DOI: 10.54254/2755-2721/12/20230331

The study of the effect of fluid field above an airfoil under 2D small disturbance equation method

Weijie Li

Department of Mechanical Engineering, The Ohio State University, Columbus, 43210, United States

li.10721@osu.edu

Abstract. This review article presents a simulation based on Python for studying fluid flow over a small disturbance (airfoil) in subsonic and supersonic conditions. The simulation revolves around the small disturbance equation method, focusing on comparative analysis under supersonic conditions. Two schemes are applied in different conditions: the central difference scheme for subsonic flows and the upwind scheme for supersonic flows. Successive-over relaxation method is also utilized for converging iterative processes. The study focuses on one influence factor in aerodynamics, the Mach number, and investigates its impact on the shark angle of an airfoil. The simulation results demonstrate that the Mach number is inversely proportional to the shark angle of an airfoil, meaning that higher Mach numbers lead to smaller shark angles of the airfoil. Therefore, the Mach number can be considered a critical parameter in designing aerodynamic applications. This finding is consistent with prior research, indicating that the designed method is reliable.

Keywords: Small disturbance equation, central difference scheme, upwind scheme, Python.

1. Introduction

Fluid dynamics has been a thriving area of study for many years, and it has significant applications in various industries, including aerospace, engineering, and environmental sciences. One of the crucial aspects of fluid dynamics is the study of the flow of fluids over surfaces, such as airfoils. The flow over an airfoil is a complex phenomenon influenced by various factors such as the airfoil's shape, speed, and the fluid's properties. Computational fluid dynamics (CFD) can mathematically predict this effect by solving the Navier-Stokes equations. Studying fluid flow over an airfoil under supersonic conditions has important applications in some fields, such as aircraft and wind turbine design. Therefore, a more comprehensive understanding of fluid dynamics in this field can lead to the development of more efficient and safer realistic applications in life.

The small disturbance equation (SDE) method has proven useful in understanding fluid flow behavior [1]. Compared with another commonly used model, the full potential equations (FPE) method, SDE is a relatively simplified version. It is mostly applied in passing through thin obstructions condition. Kuihao Huang, in his article, compared the SDE method with the Euler method and found that the former can obtain more accurate experimental results in simulating nozzle flow because the 1-D Euler method in the experiment is insufficient to perfectly handle 2-D fluid

DOI: 10.54254/2755-2721/12/20230331

simulation [2]. The research mentioned above shows the feasibility and reliability of further in-depth development of the SDE method.

This article revolves around studying the effect of the fluid field above an airfoil under a 2D small disturbance equation method and potentially trying to help develop a more accurate and efficient model for studying fluid dynamics. It will begin with a discussion of the applications of the small disturbance equation method in fluid dynamics and derivations of the SDE method. Then designing a 2-D SDE model as the foundation to simulate the subsonic and supersonic fluid field around a disturbance through Python programming. Finally, it will compare the effect results of fluid flow over an airfoil by changing the velocity of the flow.

2. Realistic applications of CFD models

With advancements in computing power and numerical algorithms, computational fluid dynamics (CFD) has become an increasingly popular tool in various industries for simulating fluid flows and analyzing their behavior. Almost all modern fluid mechanical problems can rely on it to find a better solution because it enables engineers to gain insight into complex fluid dynamics phenomena that were once too difficult or expensive to study through physical experiments alone. Scott and Richardson, in their article, shared that food process engineers can utilize CFD to analyze thermal process convection patterns in containers [3].

The example above can be considered an application of CFD, while the small disturbance equation method, as one sub-branch of Navier-Stokes equations, also has applications that can be implemented in real life. Shan listed a few specific analyses of SDE, and flutter analysis is one of the examples [4]. The author compared the experimental flutter speed data collected from the SDE method with the actual velocity data to find the vibration equation near the wing [4]. This analysis can help improve the safety hazards caused by vibrations generated by various aerodynamic forces during aircraft flight. To improve the accuracy of the unsteady aerodynamics model on aircraft, Rozov and his colleagues introduced the effects of aerodynamic engines into the SDE method [5]. The research focused on the aircraft flutter behavior caused by engines mounted under the wing and successfully reproduced the flow effects in the condition of realistic engines [5]. These cases all reflect the effectiveness and feasibility of continuing the design of the SDE method.

3. Method of SDE derived from the full potential equation

Before knowing how the small disturbance equation method works, it is necessary to understand the full potential equation method. The full potential equation (FPE) is a nonlinear partial differential equation (a simplified form of the Navier-Stokes equations) that describes inviscid, incompressible fluid flow. It assumes that the fluid is irrotational, meaning that the vorticity of the flow is zero. The small disturbance equation (SDE) is a linearized version of the full potential equation, often used in subsonic and supersonic flow analysis. The condition here is restricted under two-dimension.

The full potential equation in 2-D is given by:

$$\nabla^2 \Phi = 0 \quad (1)$$

where Φ is the velocity potential, and ∇^2 is the Laplacian operator.

Small disturbance theory assumes that the flow perturbation is smaller than the freestream flow [6]. Thus, the velocity potential can be expressed as:

$$\Phi(x, y) = \Phi_\infty + \varphi(x, y) \quad (2)$$

where Φ_∞ is the freestream velocity potential, and $\varphi(x, y)$ is the small disturbance potential.

Substitute equation (2) into (1) to linearize the full potential equation:

$$\nabla^2(\Phi_\infty + \varphi) = 0 \quad (3)$$

Because Φ_∞ is constant, its Laplacian is zero. Then the linearized FP equation becomes:

$$\nabla^2 \varphi \approx 0 \quad (4)$$

For simplicity, here, set a uniform freestream to flow along the x-axis with speed U_∞:

$$\Phi_\infty = U_{\infty x} \quad (5)$$

$$u_\infty = \frac{\partial \Phi_\infty}{\partial x} = U_\infty \tag{6}$$

$$v_\infty = \frac{\partial \Phi_\infty}{\partial y} = 0 \tag{7}$$

The small disturbance flow velocities can be expressed as:

$$u = \frac{\partial \varphi}{\partial x} \tag{8}$$

$$v = \frac{\partial \varphi}{\partial y} \tag{9}$$

For an incompressible, irrotational flow, the velocity potential satisfies:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = \frac{\partial_\varphi^2}{\partial x^2} + \frac{\partial_\varphi^2}{\partial y^2} = 0 \tag{10}$$

Express the equation above in terms of the Mach number M_∞:

$$(1 - M_\infty^2) \cdot \frac{\partial_\varphi^2}{\partial x^2} + \frac{\partial_\varphi^2}{\partial y^2} = 0 \tag{11}$$

This is the Small Disturbance Equation (SDE) for an incompressible, irrotational flow with a uniform freestream Mach number M_∞, and it can finally be written in the form:

$$\left(1 - M_\infty^2\right) \cdot \varphi_{xx} + \varphi_{yy} = 0 \tag{12}$$

4. Setup of simulation

4.1. Boundary condition

For the boundary condition:

$$-\frac{\partial \varphi}{\partial y} = (U_\infty + u)\, f^{'}(x) \approx U_\infty f'(x) \tag{13}$$

The function above is set at the bottom, where U_∞ is the magnitude of flow velocity and $f'(x)$ is the shape of the airfoil [7]. The shape is modeled as a sinusoidal wall in the domain $(1,2)$, while no obstacle exists in the other range $(0,1)$ and $(2,3)$. The uniform inflow in simulation comes from the right to left, passing through the obstacle set at the middle bottom. Figure 1 below shows the shape of the disturbance.

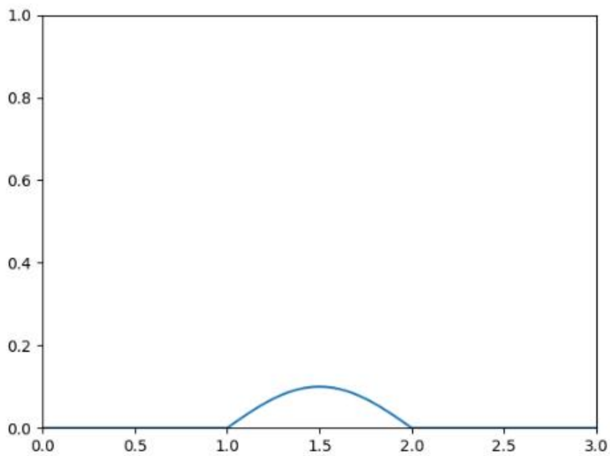


Figure 1. Boundary condition in simulation.

4.2. Two versions of the equation model

There are two different equation models used in simulation:

i. $\left(1 - M_\infty^2\right) \cdot \varphi_{xx} + \varphi_{yy} = 0$ (14)

DOI: 10.54254/2755-2721/12/20230331

ii. $\left(1-M^2\right)\cdot\varphi_{xx}+\varphi_{yy}=0$ (15)

M_∞ is a constant value in the first model, while for the second one, M is a function of φ. M here is the Mach number mentioned above, the flow velocity ratio to the local sound speed. Both model versions will be conducted to simulate subsonic and supersonic conditions, but the scheme applied to the subsonic and supersonic is not identical. The central difference scheme is applied to the subsonic condition here, while the upwind scheme is more appropriate for the supersonic condition.

A central difference scheme is a numerical approximation technique used to estimate a function's first or second derivative at a given point using the values of the function at neighboring points. It is called "central" because it uses the values of the function at points symmetrically located around the point of interest. The basic form is:

$$f'(x)=\frac{f(x+h)-f(x-h)}{2h} \tag{16}$$

where h is the spacing between the neighboring points. This approximation has a second-order accuracy, meaning that the error in the approximation decreases quadratically with decreasing h.

The upwind scheme is usually used to solve advection problems, where a velocity field transports the solution. It uses the value of the function at the point upstream of the flow direction to approximate the value at the next time step. The base form is:

$$f_i^{n+1}=f_i^n-u\left(\frac{f_i^n-f_{i-1}^n}{\Delta x}\right) \tag{17}$$

The upwind scheme ensures that the solution propagates in the correct direction and does not introduce numerical oscillations or instability that may occur when using central difference schemes for advection problems. The upwind scheme is now widely applied to second derivatives contributing to φ_{xx} when velocity is over sound speed [8].

4.3. Coefficients of discrete system

Successive-over relaxation (SOR) method will be used here to solve the Laplace equation for the potential φ. This function iteratively updates the potential at each grid point until the error falls below a certain tolerance level.

For the subsonic condition:

$$-C_0\cdot\varphi_{i,j}+\left(A_{x1}\cdot\varphi_{i+1,j}+A_{x2}\cdot\varphi_{i-1,j}+B_{y1}\cdot\varphi_{i,j+1}+B_{y2}\cdot\varphi_{i,j-1}\right) \tag{18}$$

For the supersonic condition:

$$-C_0\cdot\varphi_{i,j}+\left[B_{y1}\cdot\varphi_{i,j+1}+B_{y2}\cdot\varphi_{i,j-1}-\left(A_{x1}+A_{x2}\right)\cdot\varphi_{i-1,j}+A_{x2}\cdot\varphi_{i-2,j}\right] \tag{19}$$

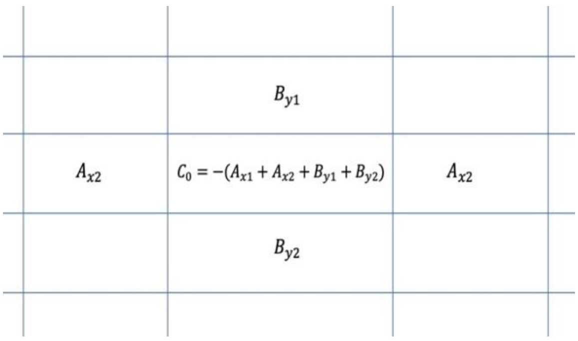


Figure 2. Coefficients of subsonic flow.

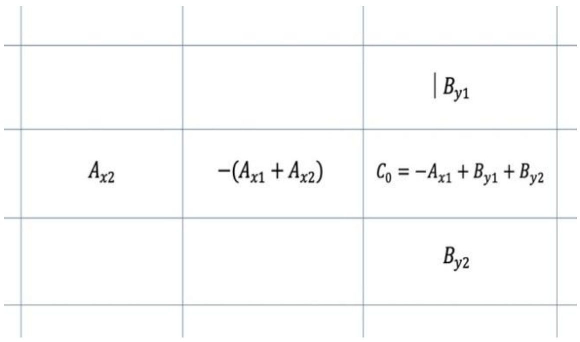


Figure 3. Coefficients of supersonic flow.

Figures 2 and 3 present the coordinate system's corresponding positions of subsonic and supersonic flow coefficients.

In model i, $\left(1-M_\infty^2\right)\cdot\varphi_{xx}+\varphi_{yy}=0$:

$$A_{x1}=A_{x2}=1-M_\infty^2\ ,\ B_{y1}=B_{y2}=1\ ,\ C_0=A_{x1}+A_{x2}+B_{y1}+B_{y2} \tag{20}$$

In model ii, $\left(1-M_\infty^2\right)\cdot\varphi_{xx}+\varphi_{yy}=0$:

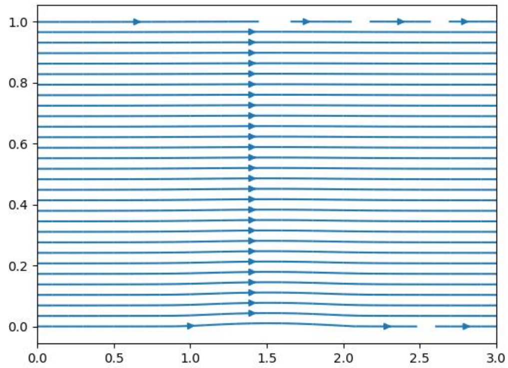
$$A_{x1}=A_{x2}=(1-\mu)\cdot k\ or\ \mu\cdot k\ ,\ B_{y1}=B_{y2}=1\ ,\ C_0=-A_{x1}+B_{y1}+B_{y2} \tag{21}$$

$$k = 1 - M_{\infty}^{2} - (\gamma + 1) \cdot M_{\infty} \cdot \frac{\varphi_{i+1,j} - \varphi_{i-1,j}}{2\Delta h} \tag{22}$$

where $\mu = 0$ for the subsonic condition, $\mu = 1$ for the supersonic condition, and take $\gamma = 1.4$ (the ratio of specific heats) [9,10].

5. Results and comparisons

For model i (2 different flow conditions with fixed $\beta = 0.1$):



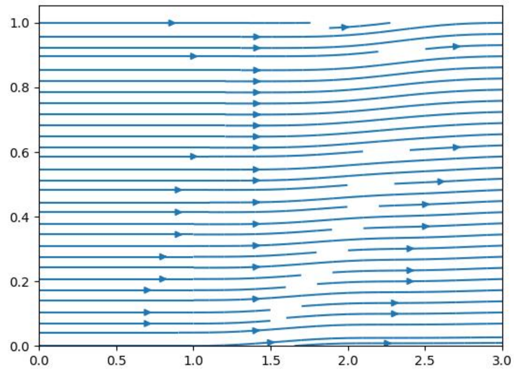
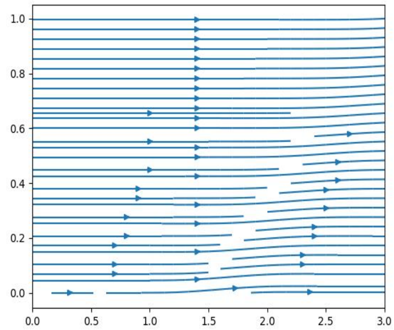
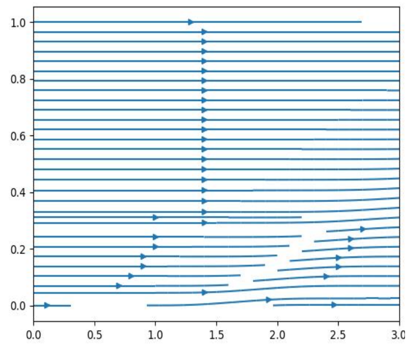


Figure 4. Subsonic flow with $M_{\infty} = 0.3$ **Figure 5. Supersonic flow with $\mathbf{M}_{\infty} = \mathbf{1.3}$**

For model ii (a supersonic condition with fixed $\beta = 0.1$):





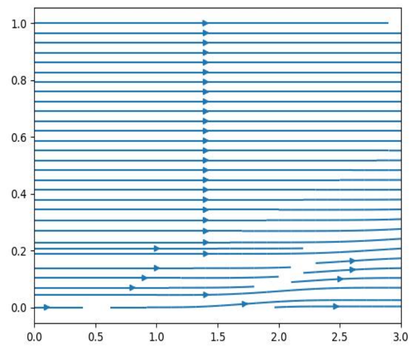


Figure 6. Mach number = 1.3 **Figure 7.** Mach number = 3.0 **Figure 8.** Mach number = 4.7

As shown above, the velocity streamlines move horizontally to the right in Figures 4 and 5. They represent subsonic flow and supersonic flow conditions separately in model *i*. However, they start changing the original moving way when they pass through the middle. This is due to the set of small disturbances within the range 1 to 2. β is the amplitude of the sinusoidal function, and the value of β will impact the scale of streamlined fluctuations. As shown in Figures 6-8, the comparison is under fixed β=0.1, changing *M* values from 1.3 to 4.7. They are under the supersonic flow condition of model *ii*. It stimulates the flow effects above the airfoil when the aircraft is accelerating. As the *M* gradually increases, the shark angle becomes smaller.

6. Conclusions

In this simulation, the streamlines visually represent the flow pattern. It can be used to analyze the flow behavior, especially supersonic flow behavior, around the obstacle based on the changing flow velocity and Mach number value. The results indicate that the shark angle of the airfoil becomes smaller with the increased Mach number. In other words, the relationship between the Mach number and shark angle is inversely proportional. This suggests that higher Mach numbers can lead to more streamlined flow patterns and potentially improved aerodynamic performance.

However, it's worth noting that the specific results of fluid flow over an airfoil in a realistic world are influenced by a range of factors beyond just velocity and Mach number, such as the airfoil's shape and angle of attack. Therefore, these possible influence factors need to be combined with the model so

DOI: 10.54254/2755-2721/12/20230331

that future simulations can present a more accurate prediction. Further research is needed to fully understand the complex interactions at fluid flow over airfoils. Meanwhile, selecting an appropriate scheme is essential because it significantly influences the results of analyzing and simulating incompressible flow over airfoils in various flow regimes.

References

[1] Anderson D, Tannehill J, Pletcher R, Munipalli R and Shankar V 1997 Computational Fluid Mechanics and Heat Transfer. Taylor & Francis
[2] Huang K 2021 J. Phys.: Conf. Ser. 2012 012087
[3] Scott G and Richardson P 1997 Trends in Food Science & Technology 8 119
[4] Shan S, Chen W and Chen Z 2021 J. Phys.: Conf. Ser. 2012 012025
[5] Rozov V, Stuhlpfarrer M, Osma M and Breitsamter C 2020 Journal of Fluid and Structures 96 103045
[6] Scholle M, Marner F and Gaskell P 2020 Water 12 1241
[7] Liu Q 2023 Highlights in Science, Engineering and Technology 37 291
[8] Jemeson A 1990 Full-Potential, Euler, and Navier-Stokes Scheme. The Amercian Institute of Aeronautics and Astronautics
[9] Opstal T, Hulshoff S and Verhoosel C 2011 the Open Aerospace Engineering Journal 4 1
[10] Osher S, Hafez M and Whitlow W 1985 Mathematics of Computation 44 1