Innovative Numerical Method to Partially Solve the Airfoil Flow

Siyuan He

Department of Naval Architecture and Ocean Engineering, Harbin Institute of Technology (Weihai), Weihai, 264209, China

siyuan.he.2021@uni.strath.ac.uk

Abstract. With the meteoric rise of modern computer science, computational fluid dynamics (CFD) has evolved into a formidable tool adept at addressing multifaceted challenges across diverse domains. Historically, CFD's maiden application was in the aviation sector, aimed at refining the aerodynamic contours of airfoils. Fast forward several decades and its deployment in this arena has reached unparalleled sophistication. This paper harnesses numerical methodologies within CFD to decipher flow dynamics around an airfoil. Central to this exploration is the small disturbance equation, a derivative of the full potential equation. The study meticulously discretizes this equation using advanced mathematical techniques. Furthermore, boundary conditions, namely the wall side and Neumann, are judiciously employed to ascertain boundary point values. Leveraging Python-based iterations, we derive numerical solutions for streamlining patterns and Mach number contours. A rigorous error analysis affirms the validity and efficacy of our results. While the study acknowledges certain inherent limitations, the findings compellingly delineate the flow characteristics around the airfoil in both subsonic and supersonic scenarios, offering valuable insights for future aerodynamic endeavors.

Keywords: Computational fluid dynamics, small disturbance equation, airfoil.

1. Introduction

The field of computational fluid dynamics (CFD) is quite active. [1]. Modern fluid mechanics, numerical fluid mechanics, and computer science are all combined in it [2]. Its study realm uses numerical methods based on computers to explore the physical laws of the motion of fluids. During this process, numerical simulation is done, which is both time-saving and energy-saving compared with traditional experiment methods. It is widely recognized that CFD has developed since the 1960s, and its development undergoes four stages: inviscid and linear, inviscid and nonlinear, the N-S equation for Reynolds averaging, and the complete N-S equation [3]. With the rapid development of computer science, CFD has become more powerful in dealing with complicated actual problems. More difficulties can be overcome using a numerical method and getting corresponding results. The numerical method of CFD was first used to solve the problems in the Aviation field [2]. People use this method to design the aerodynamic shape of aircraft and calculate the aerodynamic loads to reduce the times of wind tunnel testings. Later, CFD has advanced and is widely used in a variety of diverse fields in addition to aerodynamics. It is also applied in shipbuilding, civil engineering, water conservancy projects, environmental engineering, etc. This results from its capacity to foresee performance testing of new processes or designs before they are produced or used [4]. Due to its great contribution, more resources can be saved during the experiment process, which is important in scientific research and engineering applications [2].

In this paper, the author will focus on using numerical methods in CFD to solve some normal problems about the flow past the airfoil. It is well known that CFD has several typical numerical methods such as finite difference scheme, finite volume scheme, finite element scheme and so on. Although they have different mathematical principles, their core ideas are the same. They both rely on mathematical methods like approximation to achieve the discretization. This article will apply a finite difference scheme to explore some basic flow characteristics past the airfoil. The finite difference scheme is one of this process's most widely accepted methods. Its advantage is that the theory is based on the classical mathematical approximation theory, which is easily understood and accepted. However, when encountering a complex fluid region, it is hard and inconvenient to deal

with the boundary condition [3]. Thus, this will lead to some inevitable errors. Nevertheless, Neumann's boundary condition initially overcame this shortcoming.

Discretization of the governing equation is the most important step in studying the flow past the airfoil. The small disturbance equation (SDE), which was derived from the complete velocity potential equation, serves as the governing equation in this case. It is concluded that the Euler equation and SDE are two normal ways to solve inviscid flow [5]. It was discovered that the Euler equation simulation worked well. However, the transonic zone had numerical inaccuracies and discontinuities [5]. Moreover, the Euler equation also requires much time and money in computing [6]. Thus, the author will choose SDE because it is valid for thin wings at freestream Mach number near unity and time-saving [7].

2. SDE

The full velocity potential equation needs to be considered first to obtain SDE. It can be listed as [8]:

$$(1-M_x^2)\varphi_{xx}+\left(1-M_y^2\right)\varphi_{yy}+(1-M_z^2)\varphi_{zz}-2M_xM_y\,\varphi_{xy}-2M_xM_z\varphi_{xz}-2M_zM_y\varphi_{yz}=0 \quad (1)$$

$$M_x=\frac{\varphi_x}{c},\ M_y=\frac{\varphi_y}{c},M_z=\frac{\varphi_z}{c},\ M^2=\frac{|\nabla\varphi|^2}{c^2} \quad (2)$$

$$c^2=(\gamma-1)\left[H_0-\frac{1}{2}|\nabla\varphi|^2\right] \quad (3)$$

In this calculation, M represents the Mach number, c denotes the speed of sound, and φ is the velocity potential. Besides, H_0 means the stagnation enthalpy. Then, a small disturbance approximation can be done. It applies to tiny obstructions like delicate airfoils. Only 2D is covered in the conversation. The obstacle's small size means that it has a minimal impact on the flow. It can be considered that the perturbation to uniform flow at speed of magnitude U_∞ in the x direction. The potential can be denoted as:

$$\varphi=U_\infty(x+\emptyset) \quad (4)$$

The formulas are used to recover velocities from the potential while:

$$u=U_\infty(1+\emptyset_x) \quad (5)$$

$$v=U_\infty\emptyset_y \quad (6)$$

Here, u denotes the velocity in the x direction, and v denotes the velocity in the y direction. The equation then becomes:

$$(1-M_x^2)\varphi_{xx}+\varphi_{yy}=0 \quad (7)$$

The equation (7) can be simplified further to:

$$(1-M_\infty^2)\varphi_{xx}+\varphi_{yy}=0 \quad (8)$$

Here, M_∞ can be approximately considered as the Mach number of inflow. Equation (8) is the SDE, which needs to be discretized in the next step.

3. Discretization and Difference Equation

To realize simulation, discretization of SDE is inevitable. The problem must be separated into two parts with distinguished conditions.

When M<1, it is subsonic flow, and the SDE can be discretized as:

$$(1-M_\infty^2)\left(\varphi_{i+1,j}-2\varphi_{i,j}+\varphi_{i-1,j}\right)+\left(\varphi_{i,j+1}-2\varphi_{i,j}+\varphi_{i,j-1}\right)=0 \quad (9)$$

Then, equation (9) can be simplified as:

$$\varphi_{i,j} = \frac{(1-M_\infty^2)\varphi_{i+1,j}+(1-M_\infty^2)\varphi_{i-1,j}+\varphi_{i,j+1}+\varphi_{i,j-1}}{2(2-M_\infty^2)} \tag{10}$$

Equation (10) is the expression for $\varphi_{i,j}$ in subsonic case. When M>1, it is the supersonic case, and the SDE can be discretized as:

$$(1-M_\infty^2)(\varphi_{i,j} - 2\varphi_{i-1,j} + \varphi_{i-2,j}) + (\varphi_{i,j+1} - 2\varphi_{i,j} + \varphi_{i,j-1}) = 0 \tag{11}$$

Then, equation (11) can be simplified further to:

$$\varphi_{i,j} = \frac{2(1-M_\infty^2)\varphi_{i-1,j}-(1-M_\infty^2)\varphi_{i-2,j}-\varphi_{i,j+1}-\varphi_{i,j-1}}{(1-M_\infty^2)-2} \tag{12}$$

Equation (12) is the expression for $\varphi_{i,j}$ in supersonic case.

4. Boundary Condition

4.1. Wall Side Boundary Condition

This paper uses the function f(x) to describe the airfoil's shape. It can be described as:

$$f(x) = \beta \sin[\pi(x-1)] \tag{13}$$

This function is defined on $x \in [1,2]$. When x is out of this region, f(x) equals zero. The β should be less than 0.1 to ensure it is a small obstacle. For the boundary condition on the wall side:

$$v = (U_\infty + u)\dot{f}(x) \tag{14}$$

This can be further simplified via an approximation as:

$$v = U_\infty \dot{f}(x) \tag{15}$$

Here, $\dot{f}(x)$ represents the derivative of the shape function of the airfoil. The physical meaning of the wall-side boundary condition is that the velocity at this boundary should be parallel to the surface of the airfoil.

4.2. Neumann Boundary Condition

Here, an intermediate variable $g_{i,j}$ is introduced. For the Neumann boundary in the case of 2D, an expression can be described as:

$$g_{i,j} = \frac{\partial \varphi}{\partial h} \tag{16}$$

In equation (16), the variable h denotes the distance between two discrete points in the y direction. Thus, this equation can be further expressed as:

$$g_{i,j} = \frac{1}{2h}(\varphi_{i,j+1} - \varphi_{i,j-1}) \tag{17}$$

Through equation (17), an expression for $\varphi_{i,j-1}$ can be written as:

$$\varphi_{i,j-1} = \varphi_{i,j+1} - 2hg_{ij} \tag{18}$$

Normally, the range of i and j starts from zero. When j=0, it is found that $\varphi_{i,-1}$ appears. It is a point which is called a ghost point. And Neumann boundary condition is just applied to estimate the value of this point. Then, $g_{i,j}$ should be denoted in another way. Comparing equation (6) and equation (15), it is found that they are both the expression for v. Then, obviously, $\emptyset_y$ equals to $\dot{f}(x)$. It can also be regarded as:

$$\varphi_y = \dot{f}(x) \tag{19}$$

Considering equation (17), the right part of this equation is just the expression for φ_y. Then, it can be concluded that:

$$g_{i,j} = \mathrm{f}(\dot{\mathrm{x}}) \tag{20}$$

Finally, the expression for $\varphi_{i,j-1}$ is:

$$\varphi_{i,j-1} = \varphi_{i,j+1} - 2h\mathrm{f}(\dot{\mathrm{x}}) \tag{21}$$

5. Iteration

The basic idea for iteration is using a new variable d_p. It is in increments for $\varphi_{i,j}$. When using $d_p + \varphi_{i,j}$ to substitute for $\varphi_{i,j}$ in equation (10) and (12), the expression for d_p can be obtained. When M<1:

$$d_p = -\varphi_{i,j} + \frac{(1-M_\infty^2)\varphi_{i+1,j}+(1-M_\infty^2)\varphi_{i-1,j}+\varphi_{i,j+1}+\varphi_{i,j-1}}{2(2-M_\infty^2)} \tag{22}$$

When M>1:

$$d_p = -\varphi_{i,j} + \frac{2(1-M_\infty^2)\varphi_{i-1,j}-(1-M_\infty^2)\varphi_{i-2,j}-\varphi_{i,j+1}-\varphi_{i,j-1}}{(1-M_\infty^2)-2} \tag{23}$$

Then, by using d_p, the iteration can be realized.

6. Result and Discussion

In the subsonic case, the author chooses two different values for M_∞. And β equals 0.1 in each case. Fig. 1 shows the streamline and Mach number contour when $M_\infty = 0.4$. Fig. 2 shows the error with the times of iteration when $M_\infty = 0.4$.

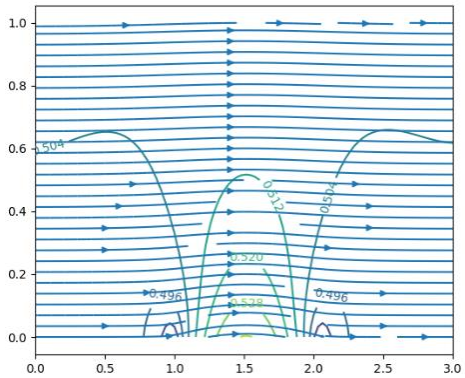


Figure 1. Streamline and Mach number contour (Photo/Picture credit: Original)

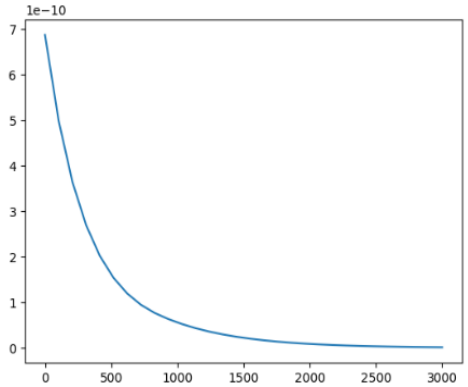


Figure 2. Error concerning iteration (Photo/Picture credit: Original)

Fig. 3 shows the streamline and Mach number contour when $M_\infty = 0.8$. Fig.4 shows the error with iteration when $M_\infty = 0.8$.

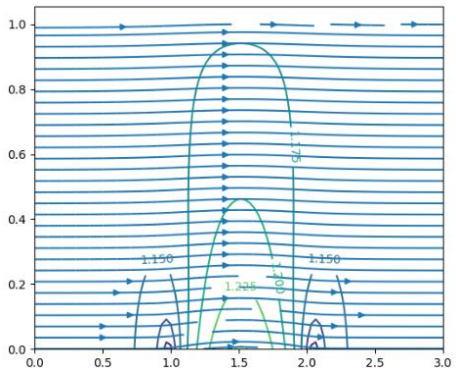


Figure 3. Streamline and Mach number contour (Photo/Picture credit: Original)

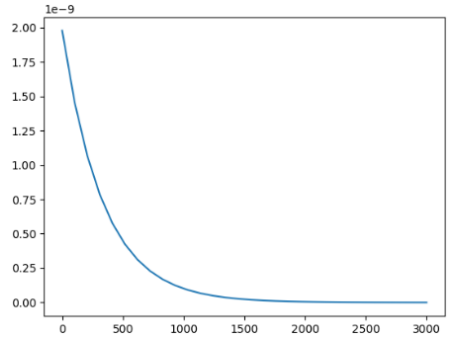


Figure 4. Error concerning iteration (Photo/Picture credit: Original)

In the supersonic case, the author also chooses two different values for M_∞. And β equals 0.1 in each case. Fig. 5 shows the streamline and Mach number contour when $M_\infty = 1.3$. Fig. 6 shows the error with iteration when $M_\infty = 1.3$.

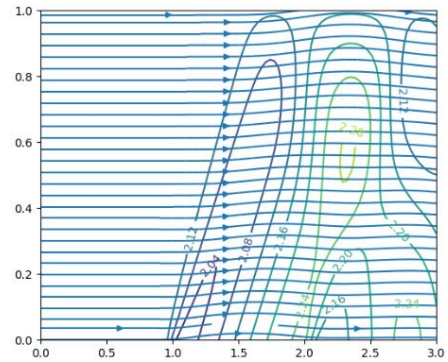


Figure 5. Streamline and Mach number contour (Photo/Picture credit: Original)

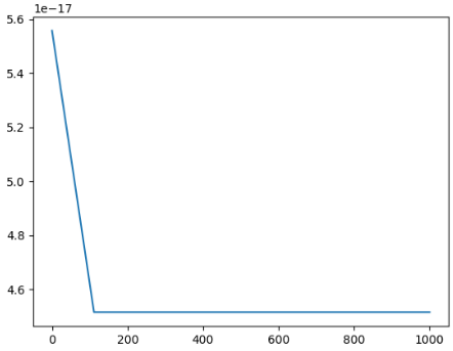


Figure 6. Error concerning iteration (Photo/Picture credit: Original)

Fig. 7 shows the streamline and Mach number contour when $M_\infty = 2$. Fig. 8 shows the error with iteration when $M_\infty = 2$.

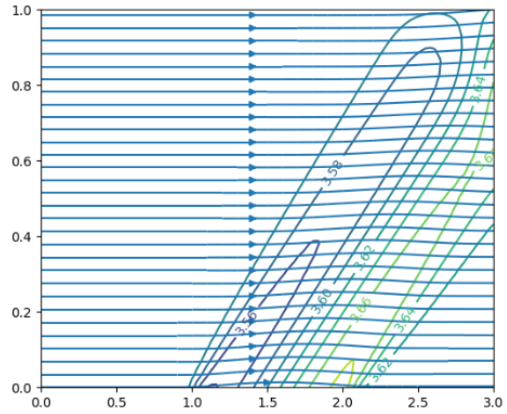


Figure 7. Streamline and Mach number contour (Photo/Picture credit: Original)

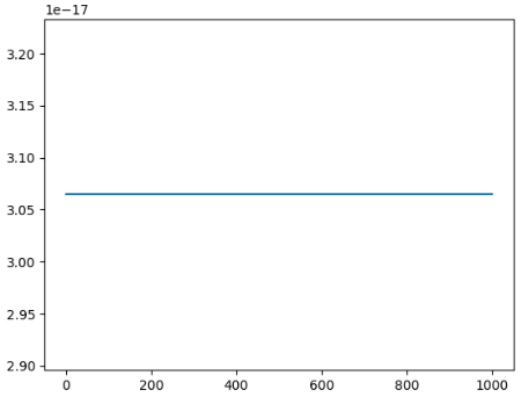


Figure 8. Error concerning iteration (Photo/Picture credit: Original)

It is shown that in each case, the error tends to decrease. In the subsonic case, the error can be less than 10^-9. In a supersonic case, the error can even reach 10^-17. Thus, the result of each case can be regarded as valid and effective. However, equation (8) is considered to be used when the Mach number is not close to unity [9]. Thus, further research should be done here to determine what happens. Furthermore, it is also pointed out that when it comes to real configurations, the full potential equation rather than the SDE is more often used [10].

7. Conclusion

This paper uses the SDE to study the flow past the airfoil. The streamline and Mach number contour are obtained via Python and discrete equations in different cases. Considering that the error is rather small, the result is reliable. Nevertheless, some problems in this process need to be solved. First, the entire potential equation can be reduced to another form of SDE when the Mach number is close to unity. Thus, further work should be done here to satisfy this condition, making the result more accurate. Secondly, SDE is undeniably limited because it can only be applied to small obstacles. Therefore, the airfoil cannot be too thick. That is why the β is limited to less than 0.1. However, the objects may not be small in real applications.

In comparison, the full potential equation does not have this limitation, so it is more widely used. Last but not least, the analysis in this publication only considers the upper surface boundary conditions and half of the airfoil, hence the results are not full. The whole airfoil must be considered to obtain a more precise solution for the complete stream field. Generally speaking, the research in this paper is valid and effective, but it is incomplete. It only partially solves the flow past the airfoil. And further study should be done in this realm.

References

[1] Cheng Fanrui, Xu Yang and Zhou Wenbo. Nozzle flow simulation by small disturbance approximation and Euler method. Journal of Physics: Conference Series, 2021, 2012 (1): 012048.

[2] Sun Zhiyi, Wu Xiaorong. Discussion and application of CFD numerical simulation technology. Water Conservancy Science and Technology and Economy, 2008, 14: 126-128.

[3] Li Zhiyin, Xiong Xiaohui, Wu Jiaming. The brief introduction of normal numerical method of CFD. Guangdong Shipbuilding, 2004, 3: 5-8.

[4] Ambarita H., Siregar M. R. and Kishinami K., et al. Application of CFD in Indonesian research: a review. Journal of Physics: Conference Series, 2018, 1005 (1): 012014.

[5] Qian Xuehui. The impact of inlet Mach number and domain shape on subsonic nozzle flow via small disturbance equation. Journal of Physics: Conference Series, 2023, 2441 (1): 012002.

[6] Kreiselmaier Erich and Laschka Boris. Small disturbance Euler equation: efficient and accurate tool for unsteady load prediction. Journal of Aircraft, 2000, 37 (5): 770-778.

[7] Holst Terry L and Ballhaus William F. Fast, conservative schemes for the full potential equation applied to transonic flows. AIAA Journal, 1979, 17 (2): 145-152.

[8] Anderson John David. Modern compressible flow: with historical perspective. New York: McGraw-Hill, 1990, 12.

[9] Kwon Jae Ryong and Vepa Ranjan. Transonic small disturbance unsteady potential flow over very high aspect ratio wings. The Aeronautical Journal, 2022, 126 (1305): 1834-1852.

[10] Hafez M.U.R.M.A.N., South J. and Murman E. Artificial compressibility methods for numerical solutions of transonic full potential equation. AIAA Journal, 1979, 17 (8): 838-844.