Flow Patterns around the MESUR Capsule Traveling at Supersonic/Hypersonic Speeds∗

Hiroaki NAKAMURA∗∗, Keiji MANABE∗∗∗ and Masatomi NISHIO∗∗∗

Flow structures around a hypersonic re-entry Mars Environmental Survey (MESUR) capsule traveling at supersonic/hypersonic speeds are investigated utilizing a new computational method based on the finite element method (FEM). In order to confirm the validity of the computational method, the results are compared with experimental ones obtained by the electrical discharge method for a hypersonic flow at Mach 10. From these results, it can be concluded that they were in good agreement. As an example of the numerical simulation, flowfields around the capsule traveling at speeds of Mach 5, 3, and 2 are investigated. By these simulations, differences between the flow structures for these Mach numbers are clarified. The result shows that the location of the separation behind the capsule is governed by the strength of the adverse pressure gradient. The reverse flow behind the capsule that determines the base pressure distribution is correlated to the strength of the re-circulation and the stability of the capsule.

Key Words: Supersonic Flow, Hypersonic Flow, Wake Structure, Re-Entry Capsule, Numerical Simulation, Electrical Discharge Method

1. Introduction

In space development, it is vitally important to clarify the flowfield around supersonic/hypersonic vehicles. Visualization experiments utilizing a gun tunnel are dominant methods for understanding hypersonic flow structures, and visualization methods such as Electron Beam Method (EBM)(1) and Laser Induced Fluorescence (LIF) have been developed. However, the elucidation for hypersonic flow structures presents various problems because of very high speed, low density, and short test duration and so on. Therefore, recent investigations of flow structures around hypersonic vehicles such as re-entry capsules have been based primarily on computational investigations(2),(3).

The studies on compressible flows using the computational fluid dynamics (CFD) are mainly carried out by means of the finite difference method (FDM) and the finite volume method (FVM). These methods have advantages of calculation times and memories compared with the finite element method (FEM), and moreover the FDM has the high-resolution scheme such as the total variation diminishing (TVD) method. Therefore, hardly analyses by the FEM were conducted.

To address these flaws, the authors and others have developed a computational method based on the FEM(4). The merit of this method is treatment of pressure term in the Navier-Stokes equations. That is, the stress tensor including the pressure term is calculated as the non-differential form in the weighted residual equation. By the use of this method for the analyses of compressible flows with shock waves, flowfields are calculated with more stable computation, less numerical error, and with smaller artificial viscosity than that of conventional FEM analyses.

In this study, the above method was applied to analyses of flowfields around a Mars Environmental Survey (MESUR) capsule. First, in order to confirm the validity of the computational method, calculated results were compared with experimental ones obtained by the electrical discharge method for a hypersonic flow at Mach 10. Subsequently, as an example of numerical simulations, flowfields around the capsule traveling at several speeds of Mach 5, 3, and 2 were calculated, and flow structures
for these Mach numbers were investigated.

**Nomenclature**

- \( \mathbf{U} \): conservative variable vector
- \( \mathbf{F}_i, \mathbf{G}_j \): numerical flux vector
- \( w \): weighting function
- \( \rho \): density, \([\text{kg/m}^3]\)
- \( u_i \): velocity vector, \([\text{m/s}]\)
- \( \epsilon \): total energy, \([\text{J/kg}]\)
- \( \epsilon_i \): internal energy, \(RT/(\kappa - 1), \text{[J/kg]}\)
- \( \kappa \): ratio of specific heat
- \( p \): pressure, \([\text{N/m}^2]\)
- \( \tau_{ij} \): heat flux vector, \([\text{W/m}^2]\)
- \( \phi \): shape function for weighting function
- \( Re \): Reynolds number
- \( R_b \): capsule radius, \([\text{m}]\)
- \( R_s \): distance between separation point and capsule axis, \([\text{m}]\)
- \( X_s \): shock standoff distance, \([\text{m}]\)
- \( R_f \): distance between free shear layer at the location of 2 Rb and capsule axis, \([\text{m}]\)

2. Finite Element Method Formulation of Basic Equations

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

\[
\frac{\partial \mathbf{U}}{\partial t} + \left( \frac{\partial \mathbf{F}_i}{\partial x_i} + \frac{\partial \mathbf{G}_j}{\partial x_j} \right) = 0, \tag{1}
\]

where \( \mathbf{U} \) is the vector of conservation variables, and the particular form of the variables for Eq. (1) is presented in Eqs. (2) and (3). Equation (1) represents the formulation for the two-dimensional and axi-symmetric problems. The underlined terms are necessary when the problem is axi-symmetric. In this paper, the summation convention is used for subscripts.

The particular form of the variables for the above Eq. (1) is

\[
\mathbf{U} = \begin{bmatrix} \rho \\ \rho u_1 \\ \rho u_2 \\ \rho e \end{bmatrix}, \quad \mathbf{F}_i = u_i \mathbf{U}
\]

\[
\mathbf{G}_i = \begin{bmatrix} 0 \\ \tau_{11} - p \delta_{11} \\ \tau_{12} - p \delta_{12} \\ u_m (\tau_{m1} - p \delta_{m1}) - q_1 \end{bmatrix}
\]

and

\[
\mathbf{G}_2' = \begin{bmatrix} 0 \\ \tau_{12} - p \delta_{12} \\ \tau_{22} - \tau_{\theta \theta} \\ u_2 (\tau_{22} - p \delta_{22}) - q_2 \end{bmatrix}
\]

Where \( \rho \) is the fluid density, \( u_i \) represents the velocity components, \( \epsilon \) is the total energy per unit mass (\( \epsilon = \epsilon_i + \frac{u_i u_i}{2} \)), \( \epsilon_i \) is the internal energy, \( q_i \) represents the components of a heat flux vector, \( \delta_{ij} \) is the Kronecker deltas and \( \tau_{ij} \) is the components of the deviatoric stress tensor. In the case of the axi-symmetric problem, Eq. (3) is considered. In Eq. (3), \( \tau_{\theta \theta} \) is the deviatoric stress of circumference direction.

In the conventional FEM for the analysis of compressible flows, the pressure \( p \) is treated in the flux vector \( \mathbf{F}_i \). However, as is shown in Eq. (2), \( p \) is included in the flux vector \( \mathbf{G}_i \) in this study. The reason for this is the following.

After multiplication of Eq. (1) by the weighting function \( w \), integrating over the domain \( A \) and applying of the Gauss’s divergence theorem, the following equation is obtained:

\[
\int_A w \frac{\partial \mathbf{U}}{\partial t} dV = - \int_A w \frac{\partial \mathbf{F}_i}{\partial x_i} dV - \int_A \frac{\partial \mathbf{G}_j}{\partial x_j} dV + \int_{A_p} \mathbf{w} \mathbf{F} dS - \int_A \mathbf{w} \tau_{ij} \phi_{ij} dV. \tag{4}
\]

In Eq. (4), \( dV = 2\pi x_2 dA \) if the problem is axi-symmetric.

As shown in Eq. (4), \( \mathbf{F}_i \) is differential form and \( \mathbf{G}_i \) is non-differential form. Thus, by applying partial integration, the rank of differentiation is reduced. When the momentum conservation law is applied to the second term on the right-hand side of Eq. (4), that is,

\[
\left[ \int_A \frac{\partial \mathbf{w}}{\partial x_1} \mathbf{G}_1 dV + \int_A \frac{\partial \mathbf{w}}{\partial x_2} \mathbf{G}_2 dV \right]
\]

this term represents nodal point force, which is equivalent to internal stress according to the law of virtual work. Therefore, when the explicit method is applied by diagonalizing the matrix of the left-hand side of Eq. (4), the momentum conservation law of Eq. (4) directly shows the relation between the equivalent nodal force and the acceleration of the nodal points, that is, (material derivative of nodal momentum)

\[
= (\text{nodal internal force}) + (\text{nodal external force}). \tag{6}
\]

Thus it can be analyzed as a Newtonian equation of motion. By obtaining this relation, simultaneous equations do not need to be solved, and also, better stability and less error can be obtained in the calculations of the FEM.

3. Validity of Computational Method

To confirm the validity of the computational method, calculated flowfields around the capsule are compared with experimental ones obtained by the electrical discharge method \(^{5-9}\) for a hypersonic flow at Mach 10.

3.1 Simulation by FEM

3.1.1 Computational conditions

Figure 1 shows the dimension of capsule model used for this study. In this simulation, the flowfield is treated as the
axi-symmetric problem, and therefore the term using the underline is considered (as shown in Eq. (1)). The computational grid contains the number of nodes is 93,032 and the number of elements is 92,400. The minimum grid spacing near the wall is $2.0 \times 10^{-3}$. The flow is assumed to be the laminar flow. For the boundary condition, the non-slip condition is applied for the surface of the model, and the surface is treated as the adiabatic wall. To compare the result with the experimental one utilizing a hypersonic gun tunnel installed in Fukuyama University, the simulation is carried out under the same conditions: the Mach number is $M = 10$, the freestream density is $\rho = 4.5 \times 10^{-3}$ kg/m$^3$, static pressure is $p = 70$ N/m$^2$, static temperature is $T = 54$ K, freestream velocity $V = 1500$ m/s, and Reynolds number $Re = 9.75 \times 10^4$ (reference length is $5.0 \times 10^{-3}$ m).

### 3.1.2 Computational results

The calculated streamlines and pressure contours around the capsule are shown in Fig. 2. As shown in Fig. 2(b), the re-compression shock wave/wake interaction can be found near by the region of $x/Rb = 3 \sim 4$, and the interaction produces the high-pressure region at the downstream behind the capsule. Where, $Rb$ is the capsule radius. However, this value of non-dimensional pressure $p/p_{\infty}$ is lower than the one at the stagnation point. From this result, it is found that the re-compression shock wave/wake interaction is comparatively weak. In addition, there is the re-circulation region that determines the base pressure distributions behind the capsule. In this simulation, the value of pressure obtained from the reverse flow, that is, the strength of re-circulation region becomes $(p/p_{\infty})_{\text{max}} = 3.0$ (This value indicates 2.4% of the non-dimensional pressure at the stagnation point). From this result, it is found that there is hardly influence on the aerodynamics characteristics of capsule.

### 3.2 Visualization by electrical discharge method

#### 3.2.1 Visualization principle of shock wave and experimental results

The shock wave was visualized by the electrical discharge method. Figure 3 shows the arrangement of a model and a pair of electrodes. A needle electrode (cathode) is installed in the freestream and a very...
thin line electrode (anode) of some $1.0 \times 10^{-4}$ m is bonded to the model surface so as not to disturb the flow. The visualization principle of the shock wave is based on the following ideals. When a sheet-shaped electrical discharge is generated across a shock wave, a dark portion at the shock position in the electrical discharge can be seen because we can make the energy of the electrons drifting in the electric field drop suddenly at the shock position. As the result, the level of electron excitation of the gas molecules at the shock position becomes very low, and therefore, the radiation intensity from the position becomes very weak. Figures 4 and 5 show the visualized shock waves over the wedge model and ahead of the capsule, respectively.

3.2.2 Visualization principle of streamline and experimental results

The streamline can be also visualized by the electrical discharge method. Figure 6 shows the arrangement of a model and a pair of electrodes. The visualization principle of the streamline is described as follows. When a columnar spark discharge is generated cross a shock wave by the application of high voltage to a pair of point and point electrodes, as shown in Fig. 6, and the application of voltage between the electrodes is continued after the spark discharge, the columnar discharge drifts with the flow, radiating light. The radiation intensity of the drifting electrical discharge changes at the position $P_n$ on the streamline coming from the intersection $P_0$ of the shock wave and the initial spark discharge. From this, the radiation contract appears between $A$ and $B$ regions where $A$ is the region above the streamline and $B$ is the one below the streamline. Therefore, the streamline can be obtained by taking a photograph of the continuous drifting columnar discharge. Figures 7 and 8 show the visualized streamlines around the models.

3.2.3 Visualization principle of wake and experimental results

The technique described above was applied to the visualization of wake behind the capsule. As shown in Fig. 9, the lead wires are inserted in the sting.

First, the flow direction behind the capsule was investigated. Figures 9 and $10^{(10)}$ show an arrangement of a model and a pair of electrode, and the distance between the tips of two electrodes is 2.5 mm. In this case, the tips are arranged just at the model surface so as not to disturb the flow. When a voltage is applied between the electrodes,
excited particles are produced near the tips. These excited particles are drifted by the flow, radiating light. Figure 10 shows the flow direction behind the capsule for a supersonic flow at Mach 10.

Subsequently, the separation point and the free shear layer behind the capsule were investigated. Figure 11 (a) shows the separation location and free shear layer obtained by generating an electrical discharge inside the vortex. On the other hand, Fig. 11 (b) shows the separation location and free shear layer obtained by generating the electrical discharge outside the vortex. From both experimental results, it is found that the separation behind the capsule occurred after the capsule shoulder.

3.3 Comparison of computational results and experimental results

Figure 12 shows the comparison of calculated results and visualized ones for flow structures around the capsule traveling at Mach 10. Where, the solid line represents the
4. Analysis by FEM

In this section, flowfields around the capsule traveling at several speeds of Mach 5, 3 and 2 are calculated by the new method based on the FEM, and flow structures for these Mach numbers are investigated. Table 1 shows the freestream conditions used for the simulations.

### 4.1 Streamlines around capsule

Figure 13 shows calculated streamlines around the capsule. The separation point moves toward the capsule shoulder and the curvature of free shear layer expands toward the capsule radius when the Mach number becomes lower.

Figure 15 shows the dependence of flow structures (as shown in Fig. 14) on freestream Mach numbers. The flow structures around the capsule, especially the shock standoff distance are greatly different when the Mach number changes 3 to 2. This tendency means that the restraint on flowfield around the capsule by the shock wave becomes weak. Therefore, we have to pay attention to the influence of the freestream including the turbulence on flow structures behind the capsule when the flowfield is investigated by utilizing a gun tunnel, etc.

As a cause of described above (the change of separation point), the effect of adverse pressure gradient near the capsule shoulder can be considered. That is, the adverse pressure gradient is supposed to be steeply when the Mach number becomes lower. This tendency means that the strength of re-circulation region, that is, the ef-

---

**Table 1  Freestream conditions**

<table>
<thead>
<tr>
<th>Mach number</th>
<th>2</th>
<th>3</th>
<th>5</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity, m/s</td>
<td>5900</td>
<td>895.0</td>
<td>1500.0</td>
<td>1500.0</td>
</tr>
<tr>
<td>Density, kg/m³</td>
<td>$8.9 \times 10^{-2}$</td>
<td>$4.0 \times 10^{-2}$</td>
<td>$1.8 \times 10^{-2}$</td>
<td>$4.5 \times 10^{-3}$</td>
</tr>
<tr>
<td>Static pressure, N/m²</td>
<td>5529.0</td>
<td>2549.0</td>
<td>1197.0</td>
<td>70.0</td>
</tr>
<tr>
<td>Static temperature, K</td>
<td>216.7</td>
<td>221.6</td>
<td>226.5</td>
<td>54.0</td>
</tr>
<tr>
<td>Reynolds number*</td>
<td>$1.8 \times 10^6$</td>
<td>$1.26 \times 10^6$</td>
<td>$9.3 \times 10^4$</td>
<td>$9.75 \times 10^4$</td>
</tr>
</tbody>
</table>

*Reference length is $5.0 \times 10^{-2}$ m

![Fig. 12 Comparison of calculated results and visualized ones for flow structures around the capsule](image)

![Fig. 13 Calculated results for streamlines around the capsule at freestream Mach numbers 5, 3 and 2](image)

![Fig. 14 Illustration of the wake structure around the capsule](image)
fect of reverse flow on capsule becomes great when the freestream Mach number becomes lower.

### 4.2 Pressure contours around capsule

#### 4.2.1 Effect of re-compression region

Figures 16 and 17 show pressure distributions around the capsule. As described above, the re-compression shock wave/wake interaction produces the high-pressure region near by the capsule axis. In the case of Mach number is 5, the non-dimensional pressure \(p/p_\infty\) becomes 1.7 (This value indicates 5.7% of the non-dimensional pressure at the stagnation point), in the case of Mach number is 3, \(p/p_\infty\) becomes 1.1 (10.0%) and in the case of Mach number is 2, \(p/p_\infty\) becomes 0.8 (15.0%), respectively. From these results, it is found that the value produced by the re-compression shock wave/wake interaction is inclined to increase when the Mach number becomes lower.

In addition, the re-compression shock wave/wake interaction produces the adverse pressure gradient at the downstream behind the capsule. Especially, the adverse pressure gradient becomes steep when the Mach number is low. Therefore, it is considered that the flow goes downstream from the separation point is difficult to flow because of the including kinetic energy is small. On the other hand, the pressure gradient near the rear stagnation point produces the flow going to the upstream from the down-stream, and the reverse flow determines the base pressure peak behind the capsule.

The base pressure peak obtained by the reverse flow
becomes higher when the Mach number becomes lower and the adverse pressure gradient at the downstream becomes steeper. From these results, we can consider that there is an obvious relationship between adverse pressure gradient near the rear stagnation point and strength of recirculation region. These base pressure distributions obtained by the difference of flowfield at the downstream indicate that the characteristics of vortex structures determine the base pressure behind the capsule, and this behavior agrees with the reports by TERAMOTO and FUJII et al. (10), (11)

4.2.2 Effect of base pressure Figures 18 and 19 show the wall pressure distributions, and Fig. 20 shows the relation between front force/back force vs Mach numbers. The wall pressure suddenly decreases near the capsule shoulder by the expansion wave, and increases at the region of $x/R_b = 1.13 \sim 1.30$ after that. The adverse pressure gradient ($x/R_b = 1.13 \sim 1.30$) becomes most steep when the Mach number is 2. From these results, it is found that the separation point changes by the effect of adverse pressure gradient near the capsule shoulder. In addition, the base pressure peak by the reverse flow occurs at the capsule neck. In this case of Mach number is 5, the non-dimensional pressure $p/p_{\infty}$ becomes 1.1 (This value indicates 3.7% of the non-dimensional pressure at the stagnation point), in the case of Mach number is 3, $p/p_{\infty}$ becomes 0.6 (5.5%) and $p/p_{\infty}$ becomes 0.5 (9.4%) at Mach number is 2. Judging from these calculated results, we can consider that the strength of recirculation region becomes great when the freestream Mach number becomes lower.

The tendency of described above also appears in pressure coefficient ($C_p$) distributions as shown in Fig. 19. That is, the pressure recovery occurs at the capsule neck by the reverse flow. However, the ratio of neck area to total surface area behind the capsule is comparatively small, and moreover the recovery ratio is also low. From these results, it is found that there is hardly influence on the aerodynamics characteristics of capsule, and the capsule mainly suffers the pressure drag. In this study, the negative pressure behind the capsule becomes higher when the Mach number becomes lower. Furthermore, as shown in Fig. 15, the flowfields around the capsule are greatly different when the Mach number changes 3 to 2. According to these calculated results, it is considered that the capsule suffers more great drag from the upstream, and the size and shape of recirculation region behind the capsule affects the drag operating on the capsule.

In this study, we could find the result that the flowfields around the capsule were considerably changed around the Mach number is 3 to 2. However, as described above, in the case of experiments utilizing a gun tunnel, etc, we can consider that the flowfield behind the capsule suffers the effect of freestream including the turbulence.
when the Mach number becomes lower. Therefore, there is possibility that the strength and size of re-circulation region are greatly change.

5. Conclusions

In this study, the flowfield around the MESUR capsule was investigated by utilizing the new method based on the FEM.

First, to confirm the validity of the computational method, calculated results were compared with experimental ones obtained by the electrical discharge method for a hypersonic flow at Mach 10. As the result, it was found that the calculated results were considerably correct.

Subsequently, flowfields around the capsule traveling at speeds of Mach 5, 3 and 2 were calculated as the examples of numerical simulations, and flow structures for these Mach numbers were investigated. The results are summarized as follows:

(1) The re-compression shock wave/wake interaction produces the high-pressure region at the downstream behind the capsule. The value is inclined to increase when the Mach number becomes lower.

(2) The re-circulation region exists behind the capsule. The location of separation moves toward the capsule shoulder when the Mach number becomes lower. As the cause, it is considered that this is due to the effect of adverse pressure gradient near the capsule shoulder. That is, the adverse pressure gradient becomes more steeply when the Mach number becomes lower.

(3) The great relation between base pressure distributions behind the capsule and size of re-circulation region was found. That is, the base pressure peak becomes higher when the curvature of free shear layer moves toward the capsule radius. In addition, the base pressure peak occurs at the capsule neck.

References