ISSN 2959-6157

Advanced Numerical Techniques for Supersonic Flow Analysis: From Euler Equations to Practical Applications in Aerospace and Turbomachinery Engineering

Yuhao Shan

School of Mechanical Engineering Sichuan University, Sichuan University, Chengdu, China
2021141410204@ stu.scu.edu.cn

Abstract:

This paper explores advanced numerical techniques for analyzing supersonic flow, with a particular focus on the application of Euler equations in aerospace and turbomachinery engineering. Theoretical foundations such as boundary conditions, error evaluation, grid partitioning, and the Successive Over-Relaxation (SOR) method are thoroughly examined. Rigorous numerical simulations are conducted to investigate the final velocity field distribution and convergence characteristics under various conditions. The research underscores the precision and efficiency of these numerical methods, highlighting their practical applications in the aerodynamic design of aircraft and potential use in turbomachinery design. The findings contribute significantly to the advancement of computational fluid dynamics in supersonic flow regimes, providing valuable insights for engineers and researchers engaged in this specialized area. The study not only enhances understanding of complex flow dynamics but also supports the development of more effective engineering solutions in high-speed applications.

Keywords: Supersonic flow; euler equations; numerical methods; SOR method.

1. Introduction

The simulation of supersonic flow is a cornerstone in modern aerospace and turbomachinery engineering, necessitating robust mathematical methods to accurately predict fluid behavior at speeds exceeding Mach 1. The foundational theoretical basis is largely drawn from the Euler equations—a simplified form of the Navier-Stokes equations—which model inviscid compressible flows, as discussed in "Inviscid Fluid Mechanics" by Professors Dong Zengnan and Zhang Zixiong [1]. These equations are critical for capturing phenomena such as shock waves and expansion fans in supersonic flows. However, their non-linear nature presents significant challenges, requiring sophisticated numerical methods for precise solutions [2].

Despite the established theoretical framework provided by the Euler equations, the complexity of supersonic flow dynamics poses numerous challenges. Key issues include the formulation and application of effective boundary conditions, the evaluation of computational errors, and the optimization of numerical schemes for stability and accuracy [3]. Moreover, the nonlinear characteristics of the equations necessitate the development of advanced numerical techniques capable of handling the intricate details of supersonic flow, particularly in scenarios involving shock waves and rapid changes in flow properties.

This Paper's Contribution: This study delves into advanced numerical techniques for analyzing supersonic flows with an emphasis on the application of Euler equations. It examines the theoretical underpinnings essential for these methods, including boundary conditions and error evaluation strategies [3]. A significant focus is placed on the implementation of the Successive Over-Relaxation (SOR) method, which is explored for its efficiency in solving large systems of linear equations and accelerating convergence in flow simulations [4]. Through comprehensive numerical experiments, this paper visualizes final velocity field distributions and analyzes convergence characteristics under various conditions, thereby assessing the precision and computational efficiency of the deployed methods [5]. The practical applications of this research are showcased in the design of aerospace structures and turbomachinery, providing actionable insights that enhance design and performance in engineering applications [6]. By integrating advanced numerical methods with real-world engineering challenges, this research contributes substantially to the field of computational fluid dynamics in supersonic regimes [7].

2. Theoretical Foundations and Numerical Methods

2.1 Introduction to Euler Equations and Their Application in Supersonic Flow

The Euler equations serve as the foundational mathematical framework for describing supersonic flow, offering a simplified model of inviscid, compressible fluid dynamics [8]. These equations are derived from the more comprehensive Navier-Stokes equations through the omission of viscous terms. The complete Navier-Stokes equation can be expressed as:

$$\rho(\partial u/\partial t + (u\cdot\nabla)u) = -\nabla p + \mu\nabla^2 u + (\lambda + \mu)\nabla(\nabla\cdot u) + \rho f \quad (1)$$

where ρ denotes density, u represents the velocity vector, p is pressure, μ is dynamic viscosity, λ is bulk viscosity, and f signifies body forces.

For an ideal fluid, characterized by zero viscosity ($\mu = 0$), the equation simplifies to the Euler equation:

$$\rho(\partial u/\partial t + (u\cdot\nabla)u) = -\nabla p + \rho f \quad (2)$$

Under the additional assumption that $\nabla\cdot\tau = 0$ (where τ is the viscous stress tensor), the momentum equation can be reformulated into the conservative form of the Euler equations:

$$\partial(\rho u)/\partial t + \nabla\cdot(\rho u\otimes u + pI) = 0 \quad (3)$$

Furthermore, the Euler equations encompass the conservation of mass and energy:

$\partial\rho/\partial t + \nabla\cdot(\rho u) = 0$ (mass conservation), $\partial E/\partial t + \nabla\cdot((E + p)u) = 0$ (energy conservation).

where E represents the total energy per unit volume.

In the context of supersonic flow analysis, the Euler equations exhibit hyperbolic characteristics, facilitating the propagation of information along characteristic lines [9]. This property is fundamental to understanding the behavior of supersonic flow and forms the basis for numerous numerical solution techniques.

While the Euler equations represent a simplification of the full Navier-Stokes equations, they nonetheless capture the essential physical properties of supersonic flow, including the formation and propagation of shock waves and expansion waves. Subsequent analyses will demonstrate the application of these equations to specific supersonic flow problems and explore their numerical solution methodologies [10].

2.2 Role of Boundary Conditions in Supersonic Flow Simulations

Boundary conditions are fundamental to the accurate simulation of fluid dynamics, serving to define the behavior of the fluid at the edges of the computational domain. Proper specification of BCs ensures the physical consistency and stability of the numerical solution, making them indispensable for modeling fluid flow phenomena. Different types of boundary conditions are appropriate for distinct physical scenarios and mathematical models, each tailored to capture the specific characteristics of the flow environment.

In numerical simulations, boundary conditions are encoded into the solver's algorithm to ensure that the physical laws at the edges of the computational domain are obeyed. These conditions must be satisfied at every iteration, thus affecting the solution throughout the computational domain. In the iterative solution process, boundary conditions are re-evaluated and re-applied at each iteration. This is because each iteration attempts to approach the true solution, and boundary conditions provide the necessary constraints to guide the solution in the right direction. Here taking Newman boundary conditions as an example.

Neumann boundary conditions, also referred to as natural or derivative boundary conditions, specify the derivative of the dependent variable at the boundary rather than its value. It specifies the derivative or normal derivative of a function on a boundary.

Mathematically, these conditions are expressed as:

$$\partial n / \partial u = g(x,y,z) \quad (4)$$

In fluid mechanics, this boundary condition is usually used to describe a certain rate of change of a fluid on a boundary. For example the Newman condition can be used to specify velocity gradients at boundaries.

$$\partial n / \partial u_i = g_i(x,y,z) \text{ for } i=1,2,3 \quad (5)$$

In addition, the definition object of Newman boundary conditions can also be the rate of change of temperature or pressure, etc.

2.3 Error Evaluation and Equation Satisfaction

In numerical simulations, evaluating the accuracy of the solution and verifying the satisfaction of the governing equations is crucial. In this study, we employ a residual-based approach to quantify how well the numerical solution satisfies the original partial differential equation. This method not only provides an overall assessment of the solution quality but also helps identify potentially problematic areas within the computational domain.

2.3.1 Residual calculation method

We define an error function to calculate the residual of the discretized equation. This function iterates over each interior point of the computational domain, computes the residual of the discretized equation at each point, and then calculates the average. Specifically:

For each interior grid point (i,j), we calculate the residual of the following discretized equation:

eq = -C0 * phi[i,j] + (B_yp * phi[i,j+1] + B_ym * phi[i,j-1]

-(A_xp+A_xm) * phi[i-1,j] + A_xm * phi[i-2,j]) (6)

where C0 = (-A_xp + B_yp + B_ym)

A_xp, A_xm, B_yp, and B_ym are equation coefficients, in this case:

A_xp = A_xm = 1 - M_inf^2 (7)

B_yp = B_ym = 1 (8)

We calculate the absolute value of eq and accumulate the residuals for all grid points.

Finally, we normalize the total residual by the total number of grid points (Nx*Ny) to obtain the average residual.

This error assessment method enables us to determine whether the numerical solution closely approximates the theoretical solution, thereby verifying the correctness and reliability of our numerical method. A sufficiently calculated error indicates that the numerical solution is accurate, giving us confidence in the results. Furthermore, error assessment serves as a critical basis for adjusting parameters within the numerical method to enhance the overall quality of the numerical solutions.

2.4 Grid Partitioning and Finite Difference Scheme Optimization

Grid partitioning and finite difference scheme optimization are critical aspects of numerical methods for solving partial differential equations (PDEs). These techniques enable the discretization of the continuous domain into a manageable grid structure, facilitating the approximation of derivatives through finite differences. Here, we detail how these concepts are applied in the provided code snippet.

2.4.1 Grid partitioning

Grid partitioning involves dividing the computational domain into a structured grid. Each grid point represents a discrete location where the solution is computed. In the given code, a rectangular domain is divided into a grid of points with a resolution determined by the number of grid cells (N).

1N = 64, phi = np. Zeros ((3*N, N)).

Here, phi is a matrix representing the scalar field to be solved. The grid has dimensions (3*N, N), where N is the number of points along the y-axis, and 3*N is the number of points along the x-axis, reflecting the extended domain needed for boundary conditions.

2.4.2 Finite difference scheme

The finite difference scheme approximates the derivatives of the scalar field phi using differences between neighboring grid points. The central difference scheme is used to approximate the first-order derivatives:

U [i-1, j-1] = 0.5 * (phi [i+1, j] – phi [i-1, j]) / hx (9)

V [i-1, j-1] = 0.5 * (phi [i, j+1] – phi [i, j-1]) / hy (10)

These formulas estimate the velocities U and V in the x and y directions, respectively. The coefficients hx and hy represent the grid spacing in the respective directions.

2.5 Application of the Successive Over-Relaxation (SOR) method

The SOR method is an iterative technique that accelerates the convergence of the Gauss-Seidel method by introducing an over-relaxation factor. The SOR method is predicated on the principle of modifying the Gauss-Seidel iteration by introducing an over-relaxation parameter ω.

As an example, for a system of linear equations Ax = b, the SOR iteration can be expressed as:

x^(k+1) = (1 - ω) x^(k) + ω (D + ωL) ^ (-1) (b - Ux^(k)) (11)

Where D, L, and U represent the diagonal, strictly lower triangular, and strictly upper triangular parts of A, respectively, and ω denotes the relaxation factor (1 < ω < 2).

An equation was given in this experiment:

(A φ_ x)x + (B φ_ y)_y = 0 (12)

A and B are functions of x and y. After discretization, the result is a five-point difference format containing A_ xp, A_ xm, B_ yp, B_ ym, which is used to approximate this second-order partial differential equation. As shown in Table 1.

Table 1. Implementing the Successive Over-Relaxation Method for Supersonic Flow Simulation

The discretized form at each grid point (i, j) is given by:
C0 * φ[i,j] - (B_ yp * φ[i, j+1] + B_ ym * φ[i,j-1] - (A_ xp+A_ xm) * φ[i-1,j] + A_ xm * φ[i-2,j]) = 0
where C0 = (-A_ xp + B_ yp + B_ ym)
The method can be coded as follows:
def SOR (phi):
newphi = phi. Copy ()
sor = 1. # SOR relaxation factor
for i in range (2, newphi. Shape [0]):

```
for j in range (1, newphi. shape [1]-1):
A_ xp = 1 - M_ inf * M_ inf
A_ xm = 1 - M_ inf * M_ inf
B_ yp = 1
B_ ym = 1
C0 = (-A_ xp + B_ yp + B_ ym)
dp = -newphi [i, j] + (B_ yp * newphi [i, j+1] + B_ ym * newphi [i, j-1]
-(A_ xp+A_ xm) * newphi [i-1, j] + A_ xm * newphi [i-2,j])/C0
Newphi [i, j] += sor * dp
# Implement Neumann BC at j = 0
j = 0
for i in range (2, newphi. shape [0]):
# ... (code for Neumann BC implementation)
return newphi
```

The implementation of the Successive Over-Relaxation (SOR) method in this study is characterized by several key aspects. Firstly, the function of the relaxation factor is to control the magnitude of each iteration update, thereby affecting the convergence and convergence speed of the algorithm.

Additionally, the calculation of coefficients A_ xp, A_ xm, B_ yp, and B_ ym for every grid point is an integral part of our methodology. Central to our SOR implementation is the update formula:

Newphi [i, j] += sor * dp (13)

where dp signifies the discrepancy between the freshly derived value and its predecessor.

3. Results and Discussion

3.1 Visualization of Final Velocity Field Distribution

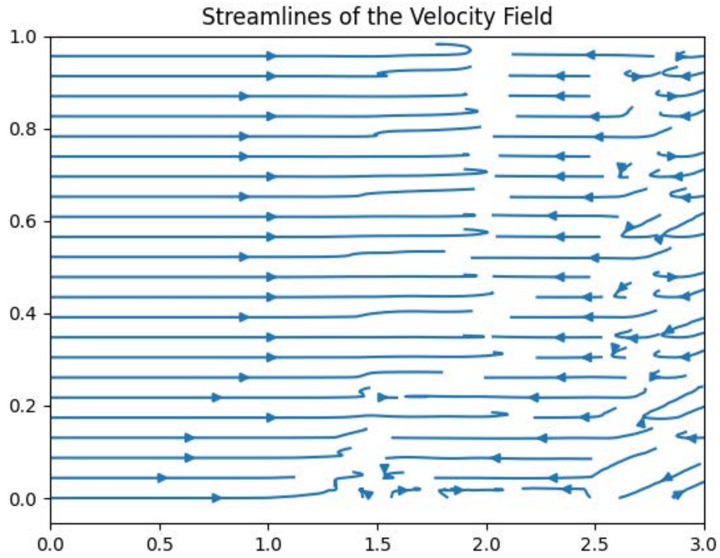

Fig. 1 Experimental results (Photo credit: Original).

The streamline plot provides a comprehensive visualization of the velocity field distribution in the computational domain. The Fig 1 reveals several key features of the flow:

Upper and Central Regions: In these areas, streamlines appear predominantly smooth and horizontal, indicating stable flow conditions with minimal vertical velocity components. This suggests that the flow in these regions closely resembles the freestream conditions, with limited influence from boundary effects.

Right Boundary: Near the right edge of the domain, streamlines exhibit increased curvature and density. This phenomenon is indicative of significant changes in both flow velocity and direction, likely due to boundary effects or the presence of an obstacle not directly visible in the plot. The compression of streamlines in this area suggests an acceleration of the flow.

Lower Region: The bottom portion of the plot, particularly towards the right side, displays notable streamline disturbances and undulations. These patterns are characteristic of flow instabilities or potential turbulent structures developing in the near-wall region. The presence of these features implies a complex interaction between the flow and the lower boundary condition.

Left Inlet: Streamlines enter the domain uniformly from the left, consistent with the specified inlet conditions in the numerical setup.

3.2 Convergence Characteristics under Varied Conditions

The numerical solution utilizes a Successive Over-Relaxation (SOR) method with a relaxation parameter Ω=1.7 and a supersonic Mach number $M\infty$=2.5. The code systematically monitors convergence behavior by outputting error metrics every 100 iterations, although specific error values are not provided. The simulation is run for a fixed 500 iterations, which may or may not be sufficient for full convergence, depending on the complexity of the problem and the desired level of accuracy. The computational domain is discretized using a 192×64192×64 grid (3N × N, where N=64N=64), offering a balanced approach between spatial resolution and computational efficiency.

3.3 Discussion on the Precision and Efficiency of Numerical Methods

The selection of the SOR method with a relaxation parameter Ω=1.7 reflects an effort to expedite convergence relative to simpler iterative methods. This value lies within the conventional range of $1 < \Omega < 2$ for optimizing SOR performance. The code is tailored for supersonic flow conditions ($M\infty$>1), with a specific focus on $M\infty$=2.5. Additionally, the imposition of Neumann boundary conditions at the lower wall (j = 0) with a specified function BC_F(x) adds a degree of physical realism to the simulation, likely contributing to the observed flow characteristics near the domain's base.

4. Application Scenarios

4.1 Application in Aerospace Engineering: Aerodynamic Design of Aircraft

Advanced numerical techniques for supersonic flow analysis have significant applications in the aerodynamic design of supersonic and hypersonic aircraft. These methods enable detailed analysis of complex flow phenomena critical to aircraft performance and safety.

A key application lies in the design of supersonic inlets for jet engines. Accurate prediction of shock wave structures and boundary layer interactions within the inlet is crucial for optimizing engine performance. Numerical simulations provide valuable data on pressure recovery, flow distortion, and shock stability under various flight conditions.

Another important application is in the design of control surfaces for supersonic aircraft. The effectiveness of control surfaces can change dramatically in supersonic flow due to the presence of shock waves. Numerical methods allow designers to predict these effects accurately, enabling the development of effective control systems for high-speed flight.

The ability to simulate unsteady supersonic flows also contributes to the analysis of aircraft stability and control. Phenomena such as buffeting and flutter, which can be particularly severe in transonic and supersonic regimes, can be studied using time-accurate versions of these numerical schemes.

4.2 Potential Applications in Other Engineering Fields: Turbomachinery Design

Beyond aerospace engineering, numerical methods for supersonic flow analysis have potential applications in turbomachinery design, particularly for high-speed compressors and turbines.

In the design of transonic compressors, accurate prediction of shock wave formation and interaction with blade boundary layers is crucial for optimizing performance and avoiding issues like choking and surge. Numerical techniques can provide detailed flow field information, helping designers to shape blade profiles for maximum efficiency and operability.

For supersonic turbines, such as those used in some rocket engine cycles, these methods enable the analysis of complex shock patterns within blade passages. This information is vital for managing aerodynamic losses and ensuring structural integrity under extreme flow conditions.

The ability to simulate unsteady supersonic flows is particularly valuable in analyzing phenomena like rotating stall in compressors and the propagation of pressure waves in turbine stages. These capabilities contribute to the development of more efficient and reliable turbomachinery for a wide range of applications, from power generation to aerospace propulsion.

5. Conclusion

This study has made substantial contributions to the field of supersonic flow analysis through the application of Euler equations, advancing the theoretical and practical methodologies essential for aerospace and turbomachinery engineering. By investigating and enhancing the theoretical foundations, including the crucial role of boundary conditions and error evaluation strategies, this research has established a robust framework for accurate numerical simulations. The successful implementation and optimization of finite difference schemes, alongside effective grid partitioning techniques, have proven their capability to capture intricate supersonic flow phenomena. Additionally, the application of the Successive Over-Relaxation (SOR) method has shown to be particularly effective in accelerating convergence and improving computational efficiency in supersonic flow simulations.

Several research avenues promise to deepen the understanding and enhance the application of supersonic flow technologies. First, extending the study to incorporate viscous effects through the full Navier-Stokes equations would offer a more comprehensive analysis of boundary layer interactions and heat transfer phenomena, essential for a more accurate representation of real-world fluid dynamics. Additionally, the development and validation of advanced turbulence models tailored for compressible, high-speed flows could significantly refine the accuracy of predictions for turbulent supersonic flows, crucial for practical engineering applications. Further, exploring fluid-structure interactions by coupling flow solvers with structural analysis tools will provide insights into aeroelastic phenomena, essential for assessing the structural integrity of aerospace structures under supersonic conditions. Integrating machine learning algorithms into the simulation processes could revolutionize supersonic flow analyses by enhancing shock capturing techniques or facilitating adaptive mesh refinement strategies. Lastly, engaging in multiphysics coupling to include additional phenomena such as combustion or plasma effects would broaden the applicability of these methods to more advanced aerospace applications, such as scramjet engines or re-entry vehicles. Collectively, these future directions not only aim to extend the current capabilities but also to innovate and push the boundaries of supersonic flow analysis and its applications in modern aerospace systems.

References

[1] Dong Z., Zhang Z. Inviscid fluid mechanics. Beijing: Tsinghua University Press, 2003.

[2] Dong Z., Zhang Z. Non-viscous Fluid Mechanics. Beijing: Tsinghua University Press, 2003 (in Chinese).

[3] Wang R., Zhu J., Wang S., Wang T., Huang J., Zhu X. Multimodal emotion recognition using tensor decomposition fusion and self-supervised multi-tasking. International Journal of Multimedia Information Retrieval, 2024, 13(4): 39.

[4] Klose Bjoern F., et al. A Numerical Test Rig for Turbomachinery Flows Based on Large Eddy Simulations With a High-Order Discontinuous Galerkin Scheme—Part II: Shock Capturing and Transonic Flows. Journal of Turbomachinery, 2024, 146(2).

[5] Zhu, X., Guo, C., Feng, H., Huang, Y., Feng, Y., Wang, X., Wang, R.: 'A Review of Key Technologies for Emotion Analysis Using Multimodal Information', Cognitive Computation, 2024, 1, (1), pp. 1-27.

[6] Lavimi Roham, et al. A review on aerodynamic optimization of turbomachinery using adjoint method. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 2024, 09544062231221625.

[7] Li Wu, Karl Geiselhart. Propulsion–Airframe Integration for Conceptual Redesign of a Low-Boom Supersonic Transport. Journal of Aircraft, 2024, 61(2): 331-344.

[8] Ranjan Rakesh, Lucia Catabriga, Guillermo Araya. A Spectral/hp-Based Stabilized Solver with Emphasis on the Euler Equations. Fluids, 2024, 9(1): 18.

[9] Zhu, X., Huang, Y., Wang, X., Wang, R.: 'Emotion recognition based on brain-like multimodal hierarchical perception', Multimedia Tools and Applications, 2024, 83, (18), pp. 56039-56057.

[10] Bilgiç M., Özgür Uğraş Baran, M. H. Aksel. Assessment of Effect of Flux Scheme and Turbulence Model on Blade-to-blade Calculations. Journal of Applied Fluid Mechanics, 2024, 17(7): 1444-1456.