A NEW CALCULATION METHOD OF COMPRESSIBLE FLOW

Hiroaki Nakamura*, Keiji Manabe, Masatomi Nishio****
***Graduate Student. Fukuyama Univ., **Fukuyama Univ.**

Keywords: *Supersonic Flow, Hypersonic Flow, Finite Element Method, MESUR Capsule, and Flowfield Stabilization Time*

Abstract

A new calculation method of supersonic/hypersonic flows based on the finite element method (FEM) has been developed. In this algorithm, the pressure term of Navier-Stokes equations was included in the flux vector expressing by the stress tensor. By doing this scheme, the pressure term could be expressed as a non-differential form. This treatment made it possible to calculate supersonic/hypersonic flowfields with little discrepancies and good stability, compared with previous simulations by the FEM. The correctness of the calculated results was confirmed by comparing shock shapes obtained by the schlieren system and by comparing streamlines obtained by the electric discharge method. As an example of this new computational method, the stabilization time of the wake produced by the hypersonic MESUR capsule model is investigated.

1 Introduction

It is very important to clarify the flowfield phenomena around supersonic/hypersonic vehicles for the plane of new vehicles and space development.[1],[2] Therefore, theoretical and experimental studies have been carried out by many researchers, and numerical analyses by means of the finite difference method (FDM) and the finite volume method (FVM), etc., have also been developed.

In the calculation of a flowfield with shock waves, artificial viscosity is usually employed to enhance stability and to avoid divergence. However, excessive artificial viscosity often gives a value far smaller than the correct solution.

To address these flaws, a new algorithm for compressible flow based on the FEM was developed.[3] The merit of this new method is to introduce a pressure term of non-differential form in the weighted residual equation. Okida, J. and Manabe, K.,[4] et al., tried this treatment for the dynamic-explicit FEM in the deformation analysis of a solid body. By the use of this method for the analysis of a compressible flowfield with shock waves, more stable computation with less numerical error became available in the calculation of a flowfield with artificial viscosity smaller than that of conventional FEM analyses.

The validity of this method was confirmed by comparing the results with the shock wave obtained by the schlieren method and the streamline obtained by the electric discharge method[5-8] for a hypersonic flow at Mach 10. In this comparison, the MESUR capsule model was used.

2 FEM Formulation of Basic Equations[3]

The Navier-Stokes equations of compressible flows can be written in the conservation form

$$\frac{\partial \mathbf{U}}{\partial t}+\left(\frac{\partial \mathbf{F}_1}{\partial x}+\frac{\partial \mathbf{F}_2}{\partial y}+\frac{\mathbf{F}_2'}{\underline{y}}\right)-\left(\frac{\partial \mathbf{G}_1}{\partial x}+\frac{\partial \mathbf{G}_2}{\partial y}+\frac{\mathbf{G}_2'}{\underline{y}}\right)=0, \qquad (1)$$

where **U** is the vector of conservation variables. The particular form of the variables for Eq.(1) becomes as follows:

$$\mathbf{U}=\begin{bmatrix}\rho\\ \rho u_x\\ \rho u_y\\ \rho e\end{bmatrix},\quad \mathbf{F}_i=u_i\mathbf{U}=\begin{bmatrix}u_i\rho\\ u_i\rho u_x\\ u_i\rho u_y\\ u_i\rho e\end{bmatrix},$$

$$\mathbf{G}_i=\begin{bmatrix}0\\ \tau_{ix}-p\delta_{ix}\\ \tau_{iy}-p\delta_{iy}\\ \sum_m u_m(\tau_{mi}-p\delta_{mi})-q_i\end{bmatrix} \tag{2}$$

and

$$\mathbf{F}_2{}'=\mathbf{F}_2,$$

$$\mathbf{G}_2{}'=\begin{bmatrix}0\\ \tau_{xy}\\ \tau_{yy}-\tau_{\theta\theta}\\ u_x\tau_{xy}+u_y(\tau_{yy}-p)-q_y\end{bmatrix}. \tag{3}$$

In the above ρ is the fluid density, u_i represents the velocity components, e is the total energy per unit mass, q_i represents the components of a heat flux vector, δ_{ij} are the Kronecker deltas and τ_{ij} are the components of the deviatoric stress tensor.

After multiplication of Eq.(1) by the weighting function w, integrating over the domain A and applying of the Gauss's divergence theorem, the pressure term in the Navier-Stokes equations is treated as a non-differential form of a weighted residual equation.

Subsequently, by using the explicit scheme and diagonalization of a matrix, the momentum conservation law directly shows the relation between the equivalent force and the acceleration of the nodal points. Namely, it can be analyzed as a Newtonian equation of motion. By obtaining this relation, better stability and less error can be obtained in the calculations of the FEM.

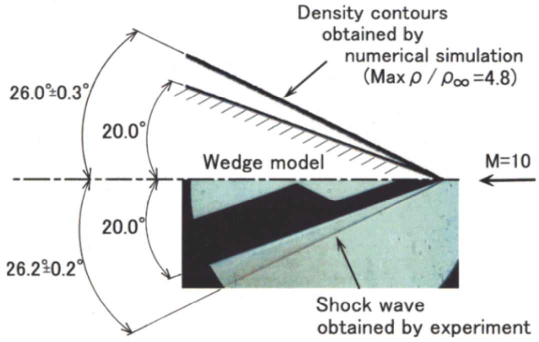


Fig.1 Comparison between the density contours near the wedge model obtained by the new calculation method and the shock wave obtained by the schlieren method at M=10.

3 Check of Calculated Shock Wave Over Wedge

To confirm the correctness of the present calculation method, the flowfield around a wedge model was calculated, and was compared with the visualized result obtained by the schlieren method. The angle of attack of the wedge model was 20-deg. In this simulation, the 4-node quadrilateral element was used. the computational grid consisted of 210×90 nodal points, that is, the number of nodes was 19201, and the number of elements was 18900. To compare the result with the experimental one utilizing a hypersonic gun tunnel, the simulation was carried out under these conditions: the Mach number was M=10, the freestream density was ρ=4.5×10^{-3}kg/m^3, static pressure was p=70Pa, static temperature was T=54K, and freestream velocity was V=1500m/s. These value were the same as the experimental ones. For the boundary condition, the non-slip condition was applied for the surface of the model. The comparison between the calculated result and visualized one obtained by the schlieren method is shown in Fig.1. As shown in Fig.1, in the case of the calculated result, the shock wave angle was 26.0±0.3 degrees. In the case of the experimental result, the shock wave angle was 26.2±0.2 degrees. Judging from this, it can be concluded that the calculated shock wave and visualized one were in very good agreement.

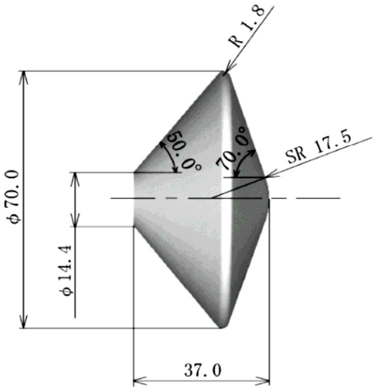


Fig.2 Model dimensions of a MESUR capsule used for the calculation and the experiment in mm.

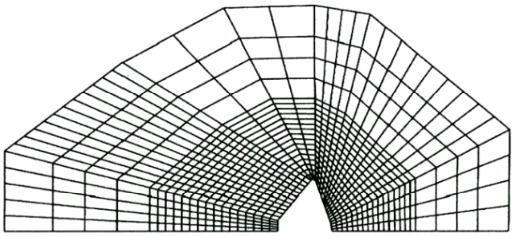
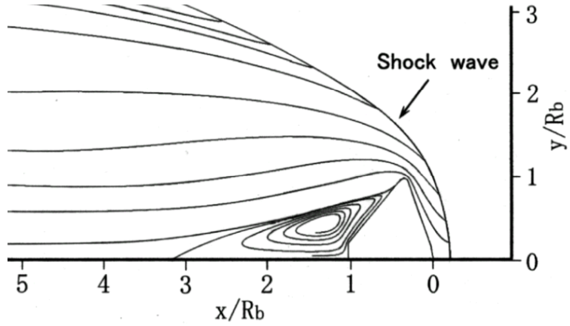
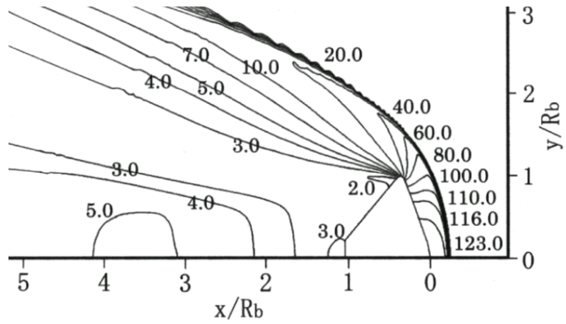


Fig.3 Computational grid around the capsule. (228×400)



(a) Streamline



(b) Pressure contours. $(P/P_\infty)_{max}$=123.0.

Fig.4 Calculated results of the Flowfield around the capsule obtained by the new calculation method at M=10. Rb:capsule radius.

4 Flowfield Calculation Around MESUR Capsule

The flowfield around a MESUR capsule traveling at a hypersonic speed of Mach 10 was calculated. By using the above idea, the flowfield around the MESUR capsule was calculated. The model shape is shown in Fig.2. The computational grid is shown in Fig.3. The computational grid consisted of 228×400 nodal points, that is, the number of nodes was 93032 and the number of elements was 92400. The simulation was carried out with the same conditions already described above. The flow was assumed to be the laminar flow. In Fig.3, these computational conditions were given from the right-hand side of the model as the uniform flow. For the boundary condition, the non-slip condition was applied for the surface of the model, and the surface was treated as the adiabatic wall. In this calculation, the value of artificial viscosity ν was 0.1. The streamline and the pressure distributions are shown in Fig.4. Figure 4 (b) shows that a re-compression wave occurred behind the capsule, although it was very weak. The existence of such a re-compression wave behind the capsule was anticipated theoretically. However, it hasn't been easy to clearly show its existence.

5 Flowfield Observation Around MESUR Capsule

In these experiments, a hypersonic gun tunnel of Mach 10 and 18 ms duration was used. The flow conditions were the same as the calculation's ones.

First, the shock wave ahead of the capsule was observed by utilizing the schlieren method. The visualized result and the calculated result are shown in Fig.5. It can be seen that both of the results were in very good agreement.

Next, the spatial streamlines and the wake pattern, such as the separation point, free shear layer, etc., were visualized by utilizing the electric discharge method. The result of the

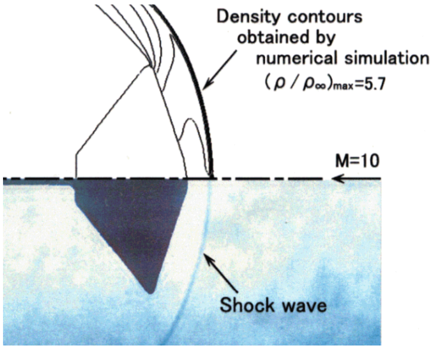
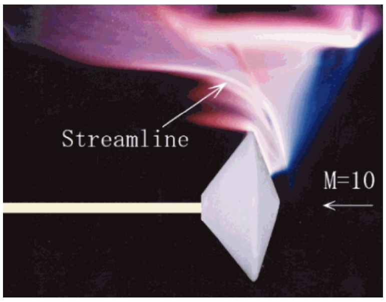
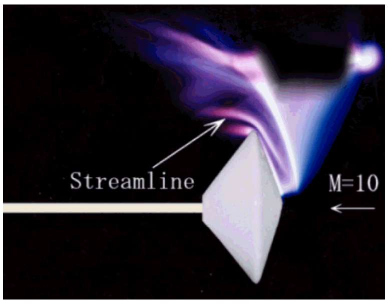


Fig.5 Comparison between the density contours around the capsule obtained by the new calculation method and the shock wave obtained by the schlieren method at M=10.



(a) Visualized streamline behind the shock wave.



(b)Visualized streamline behind the shock wave.

Fig.6 Visualized streamlines around the capsule obtained by the electric discharge method at M=10.

streamline is shown in Fig.6.[9]

To observe flow patterns, such as the separation point and free shear layer behind the capsule, radiating particles are generated by generating an electric field, as shown in Fig.7. The electric field is generated by means of a closely spaced pair of electrodes. In this case, the radiating particles are generated with a certain largeness of space according to the

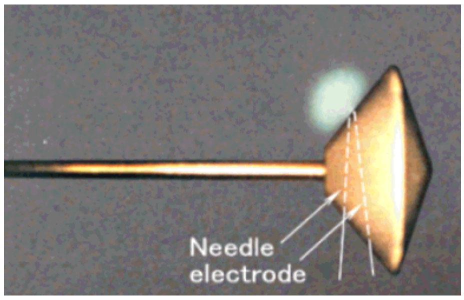


Fig.7 Visualized result with no flow.

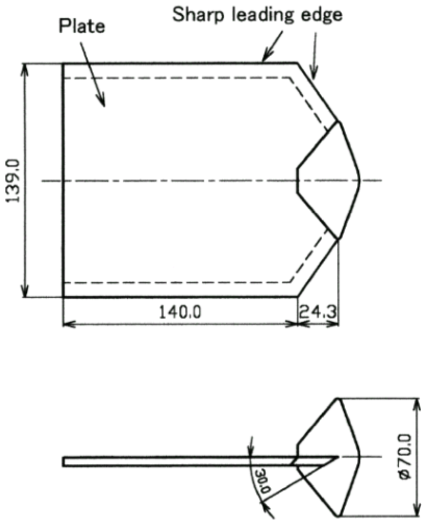
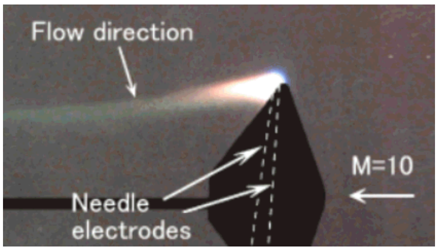
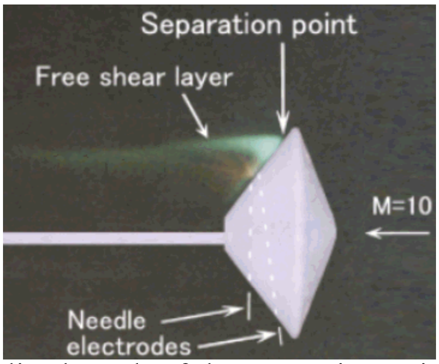


Fig.8 Capsule model with a thin plate used for investigating the wake structure in mm.

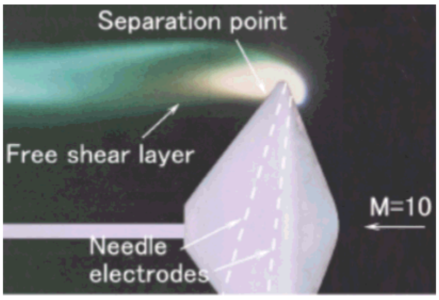
strength of the electric field. Figure 7 is an example of a case in which there is no flow. If there is a flow, the radiating particle drifts with the flow, radiating light. Thus, the flow pattern is obtainable by observing the light. The arrangement of the model is shown in Fig.8. A thin plate with shape leading edges was attached to the model so as not to disturb the flow by the lead wire. The experimental result is shown in Fig.9. The result clearly showed a spatial streamline, a separation point, and a free shear layer just after the separation, which had been very difficult to visualize without the electric discharge method. Figure 9 (b) shows the separation point and free shear layer obtained by generating an electrical discharge outside the vortex. However, Fig.9 (c) shows the separation location and free shear layer obtained by generating the electrical discharge inside the vortex. Both results agreed quite well, indicating that the visualized results were exact.



(a) Visualized result of the flow direction.



(b) Visualized result of the separation point and the free shear layer.



(c) Visualized result of the separation point and the free shear layer.

Fig.9 Visualized results of wake behind the capsule obtained by the electric discharge method at M=10.

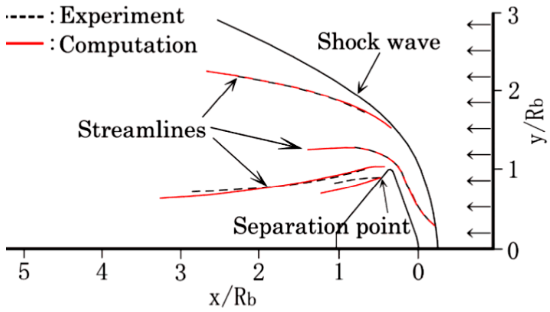


Fig.10 Comparison of the flowfield around the capsule obtained by the new calculation method and the electric discharge method at M=10.

6 Discussion

6.1 Comparison of Computation and Experiment

The computational results were compared with the experimental ones obtained above, as shown in Figs.10 and 5. Regarding the shock standoff distances Xs/Rb of the capsule, both the calculated and experimental results agreed quite well, showing that the Xs/Rb values were nearly 0.216. Also, both the streamlines and separation points agreed well. From these comparisons, it can be concluded that the calculated result by the present new method was considerably correct.

6.2 Flowfield Stabilization

It is very important to investigate the relation of the stabilization time of the flowfield when using a pulsed facility such as a hypersonic shock tunnel, ballistic range system, etc. However, it is also very difficult to examine such a relation under the condition of a very high speed, low density, and a short duration of such facilities. In this study, stabilization of the flowfield, such as the shock shape Xs, separation point Rs, and free shear layer Rf around the capsule, vs. the calculation step number, was investigated by using the present numerical method, as illustrated in Fig.11. The result of calculation is expressed numerically in Fig.12.

From Fig.12, we can see that when the shock shape seems to be almost stable at an early stage, the separation point and the free shear layer are still unstable. Also, when the separation point seems to be almost stable, the free shear layer is still unstable. Furthermore, even if the step numbers are more than 100,000, the free shear layer is still moving, and the value Rf/Rb is becoming larger and larger until almost 1,000,000 steps, although the increasing rate is very small.

From the result described above, it can be concluded that even if the shock shape seems to

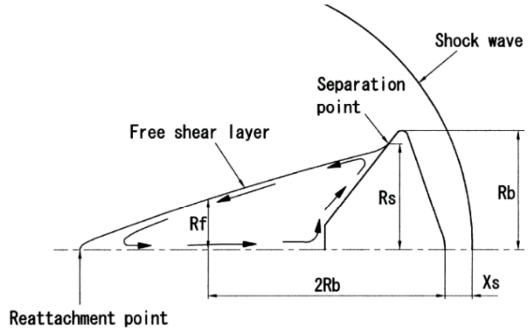


Fig.11 Illustration of wake structure around the capsule.

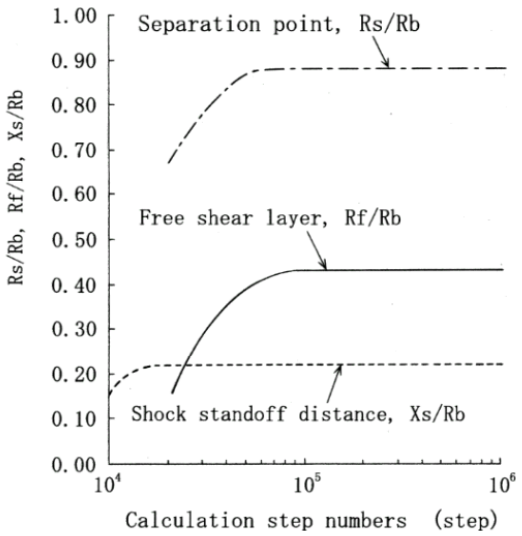


Fig.12 Stabilization of flowfield vs. calculation step numbers around the Mach 10 capsule by the new calculation method. Rb:capsule radius. Xs:shock standoff distance. Rs:distance between separation point and symmetry axis. Rf:distance between free shear layer at the location of 2Rb and capsule axis.

be stable, the wake structure is not necessarily stable. It is necessary to continue the calculation until the wake structure becomes stable, if an exact wake structure is required. In general, however, the calculation of the flowefield, including a shock wave by conventional numerical methods, would diffuse until such large calculation step numbers. Consequently, it has not been possible to obtain the exact wake structure by conventional calculation methods. However, the present new method can be used to calculate such a flowfield exactly with great stability, and, therefore, the method can be used to calculate the flowfield, including a shock wave with more than 1,000,000 steps, without the diffusion. In this sense, the present calculation method is superior to conventional methods.

7. Conclusions

A new calculation method of flowfield around supersonic/hypersonic vehicles based on the finite element method (FEM) has been presented. This method made it possible to calculate compressible flowfields with less error and better stability by using a very small artificial viscosity. The correctness of the present method was confirmed by comparing the shock shape, spatial streamline, separation point, and free shear layer. The comparisons were carried out under the condition of a Mach 10 flow by using a MESUR capsule, and the flowfield around the model was also investigated. Subsequently, the relation of the stabilization time among the shock shape, separation point, and free shear layer around the capsule was investigated by means of the present calculation method.

References

[1] Kim, M.S. Loellbach, J.M.and Lee, K.D., “Effects of Gas Models on Hypersonic Base Flow Calculations”, Journal of Spacecraft Rockets, Vol.31, pp.223-230, 1994.

[2] Olynick, D.R., Taylor, J.C. and Hassan, H.A., “Comparisons between DSMC and the Navier-Stokes Equations for Re-entry Flows”, AIAA Paper 93-2810, 1993.

[3] Nishio, M., Manabe, K, Nakamura, H. and Sezaki, S., “A New Calculation Method of Supersonic/Hypersonic Flow”, JOURNAL OF THE JAPAN SOCIETY FOR AERONAUTICAL AND SPACE SCIENCES, Vol.51, pp.683-689, 2003.(in Japanese)

[4] Okida, J., Manabe, K., and Ueda, K., “3-Dimensional Simulation of Cutting Process by Dynamin-Explicit Elastic-Plastic FEM”, Journal of Japan Society for Precision Engineering, Vol.66, pp.429-433, 2000.(in Japanese)

[5] Nishio, M., “Qualitative Model for Visualizing Shock Shapes”, AIAA Journal, Vol.30, No.9, pp.2246-2248, 1992.

[6] Nishio, M., “Method for Visualizing Streamline Around Hypersonic Vehicles by Using Electric

Discharge", AIAA Journal, Vol.30, No.6, pp.1662-1663, 1992.

[7] Nishio, M. and Hagiwara, T., "Hypersonic Flowfield Analysis of X-33 Model with the Electric Discharge Method", Journal of Spacecraft and Rockets, Vol.36, No.6, pp.784-787, 1999.

[8] Nishio, M., "Methods for Visualizing Hypersonic Shock-Wave/Boundary-Layer Interaction Using Electric Discharge", AIAA Journal, Vol.34, No.7, pp.1464-1467, 1996.

[9] Nishio, M., "Hypersonic Flow Visualization by the Radiation of Electric Discharges", AIAA-96-2392, pp.174-180, 1996.