Research on Drag Reduction Mechanism and Optimization Design of Bionic Jet Flow

K Chen1, Z G Wei1, Y S Lin1, X F Zhao1, X Qi1, C H Huang1, X H Yang1, H K Zhou1, H B Ke1,2
1Wuhan 2nd Ship Design and Research Institute, Wuhan, 430205, China
E-mail: xjtuchen@foxmail.com

Abstract. In the process of sailing, the frictional resistance of underwater objects must be reduced, which is of great significance for improving speed, range and saving energy. Based on the inspiration of the shark breathing process, this paper simplified the water spray process from the external bronchial slit of the shark breathing to the cross-flow lateral jet process. By establishing a numerical model, the influence mechanism of the jet on the flow resistance was studied, and by analyzing different jet velocities, the change of viscous resistance reveals the influence of jet velocity on viscous resistance. The results show that the side jet can reduce the boundary layer on the surface of the object, thereby having a drag reduction effect, and the drag reduction rate can be about 10%. In addition, as the jet velocity increases, the drag reduction effect is enhanced, and at the back of the jet port the vortex is formed, the outlet pressure is reduced, and the drag reduction effect is strengthened. At the same time, the study found that the arc-shaped jet port is more conducive to the reduction of resistance.

1. Introduction
Reducing resistance is very important for saving energy, especially in transportation, military, sports and other fields. Those fields have an urgent demand for developing the effective methods to reduce resistance. Among the existing drag reduction methods, the bio mimetic drag reduction method is an important branch which get the certain drag reduction characteristics by simulating the evolution of biology. It has a good drag reduction effect and has broad application prospects. For example, some researchers took experimental study on pressure distributions of the T-junction with bionic grill [1,2]. It is also found that the groove surface used in drag reduction has a similar structure to shark scales [3]. The flexible surface used in drag reduction has similar elastic characteristics to the dolphin skin [4-6]. A kind of non-smooth surface modeled on the surface of insects developed in recent years [7-9], and it has impressive performance in drag reduction. Those applications are impressive achievements in bionic research. Sharks are fast-moving aquatic animals. Researches show that the front of the body is lined with large gill plates. In addition to its close relationship with breathing, its function may also be closely related to the reduction of its own muscle strength. Some scholars have conducted studies on jet drag reduction on the gill respiration of sharks. Gu and his fellows conducted a study on the drag reduction of supersonic blunt-body reverse jet flow, focusing on the analysis of the influence of the total pressure and the size of the nozzle on the flow field modal and drag reduction [10-14]. The results show that with the change of the total pressure of the jet, two flow modes can appear, namely the long jet penetration mode and the short jet penetration mode. The jet flow can significantly reduce the resistance. The
maximum drag reduction rate can reach 51.1% within the range of the parameters studied. When the physical parameters of jet flow are unchanged, the drag-reducing rate decrease with the increase of the nozzle size. Zhao et al. proposed the concept of bionic jet technology for the drag reduction of the moving body surface [15]. Based on the establishment of the jet drag reduction model, the numerical simulation method was used to analyze the drag reduction characteristics of the bionic jet. The mechanism of jet drag reduction was analyzed in terms of pressure distribution and velocity field distribution. The drag reduction and energy saving effect of the rotating jet surface of the rotating body was analyzed. It was found that the jet velocity, mainstream field velocity, jet hole position and structure are the main factors which strongly affect the drag reduction and energy saving. The simulation results also show that the drag reduction rate can reach 11.6% under the optimal parameter combination. The above research is mainly aimed at the jet hole. There are few studies on the jet grid. Due to the interaction of multiple fluids, the flow field structure of the jet grid is more complicated and the application background is more extensive. so, it is quite necessary to conduct a fully research on the drag reduction mechanism of the jet grid.

This paper constructed a bionic jet grid, which is similar to the shark gill jet characteristics, by simplifying the morphological characteristics of shark gills. The drag reduction characteristics and analysis of the drag reduction mechanism of the shark gill jet grid are carried out by the computational fluid dynamics software ANSYS. At the same time, the global optimization of structural parameters of the bionic jet grid was studied, and the best jet structure parameters were obtained.

2. Simulation model instruction
In order to maintain a certain water level, most sharks need to keep swimming in the water. The oxygen-rich water enters the pharyngeal cavity during swimming. The gas exchange through the gills, and the oxygen-depleted water flows out of the gills to form a jet. The breathing process is shown in Figure 1. The jet fluid ejected from the external gill fissure during shark breathing changes the structure of the flow field around the gill fissure. In order to study the effect of the gill fissure jet on the frictional resistance of the shark body surface fluid, this article describe a gill fissure jet grid. The surface structure around the fissure is simplified into a flat plate, and the external gill fissure is simplified into a jet grid on the flat plate. The flat structure with a jet grid is the bionic jet surface. The breathing process of the gill is simplified as a continuous jet process. For the drag reduction effect of the jet surface, a smooth surface with the same flow field conditions is used as a control surface, and the calculation domain is established as shown in Figure 2. In Figure 2, the lower surface is a smooth surface, and the upper surface is a jet surface with a jet grid. The parameters are as follows: the flow X direction is 7.5 m, the Z direction is 0.5 m, the Y direction is 3.0 m; The cross-flow velocity = 10 m/s, the lateral jet velocity is 0.5-2.0 m/s, and the jet direction is perpendicular to the cross-flow direction.
3. Numerical simulation results and analysis

3.1. Introduce of the simulation model

The turbulence model uses the SST $k-\omega$ model. The SST $k-\omega$ model was developed by Menter so that it can be independent of the $k-\varepsilon$ model in a wide range of fields, and the $k-\omega$ model has a wide range of applications and accuracy in near-wall free flow.

In this paper, ICEM CFD14.0 is used to structure the calculation domain. The global maximum grid length is set to 4 mm, the jet hole grid length is 2 mm, and the wall first layer grid length is 0.2 mm. The calculation formula is as follows, the grid growth rate is 1.1, and the number of grid cells in the models is $(3.17 \sim 4.59)$ million.

3.2. Evaluation index of drag reduction effect

The resistance of fluid flowing through a solid surface includes friction resistance, friction resistance and differential pressure resistance. The differential pressure resistance belongs to the shape resistance. The smooth surface and the jet surface are parallel to the flow field direction. Only friction resistance is considered. The frictional resistance on the model surface includes forces in the three directions of X, Y, and Z. However, relative to the flow direction X, the component forces in the remaining two directions are negligible. The calculation method of friction resistance is as follows:

$$f = \int \tau dA = \sum_{i=1}^{n} \tau_{i} |A|$$

(1)

Where: $f$ is the viscous resistance; $\tau_{i}$ is the shear stress of the discrete element on the wall, which is the area of the discrete element on the wall. In this paper, the dimension position is normalized by the diameter of the jet grid, where the jet grid is opened at the $x/d = 0$.

**Figure 2.** The simulation model
Figure 3. The viscous force changes under different angle and jet velocity at different position

Figure 3 shows the downstream viscosity resistance changes with the jet velocity under different jet angles. The downstream position is dimensionless. The black curve is the viscosity resistance of the smooth wall without jet. When the jet velocity is small, the closer to the area of the jet outlet, the smaller the viscous resistance. When the jet velocity is higher, the area with x/d = 2 has the smallest viscous resistance, and the resistance direction is the same as the mainstream direction, which is conducive to fluid flow. The above results indicate that when the jet velocity is high, a back flow is formed in the area of twice the pipe diameter downstream. In addition, as the speed increases, the more the viscous resistance decreases, the more obvious the effect of jet drag reduction. At different jet angles, the change of the viscous resistance shows the same trend. With the change of the jet angle, the change of the viscous resistance is very small, which shows that the jet angle has little effect on the mainstream flow field. Similarly, it can be found that when the jet angle is 45 degrees, the effect of jet drag reduction is most obvious.

Then further counted the sum of the viscous resistances in all downstream areas under different speeds and different angles, and compared them. The results are shown in Figure 4, which indicate that the effect of jet drag reduction has a nonlinear relationship with the angular range. Overall, the resistance reduction effect is most obvious at 45 degrees. At the same time, it is found that the jet drag reduction effect is almost positive linear with the jet velocity range. At the same time, the internal resistance of the branch flow at different jet velocities is also analysed. As shown in Figure 5, the results show that the higher the jet velocity, the greater the internal resistance of the jet branch. The jet angle also has a certain effect on the internal resistance of the branch. When the jet velocity is high, both the vertical jet and the parallel jet are not conducive to drag reduction, and when the grid is at a certain angle of inclination with the mainstream direction, it is more conducive to drag reduction.
3.3. **Mechanism analysis of jet drags reduction**

Figure 6(a) and (b) shows the pressure and vorticity clouds at a jet velocity of 1.0 m/s and a jet angle of 45 degrees. Based on the above results, it can be seen that there is a very obvious low pressure area at the jet exit. The appearance of the low-pressure zone is very conducive to the flow of mainstream fluid, thereby reducing the fluid resistance of the wall surface. It can also be found from the vorticity diagram that a clear vortex structure occurred at the jet exit. Figure 6(c) and (d) shows the velocity cloud of the jet model. It can also be found that there are a low velocity area and a recirculation area in the jet exit area, and the low velocity at the wall zone helps to reduce the wall's viscous resistance.

Due to the obstructive effect of the jet fluid, a backflow zone is formed on the backflow surface of the jet hole. The velocity direction near the wall in the backflow zone is opposite to the incoming flow direction, and the local friction factor is negative, which has a significant impact on the drag reduction effect; and downstream of the jet hole a counter-rotating vortex pair is formed and continuously extends downstream, and a secondary vortex is induced at the wall surface, which increases the thickness of the viscous bottom layer of the boundary layer and reduces the velocity gradient, which has a drag reduction effect.
3.4. Analysis of the effect of grid shape on jet drag reduction

In addition, the influence of the shape of the grid on the drag reduction of the jet is also analyzed. The following shapes of the grid are mainly established, as shown in Figure 7. No.1 structure is a rectangle. No.2 contains two symmetric parallelograms, and the angle is 150 degree. No.3 is a transition shape, the angle is 135 degree. No. 4 is an arc, and its radian is 0.785 rad.

The resistance analysis of these different jet grids is carried out. Figure 8 shows the pressure clouds of jet grids of different shapes. The low-pressure area formed at the tail of the jet grid No. 1 is the most obvious, but the radiation range is small, and the low-pressure area formed by the grid No. 4 is obvious, and the radiation range is also large. By calculating the resistance reduction value of different shapes of grids, as shown in Figure 9, the results show that the jet grid of shape 4 has the best drag reduction effect.

![Image of pressure, velocity, and turbulent kinetic energy distributions](figure6.png)

**Figure 6.** Simulation cloud picture of the pressure, velocity and turbulent kinetic energy distribution

![Image of different shapes of jet grids](figure7.png)

**Figure 7.** Different shape of the jet grid
Figure 8. The pressure distribution of different shape of the jet grid

Figure 9. The drag reduction of different shape of the jet grid

4. Conclusions
The bionic jet structure can effectively reduce the viscosity resistance of the fluid, so as to achieve the drag reduction effect on the mainstream. Among them, the jet velocity has a greater influence on the drag reduction effect, the greater the velocity, the more obvious the drag reduction effect; at lower jet velocity, the jet angle has less effect on the jet velocity, and at higher jet velocity, the jet angle has a greater effect. Affected, the best angle is 45 degrees. The mechanism of jet drag reduction is that the shark-like gill jet thickens the viscous bottom layer of the downstream flow field at the outlet, the velocity gradient of the boundary layer fluid decreases, and the wall shear stress decreases. In addition, there is a reverse vortex downstream of the jet outlet. The stable large vortex structure makes the boundary layer fluid orderly, reducing the effect of turbulent pulsation on the wall surface.

Acknowledgement
The present work is supported by National Natural Science Foundation of China (No. 51706159).

References
[1] Huang J C and Johnson M W 2008 Boundary layer receptivity measurements on compliant surfaces Int. J Heat Fluid Fl 29 495-503
[2] Tian L M, Ren L Q and Liu Q P 2007 The mechanism of drag reduction around bodies of revolution using bionic nonsmooth surfaces J Bionic Eng 4 109-116
[3] Wang J J, Lan S L and Chen G 2000 Experimental study on the turbulent boundary layer flow over riblets surface Fluid Dyn Res 27 217-29
[4] Yin Y T, Li S C, Chen K, Wang L B, Lin M and Wang Q W 2017 Experimental study on pressure distributions around a circular cylinder in the branch of a T-junction Chemical Engineering Transactions 61 24-9
[5] Chen Y, Chen Y C and Huang W 2012 Experiment investigation of drag reduction using riblets for a slender body Journal of Experiments in Fluid Mechanics 26(2): 42-5
[6] Feng B B, Chen D R and Wang J D 2012 Riblet surface drag reduction on subsonic aircraft Journal of Tsinghua University (Science and Technology) 52(7): 967-72
[7] Hu H B, Song B W and Liu Z Y 2011 Research at the computational methods of flow fields over riblets surface Acta Aerodynamica Sinica 29(3): 348-54
[8] Qi Y C, Cong Q and Wang J Y 2012 Optimization design and drag reduction mechanism research on groove shape bionic needle Journal of Mechanical Engineering 48(15): 126-30
[9] Wang J J, Lan S L and Chen G 2000 Experimental study on the turbulent boundary layer flow over riblets surface Acta Mechanica Sinica 32(5): 621-6
[10] Gu Y Q, Zhao G and Xu G Y 2012 Experimental study on drag reduction characteristics of bionic jet surface Journal of Central South University (Science and Technology) 43 3007-12
[11] Gu Y Q, Zhao G and Zheng J X 2012 Drag reduction characteristics on jetting surface with jet angle-jet velocity coupling Journal of Xi’an Jiaotong University 46 71-7
[12] Gong W Q, Li X H and Huang S J 2002 Experiment study on the mechanism of riblets drag reductio J Eng Thermophys 23(5): 579-82
[13] Zhang C C, Ren L Q and Wang J J 2007 Experiment and numerical simulation on drag reduction for bodies of revolution using bionic scrobiculate ringed surface Journal of Jilin University of Technology (Natural Science Edition) 37(1): 100-5
[14] Xiao Y 2005 The study on mixing characteristics of a tandem multiple jet group in cross-flow [D] Nanjing: Hohai University College of Water Conservancy and Hydro power Engineering 1 49-50
[15] Zhao G, Gu Y Q and Zheng J X 2012 A testing platform based on bionics drag reduction theory for friction resistance Communications in Information Science and Management Engineering 2(5): 34-9