Unstructured Moving-Grid Finite-Volume Method for Unsteady Shocked Flows

M. Yamakawa [1], *K. Matsuno* [2]
1. Kyoto Institute of Technology, Kyoto 606-8585, JAPAN. e-mail:yamakawa@ipc.kit.ac.jp
2. Kyoto Institute of Technology, Kyoto 606-8585, JAPAN. e-mail:matsuno@ipc.kit.ac.jp

Corresponding author ***M. Yamakawa***

Abstract

Unstructured grid system is suitable for flows of complex geometries. For problems with moving boundary walls, the grid system must be changed and deformed with time if 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.

Keyword: *Compressible Flow, Unsteady Flow, Unstructured Mesh, Moving Boundary Problem*

1. Introduction

Recently, unsteady flow problem is briskly researched in CFD, since steady flow problems have reached the level of practical use. In unsteady flow problems, flows around body which is changing its shape and thus the boundary is also moving with time are very interesting. When dealing with such moving boundary 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 every moment. 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 scheme, "Moving-Grid Finite-Volume Scheme", 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 boundaries. 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[1] and unstructured grid system, which permits mergence and/or elimination of the grid cells under the condition of geometric conservation laws.

2. Unstructured Moving-Grid Finite-Volume Method

When body-wall boundaries move and change their relative positions, the body-fitted grid system must dynamically change and deform its shape according to the movement of the wall boundary. 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. 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 estimate numerical flux in this control volume. The scheme uses the variable at (n+1)-time step, and thus the scheme is completely implicit. We introduce subiteration strategy with a pseudo-time approach [2] to solve the implicit algorithm. Then flux vectors are evaluated using the Roe flux difference splitting scheme [3] with MUSCL approach, and the Venkatakrishnan limiter [4].

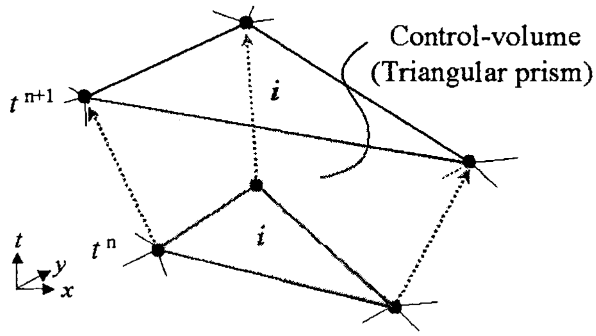
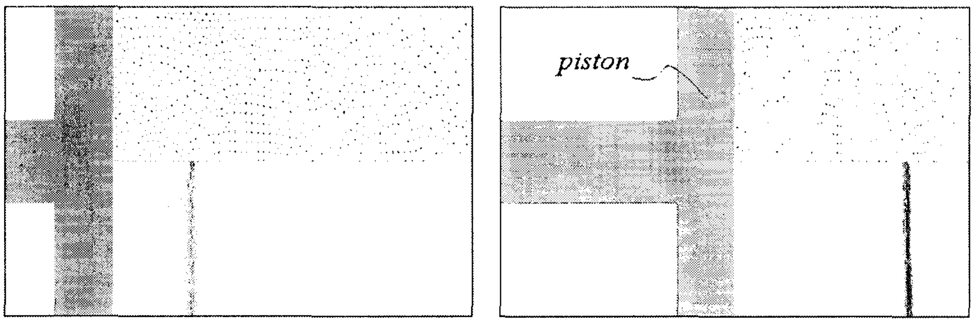


Fig.1. Control volume in space-time unified space

3. Numerical Example

Figure 2 shows an example of a two-dimensional piston problem with shock wave. 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 up to the time t=0.1 and keeps the constant speed after that. The initial grid is generated by Delaunay Triangulations [5]. According to the motion of the piston, the grid is deformed in the x-direction only by the fixed rate. Thus, the number of the element is constant at every time step. Figure 2 shows the result of the flow field and grid. The result shows that the method calculates the flow field accurately. We have applied the method to some of complicated problems and the results have shown a promising feature of the method. We will present these results in the full paper.



$t = 0.4$ $\qquad$ $t = 0.8$

Fig.2. Piston problem with shock wave on moving unstructured grid
(Upper : Deforming grid, Lower : Pressure contours)

References

[1] K. Matsuno, K. Mihara, and N. Satofuka, "A Moving-Mesh Finite-Volume Scheme for Compressible Flows with Traveling Wall Boundaries", *Computational Fluid Dynamics 2000,* Springer July, (2001) pp705-710.

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

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

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

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