@aut.ac.ir

Airships usually have low cruising speed due to their large volume and high drag level. This makes the aerodynamic design of the vehicle, including the surfaces shape, the length-to-diameter ratio and the position of the fins, all very important. Furthermore, an important parameter in the vehicle aerodynamic drag is determining the flow separation area at the rear of the air vehicle. The flow separation plays an essential role in the amount of drag and lift force, so the location of the fins and the design of the rear of the airship will be very important. By using both analytical and numerical methods, this study examines the aerodynamic efficiency of an airship in three different configurations, focusing on the location, type, and angle of attack of the fin, and compares analytical and numerical results. According to studies conducted among the types of fins, the cross-type will have the best performance among the fins in terms of lift-drag ratio. Also, moving the fins forward and distancing them from the rear of the vehicle disrupts the flow pattern at the rear of the vehicle and delays separation. This will improve aerodynamic efficiency and improve the lift-drag ratio of the vehicle.

Keywords: *Airship, Aerodynamic, Aerodynamic configuration*

Nomenclature

Description	Symbol
Mass source	S_m
Static pressure	p
Stress tensor	τ_{ij}
Viscosity	μ
Total energy	E_t
External physical forces	F_i

Introduction

The airships' development history dates back to the 18th century. But the first documents about the design of airships date back to about 1941. A document released by the United States Department of War outlining the process of aerodynamic design, aerodynamic computing, and airship configuration. In this research, the relations are analytical and computational and linearized relations have been

1 Assistant professor (corresponding author)

2 M.Sc

3 M.Sc

Receive Date: 5 June 2021

Accepte Date: 7 July 2021   Print ISSN: 1735-2134

used [1]. The fluid flow solution on an airship may seem simple in numerical form, but it has many operational complexities. Due to its aerodynamic shape and wide surface, the airship does not often have sudden fractures and deformations on the surface of the hull, and the surface changes in the hull are gradual and with a specified pitch. This causes the fluid flow to tend to laminar flow, especially in the boundary layer. As a result, the Reynolds number in airships is high due to the long size, low speed, low altitude, and low speed, which increases the tendency of the flow to be turbulent. As a result, modeling and derivation of coefficients and forces on the vehicle in a wind tunnel would face some difficulties due to the high Reynolds number. Furthermore, numerical solutions as an engineering tool should be able to predict flow patterns on vehicles. For this purpose, various parameters such as length and diameter of the vehicle will be important for the design team. According to a numerical simulation in China, increasing the vehicle length will increase the drag force with Reynolds Averaged Navier-Stokes (RANS). In fact, flow separation is a factor in increasing the drag force, and this factor is directly related to the length of the vehicle. In other words, the flow separation manner and its location has a direct effect on the amount of drag force, so length, diameter and length-to-diameter ratio have a direct effect on drag force due to changes in the separation area. Various calculations have shown that flow separation occurs at 70 to 80% of the vehicle length. The later the separation occurs, the more laminar flow the vehicle will experience on its hull, which reduces the drag force. The flow lines around the vehicle can be used to distinguish the transition location from laminar to turbulent flow. This method can provide an estimate of the separation area (Figure 1).

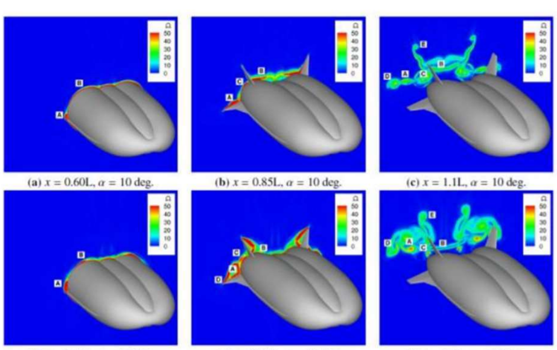

Figure 1- Visualization of the flow separation location on the Airlander airship

Examples of numerically solving fluid flow on the hull of an airship are checked in order to calculate aerodynamic forces including drag. In some of these examples, the hull of the vehicle is assumed to be rigid and the flow of fluid around the vehicle is simulated by numerical methods (Figure 2). Fluid-structure interactions have been seen in some cases, but there is no need to consider them for calculating fluid flow around the vehicle and forces and coefficients. Also, due to the wide surface of the vehicle, considering details such as fins and some jut does not affect the calculation of the total drag force.

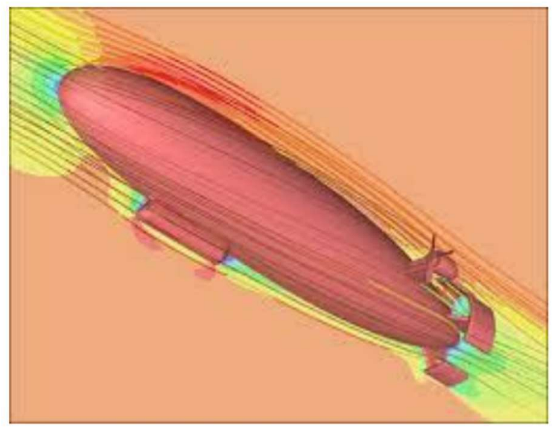

Figure 2 - Simulation of flow lines around an airship

Numerical solution of the flow around the airship carries challenges such as observing the physical phenomena of the flow. In one study, Kamal et al. (2004) Conducted a numerical simulation around an airship by using various turbulence flow models [3]. In this research, three turbulence flow models were used to observe the types of vehicle solution vortices. The results show that the VMS-LES model has a better performance than the two models of K-epsilon and large eddy simulation turbulent flow model.

In another example, Shields et al. [4] From the University of Virginia attempted to simulate a 1: 75 scaled version of an airship using numerical methods and derived the corresponding coefficients. The results of this study show that numerical solution does not provide an accurate answer in the calculation of lift force but can be effective for the design and derivation phase of processes.

In another study, Voloshin et al. (2012), analyzed three turbulence flow models on an airship using numerical solution [5]. In this study, three models of K-omega turbulence, K-epsilon and Spallart Allmaram were used. The results show that the

Spallart Allmaram turbulence flow model offers the best results compared to other models in comparing with experimental method.

In another study, Amelda et al. used a new solver to evaluate the aerodynamic characteristics of an airship [6]. This research, which has been done by analytical and experimental solution, shows that the separation of flow on the airship is important and has an effect on the stability status and coefficients of the vehicle. In addition, the effects of the dihedral angle of the fins and the effect of the vertical tail on the aerodynamics of the vehicle have been investigated.

Experimental research in the field of airship has also maintained its position. In China, for example, Ping et al. used experimental solutions to calculate the coefficients and aerodynamic forces of a ship on a smaller scale in a wind tunnel. In this study, coefficients and forces in different pitch and yaw angles were measured and the results of this study were compared with numerical solution [7].

A very important and key example has been found in articles and researches that have been done on the effects of fin shape and its location. This article, published in 2015, examines the types of fins on a fixed hull and the effects of changes in the type and arrangement of the fins on the two coefficients of lift and drag [8]. The results of this numerical solution show that the cross-type fin has a better aerodynamic performance than the two types of X-type and Y-type, and the lift-drag ratio is optimal for the cross-type in the same condition as other fin types.

In 2019, Jefferson et al. tried to calculate the coefficients and forces on the airship using numerical methods [9]. The results of this study show that numerical solution validation using experimental results obtained from wind tunnels is difficult, but this solution will be acceptable for the initial phases and the initial design.

In the present paper, using two methods of numerical and analytical solution, the fluid flow around an airship in different configuration models is solved and the results of these two solutions are compared with each other. According to what was believed, the results of the solution show that numerical solution is more efficient than analytical solution, and using numerical solution, the physics of fluid flow and how the rear of the vehicle is separated can be studied more thoroughly. We also attempted to examine the flow physics in this area with greater accuracy by analyzing the fluid flow behavior and the details of the flow around the vehicle in this research.

Different configurations of vehicle

There are different parameters for modeling an airship, but the most important parameter is the volume of the ship's compartment. With this parameter, other parameters of the airship can be derived, such as the diameter of the hull, the length of the hull, the angle of the surfaces, the location of the fins and the external configuration generalities configuration. In the present study, the volume of the airship compartment is 27,000 m^3 and therefore the dimensions of the ship can be estimated approximately. For this purpose, three different configurations have been considered for the ship, the picture of these three variants is given in Figures 3, 4 and 5. It should also be noted that configuration modeling has been done using Gambit software. The difference between the configurations is in the slenderness coefficient (length-to-diameter ratio), fin length and location. Since the main purpose of this research is to investigate the external configuration, an attempt has been made to select three configurations with different fin sizes. Also, the type of fins of the vehicle rear, their number and arrangement and the effects of these changes on the flow pattern and various parameters will be examined with these three configurations. It should be noted that the gondola is not modeled in this simulation and only the airship with the fin is modeled.

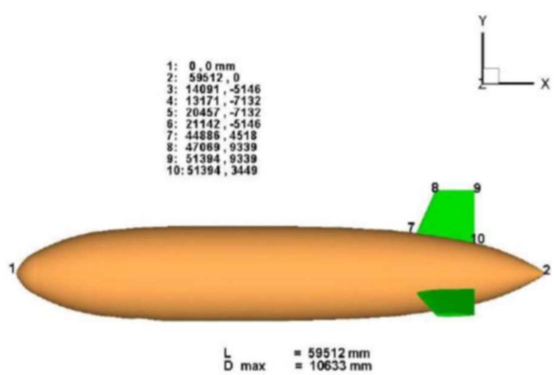

Figure 3 - Variant one

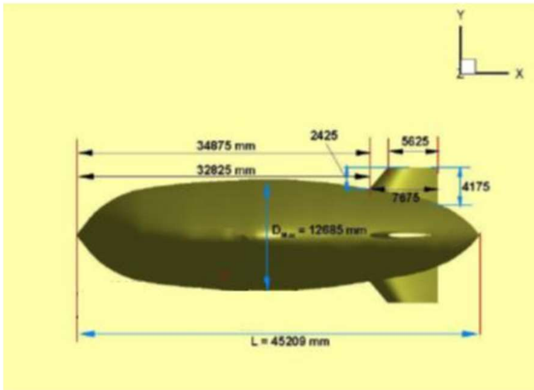

Figure 4 - Variant two

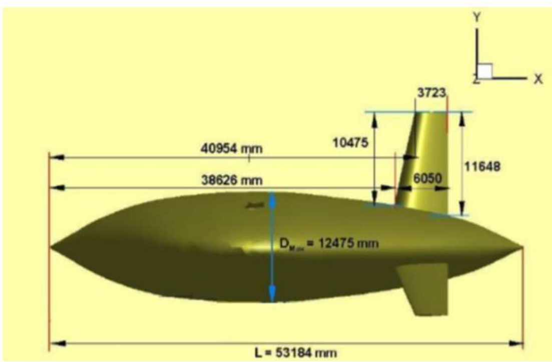

Figure 5 - Variant three

The governing equations

The equation solved in this research is a simplified form of Navier-Stokes equations. The conservation of mass equation or the continuity equation in the tensor form is written as follows:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=S_m \qquad (1)$$

The above equation is a general form of the conservation of mass equation for compressible and unstable flows. In the latter equation, the expression S_m is also a mass source.
conservation of momentum equations in tensor format are written as follows:

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial \tau_{ij}}{\partial x_i}+\rho g_i+F_i \qquad (2)$$

In that, i.j = 1.2.3. … and P is static pressure, τ_{ij} is stress tensor, ρ g_i and F_i are gravitational physical force and external physical force towards i. F_i can also be terms such as the porous medium model. For example, stress tensor τ_{ij} is as follows:

$$\tau_{ij}=\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_i}{\partial x_i}\right)\right]-\frac{2}{3}\mu\frac{\partial u_i}{\partial x_i}\delta_{ij} \qquad (3)$$

That μ is molecular viscosity and the second term on the right is the impact of volumetric expansion.
Also, the general state of the energy equation is as follows:

$$\frac{\partial E_t}{\partial t}+\nabla.E_t V=\frac{\partial Q}{\partial t}-\nabla.q+\rho B.V+\nabla.\left(\pi_{ij} V\right) \qquad (4)$$

wherein:

$$\pi_{ij}=-P\delta_{ij}+\mu\left(\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right),\ i,j,k=1,2,3 \qquad (5)$$

$$also\ if\ \ i=j\Rightarrow\delta_{ij}=1\ otherwise\ \ \delta_{ij}=0$$

$$E_t=\rho\left(e+\frac{V^2}{2}+P.E.+...\right) \qquad (6)$$

E_t is the total energy.

The expressions on the left of the energy equation include the rate of increase of total energy per unit volume and the rate of change as displacement in the control volume. Meanwhile, the expressions on the right side of the equation express the rate of heat production per unit volume due to external factors, the rate of heat changes per unit volume due to leading in control surfaces, the work done by volumetric body forces per unit volume and finally the work done by the surface forces is on the control volume per unit volume.
Fluid flow in the present problem is solved based on the K-epsilon turbulence flow model. This model of turbulence is able to provide suitable answers in both subsonic and supersonic regimes near the wall and free stream. Also, considering that the main purpose of this paper is to calculate the coefficients and forces, especially both drag and lift forces on the vehicle, this model can lead to appropriate answers [10]. Datcom software has also been used for analytical solution. The basis of this software is solving linear analytical equations [11]. The equations solved in this software are analytical.

Boundary conditions and flow solution

The flow solution for this airship occurred at an altitude of 4 km above sea level and a cruising speed of 40 km/h. Also, the maximum speed of the vehicle is 72 km/h. Due to the fact that the flow is subsonic and the maximum speed of the vehicle is 72 km/h, in choosing the solution field, more attention should be paid to the upstream flow. For this purpose, the front of the solution field is ten times the model and the back is considered the same value (Figure 6). For all variations of flow, efforts have been made to carefully observe these factors.

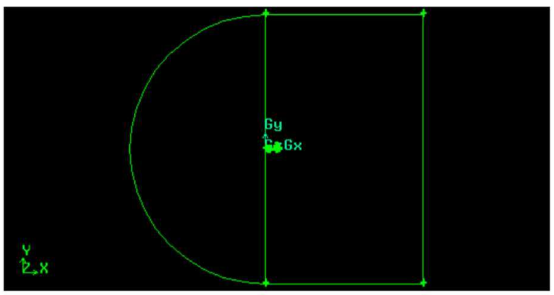

Figure 6- Side view of the flow field

Due to the fact that the vehicle configuration is symmetrical, in order to reduce the computational cost and flow solution time in modeling the airship, the boundary condition of the symmetry boundary condition has been used. For this purpose, the airship is divided in half from the middle and the middle plane of the symmetry boundary condition is considered. For around the solution field, the flow was first solved with a pressure far-field condition, but due to the very low flow velocity during several stages of solution and creating errors in the calculations, it was determined that this boundary condition is not suitable for the flow solution. And with these boundary conditions, we will see the flow back in and divergence of the solution. Therefore, the solution field was divided into upstream and downstream parts of the flow, and for the upstream part, the pressure outlet boundary condition and for the downstream part of the flow, the velocity inlet limit condition was considered. Due to having the flight condition of the ship, the inlet velocity of the flow was considered 20 m/s for the boundary condition of the entrance, and the altitude pressure of 4 km was considered for the flight altitude from the table of atmospheric conditions.

Aerodynamic configuration design of airship

The aerodynamic analysis of the airship with the approach of selecting the external configuration of the vehicle includes different parts. To form the external configuration of an airship, different parts of different perspectives must be analyzed. As a standard model for designing the exterior of an airship based on aerodynamics, the four parts of the design of body and nose, the design of fins and surfaces, the design of tail group and the locating of the engine are the main parts of the aerodynamic configuration of the airship. Figure ٧ shows these four parameters.

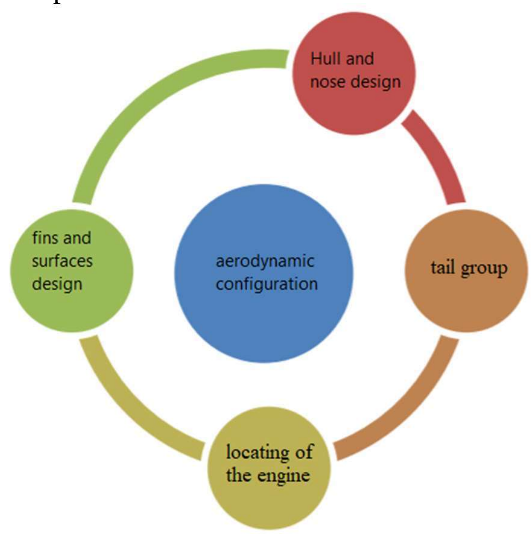

Figure 7 - Aerodynamic design and configuration framework

According to studies, slenderness coefficient is very important in body design. This coefficient is a function of aerodynamic shape and symmetry in flow lines to reduce drag force is its most important indicator. In the design of the airship, the slenderness coefficient can be changed according to the constant volume of gas inside the compartment, and this coefficient will be selected according to the requirements of the mission and with the approach of reducing the aerodynamic drag force. An example of this approach is given in Figure 8.

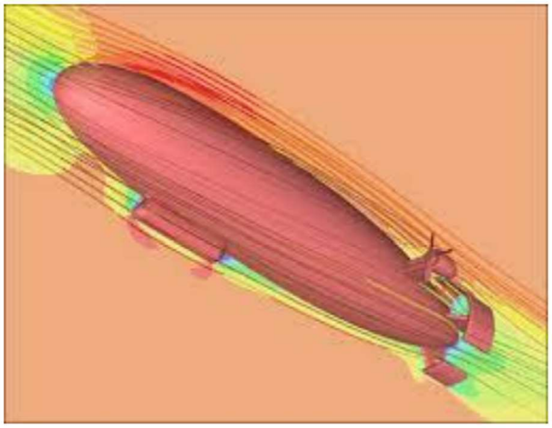

Figure 8 - Fluid flow around an airship

In designing the tai group, considerations related to flow vortices and reduction of drag force are of great importance. Also, the stability of the vehicle in maneuvers is one of the other important points in the design of the tail group. Figure 9 shows the flow vortices at the rear of the airship.

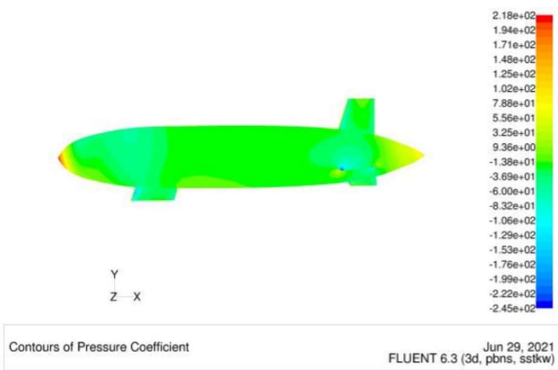

Figure 9 - velocity vector around an airship

Numerical solution results

One of the important parameters in aerodynamic simulation is the pressure coefficient and its distribution on the hull surface. By comparing the pressure coefficient as a key parameter, the flow pattern including the separation area can be estimated. In variant number one, with three fins to the conventional contour configuration, the pressure coefficient for the three attack angles of 5 and 10 degrees is shown in Figures 11 and 12. Figure 10 also shows the velocity vectors around the vehicle at an angle of attack of 5 degrees. The convergence of the vectors at the rear of the airship is shown in the Figure. This image is comparable to Figure 9 in which the rear vectors of an airship are simulated.

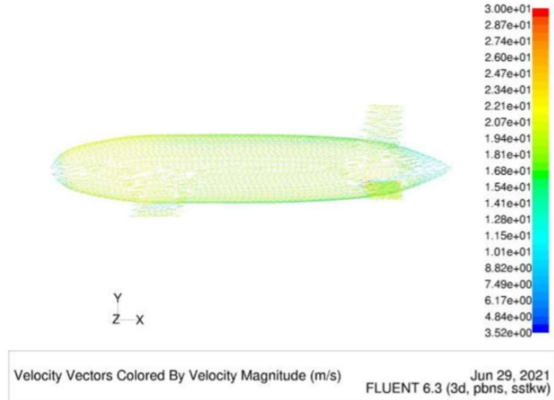

Figure 10 - Comparison of velocity vector with one of the available sources

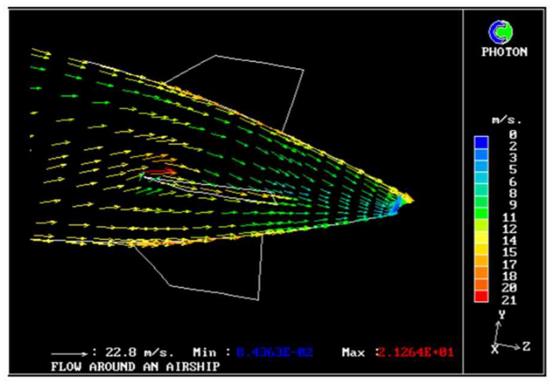

Figure 11 - Pressure coefficient on variant number 1 at an angle of attack of 5 degrees

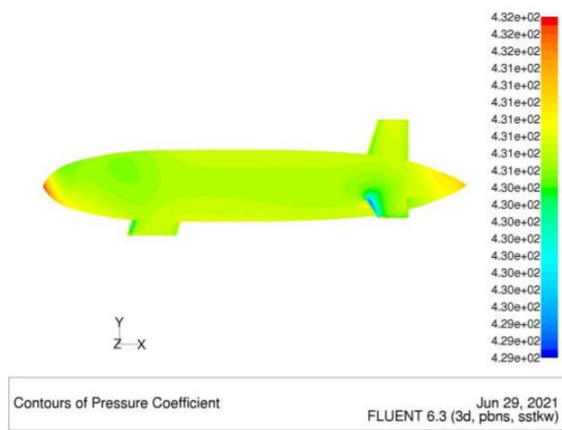

Figure 12- Pressure coefficient on variant number 1 at an angle of attack of 10 degrees

Increasing the angle of attack in variant number one at the rear of the vehicle increases the pressure in this area, especially after the tail and at the rear the hull. The pressure of the flow on the top part of the vehicle at the rear creates a turbulent area behind the airship, which creates flow vortices and improves the vehicle lift to drag conditions. Figure 10 shows the lift-drag ratio of variant 1 based on numerical solution at three different attack angles.

As in variant number one, in variant number two, the effects of the angle of attack on the pressure coefficient distribution will be investigated. The distribution of pressure coefficient on the rear of the vehicle can determine the flow pattern in this area and the separation of the flow. Increased pressure in this area is normal due to the vortices at the rear of the vehicle and the current behind the fin. Figures 13, 14 and 15 show the pressure coefficient distribution on configuration 2 at different attack angles.

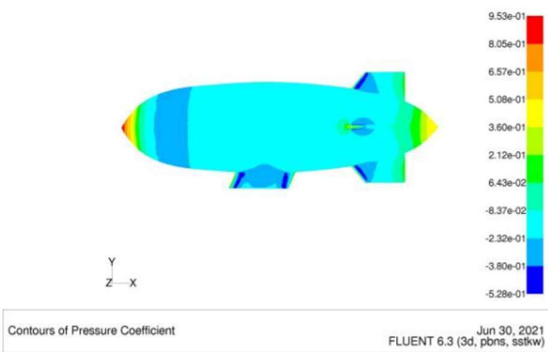

Figure 13- Distribution of pressure coefficient on the vehicle at zero degrees of attack angle in the second configuration

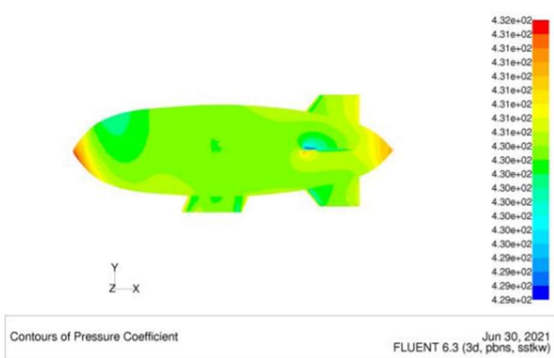

Figure 15- Distribution of pressure coefficient on the vehicle at an angle of attack of 10 degrees

In the above three diagrams, it can be clearly seen that at the rear of the airship behind the fin and at the end of the rear fins, the pressure coefficient increases with increasing attack angle, causing the flow pattern and its separation to occur in such a way that the lift-drag ratio increases. These contours are also given for variant three (Figures 16 to 18).

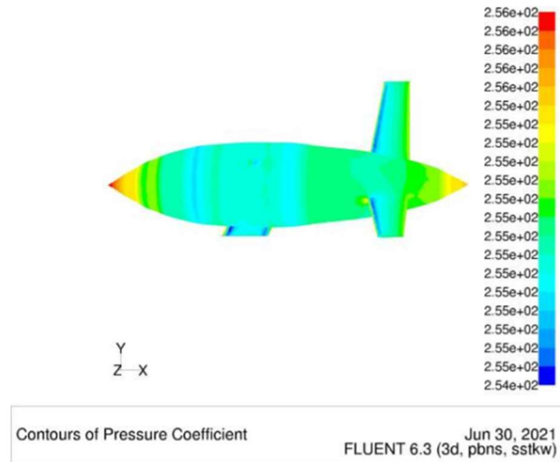

Figure 16- Distribution of pressure coefficient on the vehicle at zero angles of attack

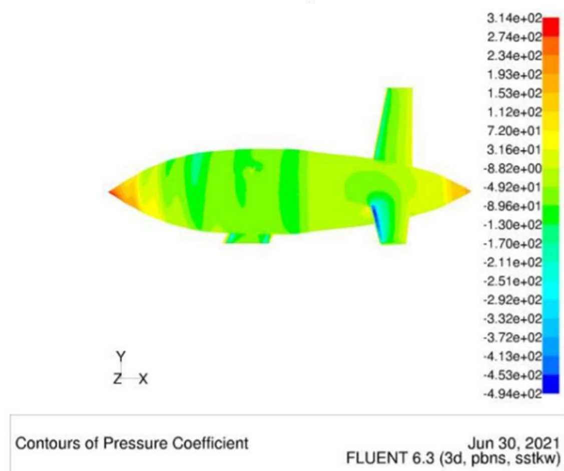

Figure 17- Distribution of pressure coefficient on the vehicle at an angle of attack of 5 degrees

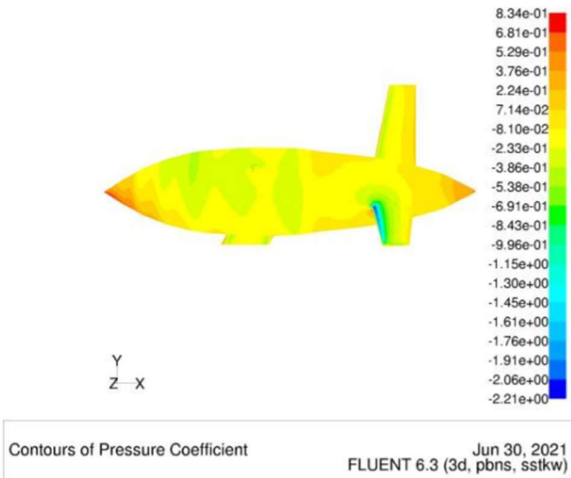

Figure 18- Distribution of pressure coefficient on the vehicle at an angle of attack of 10 degrees

Analyze the position of the fin on the hull using analytical solution

One of the most important factors in many aerodynamic parameters of an airship is the position of the airship fin, the effect of which on two coefficients of lift and drag through separation is determined by the effect on the separation of the end of the flow. In this part of the research, using the Datcom analytical solver, the effects of flow separation at the rear of the vehicle and its effects on the two coefficients of lift and drag for all three variants will be presented. In this regard, in three tables 1, 2 and 3, the effect of moving the fin in the longitudinal direction of the vehicle for three variants in different attack angles is given.

Table 1- Effect of fin movement along the vehicle on lift and drag

Place of fin installation of the from the nose of the vehicle (meters)	Angle of attack (degrees)	Lift coefficient (Cl)	Drag coefficient (Cd)	Cl/Cd
40.8	0	0	0.073	0
	5	0.367	0.104	3.534
	10	0.745	0.201	3.711
44.8	0	0	0.074	0
	5	0.352	0.103	3.414
	10	0.718	0.197	3.651
48.8	0	0	0.074	0
	5	0.328	0.101	3.246
	10	0.672	0.189	3.56

Table 2- Effect of fin displacement along the vehicle on lift and drag in variant 2

Place of fin installation of the from the nose of the vehicle (meters)	Angle of attack (degrees)	Lift coefficient (Cl)	Drag coefficient (Cd)	Cl/Cd
34.62	0	0	0.076	0
	5	0.41	0.109	3.75
	10	0.794	0.208	3.82
38.62	0	0	0.076	0
	5	0.393	0.108	3.638
	10	0.764	0.203	3.764
42.62	0	0	0.076	0
	5	0.371	0.106	3.494
	10	0.728	0.197	3.695

Table 3- Effect of fin displacement along the vehicle on lift and drag in variant 2

Place of fin installation of the from the nose of the vehicle (meters)	Angle of attack (degrees)	Lift coefficient (Cl)	Drag coefficient (Cd)	Cl/Cd
28.8	0	0	0.089	0
	5	0.209	0.106	1.97
	10	0.473	0.173	2.743
32.8	0	0	0.089	0
	5	0.201	0.106	1.901
	10	0.45	0.168	2.674
36.8	0	0	0.089	0
	5	0.182	0.104	1.752
	10	0.405	0.16	2.527

From the above data table, it can be seen that by moving the fin forward, the ratio of lift-drag, which is the main parameter in this section for measurement, increases as expected and it can be expected that moving fins forward will optimize aerodynamic behavior. There is a fundamental difference between the data obtained from the calculations of variant 3 and variant number 1, and that is the proximity of the ratio of lift-drag in the two angles of attack 5 and 10. This is due to the increasing size of the fin and increasing its effects. In fact, it can be expected that increasing the size of the fin reduces the effects of the angle of attack. In examining the effects of the angle of attack, it can be seen that by increasing the angle of attack of the vehicle in each case, the ratio of lift-drag increases, which is obvious and acceptable considering the type of configuration and the basics of flow solution. In order to validate and ensure the results considering the low speed of the vehicle, the pressure distribution coefficient on the body was qualitatively compared with similar samples. The results show that there is a good parity between the data and the range of pressure coefficient with similar samples is in a certain range.

Conclusion and analysis of the results

In the present paper, aerodynamic calculations have been performed on three selected airship configurations. The three selected configuration examples include three and four fins at different attack angles. The main purpose of this research is to find the selected configuration from the perspective of aerodynamic efficiency in order to reduce the drag force and increase the lift force. In fact, increasing the lift-drag ratio is the main goal of this research. For this purpose, analytical solution was performed by Datcom software and numerical solution was performed by Fluent software. The desired results were obtained by Fluent, but Datcom software and analytical solution were used to evaluate the changes in forces and coefficients. In fact, dimensionless numbers, such as the lift-drag ratio, are the criteria for deciding on an analytical solution. In numerical solution, due to computational constraints, symmetry boundary conditions have been used. Also, an unstructured computing network has been used by increasing the density of cells near the hull. Also in the computational network, network independence for each configuration is examined. The numerical solution is of turbulent flow type and the solution boundary conditions are adjusted based on the inlet velocity and the outlet pressure according to the client's needs. What is certain and expected is that the same results are not obtained in numerical and analytical solutions, and certainly numerical results can be cited more than analytical solutions, because Datcom software has inherent computational error and has been simplified. Also, this software has been developed to solve the flow analysis on slender body objects, but despite the existing errors, this software can be used to derive computational processes and sensitivity studies of some parameters. The results of numerical and analytical solution show that the ratio of lift-drag is optimal in variants with less diameter and with increasing diameter, the drag force increases as expected and the length of the vehicle has less effect compared to the diameter. Therefore, it is suggested that diameter reduction to be the criterion in the development of the vehicle to optimize aerodynamic performance. Also, the slenderness coefficient of the vehicle must be based on diameter reduction. Another important result is about the rear fins of the vehicle. In all three variants, moving the fins forward and distancing them from the rear of the vehicle disrupts the flow pattern at the rear of the vehicle, delaying separation. This will improve aerodynamic efficiency and improve the lift-drag ratio of the vehicle. On the other hand, the movement of the fins towards the rear of the vehicle interferes with the vortices flow behind the vehicle and the flow behind the fin. Therefore, from an aerodynamic point of view, it is appropriate for the rear fins to move as far forward as possible. Of course, this issue should also be considered from a systemic perspective, and the present study is only from the perspective of aerodynamics. Also, according to studies conducted among the types of fins, the cross-type will have the best performance among the fins in terms of lift-drag ratio. In fact, in all variants, converting the fins from X-type to cross or from the inverted Y variant to cross variant improves the aerodynamic performance of the vehicle. As a validation, the ratio of lift-drag for the three selected variants has been investigated using numerical solution and analytical solution.

Table 4: Comparison of numerical and analytical solutions

Variant	Angle of attack	Cl/Cd Numerical solution	Cl/Cd analytical solution	Error percentage
Variant 1	0	0.01	0	Near to zero
	5	2.26	3.414	33
	10	4.9	3.651	34
Variant 2	0	0.001	0	Near to zero
	5	1.86	1.901	2
	10	7.1	2.674	165
Variant 3	0	0.001	0	Near to zero
	5	4.1	3.638	12
	10	18.0	3.764	370

It can be seen from Table 4 that increasing the angle of attack will cause errors in the comparison results between numerical and analytical solutions. As mentioned, the results of numerical solution are more acceptable for this research. Among the selected solution variants, variant number 3 will see more errors due to the presence of large fins in the higher attack angle, which can be seen in the table. Of course, the solution results show that variant number 3 has a higher lift-drag ratio than the other two variants.

References:

1- Carrion, M, Steijl, R., Barakos, G., and Stewart, D. (2016) Analysis of hybrid air vehicles using computational fluid dynamics. Journal of Aircraft, 53(4), pp. 1001-1012.
2- Technical manual of airship aerodynamics, war department, February 1941. TM1-320.
3- K. E. Omari, E. Schall, B. Koobus, A. Dervieux, TURBULENCE MODELING CHALLENGES IN AIRSHIP CFD STUDIES, 2004.
4- K.Shields, CFD APPLICATIONS IN AIRSHIP DESIGN, Thesis submitted to the College of Engineering and Mineral Resources at West Virginia University, 2010.
5- V. Voloshin, Y.K. Chen, R. Calay, A comparison of turbulence models in airship steady-state CFD simulations, October 11, 2012.
6- A. D. Andan, W. Asrar, A. A. Omar, Investigation of Aerodynamic Parameters of a Hybrid Airship, Journal of Aircraft, 2012.
7- L.Ping, G.yi Fu, L.j.Zhu, X.l.Wang, Aerodynamic characteristics of airship Zhiyuan-1, Journal of Shanghai Jiaotong University (Science) 18(6):679- 687, DOI:10.1007/s12204-013-1443-9, December 2013.
8- C.Yanxiang, Y.Yanchu, Zh.Jianghua, Zh. Xiangqiang, Y.feng, Numerical Aerodynamic Investigations on Stratospheric Airships of Different Tail Configurations, Conference Paper · March 2015.
9- Jefferson L. Mendonça Junior, Jonatas S. Santos, Maurício A. Morales, Luiz C. Goes, Stojan Stevanovic and Rodrigo A. Santana, Airship Aerodynamic Coefficients Estimation Based on Computational Method for Preliminary Design, AIAA 2019-2982.
10- D. Monk, E.A. Chadwick, Comparison of Turbulence Models Effectiveness for a Delta Wing at Low Reynolds Numbers, european conference for aeronautics and space sciences, 2017.
11- THE USAF STABILITY AND CONTROL DATCOM, Volume I, Users Manual, From AFFDL-TR-79-3032, 1999.
12- Carrion, M., Steijl, R., Barakos, G., and Stewart, D. (2016) Analysis of hybrid air vehicles using computational fluid dynamics. Journal of Aircraft, 53(4), pp. 1001- 1012.