Verification of the numerical analysis using a collocated grid system directly solving the quasi-inviscid incompressible flow around a symmetric airfoil

Hiroki Suzuki¹,⁴, Masaya Watanabe², Yutaka Hasegawa³ and Shinsuke Mochizuki¹

¹Graduate School of Sciences and Technology for Innovation, Yamaguchi University, 2-16-1 Tokiwadai, Ube-shi, Yamaguchi 755-8611, Japan
²Department of Engineering Physics, Electronics and Mechanics, Nagoya Institute of Technology, Gok iso-cho, Showa-ku, Nagoya-shi, Aichi 466-8555, Japan
³Department of Electrical and Mechanical Engineering, Nagoya Institute of Technology, Gokiso-cho, Showa-ku, Nagoya-shi, Aichi 466-8555, Japan
⁴E-mail: h.suzuki@yamaguchi-u.ac.jp

Abstract. The present study describes the results of validating a high-quality computational scheme of kinetic energy conservation characteristics under a collocated grid system. In the present study, the pseudo-inviscid incompressible flow around the Kármán-Trefftz airfoil is analyzed. In this analysis, the kinetic energy conservation error is reduced by temporally evolving the governing equation using pressure increments in fractional steps. This sufficient conservation of kinetic energy is validated using an inviscid homogeneous isotropic fluctuation field. The velocity field obtained by this analysis is compared to that of the potential flow obtained using the Kármán-Trefftz airfoil. The numerical analysis is also validated using the lift coefficient profile as a function of a small angle of attack.

1. Introduction
In applications of fluids engineering, there are commonly cases that require the solution of an incompressible flow bounded by a wall. In particular, in aerodynamics, these boundaries are often curved. To analyze a flow bounded by curved walls, the governing equations are often discretized into a curvilinear grid system using a collocated grid system. An early example of a numerical scheme for an incompressible flow using a collocated grid system was developed by Rhie and Chow [1]. In addition, this collocated grid system is often used to numerically analyze the flow around an airfoil near stall [2, 3].

The incompressible flow around an airfoil is often analyzed using a curvilinear grid system as well. The collocated grid system is commonly used for this purpose [4-7]. Previous studies have examined the accuracy of a numerical analysis for incompressible flows by using a collocated grid system (e.g., [8, 9]). In these cases, using a collocated grid, the kinetic energy conservation error is usually proportional to the time step itself. The magnitude of this kinetic energy conservation error is larger than that in the analysis using a staggered grid. When a staggered grid is used, the kinetic energy conservation error can be obtained as a higher-order function of the time increment. Therefore, in this type of analysis, the kinetic energy conservation error can be reduced to be sufficiently small [10-12]. A method has also been proposed to reduce the conservation error of the kinetic energy for the
analysis using a collocated grid system [13]. In the present research, the results of numerical analysis using a collocated grid system are shown using this method.

The purpose of the present study is to validate the present numerical analysis with sufficiently high kinetic energy conservation characteristics under a collocated grid system by analyzing the quasi-inviscid flow around an airfoil. The Kármán-Trefftz airfoil is used for this analysis, and the flow around the airfoil is obtained as a potential flow. Additionally, a laminar Taylor-Couette flow is used to verify the coordinate transformation of the present numerical simulation. The flow around the Kármán-Trefftz airfoil is obtained using an O-type computational grid system. In addition, the kinetic energy conservation error of this analysis method is validated using the inviscid homogeneous isotropic fluctuation field. The flow field obtained by this analysis is then compared with that obtained as the potential flow. Finally, using the profile of the lift coefficient as a function of angle of attack, the numerical analysis method for analyzing the flow around an airfoil is validated.

2. Numerical methods

2.1. Numerical schemes

In this numerical simulation, an incompressible flow is analyzed. Thus, the governing equations are the continuity equation and the Navier-Stokes equation, as follows:

\[ \frac{\partial u_i}{\partial x_i} = 0 \quad \text{and} \quad \frac{\partial u_i}{\partial t} = -\frac{\partial u_i u_j}{\partial x_j} - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \frac{2}{Re} \frac{\partial u_i}{\partial x_i} \right] \]

where \( x_i \) is the physical space for the \( i \)th component, \( t \) is non-dimensional time, \( u_i \) is the normalized instantaneous velocity for the \( i \)th component, \( p \) is the normalized pressure, and \( Re \) is the Reynolds number. In this analysis, the flow is quasi-inviscid, so the viscous terms in Equation (1) are considered to be insignificant. Consequently, the following equations can be derived from Equation (1):

\[ \delta_i F_i/\delta x_i = 0 \quad \text{and} \quad \delta_i (F_i \bar{u}_i)/(\delta x_i) - \delta_i p/\delta x_i = 0. \]

Here, the discretization and approximation forms used in the above equation are given by the second-order difference scheme as follows:

\[ \delta_i f/\delta x_i = \left( f(x_i + h/2, x_2, x_3) - f(x_i - h/2, x_2, x_3) \right) / h, \]

\[ \delta_i f/\delta x_i = \left( f(x_i + h, x_2, x_3) - f(x_i - h, x_2, x_3) \right) / 2h, \]

\[ f_i^{1/2} = \left( f(x_i + h/2, x_2, x_3) - f(x_i - h/2, x_2, x_3) \right) / 2, \]

where \( h \) is an arbitrary value of grid spacing and the same forms are used for the other components. The velocity components \( u_i \) for the \( i \)th component and pressure \( p \) are defined at the center of the computational cell. In Equation (2), \( F_i \) is the \( i \)th velocity component defined on the surface of the computational cell. Equation (2) discretized under the collocated grid is solved using the time integration method.

A previous study [6, 13] focused on pressure discretization schemes and reduced kinetic energy conservation errors under a collocated grid system. This study considers that the previous study [13] has improved the kinetic energy conservation error in the collocated grid system. In the present study, we use a method to reduce this conservation error that analyzes the pressure increment rather than the pressure itself. This is equivalent to the SMAC method. Using the second-order Adams-Bashforth scheme, the time evolution steps for this scheme are as follows:

\[ \bar{u}_i^{n+1} = u_i^n - \left( \frac{3}{2} \Delta t \right) N_i^n + \delta t p_{i-1} / \delta x_i + \left( \frac{1}{2} \Delta t \right) (N_{i-1}^{n+1} + \delta t p_{i+1} / \delta x_i), \]

\[ \delta_i (\delta_t (\Delta t p_{i+1}^{n+1})/\delta x_i) / \delta x_i = (1 / 2 \Delta t) \delta_i \left( \bar{u}_i^{n+1,1} \right) / \delta x_i, \]

\[ u_i^{n+1} = \bar{u}_i^{n+1} - 2 \Delta t \delta_t \Delta t p_{i+1} / \delta x_i, \]

\[ F_i^{n+1} = \delta_t \Delta t p_{i+1} / \delta x_i, \]

\[ p^{n+1} = p^n + \Delta t p_{i+1}^{n+1}. \]
3

Figure 1. Schematic diagrams of computational conditions for the analysis of a laminar Taylor-Couette flow using physical and computational spaces.

In this scheme, the kinetic energy conservation error is proportional to the square of the time increment. By using this method, the conservation error of kinetic energy can be reduced to be sufficiently small.

In a collocated grid system, the characteristics of the kinetic energy conservation error differ from those of a staggered grid, even though the accurate scheme discretizes the convection terms. Specifically, the kinetic energy conservation error is proportional to the first power of the time increment. In the discretized governing equations under a staggered grid, the kinetic energy conservation error is described as a power function of the time-integration scheme for incompressible flows. This effect is caused by the discretized form of the pressure term in the kinetic energy equation under the collocated grid system. Higher-order time integration schemes are commonly used in numerical simulations of incompressible flows. Therefore, kinetic energy conservation errors for analyses using a collocated grid system are often larger than staggered-grid kinetic energy conservation errors.

2.2. Verification using the laminar Taylor-Couette flow

Before calculating the results for the flow around an airfoil, the numerical analysis method is first verified using a simple flow, namely, a two-dimensional laminar Taylor-Couette flow. In this type of flow, the velocity field can be described analytically. Therefore, the coordinate transformation in the numerical analysis can be verified. Although the governing equations used in the present study have a small magnitude for the viscous terms, these terms must be added to the equations in the case of the Taylor-Couette flow in order to derive the analytical solution. The boundary conditions of this two-dimensional laminar Taylor-Couette flow are shown in Figure 1. As shown in the figure, the inner boundary is set at the radius of $r = 1$, at which place the velocity components for the radial and circumferential directions are set to zero. Consequently, at the inner boundary, the non-slip condition is set. The outer boundary is set at $r = 2$. Here, the velocity in the circumferential direction is set to $u_\theta = 1$. In addition, the periodic boundary condition is set in the ring region, as shown in the figure. In the physical space, the number of grid points for both the radial and circumferential directions is fifty. Here, an unequally spaced grating is used for the radial direction. This computational region and the boundary conditions in the physical space are then transformed into the computational space. The boundary conditions in the computational space are shown in the figure. Here, the computational grid spacing in the computational space is uniform and unity.
Figure 2. Results for the verification using the laminar Taylor-Couette flow: (a) Grid spacing profile for the radial direction; (b) Verification results using the analytical velocity profile in the circumferential direction.

Figure 2(a) shows the profile of the grid spacing in the radial direction, and it can be seen that the finer grid spacing is used near the inner and outer boundaries. The present study considers that this numerical analysis can analyze the velocity profile of this Taylor-Couette flow with sufficient accuracy, even without setting the profile of the grid spacing to vary in the radial direction. On the other hand, as shown in the previous section, in order to analyze the flow around the airfoil with sufficient accuracy, a fine grid spacing is used near the surface of the airfoil. In order to validate this numerical analysis using a computational grid with such a grid spacing profile, a grid spacing profile that varies with respect to the radial direction is used, as shown in the figure. In the laminar Taylor-Couette flow, the velocity profile in the circular direction is analytically given. Figure 2(b) shows the results for the validation of this numerical analysis with coordinate transformation using this analytical velocity profile. As shown in the figure, the velocity profile of this analysis is in good agreement with the analytical solution, which thus validates the numerical analysis using coordinate transformation.

3. Numerical results and discussion

3.1. Computational grid and conditions

In the present study, the Kármán-Trefftz airfoil is used as a symmetric airfoil. The present aerofoil is two-dimensional. The flow around the airfoil can be given by the potential flow method. Additionally, the thickness of the airfoil is assumed to be 15% of the chord length, where the characteristic length of the present simulation is the chord length. In the analysis of the present research, the flow field is obtained by directly solving the governing equations as similar with [6]. The velocity field of the flow obtained herein is then compared to that of the potential flow obtained. The panel method is often used to study the aerodynamic characteristics of an airfoil. In the panel method, the free-stream is treated as a potential flow. On the other hand, the governing equations of the present study are the continuity and Navier-Stokes equations. From the perspective of the panel method, the present numerical results of this study are compared with those of the potential flow. In this analysis, the slip wall is set on the surface of the aerofoil. Therefore, this numerical simulation analyses the free-stream region, excluding the region of the boundary layer on the surface of the blade surface in the present analysis. Therefore, this analysis does not require the use of a turbulence model.
Figure 3. Schematic diagrams of the present computational grid: (a) Overall view of the present computational mesh. The airfoil is located at the center of the mesh; (b) Enlarged view around the airfoil. A Kármán-Trefftz airfoil with 15% thickness is used in the present study; (c) Enlarged view around the forward stagnation point for the case of AOA = 0 [deg], where $N \times N = 121 \times 121$.

Three types of the curvilinear grid system, i.e., O-type, C-type, and H-type systems, are generally used. In the present study, the slip condition is set on the surface of the airfoil, and it is therefore considered that the wake from the trailing edge is insignificant. Consequently, an O-type computational grid system is used to analyze the present flow around the airfoil. This grid system is constructed with a generally used method involving elliptic equations. A smooth curvilinear grid system can be constructed in this manner. Here, the method of Steger and Sorenson [14] is used to construct the computational grid system, in which $N_x \times N_y = 121 \times 121$. The number of grid points for the span direction is 5, and the time step is set to $2 \times 10^{-5}$. In this analysis, the kinetic energy conservation error is of the order of $O(\Delta t^2((\Delta V)^{1/3})^2)$, where $(\Delta V)^{1/3}$ is characteristic length of the grid spacing and $\Delta V$ is the volume of a computational cell. This conservation error is sufficiently small to be negligible because the accuracy order of the conservation error with respect to the time step is improved to the second order. In fact, despite the low spatial resolution of the present verification using the inviscid flow, the conservation error is negligibly small. When other numerical methods are used without these improved conservation characteristics, the effect of spatial resolution on the conservation error may not be negligible. In the collocated grid, the conservation error is considered to be of the order of $O(\Delta t((\Delta V)^{1/3})^2)$ in the use of this kind of numerical methods [8]. In addition, the Reynolds number is set to $10^7$. The value has been set as one of the practical Reynolds number values in this study. In this study, the slip wall is set on the surface of the aerofoil, and the computational region is set, excluding the turbulent boundary layer on the surface of the aerofoil. Therefore, this study has considered that the difference in set Reynolds number hardly affects the results and conclusion of this analysis unless the value of the Reynolds number is set excessively small.
Figure 3(a) shows a schematic diagram of the computational grid obtained in the present study. As shown in the figure, this airfoil is installed near the center of the computational domain. In addition, a fine computational grid is set near the airfoil. Here, a uniform inflow is set at the left-hand side of the domain, where the uniform inflow velocity is the characteristic velocity. Figure 3(b) shows a schematic diagram of the computational grid near the airfoil. As shown in the figure, the airfoil of the Kármán-Trefftz airfoil can be found. In addition, as shown in the figure, a fine calculation grid is set in the regions of the leading edge and the trailing edge region of the airfoil. Figure 3(c) shows the computational grid near the forward stagnation point under the computational condition, where the angle of attack is zero degrees.

3.2. Validation of the conservation laws

In this analysis, the kinetic energy can be conserved under the collocated grid system with sufficiently high accuracy. As shown in the previous section, the conservation error of the kinetic energy under the collocated grid system is on the order of the squares of the time step. A three-dimensional inviscid periodic homogeneous isotropic fluctuation field is used to examine the kinetic energy conservation characteristics of this analysis. In this flow, the initial kinetic energy is maintained to be constant over time due to the inviscid characteristics of the validating flow. The initial velocity field is obtained by taking the rotation of a normal white random number field and satisfies the equation of continuity. A three-dimensional periodic computational region having one side of $2\pi$ is discretized using $N^3 = 16^3$ grid points. Here, the grid spacing is uniform in three directions. The initial velocity field is dimensionless, so that its kinetic energy is unity. In the present study, the time step $\Delta t$ is set to 0.01. Here, $\Delta t$ is the time step for the dimensionless time $t$.

Figure 4(a) shows the time evolution of the spatial mean value of the kinetic energy. Here, this study uses a linear scale to show the result in Figure 4 in an easy-to-understand manner. As shown in the figure, the value of kinetic energy at each time deviates from the initial value as the time increases. This temporally evolving deviation is approximated by a linear function because the deviation is small. Additionally, the spatial mean value of the kinetic energy is in good agreement with the analytical characteristics, and the deviation of this value is sufficiently small. This result validates that the kinetic energy conservation characteristics of this numerical analysis are sufficiently high.
Figure 5. Profile of the lift coefficient as a function of angle of attack, AOA, where the theoretical linear profile is also shown in the figure.

Next, the flow around the airfoil is taken to be an incompressible flow, which has zero velocity divergence and is therefore equivalent to the continuity equation [15]. Figure 4(b) shows the time evolution of the continuity equation at the forward stagnation point of the flow around the airfoil. Here, the angle of attack is zero degrees under the calculation conditions of this flow. In this study, which increases the accuracy order of kinetic energy storage error, a numerical fluctuation may occur as an error. As shown in Figure 4(b), the amplitude of the numerical fluctuation increases with increasing time. Therefore, the set region of the time progression has been set in the range where the amplitude of the fluctuation is negligibly small. As shown in the figure, the absolute value of the continuity equation at the front stagnation point is smaller than \(10^{-7}\). The error of the continuity equation is so small that it is considered to be negligible. The equation of continuity describes the law of conservation of mass. Therefore, as shown in the figure, the mass conservation law for an incompressible flow is established with sufficient accuracy in this analysis. In this analysis, numerical analysis is performed up to \(t = 10\). Here, this time is a dimensionless time defined by using the inflow velocity and the chord length. The results of this analysis are shown at \(t = 10\). Therefore, we have considered that the convergence of the present simulation is sufficient.

3.3. Verification of an inviscid flow around an airfoil

Figure 5 shows the lift coefficient as a function of the angle of attack, AOA. This study considers that the lift characteristics for the angle of attack are very fundamental and well known, so that this characteristic is appropriate to verify the analysis of this study. The computational conditions for the flow around an aerofoil not to be separated should be set in the present simulation. Specifically, the angle of attack is set to 0 to 4 degrees. Following the thin airfoil theory, the lift coefficient is given as a function of the angle of attack, \(C_L = 2\pi \alpha\), where \(\alpha\) is the angle of attack. The above relationship holds only in the range of small angle of attack, where flow separation does not occur. This relationship is shown in the figure. As shown in the figure, the result of the lift coefficient obtained agrees well with the theoretical linear relationship, thus validating the accuracy of the numerical analysis. The results of pressure coefficient, \(C_p\), agree well with those of previous studies (not shown because of limitations of space). The results of \(C_p\) may be less sensitive to kinetic energy conservation errors. As shown in the previous numerical work, even if the kinetic energy conservation error occurs due to the use of numerical viscosity, the numerical results of \(C_p\) can agree with the experimental result [16].
Figure 6. Velocity vector field around the forward stagnation point for AOAs of 0 and 2 [deg]: The red and green arrows indicate the velocity vectors of the present results and the potential flow, respectively.

Figure 6 shows the velocity vector field near the forward stagnation point, where the angle of attack for this result is taken to be zero degrees. The results obtained using the governing equations are compared to the results for the potential flow. As shown in the figure, in the region upstream from the stagnation point, there is little difference between the velocity fields obtained using the governing equations and those for the potential flow. The small difference that does exist can be found in the results for both angles of attack. On the other hand, in the region downstream from the forward stagnation point, the velocity field is found to be different from that due to the potential flow. This difference is found in the near region of the airfoil surface. In addition, these results are found for both angles of attack. Since the velocity is obtained by solving the governing equations, the vorticity of the flow field need not be zero, as opposed to that for the potential flow. This difference may in turn have caused the difference in the velocity field in the region near the surface of the airfoil.

In this study, the flow around an aerofoil with the slip wall on the surface of the aerofoil is analysed. Therefore, this analysis is different from a numerical analysis in which the spatial resolution of the boundary layer on the surface can affect numerical results. Also, the kinetic energy conservation error in this study is of the order of \( O(\Delta t^2 ((\Delta P)^{1/2})) \). The kinetic energy conservation error in this study has a higher accuracy order with respect to time step, so the conservation error can be negligibly small even if the spacial grid spacing is not sufficiently small. This negligible magnitude of the conservation error is verified using the inviscid flow.

4. Conclusions
The purpose of the present research is to validate the numerical analysis by directly analyzing the governing equations for a pseudo-inviscid flow around a symmetrical airfoil. A collocated grid system was used to analyze this flow around an airfoil. In the numerical analysis, the conservation characteristics of the kinetic energy were enhanced to the second-order accuracy with respect to the time step. This improvement in the kinetic energy conservation characteristics was achieved by using the pressure increment solved in the fractional steps. In the present study, the Kármán-Trefftz airfoil with a thickness of 15% was used. The velocity field obtained by this numerical analysis was also
compared with that of the potential flow around the Kármán-Trefftz airfoil using an O-type computational grid.

First, the coordinate transformation of this numerical analysis was verified using the laminar Taylor-Couette flow. The kinetic energy conservation characteristics of this numerical analysis were validated using a three-dimensional homogeneous isotropic fluctuation field. The establishment of the law of conservation of mass in the flow around the airfoil was validated using the time evolution of the velocity divergence. The velocity field near the leading edge of this flow was compared with that of the potential flow. Furthermore, the profile of the lift coefficient obtained by this analysis was shown as a function of the angle of attack and was consistent with the linear characteristics. This agreement validates the present numerical analysis method for the flow around an airfoil. In this study, the conservation error of kinetic energy is so negligibly small that the spatial resolution dependence of the conservation can be negligible. In this research, the slip wall to be set on the surface of the aerofoil. Therefore, this analysis does not analyse the turbulent boundary layer on the surface of the aerofoil. An analysis that includes the analysis of the turbulent boundary layer on the aerofoil surface is future work.

Acknowledgements
The authors acknowledge Professor Y. Morinishi (Nagoya Institute of Technology) and Associate Professor T. Ushijima (Nagoya Institute of Technology) for their valuable comments on this study. The present study was supported in part by the Japanese Ministry of Education, Culture, Sports, Science and Technology through Grants-in-Aid (Nos. 18K03932 and 18H01369).

References
[1] Rhie CM and Chow WL 1983 Numerical study of the turbulent flow past an airfoil with trailing edge separation AIAA J. 21 1525-32
[2] Mary I and Sagaut P 2002 Large eddy simulation of flow around an airfoil near stall AIAA J. 40 1139-45
[3] Rosti ME, Omidyeganeh M and Pinelli A 2016 Direct numerical simulation of the flow around an aerfoil in ramp-up motion Phys. Fluids 28 025106
[4] Kajishima T, Ohta T, Okazaki K and Miyake Y 1998 High-order finite-difference method for incompressible flows using collocated grid system JSME Int. J. Ser. B 41 830-839
[5] Morinishi Y, Takemura M and Nezu I 2002 Investigation on computational scheme for mac methods with collocated grid system (in Japanese) Doboku Gakkai Ronbunshu 719 11-19
[6] Felten FN and Lund TS 2006 Kinetic energy conservation issues associated with the collocated mesh scheme for incompressible flow J. Comput. Phys. 215 465-484
[7] Lardeau S, Leschziner M and Zaki T 2012 Large eddy simulation of transitional separated flow over a flat plate and a compressor blade Flow, Turb. Combust. 88 19-44
[8] Morinishi Y, Lund T S, Vasilyev OV and Moin P 1998 Fully conservative higher order finite difference schemes for incompressible flow J. Comput. Phys. 143 90-124
[9] Ranjan R and Pantano C 2013 A collocated method for the incompressible Navier–Stokes equations inspired by the Box scheme J. Comput. Phys. 232 346-382
[10] Suzuki H, Nagata K, Sakai Y and Hayase T 2010 Direct numerical simulation of turbulent mixing in regular and fractal grid turbulence Phys. Scr. T142 014065
[11] Suzuki H, Nagata K, Sakai Y, Hayase T, Hasegawa Y and Ushijima T 2013 Direct numerical simulation of fractal-generated turbulence Fluid Dyn. Res. 45 061409
[12] Suzuki H, Nagata K, Sakai Y, Hayase T, Hasegawa Y and Ushijima T 2013 An attempt to improve accuracy of higher-order statistics and spectra in direct numerical simulation of incompressible wall turbulence by using the compact schemes for viscous terms Int. J. Numer. Meth. Fluids 73 509-522
[13] Morinishi Y 1999 An Improvement of collocated finite difference scheme with regard to kinetic energy conservation (in Japanese) Trans. JSME Ser. B. 65 505-512
[14] Steger J L and Sorenson R L 1979 Automatic mesh-point clustering near a boundary in grid generation with elliptic partial differential equations J Comput. Phys. 33 405-410

[15] Suzuki H, Hattori S and Mochizuki S 2017 Numerical investigation using an exact solution of the effects of non-solenoidality of the viscous terms on the incompressible flow J Fluid Sci. Tech. 12 Paper No.16-00257

[16] Suzuki M 2006 Investigation into accuracy of CFD for flow around isolated airfoil (in Japanese) Turbomach. 34 366-373