Numerical Study of Flow Field Over the Deck with Active Flow Control Method

Bingjie Liu

Beijing 101 Middle School, Beijing, 100000, China

Abstract. When the destroyer is sailing on the sea, the turbulent ship airwake will be formed above the deck, due to the flow separation after the airflow passes through the edge of the hangar, and the most obvious vortex structure is the recirculation area, which will seriously affect the safety of take-off and landing of the ship-borne helicopter on the deck. Based on the means of flow control in fluid dynamics, this paper improves the quality of the flow field above the deck of the destroyer by adopting different blowing ways (3 blowing angles and 5 blowing speeds). The results show that for different blowing directions, the length of recirculation zone tends to decrease with the increase of blowing speed. The optimal blowing way is the blowing speed of 10m/s in the blowing direction of -45°, and the length of recirculation zone is reduced by 59.4%. This will obviously improve the landing safety of ship-borne helicopters and reduce the pilot's control load.

Keywords: Computational Fluid Dynamics; Numerical Simulation; Ship Airwake; Active Flow Control; Recirculation Aera.

1. Introduction

In nowadays wars, the operations of shipborne helicopters become more and more significant. Bigger and more advanced amphibious assault ships have been deployed for shipborne aircrafts' take off and landing. However, when these aircrafts land on the flight deck, the complex air flow above the deck, including nature wind, vortexes, backflow and flow separation, creats a dangerous environment for the aircrafts' operations. Therefore, a clear understand of the complex features of the flow field above the flight deck is undoubtedly important and necessary to give guidelines to the pilots and ship manufactures.

At the early stage of the flow field study, researchers are mainly conducting experiments in situ or in wind tunnels. In 2000, researchers from NUAA (Nanjing University of Aeronautics and Astronautics) [1] used a Seven-hole probe to successfully map vector field above the deck where helicopters are about to land. In 2009, Hong and other scientists [2] have simulated flow field around a ship when it has different layout, showing that it is possible to improve the air flow and reduce risk of shipboard helicopters landing.

By the support of The Technical Cooperation Programme (TTCP) [3], a modelling activity aimed to foster the development of ship airwake validation database was found. The simple frigate shape (SFS) was created as a highly simplified geometry originally by TTCP, which was for the program. The updated model of SFS was released and widely used to facilitate the development of CFD in flow field studies. This updated version is termed simple frigate shape 2 (SFS 2) which adds the bow to the original structure. National Research Council of Canada (NRC) performed a collection of experiments in wind tunnels to test both SFS and SFS 2 for benchmarks to validate CFD results and codes. Bardera-Mora have processed the wind tunnel results and full-scale test results on real frigate type ship geometry [4], which shows the capability of SFS model on predicting the flow field on the flight deck of similar ships.

Some investigations were also concentrated on adding the active or passive flow control devices to the ship configuration to improve the ship airwake. Many Researchers have proved that method of flow control above the deck can reduce the turbulence and instability. Tai presented a modified SFS geometry by adding a round bow at the front end [5]. The modification avoids the complete flow separation from the hull surface. However, this modification does not improve the flow characteristics over the flight deck. Later, Shukla et al [6]. modified SFS with a round bow and a round hangar door, which decreases the impact of complex flow characteristics. Shafer's study shows the effectiveness

of active control on naval vessels for air flow [7]. Greenwell et al. mounted some inclined porous screens around the hangar door area to improve the airflow over the flight deck [8]. They found that a dense screen, mounted on the sides and roof of the hangar and inclined rearwards give the best performance. Compared to the baseline ship geometry, all the ship modifications, particularly the side-flap and the notch modification, can significantly reduce root-mean-square forces and moments [9]. Three new SFS2 model configurations with different curvatures of the hangar roof were studied in the wind tunnel, which shows that round hangar roof is able to decrease the recirculation zone behind the hangar [10]. Bardera-Mora et al. used two different configurations of dielectric barrier discharge plasma actuator to modify the flow over the SFS model [11]. Results proved that plasma actuators can reduce instabilities and turbulence over the SFS model. Recently, the flow fields over the different modified SFS models were numerically studied by Li [12], with changing the ratio of hangar height and length in the static state and pitching state. The results show that the size of recirculation zone and the location of the reattachment point have distinct differences for diverse models, and reveal the tracks of recirculation zone's center and reattachment position in a pitching period. In addition, another study shows that by varying the model size and inflow velocity [13], the changing of Reynolds number has tiny effect on the variation of the dimensionless size of the recirculation zone.

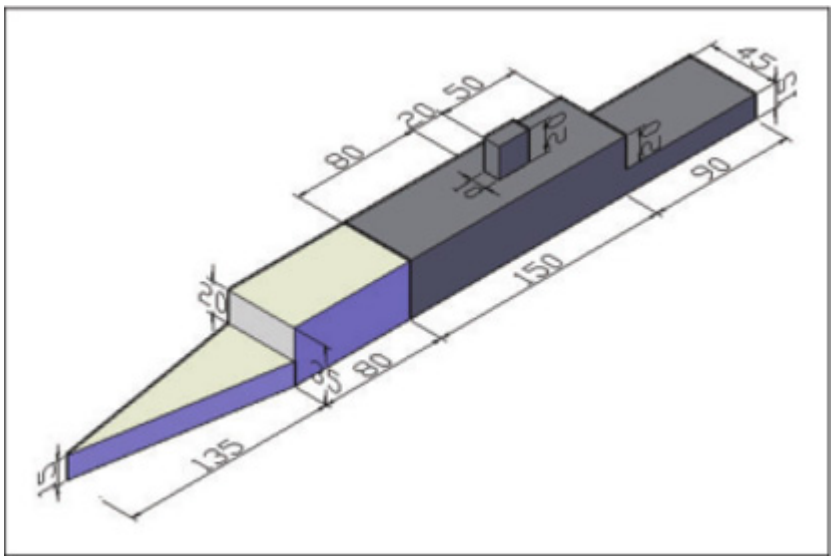


Figure 1. Schematic of SFS2 model (SFS geometry shown by darker shading) [14]

Flow control is cutting edge topic and this essay will aim at analyzing and improving the flow field over the deck and improve it by applying active flow control to find a relative better solution for minimizing the downwash area, which can provide a better air environment to reduce accidents and increase efficiency and stability of shipborne helicopters' operations.

2. Method

The inflow air speed is set to 10 m/s which is significantly lesser than 0.3 Mach, therefore, air in the study is considered uncompressible. The incompressible flow governing equations are as follows:

$$\nabla \cdot V = 0$$

$$\rho \frac{DV}{Dt} + \nabla p = F$$

where V is the velocity vector, ρ is the density of the air, $DV=Dt$ is the material derivative of V, p is the static pressure, and F is the viscous force vector. The commercial CFD solver FLUENT was used for the numerical simulation, employing RANS with the Realizable k-e turbulence model for

closure. The coupling of the pressure and velocity was handled using Semi-Implicit Method for Pressure-Linked Equation (SIMPLE) algorithm.

3. Model and Grid

SFS2 model is used in this study for the numerical simulation of flow field above the deck. The model has total length of 138 meters with its width of 14 meters, and the height of the hanger is 6 meters. In Figure 2, the active control nozzle is located at the top of the hanger wall with height of 0.3 meters (colored in red).

The section grid of SFS2 at y=0 is shown in Figure 3. The surface of the model is meshed with triangular grids grown up to 30 layers for purpose of resolving viscous boundary layer. A hexahedral mesh with its dimension of 20×14×12 ($X \times Y \times Z$ in meters) is used above the deck for more accurate simulations of the backflow region. The total number of cell number is about 4.4 million.

The active flow control method used in this paper includes blowing air at speed of 2 m/s,4 m/s,8 m/s,10 m/s, and 20 m/s, with the angle of +45°, 0°, and -45° (shown in Figure 4).

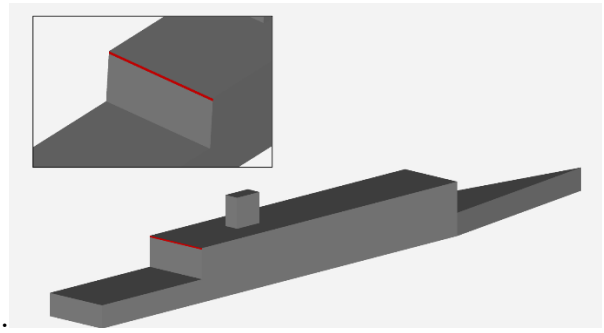


Figure 2. Schematic of SFS2 model with active control nozzle

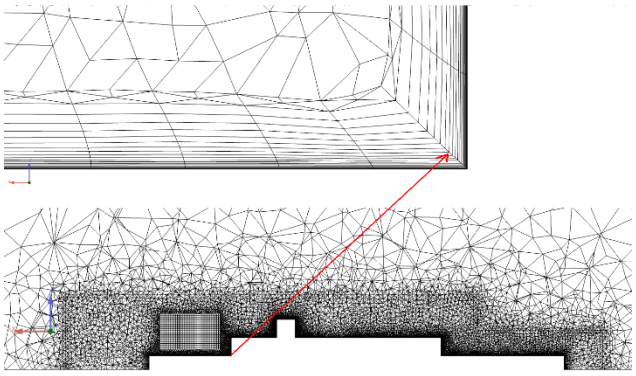


Figure 3. Section grid of SFS2 at y=0

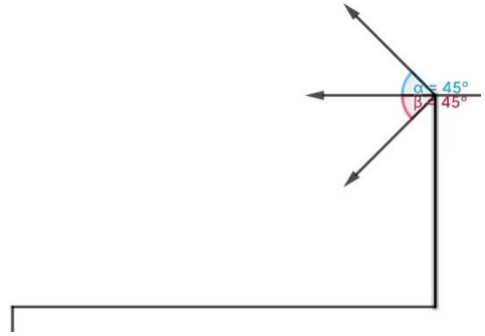


Figure 4. Direction of the blowing air

4. Results and Analysis

The contours of vertical velocity for different ways of blowing air was shown in Figure 5. It can be seen that when the speed of blowing air increases, for all three angles, the velocity magnitude of downwash increases.

For +45° state, as the blowing direction is inclined upward, the airflow will drive the recirculation area upward to form a bulge, which makes the recirculation area slightly larger, and at the same time, the vortex core gradually moves upward. When the blowing speed increases to 4m/s, the length of the recirculation zone increases obviously, and then with the increasing of blowing speed, the length of the recirculation zone decreases obviously, and the center of the recirculation zone moves up obviously. It can also be noted that the vertical velocity in the recirculation zone, including the upwash speed and downwash speed, also gradually increases.

For 0° state, because the blowing direction is horizontal, the height of the recirculation area is basically the same as that of the hangar. With the increase of blowing speed, the recirculation area is constantly expanding, the reattachment position on the deck is constantly moving backward, and the vertical speed behind the hangar is obviously increasing. When the blowing speed is increased to 20 m/s, there is a large area of strong upwash in the rear of the hangar, which will adversely affect the take off and landing operations of ship-borne helicopters.

For -45° state, as the blowing speed increases, because the blowing direction is toward the surface of the deck, it brings additional strong downwash air flow behind the hanger, and then disturbs the recirculation area. The main performance is that the downwash area of the original recirculation area will gradually approach the hangar. The extra strong downwash will hinder the full development of the initial recirculation area, and press it to the hangar, reducing the scope of the recirculation area behind the hangar. At the same time, it can be seen that the downwash speed just above the center of the deck is weakened, which will be beneficial to the take-off and landing operations of ship-borne helicopters.

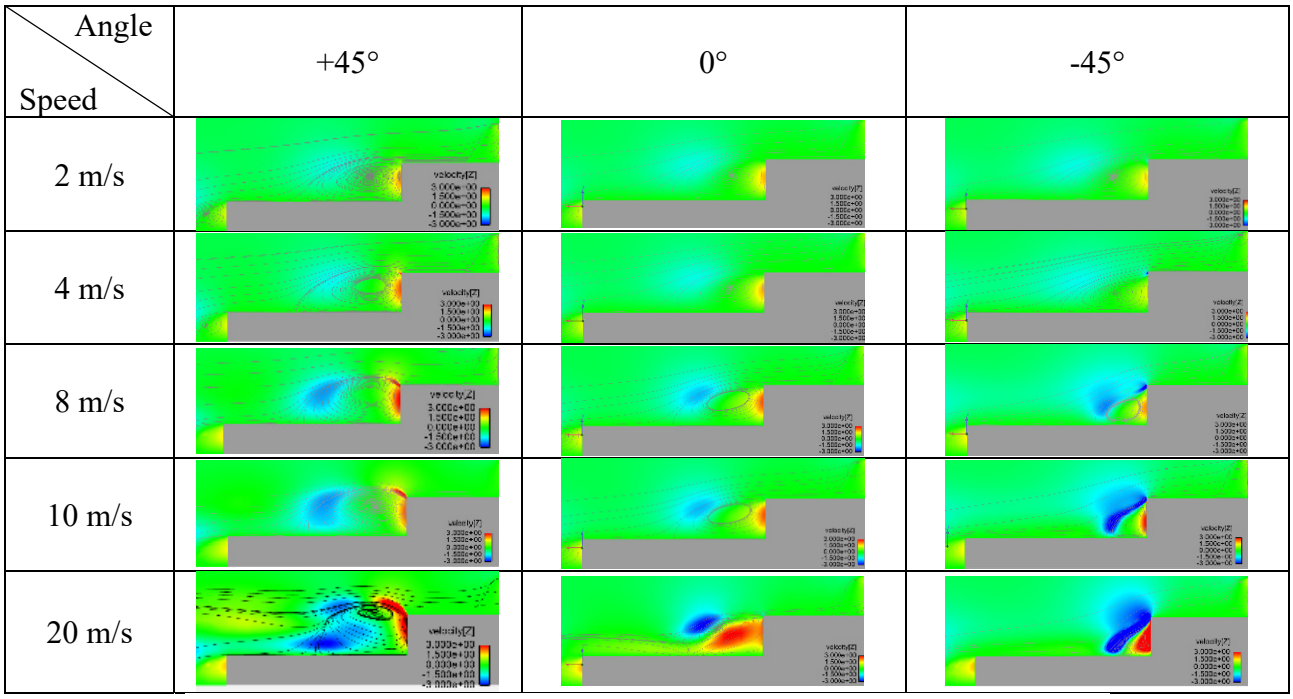


Figure 5. Contours of vertical velocity for different ways of blowing air

In Figure 6, the contours of pressure for different ways of blowing air was shown. For +45° state, with the increase of blowing speed, the pressure in the rear area of the deck is getting higher and higher, while the pressure in the area near the rear of the hangar is getting lower and lower, that is, there is an obvious "high-low pressure" distribution above the deck, which strengthens the "suction"

of the helicopter during take-off and landing, and makes it easy for the helicopter to crash into the hangar. For the 0° state, the rear of the hangar also presents a "high-low pressure" distribution, but the range of the low pressure area is small, and the "suction" produced is weaker than that in the +45° state. At the same time, it can be found that with the increase of blowing speed, the high pressure area gradually moves to the hangar. For the state of -45°, the low pressure area behind the hangar is very small, and the high pressure area on the deck moves forward obviously with the increase of blowing speed, which effectively weakens the "suction" effect of the backflow area behind the hangar on the helicopter.

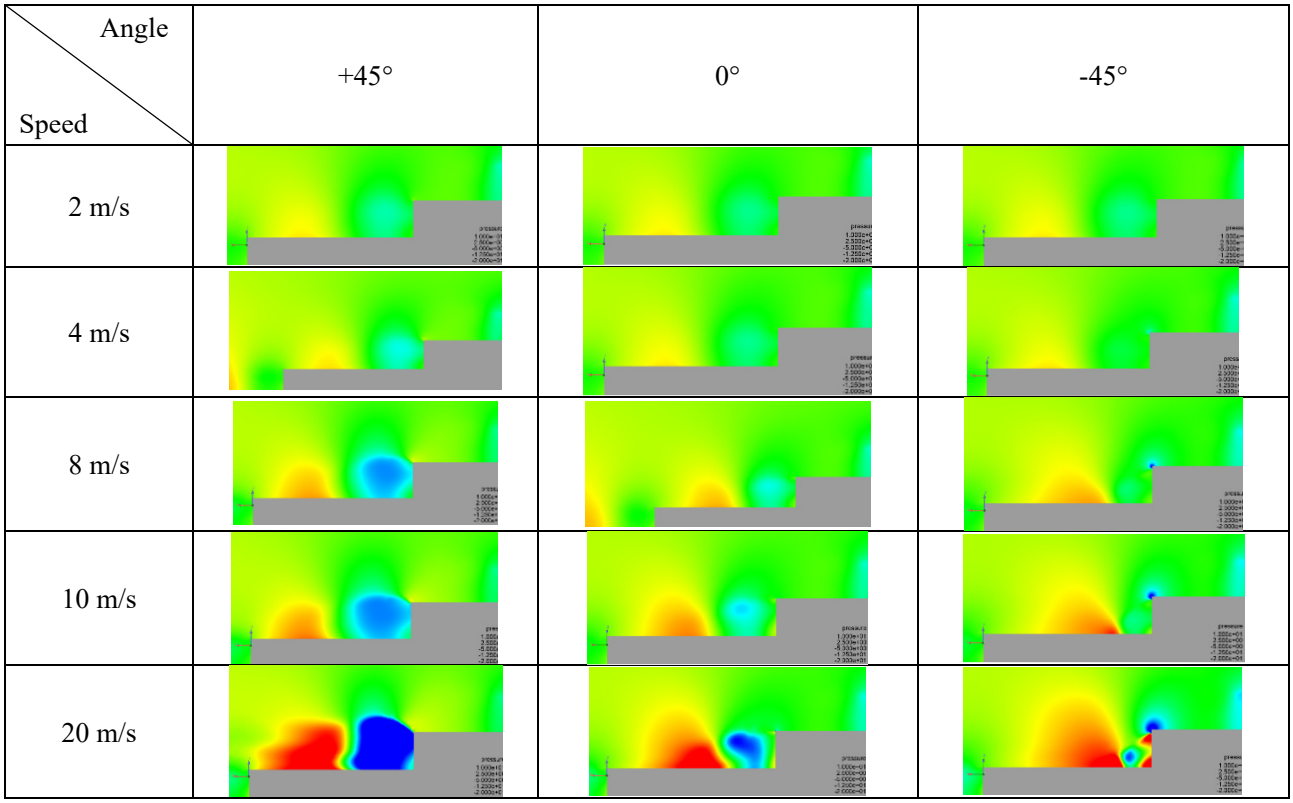


Figure 6. Contours of pressure for different ways of blowing air

Figure 7 shows the variation of recirculation zone length with blowing speed in three blowing directions. From the aspect of blowing direction, the control effect of blowing air (-45°) to the deck surface is the best, which makes the length of the recirculation zone obviously reduced compared with that of +45° and 0° control methods, while the control effect of 0° is in the middle. From the point of blowing speed, for three different blowing directions, the length of recirculation zone will decrease with the increase of blowing speed. For -45° blowing direction, when the blowing speed is 10 m/s, the length of recirculation zone decreases most obviously, which decreases by 58% and 53%, respectively, compared with +45° and 0° control methods, and decreases by 60% compared with 2 m/s blowing speed. Considering that the blowing device needs to consume energy, the flow control effect is the best when the blowing direction is -45° with the blowing speed of 10m/s. The specific value of recirculation zone length is given in Table 1, and the corresponding variation trend of recirculation zone length can also be obtained.

Table 2 shows the effect of this flow control method, that is, the deviation of the length of the recirculation zone compared with that without blowing is obtained under different blowing ways, in which the length of the recirculation zone without blowing is 15.785 meters. According to the results in Table 2, compared with the non-blowing way, the length of the recirculation zone decreases by 13.3% at most in the state of +45°, when the blowing speed is 20 m/s, while the length of the recirculation zone increases by 6.5% when the blowing speed is 4 m/s, which is contrary to the purpose of adopting flow control measures in this paper. In the state of 0°, the length of recirculation

zone is reduced by 28.7% at most, and the blowing speed is 20 m/s. Under the condition of -45°, the length of the recirculation zone is reduced by 61.5% at most, and the blowing speed is 20 m/s, while the length of the recirculation zone is reduced by 59.4% when the blowing speed is 10 m/s, and these two blowing speeds have similar improvement effects on the flow field. Therefore, according to the improvement effects brought by different flow control methods, the conclusion is that the length of recirculation zone decreases the most at -45°, and considering the energy consumption of the control device, the optimal blowing way is the blowing speed of 10m/s at -45°.

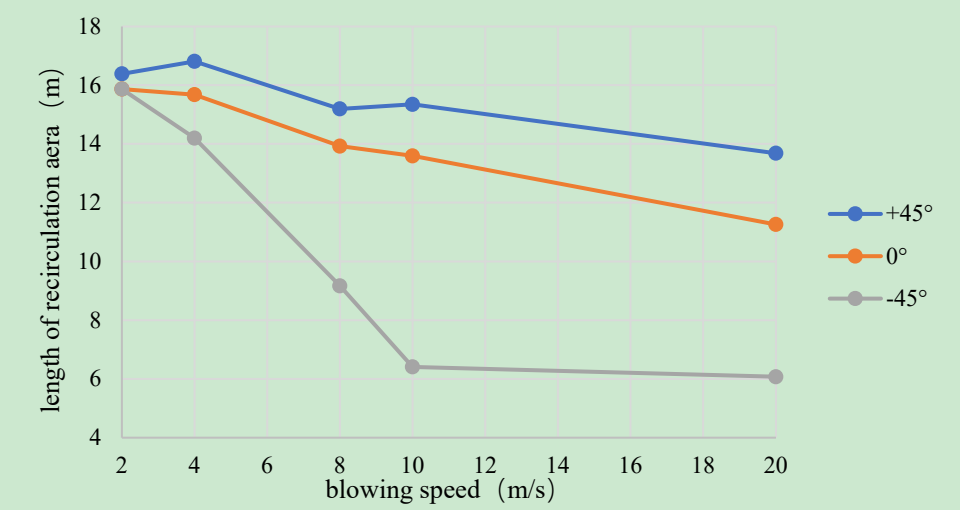


Figure 7. Variation of recirculation area with blowing speed

Table 1. Length of recirculation area for different ways of blowing air

Angle \ Speed	2 m/s	4 m/s	8 m/s	10 m/s	20 m/s
+45°	16.383	16.812	15.192	15.346	13.682
0°	15.868	15.677	13.924	13.591	11.26
-45°	15.868	14.196	9.167	6.41	6.075

Figure 8 and figure 9 show the comparison between the contour of the non-blowing result and the contour of the optimal result, including the vertical velocity and pressure distribution. In the vertical velocity contour, it can be seen that compared with the result of non-blowing, under the optimal blowing way, the range of the recirculation area is obviously reduced, and the backflow area and vertical airflow are concentrated in the range of about one hangar height behind the hangar, which makes the streamline in most areas above the deck more stable, creating a favorable environment for the helicopter to take off and land. Similarly, in the pressure contour, compared with the result of non-blowing, under the optimal blowing way, because the ejected high-speed airflow hits the deck, there is an obvious high-pressure area above the deck, and the low-pressure area behind the hangar is obviously reduced, which weakens the "suction" effect of the vortex area behind the hangar on the helicopter, and reduces the pilot's control load at the same time.

Table 2. Effect of different ways of blowing air

Angle \ Speed	2 m/s	4 m/s	8 m/s	10 m/s	20 m/s
+45°	+3.8%	+6.5%	-3.8%	-2.8%	-13.3%
0°	+0.5%	-0.7%	-11.8%	-13.9%	-28.7%
-45°	+0.5%	-10.1%	-41.9%	-59.4%	-61.5%

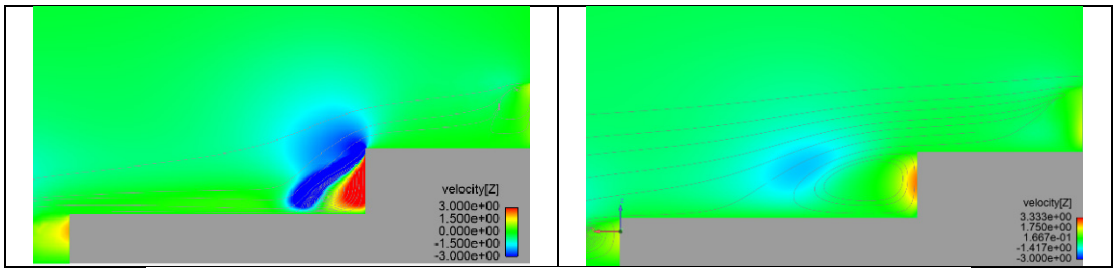


Figure 8. Contours of vertical velocity with/without blowing air

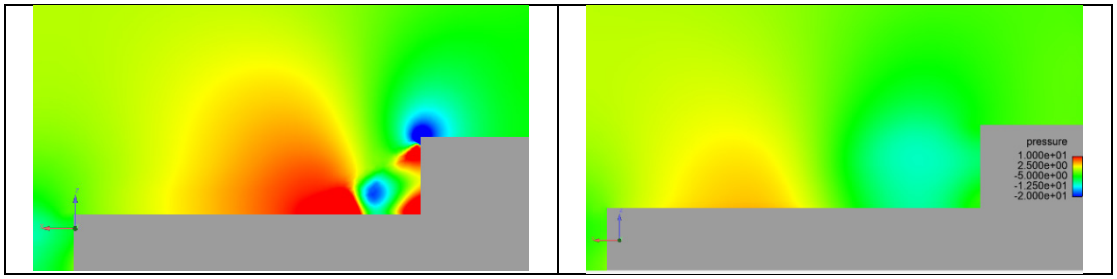


Figure 9. Contours of pressure with/without blowing air

5. Conclusion

Based on the computational fluid dynamics method, the flow field quality above the deck of the destroyer was improved by active flow control method. According to different blowing ways (3 blowing angles and 5 blowing speeds), the conclusions obtained in this paper are as follows:

1. For non-aviation ships, the recirculation area behind the hangar will have a direct adverse impact on the take off and landing of helicopters;

2. By adding blowing device to the rear wall of the hangar, the position and size of the recirculation area behind the hangar can be obviously changed;

3. For different blowing directions, the length of recirculation zone tends to decrease with the increase of blowing speed;

4. Under the condition of -45°, the length of recirculation zone decreases the most. Considering the energy consumption of the control device, the optimal blowing way is the blowing speed of 10m/s with the direction of -45°, and the length of recirculation zone decreases by 59.4%.

References

[1] Lu W. (2014). Flow field properties flight deck of destroyer investigations on flow control methods. Nanjing University of Aeronautics and Astronautics the Graduate School College of Aerospace Engineering.

[2] Hong WH, Jiang ZF, Wang T. (2009). Influence on Air-wake with Different Layout of Ship Superstructure. Chinese Journal of Ship Research, 4(2).

[3] Forrest JS, Owen I. (2010). An investigation of ship airwakes using detached-eddy simulation. Computers & Fluids, 39(4), 656–673.

[4] Bardera-Mora R. (2014). Experimental investigation of the flow on a simple frigate shape (SFS). The Scientific World Journal, 2014, 1–8. https://doi.org/10.1155/2014/818132.

[5] Tai T. (2001) Airwake simulation of modified TTCP/SFS ship. In: RTO AVT Symposium on Vortex Flow and High Angle of Attack, 7-11 May 2001; Loen, Norway.

[6] Shukla S and Singh SN. (2015) A Computational Study of Modified TTCP/SFS Ship Airwakes. In: Proceedings of the 4th International Conference on Ship and Offshore Technology, 10 - 11 December 2015; IIT Kharagpur, India.

[7] Shafer D, Ghee T. (2005) Active and passive flow control over the flight deck of small naval vessels. 35th AIAA Fluid Dynamics Conference and Exhibit. https://doi.org/10.2514/6.2005-5265.

[8] Greenwell DI and Barrett RV. (2006) Inclined Screens for Control of Ship Air Wakes. In: Third AIAA Flow Control Conference, 5-8 June 2006, San Francisco, CA.

[9] Kääriä CH, Wang Y, White MD, et al. (2013) An experimental technique for evaluating the aerodynamic impact of ship superstructures on helicopter operations. Ocean Eng , 61: 97-108.

[10] Bardera-Mora R and Meseguer J. (2015) Flow in the near air wake of a modified frigate. Proc. Inst. Mech. Eng. Part G J. Aerosp. Eng, 229: 1003-1012.

[11] Bardera-Mora R, Conesa A and Lozano I. (2016) Simple frigate shape plasma flow control. Proc. Inst. Mech. Eng. Part G J. Aerosp. Eng , 230: 2693–2699.

[12] Li T, Wang YB, Zhao N. (2021) Influence of Ship Motion on Flow Field over Modified Simple Frigate Shapes [J]. Transactions of Nanjing University of Aeronautics and Astronautics, 38(3):520-534.

[13] Li T, Wang YB, Zhao N. (2020). Numerical Study of the flow over the modified simple frigate shape. Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, 235(12), 1551–1565.

[14] Yuan W, Wall A, Lee R. (2018). Combined numerical and experimental simulations of Unsteady Ship airwakes. Computers & Fluids, 172, 29–53.