DOI: 10.54254/2753-8818/56/2024.18488

Simulation and Flow Field Analysis of Objects in Fluids

Zirun Wu, Yiran Chen
Experimental High School Attached to Beijing Normal University, Beijing 100000, China
Chongqing Depu Foreign Language School, Chongqing 400000, China

zirunwu22@gmail.com
abc2843918@qq.com

Abstract. This project mainly focuses on the investigation of the distribution of the streamlines around an object in a fluid like air or water and verifies the effectiveness of the shape of the objects based on the pressure and drag distribution to reduce drag and energy expenditure. This paper uses the simulation of streamlines to simulate the behavior of the streamlines of a certain fluid and analyzed the curvature and density of the streamlines to study the pressure and drag experienced by certain parts of the objects. This study is mainly divided into two parts: rocket and swimmer. The common result is that it is better for an object to be streamline shaped to minimize drag and save more energy.

Keywords: streamlines, drag, curvature, streamline shaped

1. Introduction: Overview of the project

Fluid dynamics enables engineers to optimize designs of lower energy consumption, higher efficiency,greater safety and reliability. Understanding the drag suffered by objects like submarine and rocket will effectively guide engineers to opt for more proper materials to prevent explosion. Also,it can make the design better to develop more efficient shape that can combat with the drag, and thus reduce the over-consumption of fuel, leading to a more environmentally friendly future.

There were several pioneering studies on this topic, like Hermann Schlichting's research used wind tunnel experiments to study fluid flow around objects and its effect on drag, analyzing boundary layer development and separation to illuminate drag reduction. Dr. Robert H. Goddard used wind tunnel tests to study rocket shapes and heat management. He found that streamlined shapes reduce drag and that using heat-resistant materials and cooling methods is essential for managing thermal loads.

In this study, we want to verify the effectiveness of the shapes.

This work concentrated on the study of the distribution of streamlines around an object in a certain fluid and what role the shape plays in the allocation of those streamlines using simulation. Here are two examples on aerospace engineering and swimming:

The significance of optimising the shape of a rocket is undeniable since it can approach the best-fit one to minimize the drag and maximize the lift, increasing the rocket's efficiency in terms of fuel economy, payload capacity and so on. The less the consumption of the fuel, the more environmentally friendly it is, leading to a greener aerospace practice. Although the current rockets are already well-designed, we are still at liberty to verify the effectiveness of it and develop better understanding of this field.

In swimming, minimizing the drag caused by water is always a goal that every athlete longs for. By analyzing and interpreting the streamlines of fluid field around swimmer, we are able to improve current swimming equipment depends on simulation. This exploration is worthwhile, since a tiny improvement could lead to a record-breaking, especially in short distance. It can also allow divers perform better underwater, which may contribute to the development of diving.

2. Theoretical background

We begin by writing the components of the velocity of the fluid relative to the Cartesian coordinate system:

$$v = v_x\hat{x} + v_y\hat{y}, \nabla \cdot v = 0, \tag{1}$$

We can express v in terms of a scalar function, (known as the stream function), which we define as follows:

$$v_x = \frac{\partial \psi}{\partial y} \quad v_y = -\frac{\partial \psi}{\partial x} \tag{2}$$

Noting that our fluid is considered to be incompressible [1] and irrotational [2], this means that the density is constant throughout the entire fluid, and that there is no net "rotation" of the fluid. Mathematically, we can express these two statements in the following way:

$$\nabla \cdot \boldsymbol{v} = 0 \tag{3}$$

$$\nabla \times \boldsymbol{v} = \boldsymbol{0} \tag{4}$$

It is easily seen that the definition of given by Equation (2) automatically satisfies Equation (3). By substituting (2) into (4), we can show that must also satisfy Laplace's [3] equation:

$$\nabla^2 \psi = \frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0 \tag{5}$$

In Cartesian coordinates, Equation(4)can be written as:

$$\begin{pmatrix} 0 \\ 0 \\ \frac{\partial v_y}{\partial x} - \frac{\partial v_x}{\partial y} \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \end{pmatrix} \tag{6}$$

The conclusion here is that Laplace's equation may be used to describe the stream function,Ψ, of an irrotational incompressible flow of an ideal fluid in two dimensions. Laplace's equation is crucial in potential flow theory and various applications in fluid dynamics, as it defines regions of flow with no sources or sinks. Only when the flow satisfies the Laplace's equation could we do further exploration on the problem.

2.1. Solutions to Laplace's Equation Using Separation of Variables

We will solve Equation(5) by using the method of seperation of variables. First, we assume a solution of the form:

$$\psi\text{(x,y)=X(x)Y(y)}$$

Then substitutes it into the Laplace's equation:

$$X''(x)Y(y) + X(x)Y''(y) = 0$$

And then, we divide it by X(x)Y(y):

$$\frac{X''(x)}{(x)} + \frac{Y''(y)}{y} = 0$$

This implies:

DOI: 10.54254/2753-8818/56/2024.18488

$$\frac{X''(x)}{X(x)} = -\lambda, \quad \frac{Y''(y)}{Y(y)} = \lambda$$

Here we assume λ to be a seperation constant.

Then we solve for X(x) The equation of X(x) is:

$$X''(x)\text{+}\lambda\text{X(x)=0}$$

This is a second-order linear differential equation. The general solution depends on the sign of λ For λ>0:

$$X(x) = A\cos(\sqrt{\lambda}x) + B\sin(\sqrt{\lambda}x)$$

For λ =0:

$$X(x) = A(x) + B$$

For λ<0(let λ=-μ^2):

$$X(x) = Ae^{\mu x} + Be^{-\mu x}$$

By doing the similar process, we can get the general solution for Y(y): For λ >0:

$$Y(y) = Ce^{\sqrt{\lambda}y} + De^{-\sqrt{\lambda}y}$$

For λ =0:

$$Cy + D$$

For λ <0(let λ=-μ^2):

$$Y(y) = C\cos(\mu y) + D\sin(\mu y)$$

The general solution is a sum of products of X(x) and Y (y)for different separation constants. For example, we take λ >0. Combining two solutions, we get: $\psi(x, y) = \sum_n \left(A_n\cos(\sqrt{\lambda_n}x) + B_n\sin(\sqrt{\lambda_n}x)\right)$

$\left(C_n e^{\sqrt{\lambda_n}y} + D_n e^{-\sqrt{\lambda_n}y}\right)$

This is a general solution and we will need boundary condition [4] to determine the specific form and coefficients of the solution.

2.2. Physical Motivation for Boundary Conditions

In reality, ideal fluid does not exist. However, in order to solve complex questions in fluid dynamics, this paper simplify our problem by assuming the fluid we discuss is inviscid, no viscosity. It has no internal friction and no suffer no fraction from the surface of obstacles. Under this circumstance, two types of boundary condition are involved. First, no-slip [5] boundary condition:

$$\mathbf{v} = \nabla\psi \tag{7}$$

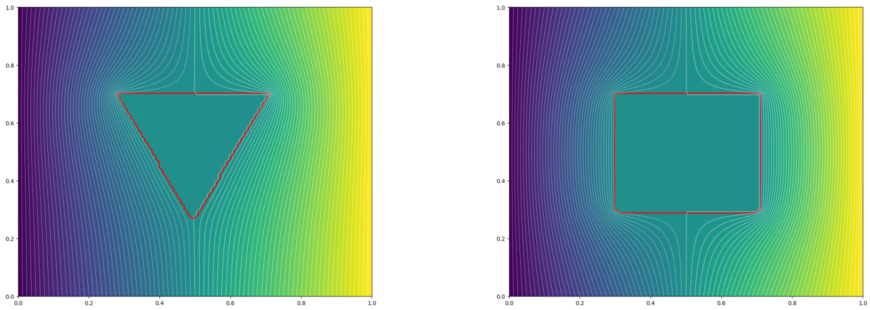
$$\begin{pmatrix} v_x \\ v_y \end{pmatrix} = \begin{pmatrix} \frac{\partial\psi}{\partial y} \\ \frac{\partial\psi}{\partial x} \end{pmatrix} \tag{8}$$

In this condition ψ=0 or constant means at any point on the surface of the obstacle, the velocity will equal to 0. Second, no-penetration [6] condition: This condition indicates the water moves along the obstacle but never goes into the boundary. The algorithm used to define two boundary conditions is presented in Table.1.

DOI: 10.54254/2753-8818/56/2024.18488

Table 1: code used for defining two boundary conditions

no-slip boundary condition	$\psi[obstacle\_mask]$ = 0.0
no-penetration boundary condition	$\psi[0,:] = -1.0$ $\psi[-1,:] = 1.0$



(a) (b)

Figure 1: Streamlines around triangle and rectangle

3. Results

This section will present the results and analysis of simulation of several obstacles. However, there are something about streamlines that need to be clarified beforehand, including what role the density or the curvature of the streamlines play to influence the pressure. The streamline, by definition, is the speed of motion of fluid particles, so it is conspicuous to conclude that the denser the streamlines, the higher the speed, and thus the lower the pressure according to the Bernoulli's equation [7]. As for the curvature of the streamline, which is another factor reflecting the pressure of the fluid. The more it bends, the higher the pressure. So, there might be a dilemma where if the curvature is large (large pressure) and the density of the streamline is high as well (low pressure), which has greater effect? We can currently priories the effect of curvature. Of course, there will be turbulent flow in certain areas that will cause drag, but we won's go to that here for the reason that turbulent flow is not the pivot in this paper.

3.1. Simulation of Rocket

In order to verify the effectiveness of the current shape of the rocket, we will begin with the composition of a standard rocket, which is basically a triangle on the top with a rectangle forming its body parts viewing from the two dimensional world(here we temporarily ignore the effect of the shape of the fuel).

3.1.1. Choice for head/body In this subsection, the benefits and drawbacks of both triangle and rectangle are going to be analysed using simulation figures of them staying in a certain fluid, causing the distribution of streamlines behave differently.

So firstly is the study of the distribution of streamline of a triangle as shown in Figure 1 (a). It is known that the denser the streamlines, the higher the speed, the lower the pressure. And also the curvature of streamlines in the base vertices is quite large compared to that in the vertex on the lower position. So we can say that the net pressure, and thus force(equals to area times pressure, and since the area on the top is also larger than on the lower part, which can better support the fact that the net force is downward), and since an object tends to move down the pressure gradient towards places with lower pressure, so this shape can generate some drag force due to the pressure difference, even.

DOI: 10.54254/2753-8818/56/2024.18488

Next is the analysis of a rectangle. It is noticeable in Figure 1 (b) that the streamlines are distributed more evenly along the side body of the rectangle compared to the triangle since the rectangle is a perfectly symmetrical-shaped one and we can naturally think that the net force is zero because forces from all directions are cancelled out due to the property 'symmetrical', exposing the remarkable benefit and drawback when it comes to the comparison with the triangle. On the one hand, the merit is that the evenly allocated pressure(force) can make the heating effect of on the side body more uniform, preventing cases like explosion or problems caused by welding different materials to combat with various levels of heating effect, which will reduce costs as a natural consequence. On the other hand, however, from the sole perspective of the simulation figure, we can see that the force along the motion of the rocket is not as large as that of a triangle, since the triangle is more streamline shaped and we can see that on both the up and down sides of the rectangle, there are a lot of flow separations, which is simply the streamlines detach from the body, causing unignorable amount of pressure, and thus drag on the head. Taking everything into account, triangle is a preferred option for the head and rectangle is better for the body. The conclusion is that the optimised choice for our rocket is triangle being the head and rectangle

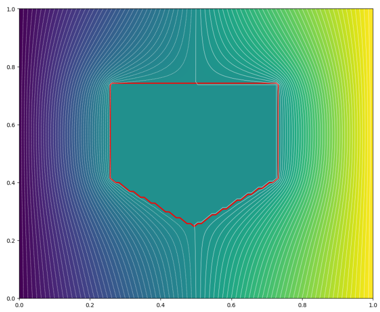
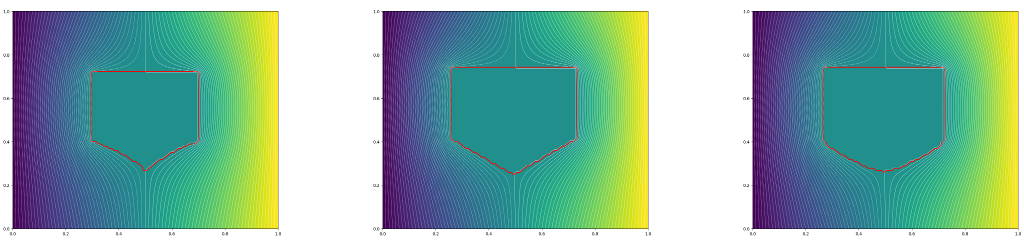


Figure 2: Streamlines of a rocket

being the body, so that it can take advantage of more evenly distributed pressure on its body and less resistance of its head. The simulation of the rocket is shown in Figure 2.

3.1.2. Choice for the curvature of the head This part will discuss the choice for the head of the rocket from three options, outward-curved triangle, inward-shaped triangle and a straight-lined triangle.

Here, we focus on the nose (triangle part) of them using the results given by simulations in Figure 3. The first focus is flow separation. The outward shaped nose helps minimize flow separation by making



(a) (b) (c)

Figure 3: Streamlines of rockets with different noses

DOI: 10.54254/2753-8818/56/2024.18488

progressively change of its surface to maintain the flow attach the surface over a larger surface area.The outward bended one has the streamlines attaching it in the most closely, which will generate less pressure, and thus less drag, protecting the head of the rocket. The second focus is the distribution of pressure. The outward head creates a more gradual change of the pressure, avoiding abrupt or sudden ones.The design of an outward bended one allows for more gradually change of pressure, which contributes to less flow separation. To sum up, the outward-curvature nose allows for smoother pressure declining gradient and thus avoids abrupt change, which leads to less flow separation and more evenly distributed pressure and makes its head experience a more uniform drag. The third focus is that the blank area in the lowest position of the vertex of the head of the outward one is the largest, so the pressure/drag in there is the lowest.

To summarise, the outward curved one has striking benefits, including smoother pressure distribution, less flow separation and abrupt pressure change as well as less drag in the vertex of the head. Therefore, it was chosen for the rocket head.

3.2. Simulation of swimmer

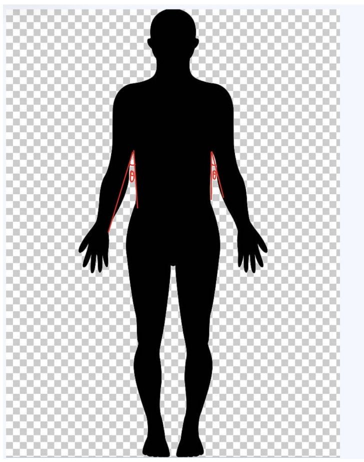


Figure 4: Swimmer silhouette

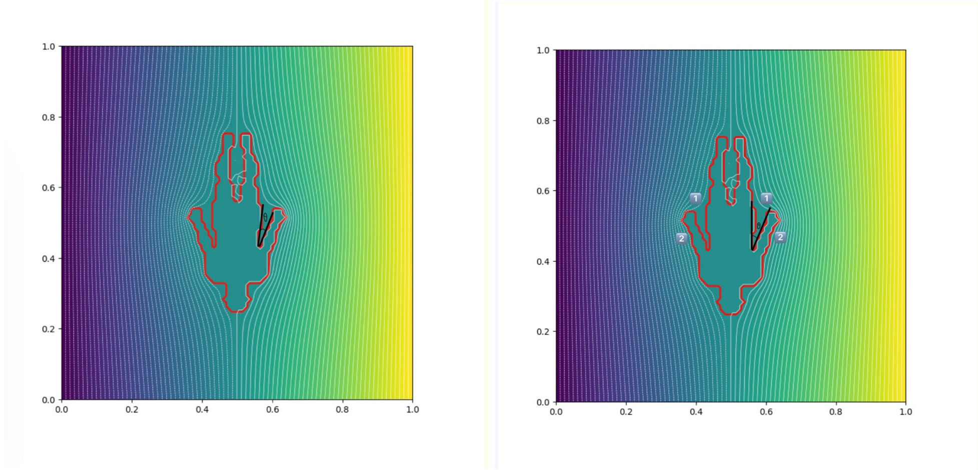
3.2.1. Algorithm method for simulation For the swimmer part, this project used the SOR iterative import a relaxation parameter to accelerate the convergence of the iterative solution.

Here, we import the image 'Swimmer_silhouette', which is shown in Figure 4, into the code and use SOR method to simulate the fluid field around it. We introduce two kinds of swimmer, and their difference is the angle θ between arm and torso. We define swimmer 1 has then $\theta>0$, whereas swimmer 2(an improvement of swimmer 1) has the $\theta=0$.

3.2.2. Analysis of the simulation results of the swimmer 1 The results of simulation are shown in Figure 5. Streamlines [8] at ②(arms and wrists) are much more dense than the streamline at ①(thighs), which means that the flow at arms and wrists has greater velocity than at things. From the curvature aspect, streamlines at ②are slightly curved and almost equal to a straight line. However, these at ①are largely curved. The greater streamlines bend, the larger forces will act on the swimmer, as large curvature leads to large pressure gradients.

This indicates that this swimmer suffer from greater pressure at ②than at ①. The difference in pressure gives this swimmer a force acting opposite to the relative motion of the swimmer, which we call it drag. The larger the difference in pressure is, the larger drag the swimmer will encounter. Besides, since the lateral streamlines are also in great curvature, the swimmer will hence face a great lateral force.

Proceedings of the 2nd International Conference on Applied Physics and Mathematical Modeling
DOI: 10.54254/2753-8818/56/2024.18488



(a) (b)

Figure 5: Streamlines around swimmer1

After a closer look, We can see from the chaotic streamlines that there are turbulent drags [9] between the swimmer's two legs. Although our paper will not focus on turbulent flow, it is still necessary to briefly explain the affect of turbulent drag [10] on the simmer's overall performance.

In reality, the degree of impact from turbulent drag might vary among different categories of swimmers, since athletes often have different body configurations that may alter the interaction of turbulent drag with their techniques and strategies. If we take turbulent drag into account, there will be a decrease in the swimmer's overall performance than we previously expected, since the drag may increase energy expenditure and negatively affect the swimmer's propulsion efficiency. Drag will be influenced by velocity. There relationship is expressed mathematically as:

$$F_D = \frac{1}{2}\rho v^2 C_D A \tag{9}$$

Where F_d is the drag force, ρ is the density of the fluid, A is the cross sectional area, and C_d is the coefficient-a dimensionless number. The equation indicates that as velocity increases, the drag force increases with the square of the velocity. [11]

Also, higher speed of swimmers will increase the turbulent drag due to the chaotic fluid motion around their torsos and limbs, which may significantly contribute to overall drag force.

3.2.3. Analysis of the simulation results of the swimmer 2 In order to reduce the drag, we now introduce Figure 6, which is an improvement of Figure 5. Compare with swimmer 1(Figure 5), swimmer 2(Figure 6) has the angle θ=0. The difference of pressure at ④(arms and wrists) and ③(thighs)is reduced. Also, the curvature at ③also decreased a lot. Therefore, swimmer 2 face less overall drag and smaller lateral force underwater.

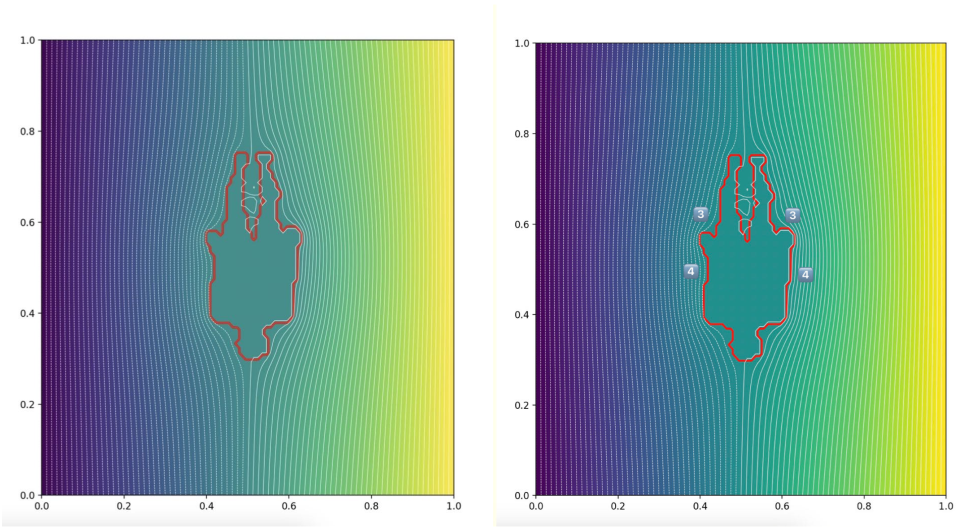
The comparison shows that increasing how streamlined a swimmer is can significantly increase the overall efficiency- the more streamlined the less energy needed to overcome the drag due to water, and hence the faster the swimmer will go.

This finding can contribute to the improvement of swimming suits and equipment, and has the potential to give athlete a significant competitive advantage.

4. Conclusion

This work involved simulating the streamlines of a rocket and a swimmer within fluids like air and water, comparing pressures with different shapes to understand, verify, and improve the effectiveness of current

DOI: 10.54254/2753-8818/56/2024.18488



(a) (b)

Figure 6: Streamlines around swimmer2

designs.

Assumptions of incompressible and irrotational fluid were made to validate that Laplace's equation applies to the stream function, and the separation of variables method was used to solve it. The boundary conditions applied were no-slip and no-penetration. These principles were then used to determine the contour of the obstacles.

Python was used to simulate the swimmer and rocket in water and air, respectively. The code updates each point in a two-dimensional flow field,which exclude boundary and obstacle areas) using the SOR iterative method and visualizes the flow field and obstacles.

For the rocket, the effectiveness of the current shape was verified by dividing it into two two-dimensional parts: a triangle for the head and a rectangle for the body. The results showed that a triangle experienced less drag compared to a rectangle due to its streamlined shape. The rectangle experienced more evenly-distributed drag on the sides, reducing the risk of overheating and preventing pressure buildup or explosion due to sudden temperature spikes.

For the swimmer, reducing the angle between arms and thighs to achieve a more streamlined position resulted in less curvature in the streamlines and lower energy consumption to overcome drag and lateral pressure.

Author Contribution

Zirun Wu and Yiran Chen contributed equally to this work and should be considered co-first authors.

References

[1] Ronald L Panton. *Incompressible flow*. John Wiley & Sons, 2024.

[2] Jeffrey S Marshall. *Inviscid incompressible flow*. John Wiley & Sons, 2001.

[3] George H Shortley and Royal Weller. The numerical solution of laplace's equation. *Journal of Applied Physics*, 9(5):334–348, 1938.

[4] H Feshbach and EL Lomon. The boundary condition model of strong interactions. *Annals of Physics*, 29(1):19–75, 1964.

[5] S Richardson. On the no-slip boundary condition. *Journal of Fluid Mechanics*, 59(4):707–719, 1973.

[6] Reindorf Nartey Borkor, Magnus Svärd, and Peter Amoako-Yirenkyi. A stable scheme of the curvilinear shallow water equations with no-penetration and far-field boundary conditions. *Computers & Fluids*, 269:106136, 2024.

[7] Ruqiong Qin and Chunyi Duan. The principle and applications of bernoulli equation. In *Journal of Physics: Conference Series*, volume 916, page 012038. IOP Publishing, 2017.

DOI: 10.54254/2753-8818/56/2024.18488

[8] Marco Roberto Thiele, Margot G Gerritsen, and Martin J Blunt. *Streamline simulation*. Society of Petroleum Engineers, 2011.

[9] Peter Bradshaw and AD Young. Effects of streamline curvature on turbulent flow. Agard Paris, 1973.

[10] Lu Chen, Shao Gang Liu, Dan Zhao, Hongtao Guo, Jundong Wei, and Yuxin Liu. Stability and drag reduction in turbulent flow of skin with quasi-periodic elastic supports. *Proceedings of the Institution of Mechanical Engineers, Part M: Journal of Engineering for the Maritime Environment*, 236:1057 – 1068, 2022.

[11] Armin Zare, Binh K. Lieu, and Mihailo R. Jovanović. Turbulent drag reduction by streamwise traveling waves. In *2012 IEEE 51st IEEE Conference on Decision and Control (CDC)*, pages 3122–3126, 2012.