

Development of Universal Solver for Incompressible and Compressible Flow Simulation

Yoshihiro YAMAGUCHI*

ABSTRACT

A numerical methodology that decomposes a fluid motion into the convection due to the flow material velocity and the transportation of the sound wave was newly developed and then a unified flow solver for all-speed flow regime was conducted. The developed flow solver has been validated using bench mark problems. Furthermore, the solver has been extended to practical one for an engineering purpose, such as three-dimensional turbulent flow, by implementing an unstructured grid method, a parallel computation technique, and a turbulence model. Availability as a unified flow solver in practical use has been confirmed by solving a flow field around three-dimensional wing.

1. Introduction

Until recently, advances in Computational Fluid Dynamics (CFD) have mainly proceeded along two tracks—solutions for incompressible problems and solutions for compressible problems—in terms both of the analytical methods developed and of the groups of researchers developing them. The reasons for this are not simply the difference between the two research objects. As shown in Fig. 1, the different methods of solution must be used for different conditions, determined by fluid thermal properties; it is simply not valid to use one method in a region that requires the other method. The border between the compressible and the weakly compressible regions is called the “low Mach number limit.” Still, every fluid, whether liquid or gas, is actually compressible to at least some degree. Therefore, the term “incompressible flow” has been used to signify a fluid that is modeled with incompressible solution methods under the condition of zero changes in density. The researcher must bear in mind that this is only an approximation of the actual fluid. For the discussion in the present study, fluids with compressibilities or heat-expansion ratios of less than 5% were defined as weakly compressible.

As stated above, so far, designers have used different solution methods as appropriate to the flow phenomena under consideration. When handling conditions that extend across the

low Mach number limit, however, they have had to make special assumptions, construct special models, or endure difficulties of convergence and questionable reliability in existing models.

Recent developments in computation capabilities have enabled solution of large and complicated systems. For example, complete simulations of gas turbine engines and reciprocating engines, from inlet to exhaust, have been constructed. Inside, slow- and high-speed flows intermix, and even the low-speed flows see density fluctuations due to combustion-caused temperature differentials. Previous

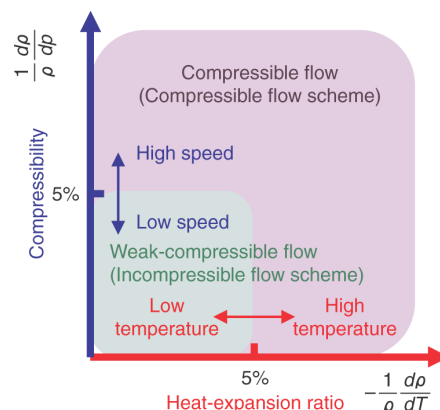


Fig. 1 Schematic of relation between flow phenomena and numerical method

* Wako Research Center

methods had to divide the models into distinct mechanical components and employ different analytical methods appropriate to each component. However, it is difficult to distinguish each component's interference with adjoining components, and boundary conditions are vague and contradictory; consequently, the precision of conventional models has been pretty low.

A simulation method that joins all the components of a system and allows large and complicated problems to be solved would constitute a powerful tool that also eliminates all of the problems described above. It would allow selection of specific inlet and outlet atmospheric conditions and other boundary conditions, without the introduction of models or assumptions. Such a method would also provide highly reliable solutions for the capabilities of every component and of the entire system.

The object of this research was to construct a CFD scheme with a unified algorithm capable of handling mixed flows of weakly compressible and compressible fluids and the flow phenomena occurring around the low Mach number limit. The following three topics were addressed, with the aim of applying this scheme to complex and large-scale problems: (1) employing of a turbulence model; (2) adaptability to hybrid grids to accommodate complicated shapes; and (3) capability for parallel computation to reduce the calculation load of large-scale problems. A code with all these capabilities was developed. This report describes validation of the code and the good results obtained when it was applied to a practical problem.

2. Comparison with Conventional Method

The following summarizes why methods for incompressible problems and methods for compressible problems may not be applied to each other's situations and describes the advantages of the present solver.

- (1) Reasons incompressible solutions are unsuitable for compressible flows:

A simplified governing equation (conservation of mass, conservation of momentum) is derived from the complete Navier-Stokes (NS) equations and solved under the assumption of constant density. Because energy-conservation equations are not treated as simultaneous equations, compressibility cannot be considered.

- (2) Reasons compressible solutions are unsuitable for weakly compressible flows:

The full NS equations are used as the governing equations. These are composed of the continuity equation, the momentum equation, and the energy equation, and account for weakly compressive phenomena. They describe macroscopic flow phenomena such as density and momentum and regard fluid materials as continuous body for microscopic phenomena of advection or pressure wave propagation of the fluid molecules and heat generation due to molecule conffliction. It can be comprehended by eigenvalue analysis of NS equations proving that the advection of physical quantities of fluid is due both to the flow velocity and to acoustic waves (pressure waves). However, compressible solutions require simultaneous solution of advection equations for flow velocity and the speed of sound. In analysis of flows whose velocity scales are much less than the speed of sound, numerical errors build up due to scale differences. These can delay or even prevent convergence of calculations in such slow, weakly compressible flows.

- (3) Characteristics of this solution:

Handling differences in scale between flow velocity and the speed of sound is the key to development of a universal solver. Preconditioning methods^{(1),(2),(3)} have been proposed by inserting a preconditioning matrix for accommodating all scales into the equations. It has also been suggested to separate the calculations for advection and acoustics^{(4),(5)}. Both of these approaches are still under study, and almost no techniques are presently in use for general problems.

The method described here employs the full NS equations as the constitutive equations and represents a new viewpoint of the properties of the full NS equations. The following two sentences discuss the two roles of flow velocity and acoustic waves are decomposed in the weakly compressive regime, as described above. Advection due to flow velocity can be expressed with a unidirectional, upstream-to-downstream hyperbolic equation. In contrast, since acoustic waves have the characteristics of a standing wave with a pressure-wave source and boundaries, they can be expressed using elliptic equations.

This analysis will separate the flow velocity-based effects from the acoustic wave-based effects to utilize the characteristics of each type of equation in the universal solver constructed here. In the first state, advection due to flow velocity is evaluated to exploit current knowledge from compressible-flow solutions. Next, propagation effects due to acoustic waves are provisionally used to approach the boundary value problem. It can be proven that the contribution of the pressure in the momentum equation is due both to flow velocity and to acoustic waves⁽⁶⁾, so the contribution by pressure is separated, and high-precision calculations for pressure are performed.

3. Governing Equations

Nonconservative forms of the full equation (1), the momentum equation (2), and the temperature equation (3) were derived from the complete NS equations to form the governing equations. Here, the temperature equation is derived by substituting the continuity equation and the momentum equation into the energy equation:

$$\frac{\partial \rho}{\partial t} + \mathbf{u} \cdot \nabla \rho = -\rho \nabla \cdot \mathbf{u} \quad (1)$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = \frac{1}{\rho} \nabla \cdot (-p\mathbf{I} + \Pi) \quad (2)$$

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \frac{1}{c_v \rho} \{-p(\nabla \cdot \mathbf{u}) - \nabla \cdot k \nabla T + \Pi : \nabla \mathbf{u}\} \quad (3)$$

Here, ρ , $\mathbf{u}$, p , and T represent density, velocity vector, pressure, and temperature, respectively. c_v , Π , and k represent the constant-volume specific heat, the viscosity tensor, and the thermal conductivity. $\mathbf{I}$ is the identity matrix.

In addition, the k - ω turbulence model^{(7),(8)} is used for practical calculations. Here, k is the turbulent energy.

4. Separation of Variables

Equations (1) – (3) were solved in a multistage sequence with time steps proceeding as $n \rightarrow n + 1$. The initial stage was called the advection stage (time step $n \rightarrow *$), in which

physical quantities of the flow translated under the influence of the flow velocity only. The second stage is called the quasi-isentropic stage (time step $n \rightarrow n+1$), in which small physical quantities were subjected to small disturbances under pressure waves according to the equation governing sound waves. Viscosity and heat flux were evaluated in the third stage, the viscid stage (time step $n+1 \rightarrow n+2$). The pressure equation was derived to evaluate the boundary value problems in the quasi-isentropic stage. Here, the time fluctuations in the pressure gradient term in the $(n \rightarrow n+1)$ time step for the momentum equation are separated using $\nabla p^{n+1} = \nabla(p^n + \phi)$. Here, ϕ was the value $\phi = p^{n+1} - p^n$, and a small disturbance of the pressure. The following two results of separation were observed. First, when the pressure equation was solved, the fact that pressure was not a direct variable moderated the accumulation of numerical errors. Second, the contribution of the pressure gradient term in the flow speed time scale was separated from its contribution in the acoustic speed time scale; this allowed calculations of each to be done in an appropriate manner⁽⁹⁾.

The decomposition in the three stages are shown in Equations (4) — (6).

Advection stage:

$$\frac{\partial}{\partial t} \begin{pmatrix} \rho \\ \mathbf{u} \\ T \end{pmatrix}^* + (\mathbf{u}^n \cdot \nabla) \begin{pmatrix} \rho \\ \mathbf{u} \\ T \end{pmatrix}^* = \begin{pmatrix} 0 \\ -\nabla p^n \mathbf{I} / \rho^* \\ 0 \end{pmatrix} \quad (4)$$

Adiabatic stage:

$$\frac{\partial}{\partial t} \begin{pmatrix} \rho \\ \mathbf{u} \\ T \end{pmatrix}^{**} = \begin{pmatrix} -\rho^* \nabla \cdot \mathbf{u}^{**} \\ -\frac{\nabla \phi \mathbf{I}}{\rho^*} \\ -\frac{p^* \nabla \cdot \mathbf{u}^{**}}{C_v \rho^*} \end{pmatrix} \quad (5)$$

Viscid Stage:

$$\frac{\partial}{\partial t} \begin{pmatrix} \rho \\ \mathbf{u} \\ T \end{pmatrix}^{n+1} = \begin{pmatrix} 0 \\ \frac{\nabla \cdot \mathbf{I}^{n+1}}{\rho^*} \\ \frac{\nabla k \cdot \nabla T^{n+1} + \mathbf{I}^{**} : \nabla \mathbf{u}^{**}}{C_v \rho^*} \end{pmatrix} \quad (6)$$

The next sections describe the solutions in each stage.

4.1. Advection Stage

The methods for solving advection calculations has developed with the methods for solution of compressible flows, where advection effects are dominant. This is called the approximate Riemann problem and features the capability to capture shock waves, which are physically discontinuous, in a sharp manner and without oscillation. The present method exploits the advantages of the approximate Riemann problem, since it involves flows in which shock waves occur. As shown in Eq. (4), the variable is not a conservative one but a primitive one which also is nonconservative; the approximate Riemann problem cannot be applied here. In addition, strict mathematical principles interfere with handling of shock waves⁽¹⁰⁾. Therefore, the key to these analyses is how to make an upwinding scheme for a wide range of flow speeds, from low to high. Equation (4) shows that the motion of physical quantities depends only on flow speed $\mathbf{u}$. Nonetheless, the pressure gradient term remains on the right side of the momentum equation, as a term

independent of the flow velocity. The Advection Upstream Splitting Method (AUSM)⁽¹¹⁾, an approximate Riemann problem, was used for obtaining flow velocity flux and pressure gradient terms in the upwinding scheme. The upwinding scheme contributed greatly to the precision of pressure calculations.

4.2. Quasi-isentropic Stage, Viscid Stage

Here, the authors take an asymptotic solution for the effects of advection due to acoustic waves, as boundary values problem. Equation (5) was substituted into the total time differential of the equation of state to derive the pressure equation:

$$\nabla \cdot \left(\frac{1}{\rho^*} \nabla \phi \right) - \frac{\phi}{\gamma \Delta t^2 p^*} = \frac{\nabla \cdot \mathbf{u}^*}{\Delta t} \quad (7)$$

Equation (7) can be solved with SOR, the residual cutting method⁽¹²⁾, or other general matrix solution methods. Once the pressure is determined, the density, velocity, and temperature are calculated.

In the viscosity stage, the time step is advanced by one after the calculations have been completed using the values found in the quasi-isentropic stage.

5. Discretization

5.1. Spatial Discretization

Two methods can be considered for locating physical amounts of fluid on the calculation grid. One is to collocate all mass properties at each grid point, and the other is staggering, in which the density, pressure, temperature, and other mass properties are located in the center of a cell while the velocities are located at the grid surfaces. Considering the process of deriving the pressure equation, the staggered approach seemed preferable for this method. Nevertheless, collocation was used, because other objectives were to maintain the conservative properties of the discretization and to avoid complications during development of the program. A hybrid grid of two- and three-dimensional elements was used for flexibility in calculations for complex shapes. It was based on the finite volume method (FVM). The spatial resolution of the grid used in the advection stage greatly affects the sharpness of the calculations of shock waves. Still, since it is not possible to make stencils of wide areas in FVM, a Monotone Upstream-centered Scheme for Conservation Laws (MUSCL) was employed to improve second-order spatial accuracy.

5.2. Time Discretization

To shorten calculation time for steady-state calculations, local time stepping was performed with first-order time accuracy. For stability of calculations with progress of time, the lower-upper symmetric Gauss-Seidel (LU-SGS)⁽¹³⁾ method with implicit calculations was employed in the advection and viscid stages. Meanwhile, for unsteady flow calculations, time accuracy was improved to second-order accuracy with the Crank-Nicolson method. LU-SGS was combined with the Newton-Raphson method in the advection and viscid stages and accuracy of time was maintained using dual-time stepping⁽¹⁴⁾.

6. Parallel Computations

This solver poses about the same calculation load as AUSM for compressive flow solutions in the advection stage; since the compressive equations were added in the quasi-

isentropic stage, the calculation load was higher than for the compressive solution. Parallel computation was employed to spread the considerable calculation load and to shorten the turnaround time.

Domain decomposition with memory allocation of the variables was used as the parallel computation method. This allows large-scale processing since the demanded memory can be distributed among all the nodes in shared-memory parallel processing. METIS⁽¹⁵⁾ was used to code the domain decomposition, balancing the memory load in each node. Data exchange between adjacent regions was carried out only for physical quantities in dummy cells located on region boundaries. This method minimized the volume of data exchanged among nodes and the data communication time. Figure 2 shows the measured and theoretical dimensionless calculation times for a single CPU. About 1.1 million nodes were used for measurements; it was seen that the parallel processing approach was effective, with the actual calculation time approaching the theoretical time for the actual number of nodes.

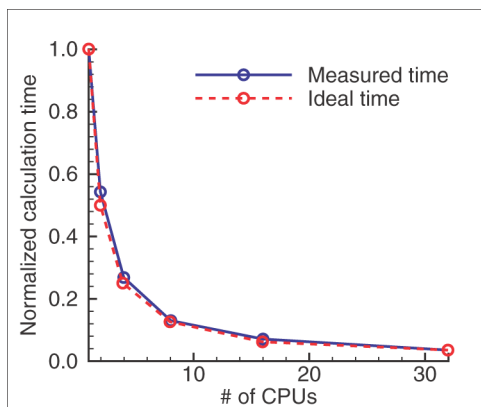


Fig. 2 Normalized calculation time between measured and ideal time

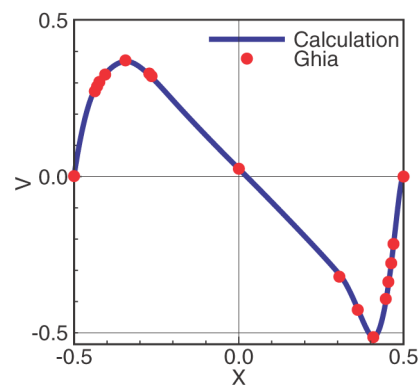
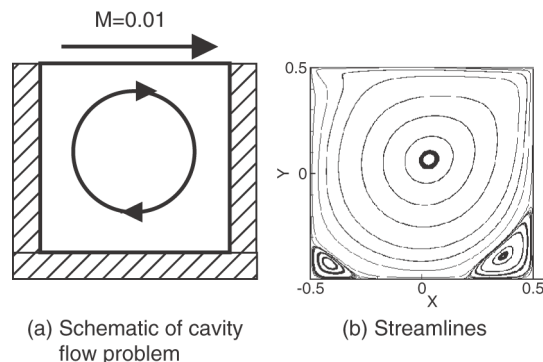
7. Validation

This solver was validated using typical problems in weak-compressible and compressible flow.

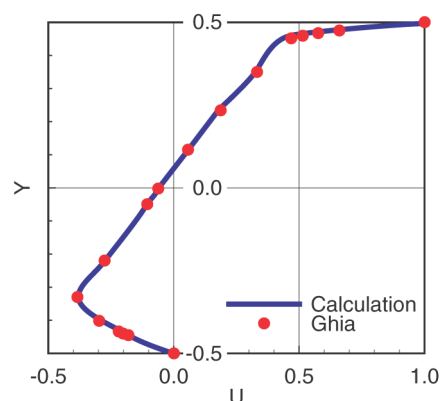
7.1. Validation of Weak-compressible Flow Solution

The weak-compressible flow model was validated by analyzing an incompressible, viscous cavity flow. The Reynolds number of the flow was 1 000 and the inlet Mach number was 0.01. Figure 3(a) is the analytical image. Figure 3(b) shows the calculated streamlines, Fig. 3(c) the distribution of velocity v at $y = 0$, and Fig. 3(d) the distribution of velocity u at $x = 0$. These velocity distributions are compared with the velocity profile validation data reported by Ghia et al.⁽¹⁶⁾

The velocity distribution in the primary vortex matches the validation results quite well. The secondary vortices were well captured, as they were in Ghia's results. These findings indicate that this method is appropriate for analysis of weak-compressible flow regions.



(c) Profile of v -velocity along $y=0$



(d) Profile of u -velocity along $x=0$

Fig. 3 Validation results for cavity flow at $Re=1\,000$, $Min=0.01$

7.2. Validation of Compressible Flow Solution

The method was verified in a compressible flow by analyzing Sod's one-dimensional shock tube problem⁽¹⁷⁾. In this problem, the initial conditions are static with a 10:1 pressure ratio across the partition between the left and right sides of the tube and a density ratio of 8:1 in the same orientation for the gas densities. The test was carried out by comparing the model-calculated results with the exact solutions for the unsteady supersonic flow occurring after removal of the partition. Figure 4(a) diagrams the initial values in the shock tube, and (b) and (c) present the distribution of pressure and velocity, respectively, 0.14154 s after removal of the partition.

When nonconservative governing equations are used, the weak solution is incapable of recognizing the entropy jump in the shock wave, so it predicts the wave position somewhat later than the theoretical model does⁽¹⁰⁾. Almost no delay is seen by this solver, however; good results were obtained.

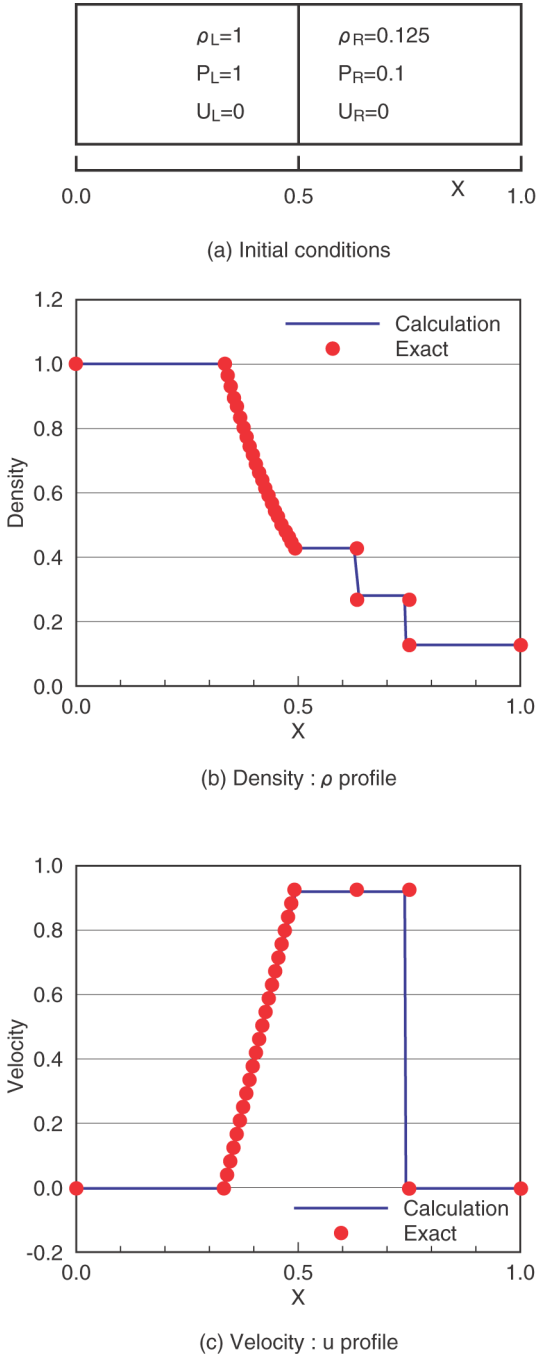


Fig. 4 Validation results for the Sod's problem

7.3. Validation of Flow around a Two-Dimensional Airfoil

The two tests described above showed that the present method is usable for both weakly compressible and compressible internal flows. Next, a simulation of inviscid flow around a two-dimensional NACA0012 airfoil was performed as an example of external flow. Another type of universal solver, local preconditioning, requires some handling by the operators, because of its dependence on

certain parameters at stagnation points^{(1) - (3)}. Therefore, solutions from weakly compressible and compressible flows were sought in the vicinity of stagnation points to check for predictions of physically impossible behavior. The main flow speeds observed were weakly compressible flow at $M_\infty = 0.005$ [Fig. 5(a)], high subsonic flow at $M_\infty = 0.6$ [Fig. 5(b)], and transonic flow at $M_\infty = 0.9$ [Fig. 5(c)]. The angle of attack in all simulations was 0° . The contours in the figures are absolute flow speed values normalized to M_∞ ; consequently, the colors of the contours in Figs. (a) – (c) cannot be compared directly.

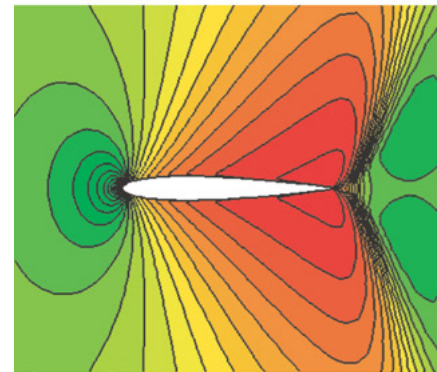
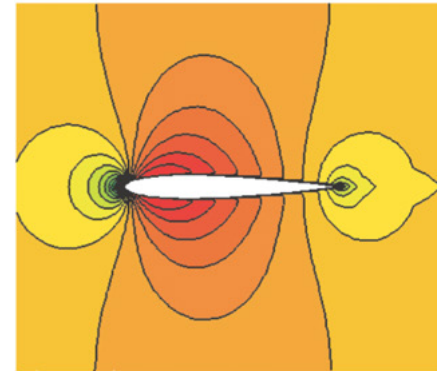
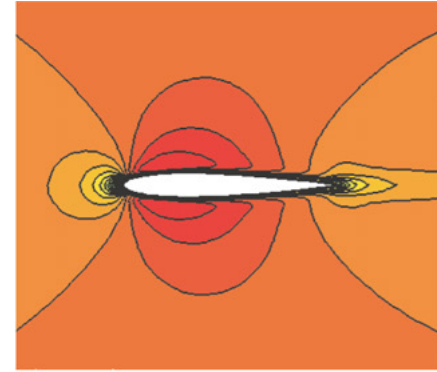


Fig. 5 Contours of velocity magnitude around NACA0012 airfoil

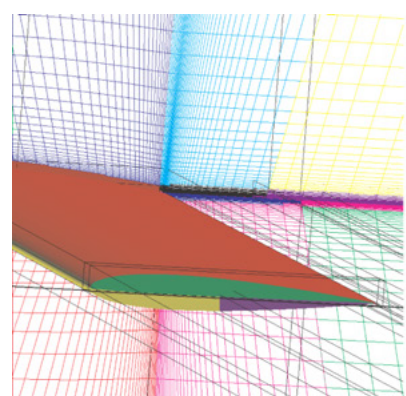
In the incompressible case, the same calculation results were found as in the weakly compressible case in Fig. 5(a), whereas this was not possible in the cases shown in Figs. (b) and (c), where compressibility effects were present. At the same time, although conventional compressive flow analysis shows the same results as in Figs. (b) and (c), it fails to converge in the case of Fig. (a), weakly compressible flow, a physically impossible “solution”. In contrast, this method provided convergent solutions for a reasonable flow velocity distribution, from low speeds to shock wave speeds, in weakly compressible flows. These results indicate that the solver may also be applied to flows at a broad range of speeds around physical objects at which stagnation points form.

8. Analysis around a Three-dimensional Wing Flow

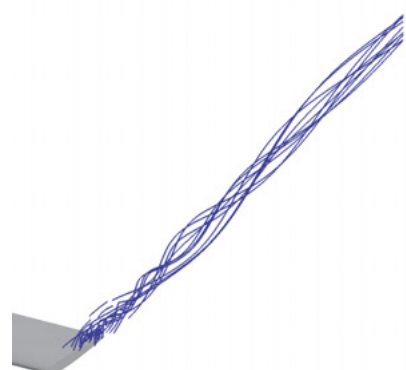
An analysis of turbulent flows around a three-dimensional NACA0012 airfoil was carried out to test this solver with a practical problem. The analytical conditions were low speed and high angles of attack, reflecting conditions at takeoff or landing. The reason for selecting high angles of attack was to raise the wing loading at low speed. The pressure difference across the upper and lower wing surfaces causes intense wingtip vortex and become wake vortex above an airport are an important reason for the delay between takeoffs and landings, as they interfere with the flight of planes passing through them. Thus, the safety issue and the desire to increase the volume of planes using an airport have inspired a large number of attempts to predict wingtip vortices using CFD. Since previous analyses have addressed speed regimes in the vicinity of the low Mach number limit, however, they have had problems with convergence, and the solutions have not been very reliable. It has been suggested that the influence of physically impossible viscosities in numerical treatment (numerical viscosity) causes the predicted wake vortex diffusing earlier than they actually do, with some dependence on the calculation method and the grid⁽¹⁸⁾. Accordingly, an analysis was performed, exploiting the present method’s universal solver and high-precision approach to calculating pressure.

The total number of nodes in the grid was 1 157 856; all cells were hexahedral to reduce influences from numerical diffusion. The calculation domain was divided into 16 regions for parallel processing. Figure 6(a) shows the regions in color coding. The wing aspect ratio was 3, the flow Mach number was 0.12, the angle of attack was 10° and the Reynolds number was 0.9×10^6 .

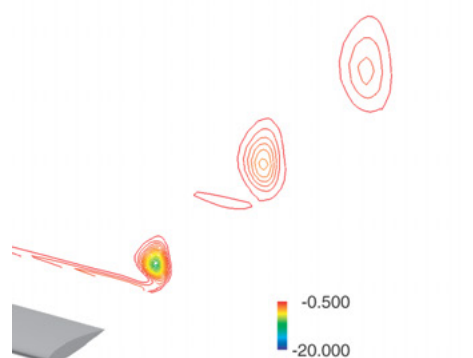
Figure 6(b) shows the streamlines of a wingtip vortex. This vortex is seen to arise on the upper surface of the wing and gradually diffuse due to viscosity effects, but it extends over 10 chord lengths downstream from the wing. This indicates that numerical diffusion was low. Next, Fig. 6(c) shows helicity contours. The contour locations correspond to 2, 4 and 6 chord lengths downstream. The helicity h is defined $h = \mathbf{u} \cdot \text{curl}(\mathbf{u})$; the maximum absolute value defines the center of the vortex, and the sign indicates the direction of vortex rotation. A negative sign means counterclockwise rotation, looking from upstream, and vice versa. Since the helicity is nearly zero in all locations other than wingtip vortices and downstream, the figure has been scaled for greater legibility. The vortices were negative at locations corresponding to wingtip vortices, and as in the previous streamline diagrams, counterclockwise vortices were clearly



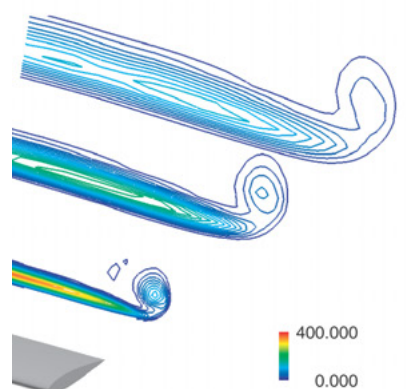
(a) Calculation grid divided by METIS method



(b) Streamlines around NACA0012 rectangular wing by turbulent calculation



(c) Helicity contours at 2, 4, and 6-chord downstream from trailing edge by turbulent calculation



(d) Contours of turbulent viscosity at 2, 4, and 6-chord downstream from trailing edge

Fig. 6 Results of a wingtip vortex calculation around NACA0012 rectangular wing

occurring. Contours of turbulent flow viscosity ratio in the same locations as those for helicity are shown in Fig. 6(d). Turbulence models cannot be avoided, if the researcher wishes to obtain analytical results closely resembling real phenomena, at realistic calculation loads. Turbulent viscosity was clearly affecting this flow in the proper manner, judging from the flow cross sections at the wingtip and the downstream locations. This universal solver code was thus found to be useful for analyzing large-scale, complicated, three-dimensional phenomena.

9. Conclusion

A universal solver for numerical analysis of incompressible and compressible flows was constructed. Basic validation tests confirmed that this solver predicts both internal and external flows in the weakly compressible and compressible regimes. Capabilities for turbulent flow analysis and flow around complicated shapes were then shown, and finally the more realistic problem of flow around a three-dimensional airfoil was examined. This solver was shown to be useful for examining practical problems. The capability for parallel processing was also added, enabling efficient distribution of the calculation load, which nearly equaled the theoretical value. This solver should be fully developed in the future. It can be applied to analysis of flows in a gas turbine and a reciprocating engine and also used for studies of phenomena influenced by pressure fluctuations, such as combustion and environmental noise.

References

- (1) Turkel, E.: Preconditioned Methods for Solving the Incompressible and Low Speed Compressible Equations, *J. Comput. Phys.*, Vol.72, No. 2, p. 227-298 (1987)
- (2) Choi, Y. H., Merkle, C. L.: The Application of Preconditioning in Viscous Flows, *J. Comput. Phys.*, Vol. 105, p. 207-223 (1993)
- (3) Turkel, E., Vatsa, V. N., Radespiel, R.: Preconditioning Methods for Low-Speed Flows, *AIAA paper*, 96-2460 (1996)
- (4) Hah, C.: Navier-Stokes Calculation of Three-Dimensional Compressible Flow Across a Cascade of Airfoils with an Implicit Relaxation Method, *AIAA paper*, 86-0555 (1986)
- (5) Yabe, T., Wang, P. Y.: Unified Numerical Procedure for Compressible and Incompressible Fluid, *J. Phys. Soc. Japan*, Vol. 60, No. 7, p. 2105-2108 (1991)
- (6) Rehm, R. G., Baum, H. R.: The Equation of Motion for Thermally Driven Buoyant flows, *J. of Research of the National Bureau of Standards*, Vol. 83, p. 297 (1978)
- (7) Wilcox, D. C.: Simulation of Transition with a Two-Equation Turbulence Model, *AIAA Journal*, Vol. 32, No. 2, p. 247-255 (1994)
- (8) Wilcox, D. C.: Reassessment of the Scale-Determining Equation for Advanced Turbulence Models, *AIAA Journal*, Vol. 26, No. 11, p. 1299-1310 (1988)
- (9) Yamaguchi, Y., Arima, T., Obayashi, S., Nakahashi, K.: Simulation of Aircraft Wake Vortex Using Pressure-based Unified Numerical Approach for Incompressible and Compressible Flows, *AIAA Paper*, 04-0077 (2004)
- (10) Leveque, R. J.: Finite Volume Methods for Hyperbolic Problem, *CAMBRIDGE TEXTS IN APPLIED MATHEMATICS* (2002)
- (11) Liou, M.-S.: Progress Towards An Improved CFD Method: AUSM+, *AIAA Paper*, 95-1701-CP (1995)
- (12) Tamura, A., Kikuchi, K., Takahashi, T.: Residual Cutting Method for Elliptic Boundary Value Problems: Application to Poisson's Equation, *J. Comput. Phys.* No. 137, p. 247-264 (1997)
- (13) Jameson, A., Yoon, S.: Lower-Upper Implicit Schemes with Multiple Grid for the Euler Equations, *AIAA Journal*, 30, No. 11 (1992)
- (14) Pulliam, T. H.: Time Accuracy and the Use of Implicit Methods, *AIAA paper*, 93-3360 (1993)
- (15) <http://www-users.cs.umn.edu/~kaypis/metis/>
- (16) Ghia, U., Ghia, K. N., Shin, C. T.: High Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method, *J. Comput. Phys.*, No. 48, p. 387-411 (1982)
- (17) Sod, G. A.: A Survey of Several Fluid Difference Methods for System of Nonlinear Hyperfsonic Conservation Laws, *J. Comput. Phys.* No. 27, p. 1-31 (1978)
- (18) Murayama, M., Nakahashi, K., Obayashi, S., Kato, T.: Numerical Simulation of Vortical Flows Using Vorticity Confinement Coupled with Unstructured Adaptive Grid Refinement, *Computational Fluid Dynamics Journal*, Vol. 10, p. 28-36 (2001)

■ Authors ■



Yoshihiro YAMAGUCHI