UNSTRUCTURED MOVING-GRID FINITE-VOLUME METHOD FOR UNSTEADY SHOCKED FLOWS

M. Yamakawa[1*] and K. Matsuno[2]

Unstructured grid system is suitable for flows of complex geometries. For problems with moving boundary walls, the grid system must be time-dependently changing and deforming according to the movement of the boundaries when we use a body fitted grid system. In this paper, a new moving-grid finite-volume method on unstructured grid system is proposed and developed for unsteady compressible flows with shock waves. To assure geometric conservation laws on moving grid system, a control volume on the space-time unified domain is adopted for estimating numerical flux. The method is described and applied for two-dimensional flows.

Keywords: Compressible Flow, Unsteady Flow, Unstructured Mesh, Moving Boundary Problem

1. INTRODUCTION

Recently, unsteady flow problems are briskly researched in CFD, since steady flow problems have reached the level of practical use. In unsteady flow problems, moving boundary problems have high needs. Especially, flows around a body which changes its shape with time, or bodies penetrating problem such as a flow around docking or separating bodies, are very interesting. The most popular method of calculating such moving boundary problems is an overset grid method.[1] This method, however, breaks the conservation laws at exchanging flow variable between a main grid and a sub grid. Thus, when we deal with such problems, we have to overcome two important issues. The one is for numerical methods. It is important for methods to satisfy a geometric conservation laws when the grid is moving and deforming at every time-step. It is essential for simulations of compressible flows to satisfy both geometric conservation laws and physical conservation laws. Otherwise we cannot obtain the correct Rankin-Hugoniot relation. As for this issue, we proposed a new method, "Moving-Grid Finite-Volume Scheme,"[2] which adopts a control volume in the space-time unified domain. The method is implicit and is solved iteratively at every time-step in order to assure both the geometric conservation laws and numerical accuracy. Another issue to be overcome is grid system for complicated geometries. It is especially difficult to generate a single body-fitted grid in the case of mergence of two bodies. An unstructured grid system is flexible and thus suitable to such problems.

The purpose of this paper is to present a new finite-volume method on the moving unstructured grid system. The present method combines the moving-gird finite-volume method with unstructured grid system and introduces a new algorithm, which permits addition and/or elimination of the grid cells under the condition of geometric conservation laws.

This paper is composed of 5 sections. Section one is the introduction. In section two, the governing equation is described. Section three the efficiency of this method when applied to a two-dimensional test problem and simple piston problem are described. In section four the penetration problem with addition and elimination of the calculation elements is developed. Finally in section five we give the conclusion.

Received: July 9, 2004, Accepted: February 25, 2005.
1 Kyoto Institute of Technology, Kyoto 606-8585, JAPAN.
2 Kyoto Institute of Technology, Kyoto 606-8585, JAPAN.
* Corresponding author. Email: yamakawa@kit.ac.jp

2. UNSTRUCTURED MOVING-GRID FINITE VOLUME METHOD

2.1 GOVERNING EQUATION

The two-dimensional Euler equation can be written in the conservation law form as follow:

$$\frac{\partial q}{\partial t}+\frac{\partial E}{\partial x}+\frac{\partial F}{\partial y}=0 \tag{1}$$

where

$$q=\begin{pmatrix}\rho\\ \rho u\\ \rho v\\ e\end{pmatrix},E=\begin{pmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ u(e+p)\end{pmatrix},F=\begin{pmatrix}\rho v\\ \rho uv\\ \rho v^2+p\\ v(e+p)\end{pmatrix} \tag{2}$$

The unknown variables ρ, u, v, and e represent the density, velocity components in the x and y directions, and total energy per unit volume, respectively. The working fluid is assumed to be perfect, and the pressure p is defined by

$$p=(\gamma-1)\left\{e-\frac{1}{2}\rho\left(u^2+v^2\right)\right\} \tag{3}$$

where the ratio of specific heats is typically 1.4.

2.2 NUMERICAL ALGORITHM

When body-wall boundaries move and change their relative locations, the body-fitted grid system must dynamically change and deform its shape according to the movement of the wall boundaries. In this case, it is important to assure the geometric conservation laws at every time step. Thus, we adopt a control volume on the space-time unified domain (x, y, t), which is three-dimensional for two-dimensional flows, in order that the method satisfies the geometric conservation laws. Now Eq.(1) can be written in divergence form in the space-time unified domain:

$$\tilde{\nabla}\tilde{F}=0 \tag{4}$$

where

$$\tilde{F}=E\vec{e}_x+F\vec{e}_y+q\vec{e}_t\ , \tag{5}$$

$$\tilde{\nabla}=\vec{e}_x\frac{\partial}{\partial x}+\vec{e}_y\frac{\partial}{\partial y}+\vec{e}_t\frac{\partial}{\partial t} \tag{6}$$

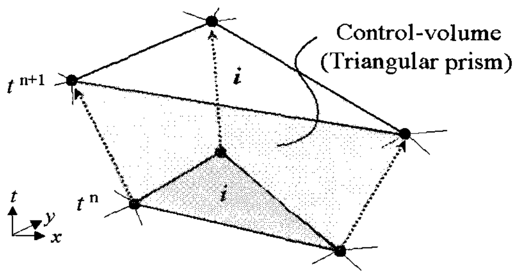

Fig. 1 Control volume in space-time unified space

Here $\vec{e}_x$, $\vec{e}_y$ and $\vec{e}_t$ are unit vectors in x, y and t directions respectively.

The present method is based on a cell-centered finite-volume method and, thus, we define flow variables at the center of cell in unstructured mesh. The control volume becomes a triangular prism in the (x, y, t)-domain as shown in Fig. 1.

We apply volume integration to Eq.(4) with respect to this control volume. Then Eq.(4) can be written in surface integral form as, using the Gauss theorem:

$$\int_V \tilde{\nabla}\tilde{F}dV=\oint_S \tilde{F}\cdot ndS=\sum_{l=1}^{5}\left(En_x+Fn_y+qn_t\right)_l=0 \tag{7}$$

where n is a outward unit normal vector of control volume surface. V is the triangular prism control volume and S is its boundary. $n_l=(n_x, n_y, n_t)_l$ (l = 1,2,...5) is the normal vector of control volume surface, and the length of the vector equals to the area of the surface. The upper (l = 5) and bottom (l = 4) surfaces of the control volume are perpendicular to t-axis, and therefore they have only n_t component and correspond to the areas in the (x, y)-space at time t^{n+1} and t^n, respectively. Thus, Eq.(7) can be expressed as,

$$q^{n+1}(\mathbf{n}_t)_5+q^n(\mathbf{n}_t)_4 + \sum_{l=1}^{3}\left\{\left(E^{n+1/2},F^{n+1/2},q^{n+1/2}\right)\cdot\mathbf{n}\right\}_l=0 \tag{8}$$

Here, the conservative variable vector and flux vector at (n+1/2)-time step are estimated by the average between n-time and (n+1)-time steps. Thus, for example, $E^{n+1/2}$ can be expressed as,

$$E^{n+1/2}=\left(E^n+E^{n+1}\right)/2 \tag{9}$$

The flux vectors are evaluated using the Roe flux difference splitting scheme[4] with MUSCL approach, as well as the Venkatakrishnan limiter [5].

The method uses the variable at (n+1)-time step, and thus the method is completely implicit. We introduce sub-iteration strategy with a pseudo-time approach[3] to solve the implicit algorithm. Now by defining that the operating function $L(q^{n+1})$ as Eq.(10), the pseudo-time sub-iteration is represented as Eq.(11).

$$L\left(q^{n+1}\right)=\frac{1}{\Delta t(\mathbf{n}_t)_5}[q^{n+1}(\mathbf{n}_t)_5+q^n(\mathbf{n}_t)_4+\sum_{l=1}^{3}\left\{\left(E^{n+1/2},F^{n+1/2},q^{n+1/2}\right)\cdot\mathbf{n}\right\}_l] \quad (10)$$

$$\frac{dq^{n+1(v)}}{d\tau}=-L\left(q^{n+1(v)}\right) \quad (11)$$

where v is index for iteration, and is pseudo-time step. To solve Eq.(11), we adopted the Runge-Kutta scheme to advance pseudo-time step. When inner iteration is converged, we can get (n+1)-time step solution, q^{n+1}.

3. NUMERICAL EXAMPLE

3.1 TEST PROBLEM

At first it is necessary to check that the method satisfies the geometric conservation laws, when the grid is moving and deforming. We try to calculate a uniform flow on moving and deforming grid. At this test problem, boundary points are fixed and inner points (x_i, y_i) are moved from initial grid (x_i^0, y_i^0) by Eq.(12-14).

$$x_i^n = x_i^0 + 0.3\Delta s\cos\theta^n, \quad (12)$$

$$y_i^n = y_i^0 + 0.3\Delta s\sin\theta^n, \quad (13)$$

$$\theta^n = \frac{\pi}{2}\left(x_i^0+y_i^0\right)\frac{3}{40}n \quad (14)$$

where Δs is the shortest grid spacing in the initial grid. n is time step. The calculation domain is a square of unit length. The initial condition is uniform flow: $\rho_\infty = 1.0$, $p_\infty = 1.0/\gamma$ (γ=1.4), u_∞ = 1.0, v_∞ = 1.0.

Fig. 2 shows the movement of the grid at each times (t = 0.0 : initial grid). We calculate until t = 10.0 (1000 time step with Δt = 0.01).

Fig. 3 shows the history of L2-Error of density.

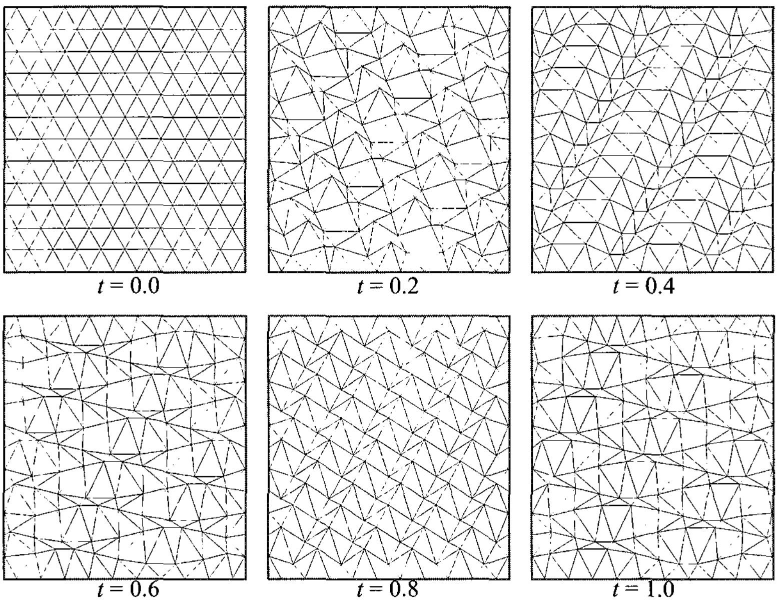

Fig. 2 Moving grid

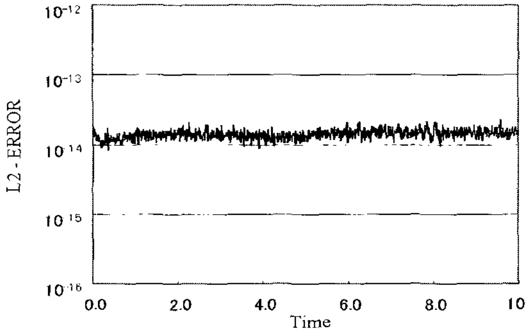

Fig. 3 History of L2-ERROR of density

The order of L2-Error is 10^{-14}, which is machine zero. And we had gotten the same results on pressure and velocity. Thus they prove that this method catch the uniform flow perfectly and satisfy a geometric conservation laws even if the grid is moving and deforming.

Here, L2 Error is defined as;

$$\text{L2-ERROR} = \sqrt{\frac{\sum_{i=1}^{i_{max}} (\rho - \rho_\infty)_i^2}{i_{max}}} \quad (15)$$

3.2 VALIDATION OF THE METHOD

The method is applied to a two-dimensional piston problem with shock wave and compared with theoretical data. Fig. 4 illustrates the problem. The initial length is 1.5 times of the height, and a piston is traveled toward the other end-wall of the cylinder. The piston begins to move at time t = 0 and accelerates at constant rate of 10.0 up to the time t = 0.1 and then keeps the constant speed. The initial condition of density, pressure, velocity components in the x and y directions are given by:

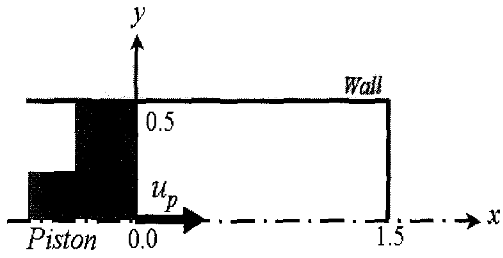

Fig. 4 Outline of Piston problem

$\rho = 0.0$, $p = 1.0/\gamma$ ($\gamma = 1.4$), $u = 0.0$, $v = 0.0$. And we calculate until t = 1.2 (4800 time step with $\Delta t = 0.00025$).

The initial grid was generated by Delaunay Triangulations.[6] The number of elements is 3833. Due to the movement of the piston, the grid is deformed in the x-direction only by fixed rate. Thus, the total number of the element is constant at every time step. Fig. 5 shows the result of the flow field and grid at t = 0.4 and 0.8. We can see the flow field with traveling shock wave on moving unstructured grid.

Fig. 6 shows the comparison of the computed shock position and exact solution. The error of the computation from the exact solution is in 0.9 – 1.3%. Thus, the result shows that the method calculates the flow field accurately.

4. APPLICATION TO COMPLICATED PROBLEM

In this section, we will try to calculate a penetration problem of two solid bodies in supersonic flow. In this case, it is necessary to

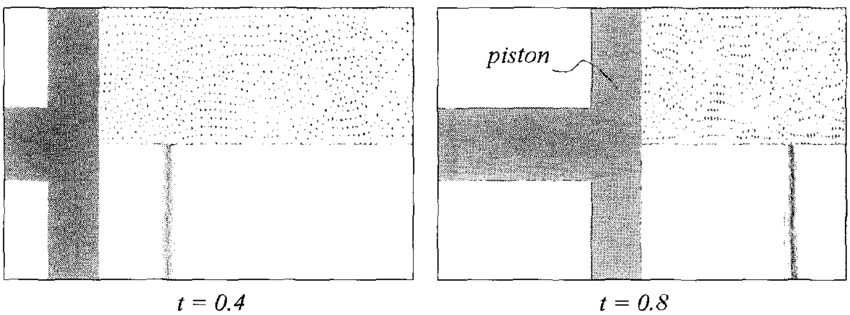

Fig. 5 Piston problem with shock wave on moving unstructured grid
(Upper : Deforming grid, Lower : Pressure contours)

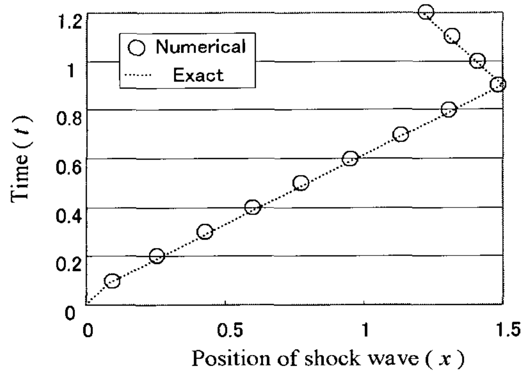

Fig. 6 Comparison of the position of shock wave

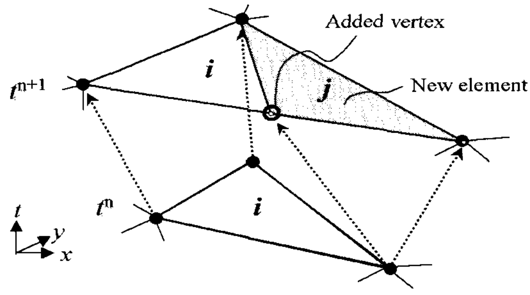

Fig. 7 Control volume with addition of element

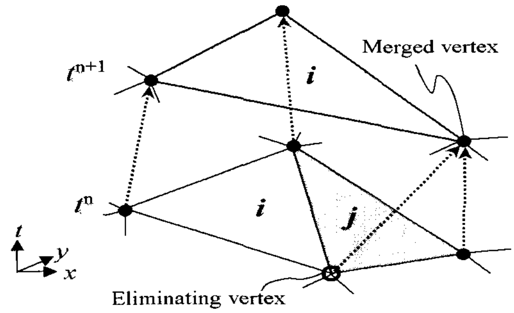

Fig. 8 Control volume with elimination of element

deform the flow region according to the movement of the body. Thus, not only movement of grid but also eliminating and addition of calculation elements must be taken into consideration.

4.1 UNSTRUCTURED MOVING-GRID FINITE-VOLUME METHOD WITH ELIMINATING AND ADDING ELEMENTS

The present moving-grid finite-volume method includes new algorithm for eliminating and adding elements according to a change of geometry. The method can also make it possible to calculate a flow around body such that the body is penetrated into another body. For the present method, the control volume for discretization is considered in the space-time unified space, and thus it can assure both physical and geometrical conservation laws even if the two of the elements are merged or a new element is added.

At the case of addition of a element, we will divide an element into two elements by cutting on grid line. Fig. 7 shows a control volume of a triangular prism that is formed by a element at n step and elements at n+1 step, which an element i is divided into an element i and a new element j by added vertex on grid line.

In order to assure the geometric conservation laws as well as physical conservation law, it is necessary to apply individually the finite-volume method to control volumes of the pentahedron i and the tetrahedron j, as shown in Fig. 7. Thus, for the pentahedron i, Eq.(8) is applied. While, for the tetrahedron j, Eq.(8) is replaced as following equation, since there is no bottom surface($(n_t)_4 = 0$) of a control volume,

$$q^{n+1}(n_t)_5 + \sum_{l=1}^{3}\left\{\left(E^{n+1/2},F^{n+1/2},q^{n+1/2}\right)\cdot n\right\}_l = 0 \qquad (16)$$

On the other hand, at the case of elimination of a

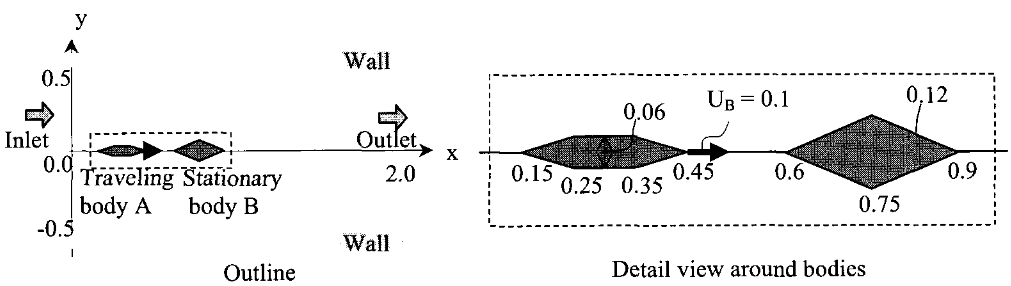

Fig. 9 Outline of penetration problem

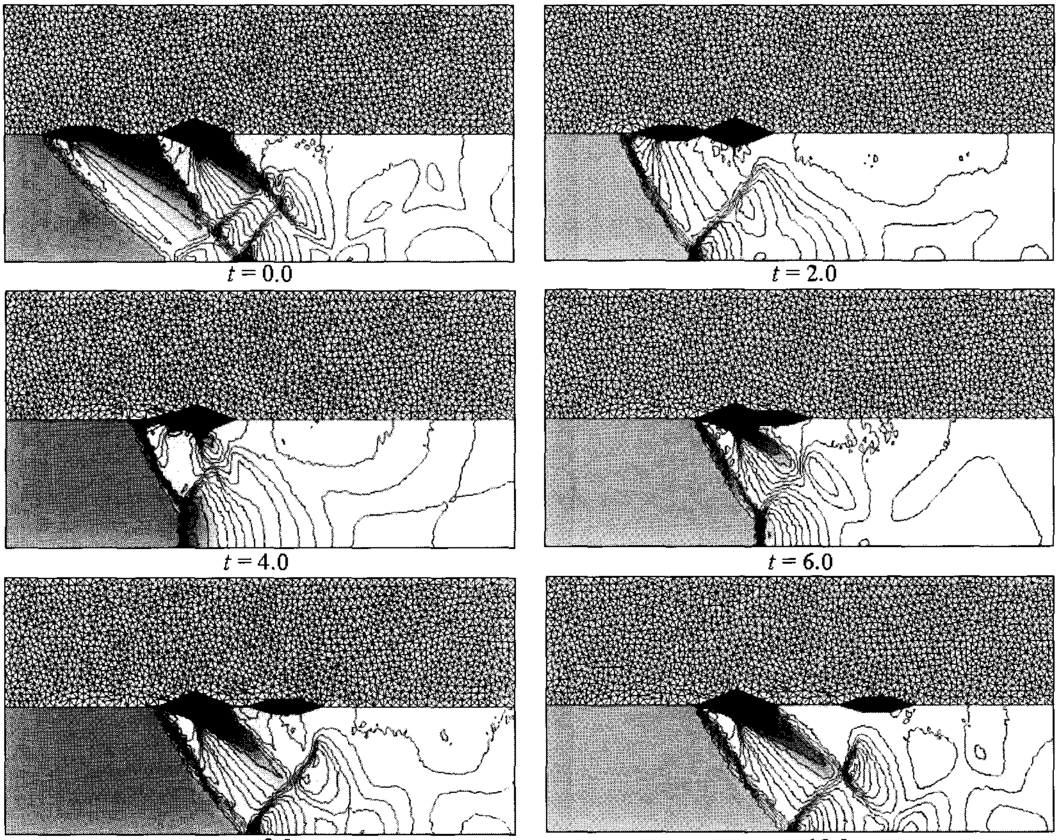

Fig. 10 Penetration problem in supersonic flow (Upper : Deforming grid, Lower : Pressure contours)

element, we consider that the elements of next doors is combined. Fig. 8 shows a control volume in the case of elimination of element, in which a vertex on line of the elements of next doors is merged.

At this case, we consider the bottom surface of the control volume at time tn to be sum of the elements i and j of the hexahedron. Thus, Eq.(8) is replaced as;

$$q_i^{n+1}\{(n_t)_6\}_i + q_i^n\{(n_t)_5\}_i + q_j^n\{(n_t)_5\}_j + \sum_{l=1}^{4}\{(E^{n+1/2}, F^{n+1/2}, q^{n+1/2})\cdot n\}_l = 0 \qquad (17)$$

For this new algorithm, we have proved the method to satisfy the geometric conservation low using same type of test problem of Fig. 2, with eliminating and adding grid elements.

4.2 APPLICATION TO BODY-PENETRATION PROBLEM

The method was applied to a penetration problem where two solid bodies travel in supersonic flow and the body penetrates another body (Inlet mach number is $M_\infty = 2.0$). Fig. 9 illustrates the model. The traveling solid body A is placed on an upstream side in the duct, and the solid body A is traveled toward the another stationary solid body B by a constant speed ($U_B = 0.1$). The steady flow is given as an initial condition. And we calculate until t = 10.0 (20000 time step with $\Delta t = 0.0005$).

We assume that each shapes of these bodies are not changed when the body A penetrates the body B. In the other word, properties of bodies are not elasticity or plasticity. This means two bodies are only overlap.

Fig. 10 shows the result of the flow field and related grids at t = 0.0, 2.0, 4.0, 6.0, 8.0 and 10.0 respectively. The traveling body A penetrated the stationary body B. Then shock waves moved with body motion, and it shows a complex flow field. As the body A approaches to the stationary body B (from t = 0.0 to 2.0), we can see that the five grid elements between two bodies are eliminated. When the body separates into two parts (from t = 8.0 to 10.0), added new ten elements can be seen.

5. CONCLUSION

In this paper, a new moving-grid finite-volume method on unstructured grid system with the new algorithm of elimination and addition of the grid point has been proposed and applied to the unsteady compressible flows with shock waves. With the simple test problem, it was confirmed that the method assures both geometric conservation laws and physical conservation laws on moving gird system. The result of the piston problem showed that the method calculates the flow field accurately and the position of the shock wave is correct. The present approach was confirmed applicable for complicated problems, for instance, body penetration problem, in which addition and/or elimination of the grid cells were conducted under the condition of geometric conservation laws. Therefore, we can conclude that the unstructured moving-grid finite-volume method is effective to calculate complicated moving boundary problems.

REFERENCES

[1] Steger, J.L., Dougherty, F.C., Benek, J.A., Ghia, K.N. and Ghia, U., 1983, "A Chimera Grid Scheme; Advances in Grid Generation," *ASME FED-5*, pp.59-69.

[2] Matsuno, K., Mihara, K. and Satofuka, N., 2001, "A Moving-Mesh Finite-Volume Scheme for Compressible Flows with Traveling Wall Boundaries," *Computational Fluid Dynamics 2000*, Springer July, pp.705-710.

[3] Rumsey, C.L., Sanetrik, M.D., Biedron, R.T., Melson, N.D., and Parlette, E.B., 1996, "Efficiency and Accuracy of Time-Accurate Turbulent Navier-Stokes Computations," *Computers and Fluids*, Vol.25, pp.217-236.

[4] Roe, P.L., 1981, "Approximate Riemann Solvers Parameter Vectors and Difference Schemes," *Journal of Computational Physics.*, Vol.43, pp.357-372.

[5] Venkatakrishnan, V., 1993, "On the Accuracy of Limiters and Convergence to Steady State Solutions," *AIAA Paper*, pp.93-0880.

[6] Bryant, C.F., 1985, "Two Dimensional Automatic Triangular Mesh Generation," *IEEE Transactions on Magnetics,* Vol.MAG-21, No.6, pp.2547-2550.