Simulation of Airflow Characteristics of a Seabird Following a Ship Based on Steady States

Lu Chao, Liu Guang Wu, Zhang Wenguo and Wang Jian

November 24, 2022

Simulation of Airflow Characteristics of a Seabird Following a Ship Based on Steady State

Chao Lu[1], Guangwu Liu[2], Wenguo Zhang[1] and Jian Wang[1]

[1] China Ship Development and Design Center, Wuhan 430064, China
[2] Green & Smart River-Sea-Going Ship, Cruise and Yacht Research Center, Wuhan University of Technology, Wuhan 430063, China
gliu@whut.edu.cn

Abstract. The seabirds have always been observed following the ships while sailing in a static condition. Despite the biological reason, there should be some physical profits for these following activities. This paper is trying to start an initial study of a seabird following a ship by Computational Fluid Dynamics (CFD) simulations on steady states. We have chosen a standard frigate simplified 2(SFS2) standard computation model as simulation ship, while a classic seagull wing to simulate a seabird. The paper has inspected the accuracy of CFD with typical wind tunnel tests' and CFD simulation examples' results such as 7.62m/sec in wind over deck（WOD）0 °and 10°. While the tests of wind tunnel had executed in the scale of 1:120, so the inspection geometric model is generated in the same scale. The results of inspection CFD are fitting well with the typical wind tunnel tests' and CFD simulation examples' results. Meanwhile a full scale SFS2's geometric model and a seabird wing's geometric model have been generated. A series of airflow simulations have been carried out then in steady states. Initial study of these simulations shows that in 0°WOD, the drag of the seabird's wing was reduced while following after the ship, compared with which situation that the ship was not followed by the seabird's wing. This paper has just started few typical studies of this interesting behaves of the seabirds, the following studies and simulations are being considered.

Keywords: S**imulation, Airflow, Seabird, Steady States**

1 Introduction

This paper focuses on the connection between airflow characteristics and seabirds biological phenomenon. While sailing on the ocean the ships are always been found followed by seabirds, even single on in groups. Despite the profit of predations, the following airflow should be a potential physical reason. The seabirds following behaves as show in Fig. 1.

Fig. 1. Seabirds group following the ship in Liaoning province

To prove the hypothesis, a small group from China Ship Development and Design Center (CSDDC) and Green& Smart River-Sea-Going Ship, Cruise and Yacht Research Center of Wuhan University of Technology (WUT) have started an initial work with Computational Fluid Dynamics (CFD) simulations and bionics references. The initial work should include two steps. First, Inspection. The paper has inspected the accuracy of CFD with typical wind tunnel tests' and CFD simulation examples' results such as 7.62m/sec in wind over deck（WOD）0 °and 10°.While the tests of wind tunnel had executed in the scale of 1:120, so the inspection geometric model is generated in the same scale. The results of inspection CFD are fitting well with the typical wind tunnel tests' and CFD simulation examples' results. Second, full scale simulation with seabird’s wing model. A full scale SFS2's geometric model and a seabird wing's geometric model have been generated. A series of airflow simulations have been carried out then in steady states. Initial study of these simulations shows that in 0°WOD, the drag of the seabird's wing was reduced while following after the ship, compared with which situation that the ship was not followed by the seabird's wing.

Initial conclusions have been made at the end of the paper presents that the bird’s following the ship could reduce the drag during flying.

2 Ship and Seabird Geometries

2.1 Ship Geometry

To investigate the effect of typical ship to the birds, the frigate class was considered as shown in Fig. 2. Instead of complex shape of the above ship structure, the so-called simplified frigate shape (SFS2) [2] was used in the wind tunnel test and the computational fluid dynamics (CFD) model. This geometry is a commonly used geometry for CFD benchmarking. The geometry was made into 1:120 scale model for numerical simulation matching the scale of the wind tunnel test model. Next, to simulate the airflow of frigate ship model with the commercial CFD code for current study.

Fig. 2. Frigate Ship Geometry

While in full scale simulation step, a full scale ship geometry has been adopted for the following condition. The full scale SFS2 geometry as show in Fig. 3 [4].

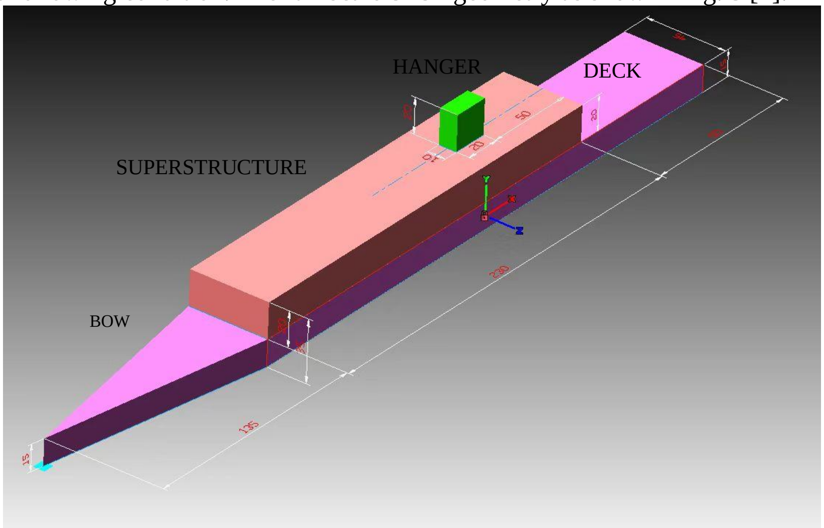


Fig. 3. Full-Scale Geometry of SFS2(in Feet)

Base on the specification, this paper has generated a ship geometry as shown in Fig. 4 which could be scaled in 1:120 and full scale for two steps simulations.

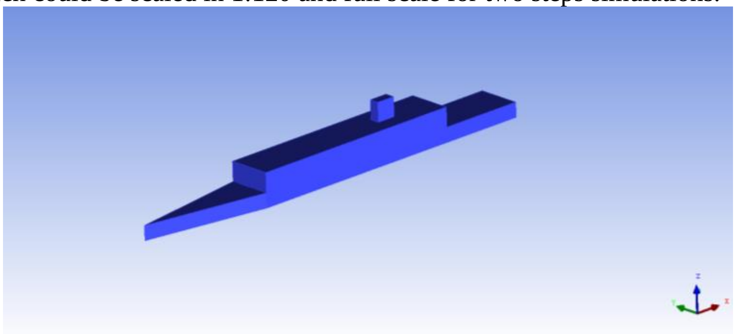


Fig. 4. SFS2 Geometry Generated in This Paper

2.2 Seabird Wing Geometry

To represent typical phenomena, this paper has chosen the seagull as the seabird, and simplified to a wing symmetry model. The wing type is a bionic seagull airfoil in full scale [3] with its section shown as in Fig. 5.

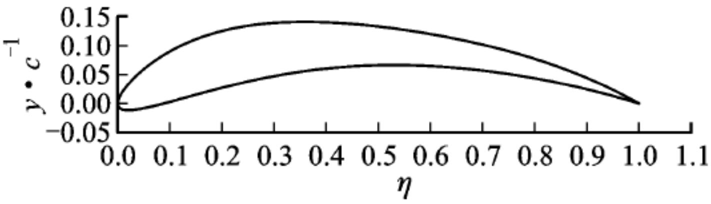
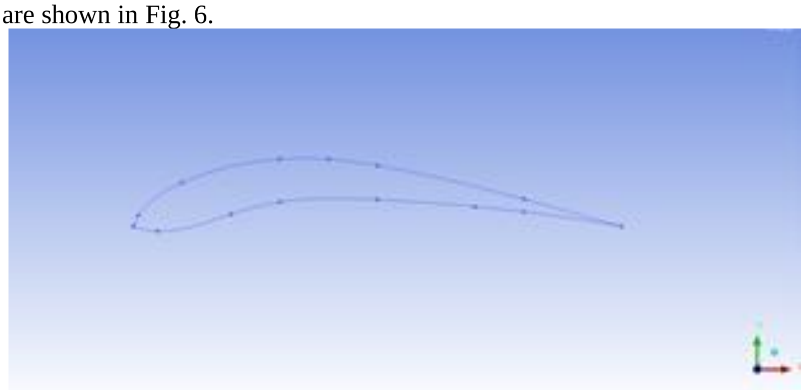


Fig. 5. Seagull Airfoil Typical Wing Section [3]

In this paper, a full-scale seabird wing model with 1 meter span and 3° sweepback like a flying seagull spread is drawn. The wing section and full-scale seabird wing model are shown in Fig. 6.



(a) Wing Section

(b) Full Scale Seabird Wing Model

Fig. 6. The Wing section (a) and Full-Scale Seabird Wing Model (b)

As metric system is a standard for international technologies, the geometries and model units are transferred in metric system as well as the calculation later [5].

3 Turbulence Type and Calculation States

3.1 Turbulence Type

The seabird following phenomenon is always happening in relatively long steady period. In this condition, the simulation should be in steady states. For ships airflow, the viscosity flow field could be set as a steady ship with steady turbulence flow. In equation (1), it shows, in steady states, forces such as gravity and trust or drugs.

$$\frac{\partial(u_i u_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}[\nu(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i})] + g_i \qquad (1)$$

In the equation, u_j represents a velocity component in coordinate x_j. $g_1 = 0$, $g_2 = 0$, $g_3 = -g$, g is gravity, t (s) is time step, p (Pa) is flow pressure, ρ is flow density, ν is velocity coefficient, f_i (N) represents volume force [6].

In engineering simulations, there are few turbulence types for common use. As classic turbulence type, a realizable $k-\varepsilon$ two-equation turbulence model were used with a pressure-velocity coupling scheme [8].

3.2 Calculation States

During inspection simulation, it is needed to compare the results with the example of SFS2 which had been tested in wind tunnel with PIV measurement and CFD calculation in specific position of specific states [2] Such as 7.62m/sec, wind over deck (WOD) with 0°, 5°, 10°, 15°. The coordination of the calculation has been described in Fig. 7.

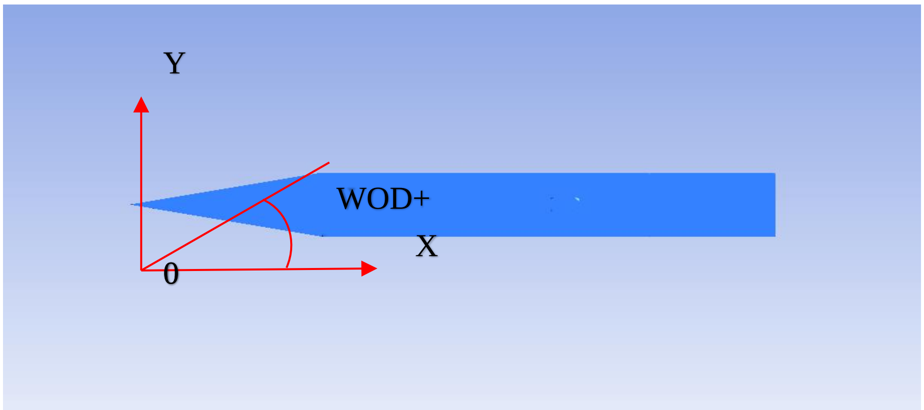


Fig. 7. The Coordination of the Calculation

This paper chose two typical states for both inspection simulations and full-scale simulations, both in steady states. The calculation states of this paper are shown in Table 1.

The wing fwd is set for calculation without the effect of the ship [6], and the wing aft is set after the ship. Both wings are 10m high from bottom and 20m before or after the ship.

Table 1. Summary of CFD States

CFD STATES	WIND X SPEED (M/SEC)	WOD (°)	SCALE
INSPECTION 1	7.62	0	1/120
INSPECTION 2	7.62	10	1/120
FULL WITH 2 WINGS	7.62	0	1/1
FULL WITH WING	7.62	10	1/1

The boundary conditions are kept same as the CFD examples shown in Table 2.

Table 2. Summary of Boundaries States

REGIONS	BOUNDARY CONDITION	VALUES	REF.
INLET	Velocity inlet	x-direction:7.62	
OUTLET	Pressure outlet	Ref. P=0	
SHIP	Wall	No slip	
LEFT/RIGHT	Wall	No slip	
UPPER/BOTTOM	Wall	No slip	
WING FWD	Wall	No slip	With U-flow
WING AFT	Wall	No slip	With U-flow

4 Inspection Simulations

The size and mesh structure of calculation domain of inspection simulations is shown in Fig. 8.

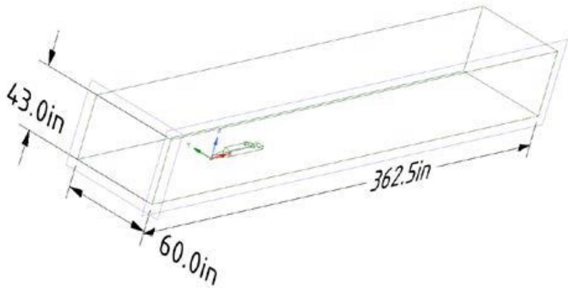


Fig. 8. The Size and Mesh Structure of Calculation Domain of Inspection Simulations

The example measurement locations have 3 plants above the helicopter deck as WP1, WP2 and WP3. WP1 was chosen for comparing, and the u speed would be measured at line x/H=0.95. The WP1 location is shown in Fig. 9.

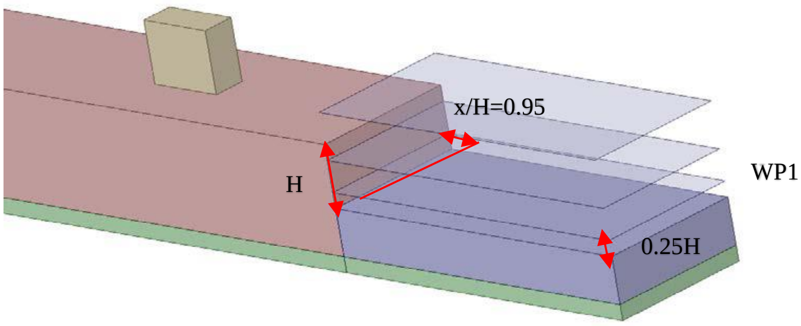
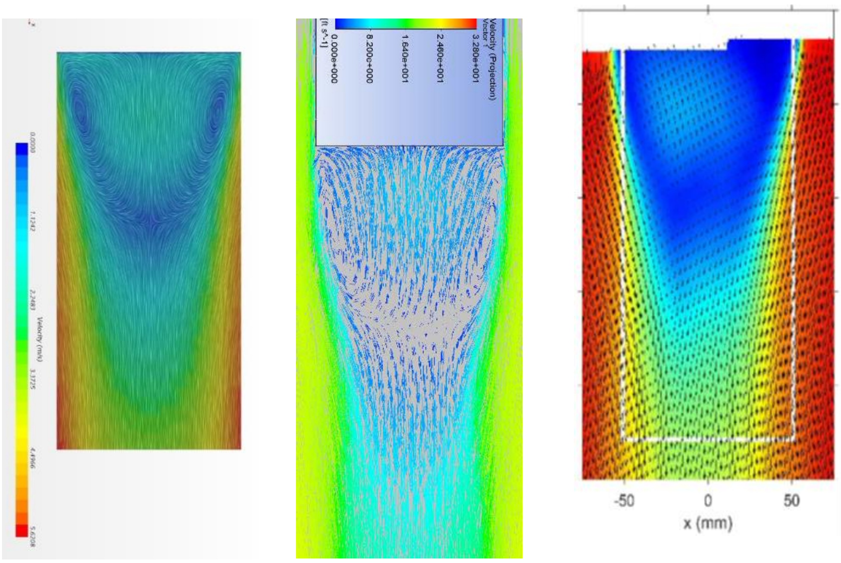


Fig. 9. WP1 Plant Location

4.1 Inspection Simulation 1

The inspection simulation 1 is corresponded with the test example with WOD=0°, the elements of the inspection simulation 1 is about 660 thousand. After solver, the velocity vectors are compared between Inspection Simulation 1, example CFD and example's PIV [2] as shown in Fig. 10.

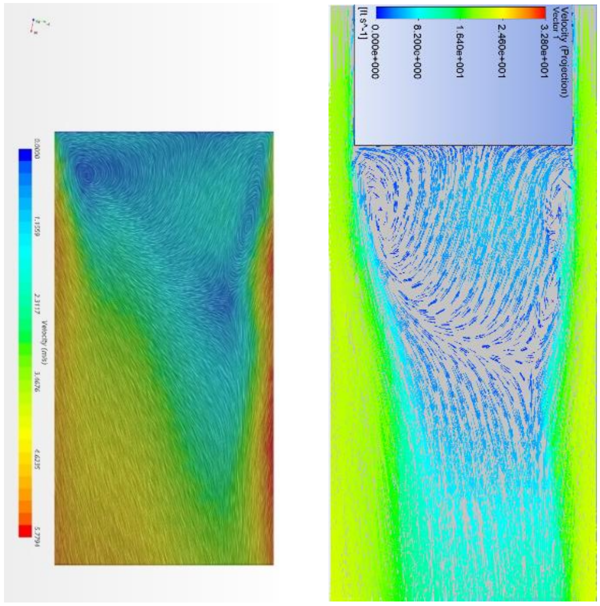


(a) (b) (c)

Fig. 10. Comparison between Simulation 1 (a), example CFD (b) and example PIV (c)

4.2 Inspection Simulation 2

The inspection simulation 2 is corresponded with the test example with WOD=10°, the elements of the inspection simulation 2 is about 670 thousand. After solver, the velocity vectors are compared between Inspection Simulation 2 and example CFD [2] as shown in Fig. 11.



(a) (b)

Fig. 11. Comparison between Inspection Simulation 1 (a) and Example CFD (b)

The u velocity magnitudes are collected on line x/H=0.95, and compared with wind tunnel (WT) test results, as shown in Fig. 12.

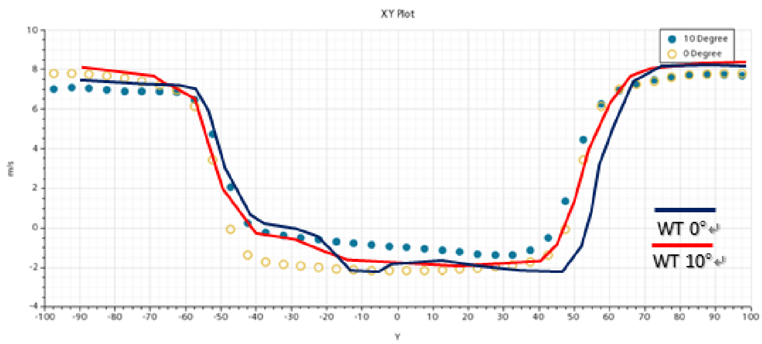


Fig. 12. The u Velocity Magnitudes Comparing with Wind Tunnel Tests (WT)

As show in Fig. 10 and Fig. 11, the air flow distribution form between Inspection Simulations, example CFD and wind tunnel PIVs are similar with each other. While the curves of Inspection Simulations comparing with wind tunnel tests also have similar shapes.

With the inspections above, the simulation code and programs should have the feasibilities to carry out further calculations with the seabird wings.

5 Full-Scale Simulations

As the wind speed and ship condition are in low Re scope, the scale has few effects for airflow trend [7], especially for the steady states. So calculations in full-scale with consideration of relatively small dimensions of seabird wings are carried out in this paper.

5.1 Full-Scale Simulation 1

The full-scale simulation 1 has two wings before (wing fwd) and after (wing aft) the ship, both 10m high from bottom. The wing fwd is 20m before the bow to avoid the flow's effect on the ship.

The geometries of full-scale simulation 1 are shown in Fig. 13.

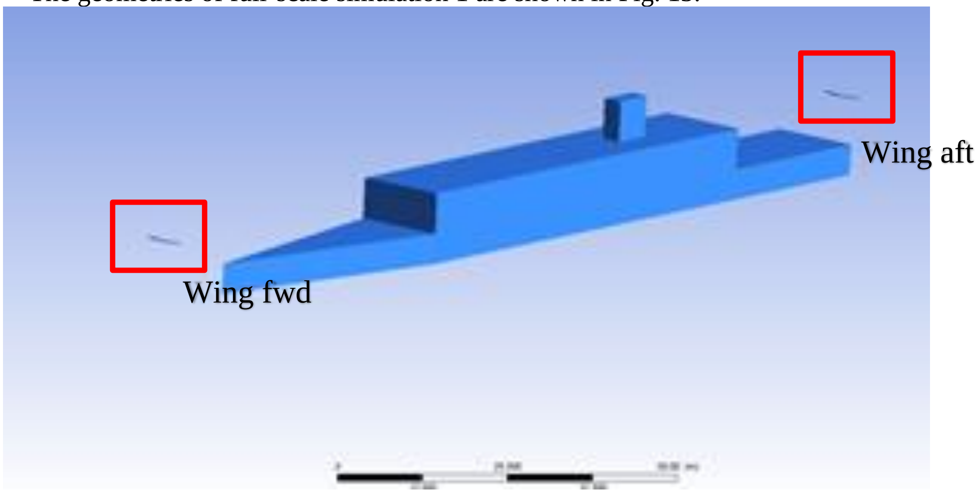
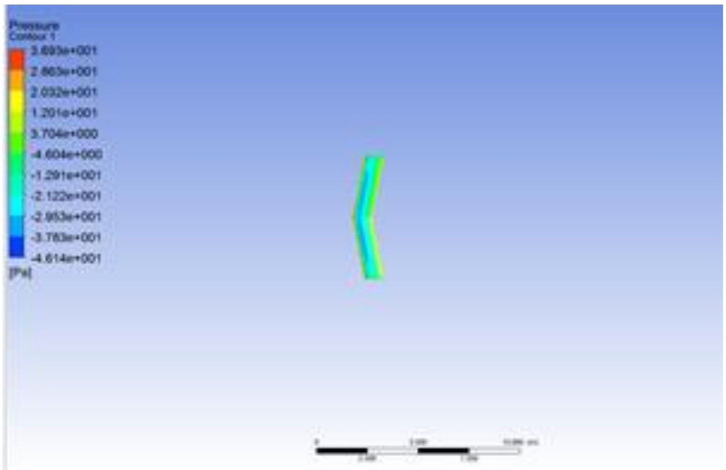
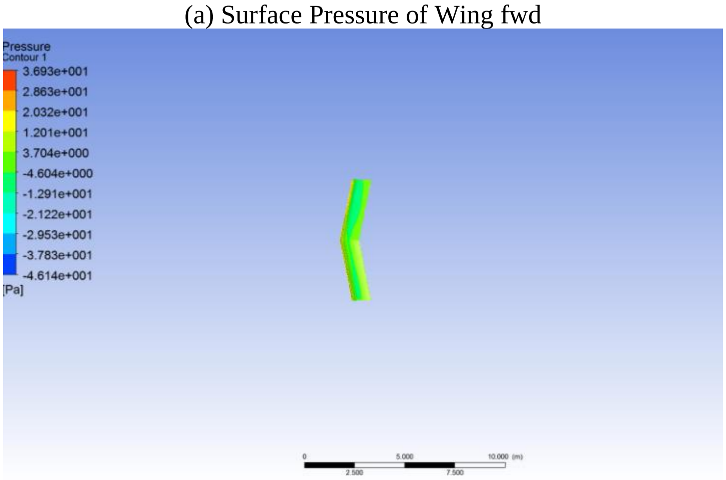


Fig. 13. The Geometries of Full-Scale simulation 1

The elements of full-scale simulation 1 is about 3.3 million. After solver, the surface pressures of wing fwd and wing aft are compared as shown in fig. 14.



(a) Surface Pressure of Wing fwd



(b) Surface Pressure of Wing aft

Fig. 14. Surface Pressures Wing fwd and Wing aft

Then a total pressure of XZ section plane has been generated to compare the deference flow condition of the wing fwd and wing aft as shown in Fig. 15.

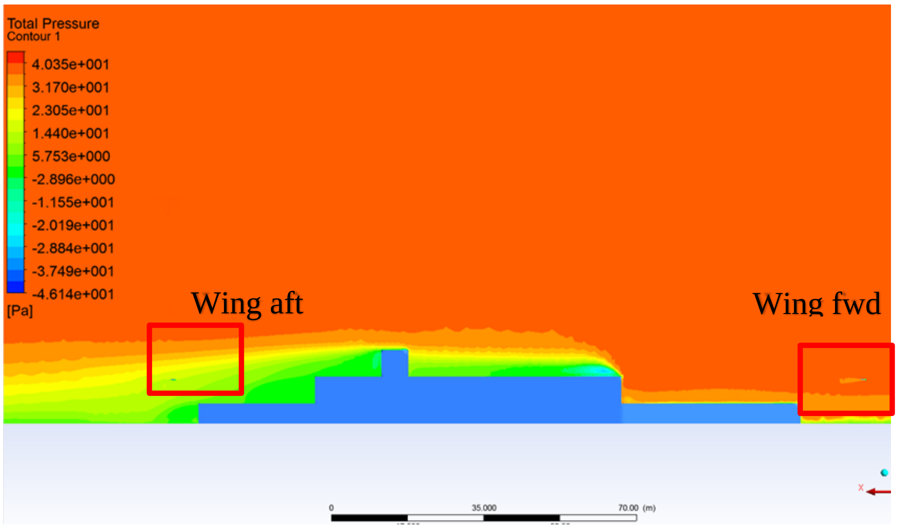


Fig. 15. Total Pressure of XZ Section Plane

A u drag force has been measured for both wing fwd and wing aft. Since the steady state has stabled aft 400 steps, the data from step 400 to step 500 was figured out in this paper, and average value has been calculated.

The solver monitor of u drag force is shown in Fig. 16.

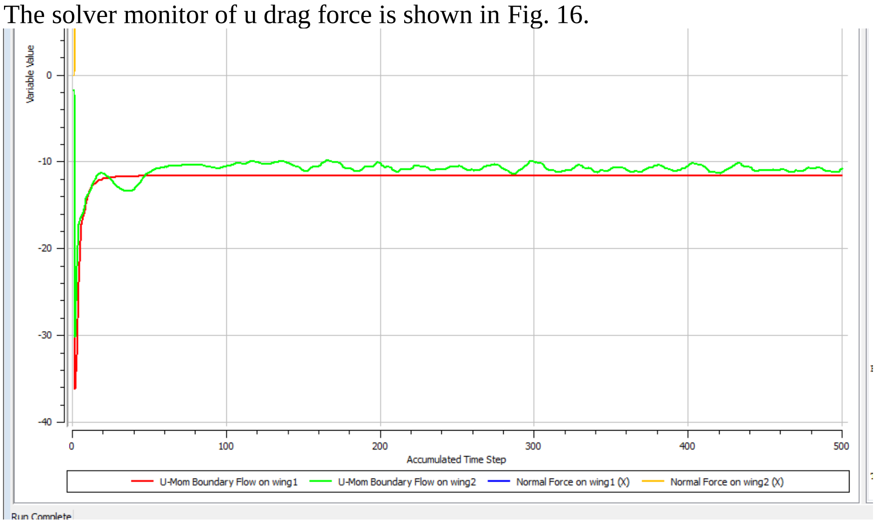


Fig. 16. The Solver Monitor of u Drag

5.2 Full-Scale Simulation 2

The full-scale simulation 2 has one wing after the ship (wing aft). When the ship has the WOD=10° the elements of full scale simulation 1 is about 3.2 million. The geometry of the ship and wing aft is shown in Fig. 17.

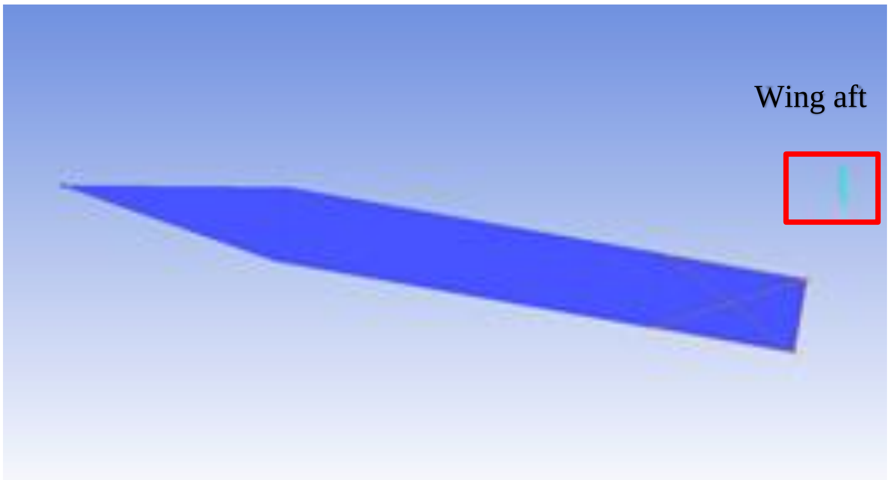


Fig. 15. The Geometry of the Ship and Wing aft at WOD=10°

Then a total pressure of XZ section plane has been generated to show the different flow condition of the wing aft as shown in Fig. 18.

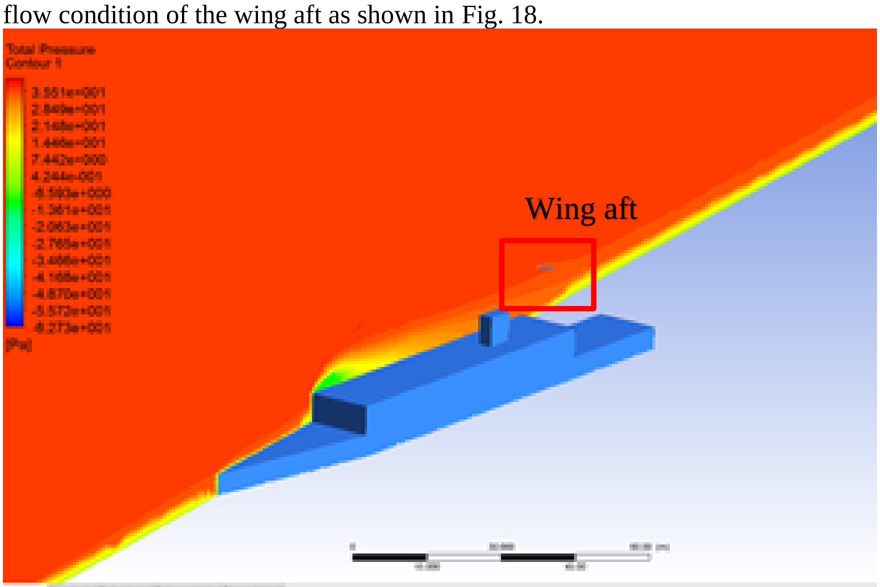


Fig. 16. Total Pressure of XZ Section plane at WOD=10 °

A u drag force has been measured for wing aft. Since the steady state has stabled after 400 steps, the data from step 400 to step 500 was figure out, and average value has been calculated. The solver monitor of u drag force of wing aft is shown in Fig. 19.

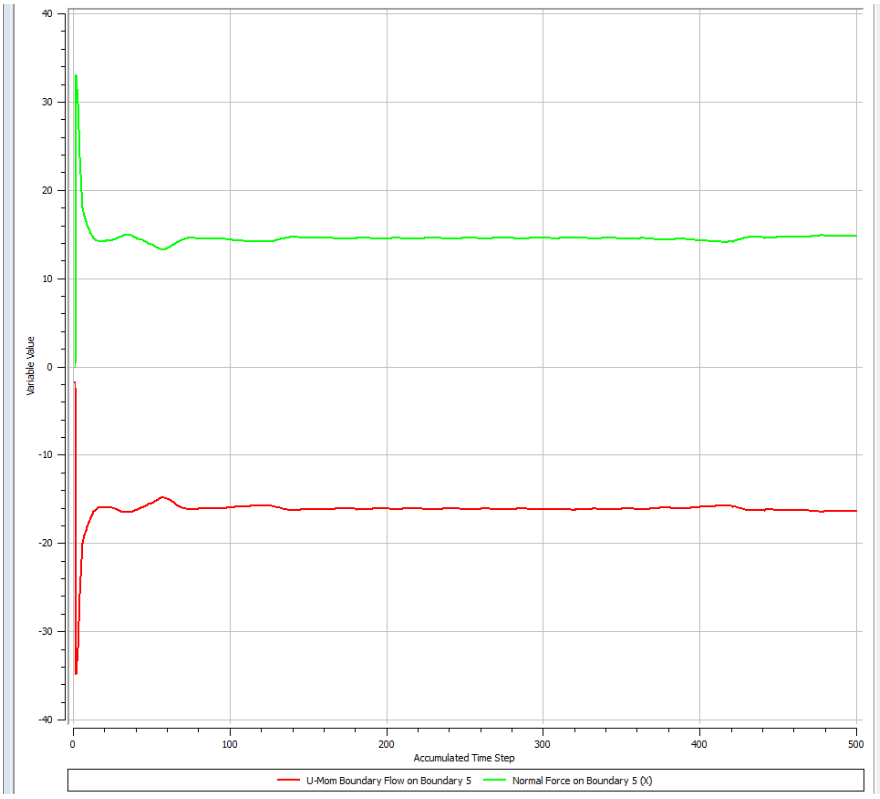


Fig. 17. The Solver Monitor of u Drag at WOD=10°

The u drag average values are summed in Table 3.

Table 3. The u Drag Average Values

CONDITIONS	WOD(˚)	AVERAGE VALUES(N)
WING FWD	0	11.63
WING AFT	0	10.92
WING AFT	10	15.97

As compared above in full-scale simulation 1, the u drag is reduced about 6.1% when the seabird flies following the ship. But when the ship is turning 10°, the situation would be changed and the u drag would be increased.

6 Conclusions

This paper is the first attempt to study the phenomenon of seabirds following the ship. The initial work should include two steps. Firstly, the accuracy of CFD with typical wind tunnel tests and CFD simulation example results of velocity 7.62m/sec and wind over deck（WOD）0° and 10° was inspected. Because the test of wind tunnel had executed in the scale of 1:120, the inspection geometric model is generated in the same scale. The results of inspection CFD are fitting well with the typical wind tunnel tests and CFD simulation examples results. Secondly, full-scale simulation with seabird's wing model was done. A full-scale SFS2's geometric model and a seabird wing's geometric model have been generated. A series of airflow simulations have been carried out in steady states. Initial study of these simulations shows that in 0° WOD, the drag of the seabird's wing was reduced while following the ship, compared with the situation that the ship was not followed by the seabird's wing.

Such conclusions should be made in this initial step:

1. A CFD method could be a reasonable way to simulate the airflow relationship between the ship and seabirds.

2. The seabird's following the ship in WOD=0° could reduce the drag of the flying which means to save the energy of the seabirds during long distance fly. But other WOD conditions need to be further studied.

References

1. Li Xiangtao, Gu Xiaojun: Active view of Chinese ocean: seabirds. Forest and Humankind 02, 36-51(2020).
2. Jongho Choi, David. S. Miklosovic: LES Simulations using the Moving Mesh Method with Comparison to Experimental Results for a Periodic Ship Airwake. In: AIAA AVIATION 2020 FORUM, pp. 1-23, (2020).
3. Wu Liming, Wang Lei, Liu Xiaomin, Ma Lie,Xi Guang: Numerical Simulation On The Static And Dynamic Aerodynamic Characteristics Of Bionic Seagull Airfoil. Journal of Xi'an Jiaotong University 12, 88-97 (2020).
4. Reddy, K. R., Toffoletto, R., Jones, K.R.W.: Numerical Simulation of Ship Airwake. Computer and fluids 29, 451-465 (2000). doi: 10.1016/S0045-7930
5. Mora, R. B.: Experimental Investigation of the Flow on a Simple Frigate Shape (SFS). The Scientific World Journal, 1-8 (2014). doi: 10.1155/2014/818132
6. Weixing Yuan, Alanna Wall, Richard Lee: Combined Numerical and Experimental Simulations of Unsteady Ship Airwakes. Computers & fluids, 30(172), 29-53 (2018). doi: 10.1016/j.compfluid.2018.06.006
7. Kang, H., Snyder, M. R., Miklosovic, D. S., Friedman, C.: Comparisons of In Situ Ship Air Wakes with Wind Tunnel Measurements and Computational Fluid Dynamics Simulations. Journal of the American Helicopter Society, 2(61), 1-16 (2016). doi: 10.4050/JAHS.61.022001
8. Mostafa Rahimpour, Peter Oshkai: Experimental Investigation of Airflow Over the Helicopter Platform of a Polar Icebreaker. Ocean Engineering, 15(121), 98-111 (2016). doi: 10.1016/j.oceaneng.2016.05.02.