ISSN 2959-6157

Optimization and Analysis of Flow Field Simulations Across Different Mach Regimes Using Discrete Numerical Methods

Zhuoxuan Li

Institute of Aeronautics and Astronautics, Zhejiang University, Hangzhou, China
3210100639@zju.edu.cn

Abstract:

This article explores the development of flow field models in steady-state environments utilizing Euler equations and potential flow equations, with verification processes conducted using Python. The models demonstrate stability and high accuracy in low-velocity scenarios, capturing essential dynamics effectively. However, as the conditions transition to supersonic speeds, the models begin to exhibit increased errors. This discrepancy highlights the challenges faced in simulating high-speed aerodynamics accurately. The research underscores the importance of improving model fidelity in diverse Mach regimes, particularly in supersonic conditions where traditional methods struggle. Future research directions identified include the development of unsteady flow field models, which are crucial for dynamic analyses, optimization of grid structures for three-dimensional complex fields to improve computational efficiency, and the creation of extensive model libraries. These advancements aim to enhance the accuracy, reliability, and practicality of flow field simulations, extending their applicability in both academic studies and industry applications, particularly in aerospace engineering where precise flow modeling is critical.

Keywords: CFD; Potential Equation; Stable Flow.

1. Introduction

Computational Fluid Dynamics (CFD) has evolved significantly since its inception, undergoing extensive trial and error, refinement, and validations. The discipline made early strides in 1973 when the CFD group at Imperial College London began a project aimed at predicting simple shear flows, free and confined jets using the stream function-vorticity solution algorithm. This initial work led to the development of the SIMPLE semi-implicit solution algorithm that simplified the Navier-Stokes momentum equations using velocity and pressure as primary variables [1-4]. Over the years, CFD technology has expanded to include complex, high-dimensional problem-solving capabilities, along with applications in diverse fields such as hemodynamics and wind tunnel system design [5-7].

Research Problem: Despite advancements, CFD technology continues to confront significant challenges that impede its broader application and effectiveness. These include computational limitations on hosts, inadequate computational modeling of discretized flow fields, and difficulties in managing irregular boundaries and wall conditions with simple orthogonal grids. Other issues encompass slow convergence, numerical diffusion, and challenges related to performing complex three-dimensional geometric operations and time-related computations [8]. These challenges highlight the need for further development and refinement of CFD methodologies to enhance their accuracy and applicability across various scientific and engineering domains.

This Paper's Contribution: This article embarks on an exploration of discrete flow field model construction using Euler's equations and the potential flow equations, demonstrating the construction and validation of flow field models under various sound speeds in steady environments using Python. The study aims to establish the basic consistency and feasibility of the models employed, and also verifies the correctness of these basic models through MATLAB software employing the vorticity stream function method [1-4]. By addressing the complexities of model construction and the intricacies of computational fluid dynamics, this work contributes to the ongoing enhancement of CFD capabilities, paving the way for more accurate and practical applications in fields that require precise fluid dynamics simulations.

2. Methodology

Navier-Stokes equations (N-S equations) derived from the Euler equations [9].

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=\rho f_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \quad (1)$$

Considering the stability of the flow field and constant temperature, the equation is simplified to a simple continuity equation with boundary conditions.

$$div\left(\rho\nabla\boldsymbol{\varphi}\right)=0(boundarycondition v_n=\frac{\partial\varphi}{\partial n}=0) \quad (2)$$

There are many methods for grid selection and optimization. In addition to the basic vertical coordinate grid division, the grid can be specifically divided according to the geometric shape of the boundary conditions of the specific research flow field or the shape of the test model in the flow field, such as the commonly used rectangular grid, quadrilateral grid, triangular grid, etc [10]. Considering that the basic characteristics of the simulated flow field model do not require a large amount of calculation for irregular model grid division, the discrete grid division adopts the traditional coordinate grid method, which can maximize the speed and stability of operation, and also provide a theoretical basis for unit model research such as flexible pipe flow field model construction.

In order to consider the different requirements of discretized difference schemes in subsonic and supersonic environments, as well as the continuity requirements in transonic flow fields, this article adopts a combination of forward difference and central difference methods to construct partial differential equations, and gives multiple feasible transition coefficients to meet the needs of transonic flow fields, and provides a variety of different coefficient μ value methods, comparative verification results.

$$K\varphi_{xx}+\varphi_{yy}=0$$

$$\mu_{i,j}K_{i-1,j}\left(\phi_{i,j}-2\phi_{i-1,j}+\phi_{i-2,j}\right)+\left(\phi_{i,j+1}-2\phi_{i,j}+\phi_{i,j-1}\right)+\left(1-\mu_{i,j}\right)K_{i,j}\left(\phi_{i+1,j}-2\phi_{i,j}+\phi_{i-1,j}\right)=0 \quad (3)$$

$$K_{i,j}=1-M^2-\left(1+\gamma\right)M^2\frac{\left(\phi_{i+1,j}-\phi_{i-1,j}\right)}{2}$$

Furthermore, considering the total potential energy equation for steady flow.

$$\frac{\rho}{\rho_0}=\left(1-\frac{|\nabla\varphi|^2}{2H_0}\right)^{1/(\gamma-1)} \quad (4)$$

$$c^2=\left(\gamma-1\right)\left[H_0-\frac{1}{2}|\nabla\varphi|^2\right] \quad (5)$$

$$\left(1-M_x^2\right)\varphi_{xx}+\left(1-M_y^2\right)\varphi_{yy}-2M_xM_y\varphi_{xy}=0 \quad (6)$$

$$M_x=\frac{\varphi_x}{c},M_y=\frac{\varphi_y}{c},M^2=\frac{|\nabla\varphi|^2}{c^2} \quad (7)$$

The flow field model and corresponding parameters under the corresponding state can be obtained. Among them, H_0 is the stagnation enthalpy, and c is the speed of sound. For cases where $Ma>1$, the original partial differential equation

$$\left(\rho\varphi_x\right)_x+\left(\rho\varphi_y\right)_y=0 \quad (8)$$

Make changes in coefficient

$$\left(\bar{\rho}\varphi_x\right)_x+\left(\bar{\bar{\rho}}\varphi_y\right)_y=0 \quad (9)$$

$$\bar{\rho}=\rho-\mu\rho_x\Delta x,\bar{\bar{\rho}}=\rho-\mu\rho_y\Delta y$$

$$\bar{\rho}_{i+\frac{1}{2},j}=\begin{cases}\rho_{i+\frac{1}{2},j}-\mu_{i,j}\left(\rho_{i+\frac{1}{2},j}-\rho_{i-\frac{1}{2},j}\right) & u_{i+\frac{1}{2},j}>0\\ \rho_{i+\frac{1}{2},j}+\mu_{i+1,j}\left(\rho_{i+\frac{1}{2},j}-\rho_{i+\frac{3}{2},j}\right) & u_{i+\frac{1}{2},j}<0\end{cases} \quad (10)$$

$$\bar{\bar{\rho}}_{i,j+\frac{1}{2}}=\begin{cases}\rho_{i,j+\frac{1}{2}}-\mu_{i,j}\left(\rho_{i,j+\frac{1}{2}}-\rho_{i,j-\frac{1}{2}}\right) & u_{i+\frac{1}{2},j}>0\\ \rho_{i,j+\frac{1}{2}}+\mu_{i,j+1}\left(\rho_{i,j+\frac{1}{2}}-\rho_{i,j+\frac{3}{2}}\right) & u_{i+\frac{1}{2},j}<0\end{cases}$$

3. Results and Analysis

3.1 Poisson Equation

Set the B.C. as follows.

$$f\left(x\right)=\beta cos\left(\pi\left(x-1\right)\right),1<x<2 \quad (11)$$

As for coefficient of the upwind and center differential equation μ, there are 3 definitions:

$$\mu_1=\begin{cases}0 & subsoniccondition(Ma<1)\\ 1 & supersoniccondition(Ma>1)\end{cases}$$

$$\mu_2=\begin{cases}0 & Ma\leqslant 0.8\\ 2Ma-1.6 & 0.8<Ma<1.3\\ 1 & Ma\geqslant 1.3\end{cases} \quad (12)$$

$$\mu_3=1-\frac{M_c^2}{Ma^2}$$

The figure 1 is after 3000 steps.

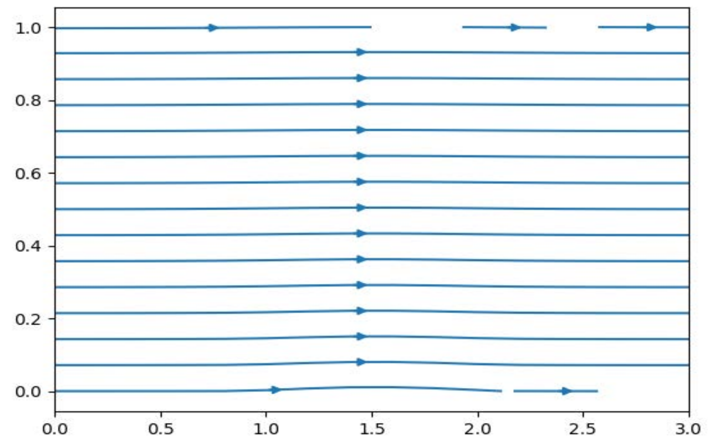

Fig. 1 Simple stable flow with $Ma = 0.8$ and $\mu = \mu_2$ (the error comes to 9.078537206574691e-10) (Photo credit: Original).

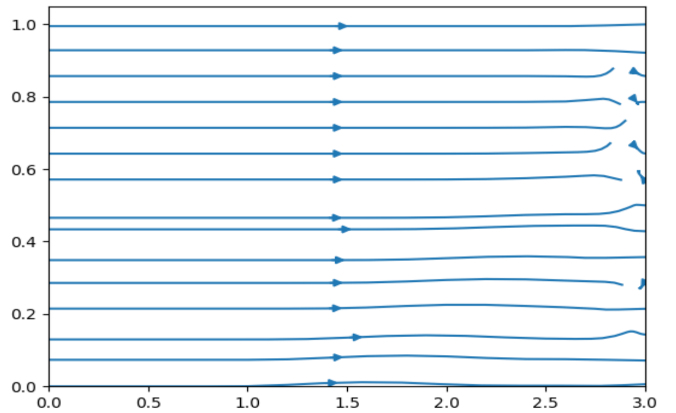

Fig. 2 Simple stable flow with $Ma = 2.5$ and $\mu = \mu_2$ (the error comes to 0.004923203387689582) (Photo credit: Original).

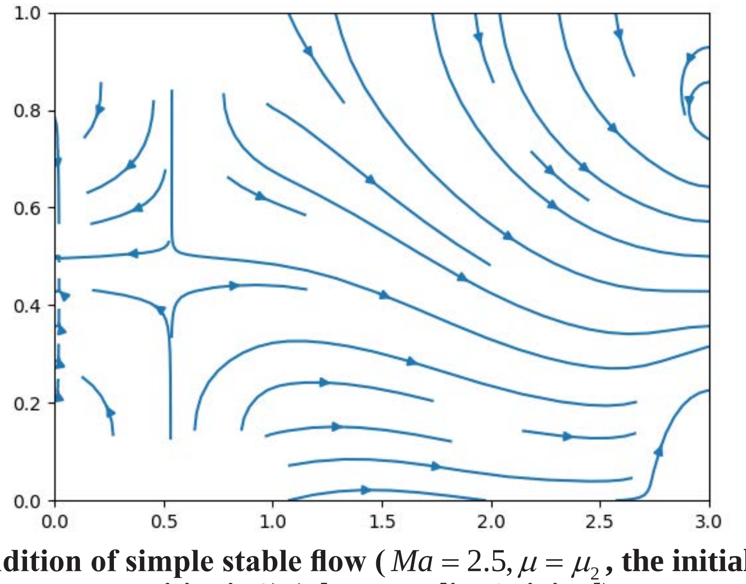

Fig. 3 Special condition of simple stable flow ($Ma = 2.5, \mu = \mu_2$, the initial potential of two cavities is 0) (Photo credit: Original).

As show in the fig.1 to the fig. 3. It is clear that under low-velocity flow conditions, the flow field tends to be stable; when the Mach number increases to supersonic, although the flow pattern in the flow field is basically consistent, the stability of the flow field significantly decreases and the computational error rapidly increases.

3.2 Full Potential Equation

Set similar conditions and analyze the flow field using the total potential function, taking subsonic speed as an example. As show in the fig. 4.

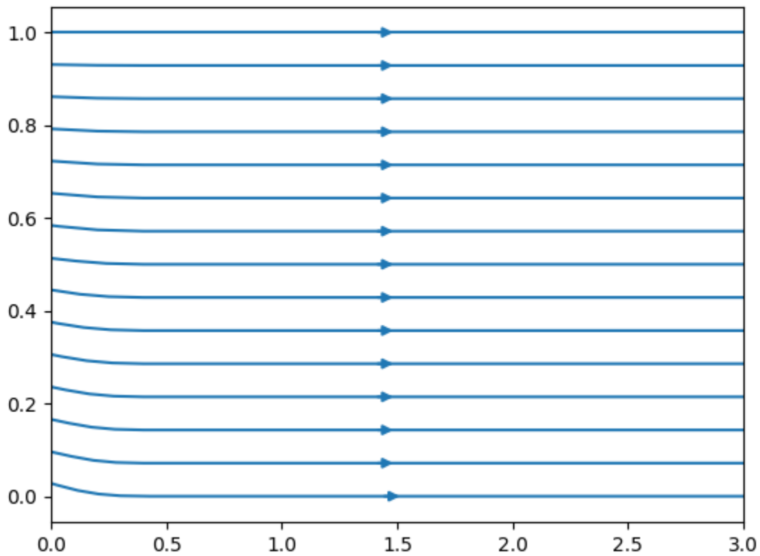

Fig.4 Stable flow by full potential equation ($Ma = 0.6, \mu = \mu_3$) (the error comes to 4.741577501003272519e-17) (Photo credit: Original).

The results obtained are similar to the experimental results mentioned earlier, with the flow field being more stable and the errors smaller. After improving the code, more intuitive visualizations were obtained through plotting with MATLAB software. As show in the fig. 5.

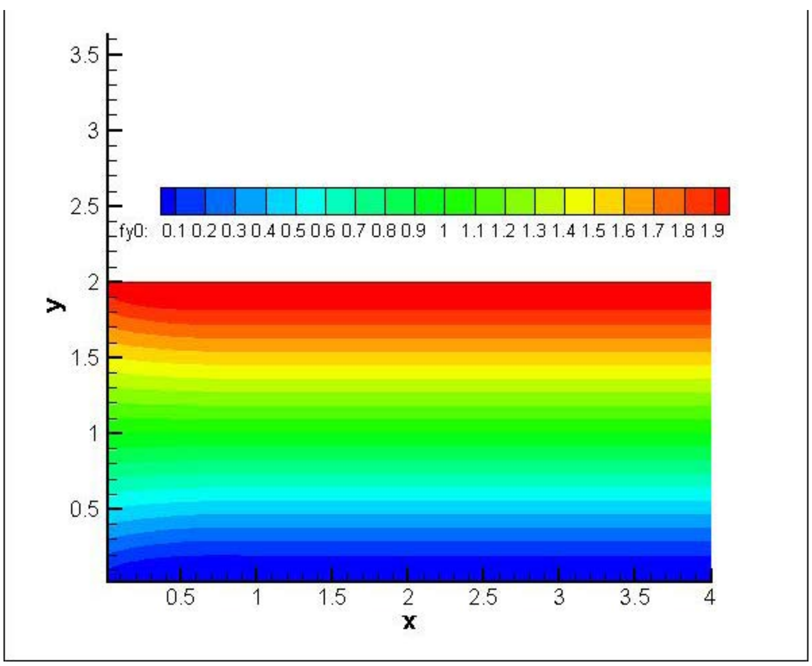

Fig. 5 Colored graph of the flow (have difference in the length of frame) (Photo credit: Original).

4. Future Improvement Ideas

The construction of basic models for simple flow fields not only provides theoretical support for further scientific research but also reveals many potential and feasible directions for further research. These include the construction of basic models for unsteady flow fields or special fluids (such as Newtonian fluids), basic modeling of three-dimensional simple or complex fields, reduction and optimization of errors (such as high-order discrete equations), and exploration of grid construction and models for flow fields with commonly used complex geometric shapes or boundaries. These research directions are highly prospective and practical. Additionally, considering the increasing demand for larger models and the contradiction between insufficient computing power, accuracy, and the lack of advanced software, a model library for basic flow field models is needed in the field of fluid mechanics.

5. Conclusions

This article has successfully developed a foundational model for simple flow fields, utilizing Euler equations and potential flow equations to construct flow field models across various sonic speeds within a steady environment. The research thoroughly examines the computational complexities and trends exhibited by different flow fields, effectively demonstrating the basic consistency and feasibility of the developed models. Findings indicate that in low-velocity conditions, the flow field remains stable, while in supersonic conditions, despite a general consistency in flow patterns, there is a noticeable decrease in stability and an increase in computational errors.

Several promising avenues for future research have been identified to enhance the scope and efficacy of current flow field modeling techniques. There is a critical need to explore unsteady flow fields to understand dynamic fluid behaviors better. Additionally, the modeling of special fluid flow fields could provide insights into less conventional and more complex scenarios. Optimizing grid construction and refining the models for three-dimensional complex fields are essential steps toward improving the accuracy and practicality of the models. These advancements will not only address the current limitations observed in supersonic conditions but also broaden the applicability of computational models in both academic research and practical engineering contexts.

References

[1] Núñez Marc, López Iñigo, Baiges Joan, & Rossi Riccardo. An embedded approach for the solution of the full potential equation with finite elements. Computer Methods in Applied Mechanics and Engineering, 2022, 388: 114244.

[2] Wang Xiaojia, & Christov Ivan C. Reduced models of unidirectional flows in compliant rectangular ducts at finite Reynolds number. Physics of Fluids, 2021, 33(10): 102004.

[3] El-Emam M. A., Zhou L., Shi W., et al. Theories and applications of CFD–DEM coupling approach for granular flow: A review. Archives of Computational Methods in Engineering, 2021, 1-42.

[4] Girfoglio M., Quaini A., & Rozza G. A POD-Galerkin reduced order model for the Navier–Stokes equations in stream function-vorticity formulation. Computers & Fluids, 2022, 244: 105536.

[5] Africa P. C., Fumagalli I., Bucelli M., et al. lifex-cfd: An open-source computational fluid dynamics solver for cardiovascular applications. Computer Physics Communications, 2024, 296: 109039.

[6] Zong Q., & Wu X. A review of computational fluid dynamics (CFD) methodology and analysis on airflow and sand transport over aeolian landforms. Catena, 2024, 241: 108010.

[7] Cheng Xiaoxun, Zhao Ling, Ge Yongjun, et al. Full-Scale/Model Test Comparisons to Validate the Traditional Atmospheric Boundary Layer Wind Tunnel Tests: Literature Review and Personal Perspectives. Applied Sciences, 2024, 14(2): 782.

[8] Khalil Essam E. CFD history and applications. 2021.

[9] Liu Ming, Li Xiaoyan, & Zhao Qian. Exact solutions to Euler equation and Navier–Stokes equation. Zeitschrift für angewandte Mathematik und Physik, 2019, 70: 1-13.

[10] Holst Thomas L. Numerical computation of transonic flow governed by the full-potential equation. 1983, 24-26.