Application of Two-way Fluid-Thermal-Structure Coupling Method in the Study of Thermal and Structural Response of Hypersonic Vehicle

Zhang ZhunHyok1, Ri CholUk1, Oh SeHyok1, Kim RyongSop1, Kim CholJin2, Kim JaeHun1, Ho PongKuk1
1School of Mechanical Technology, KimChaek University of Technology, Pyongyang 950003, Democratic People’s Republic of Korea
2KimIlSung University, Pyongyang 999093, Democratic People’s Republic of Korea

ABSTRACT

Accurate prediction of its thermal and structural response characteristics, when a hypersonic vehicle flies at high speed, is a very important issue in its design and manufacturing. Complex thermal and structural behavior is produced in the structure due to the interaction of aerodynamic force and aerodynamic heating applied to the vehicle, which in turn changes the flow field around the vehicle. In this study, as a Two-way fluid-thermal-structure coupling method, an ANSYS and CFX coupling method by ANSYS’s MFX Multi-field Solver was established and applied to the study of thermal and structural response of hypersonic vehicles. In order to verify the validity of the coupling method proposed in this study, a comparison with NASA Langley’s cylindrical leading-edge model test results is conducted. By analyzing the calculation results of the flow field, temperature field, and structure field of the cylindrical leading-edge model, it was concluded that this coupling method is effective with high accuracy. The numerical calculation results can provide accurate technical support for the thermal protection design and structural design of hypersonic vehicles.

Keywords - Hypersonic vehicle, Two-way fluid-thermal-structure coupling, Thermal and structural response, thermal protection design, thermal structural design

I. INTRODUCTION

When a hypersonic vehicle flies at a high speed, thermal and structural response characteristics of the vehicle structure change due to the combination of aerodynamic force and aerodynamic heating, ie, the temperature rise of the structure, material properties, stress and deformation, mode and stiffness change occur.

It also causes a change in the aerodynamic force and aerodynamic heating of the vehicle as an anti-coupling action. Physically, this problem can be seen as a two-way fluid-thermal-structure coupling problem, and mathematically, it corresponds to a combination of very complex nonlinear partial differential equations.
Therefore, applying the two-way fluid-thermal-structure coupling method in the design of a hypersonic vehicle to study its thermal and structural response characteristics is one of the important problems to improve the reliability and accuracy of the design.

The two-way fluid-thermal-structure coupling problem of hypersonic vehicle is a kind of multidisciplinary problem. It includes a close coupling of several physical fields such as flow field, temperature field, and stress field (displacement field), and the physical process is very complex and has strong nonlinearity. The numerical solution of this coupling problem encompasses multiple disciplines such as computational flow dynamics, computational heat transfer, computational structural dynamics, and has high analysis difficulty and computational cost, and is recognized as one of the most difficult problems in the current aerospace field.

Thornton and Dechaumphai [1] are the predecessors of two-way coupling studies. Thornton [2] et al. conducted numerical analysis on stainless steel cylinders placed in a hypersonic flow field environment, raised the hypersonic fluid-thermal-structure multi-physical field coupling problem, and analyzed the flow field and the response of stainless steel cylinders. Dechaumphai [3] et al. analyzed the structural response of a calculation model in a wind tunnel environment of a stainless steel cylinder built using a numerical method, and conducted a wind tunnel test and contrast verification. Culler and McNamara [4] conducted a meticulous study on the coupling relationship of the hypersonic fluid-thermal-structure coupling problem and the one/two-way coupling method. They used numbers to indicate different forms of association, and the specific meaning of numbers is shown in Figure 1. In the figure, $T_w$ is the wall temperature, $h_e$ is the convective heat transfer coefficient, $T_{aw}$ is the adiabatic wall temperature, $p$, $T$, $Ma$ are pressure, temperature, and Mach number of the boundary layer’s outer edge, respectively. $P_{wef}$ is the wall pressure. The study results showed the importance of two-way coupling in the fluid-thermal-solid coupling analysis.

![Figure 1. Schematic diagram of Two-way Fluid-Thermal-Structure coupling method](image)

Culler and McNamara [5] found that the thermal response characteristic time was about $10^2$ times greater than that of the structural response characteristic time through the study of the C/C composite plate. Also, the structural response characteristic time is about $10^2$ times larger than the flow field characteristic time. Through the established quasi-steady coupling analysis strategy, the effects of one/two coupling, time steps of each physical field, and the number of repeats on the coupling analysis result were analyzed. Lohner [6] et al. for the first time, proposed the concepts of tight coupling and loose coupling, and suggested possible coupling methods based on detailed analysis of the governing equations related to the coupling process and their solving methods. Based on this, a loose coupling calculation strategy for the multi-physical field coupling of aero/thermal/structure was proposed, and the coupling calculation was realized by matching the existing CFD and CTSD programs. Lamorte [7] et al. established a fluid-thermal-structure coupling model using a numerical method and studied the aerothermo-elastic response of airplane wings under different flight conditions. Studies have shown that the external
flow field transitions from laminar flow to turbulent flow and thermal stress have a certain effect on the stability of the wing. Miller [8-9] et al. considered the time scale problem between the flow field and the solid response, and applies a loosely coupling method to calculate the calculation accuracy of the flow field zone and the solid zone, and a fluid-thermal-structure multi-physical field coupling model. Dumas [10] et al. applied the maxwell-smoluchowki model to analyze the velocity slip and temperature jump conditions, and studied the coupling between the non-equilibrium flow field and the wall. Currao[11] et al. built a fluid-solid coupling model by applying a numerical method, studied the response of the protruding plate under hypersonic environment, and verified the effectiveness of the model by comparing the calculation results with the ground test results. Zhang[12-14] et al. set up a numerical simulation model of multi-physical field coupling in a vehicle thermal environment and analyzed the combined heat transfer characteristics and rules between the flow field and the structure. Haupt[15] et al. established a numerical platform applied to the multi-physical field coupling analysis by applying a loosely coupling method under the requirements of the IMENS item led by the Deutsche Land Aerospace Center (DLR), and This platform combines MpCCI as the basis for data exchange, structural finite element program ANSYS, MSC/NASTRAN, and unstructured CFD program Tau. In addition, the effectiveness of the verification evaluation was verified by conducting a calculation example analysis on the ejection pipe and the head cover. Miller and McNamara [16] analyzed the time accuracy of the discrete scheme of each physical field and established a method for promoting data exchange with second order accuracy from the basis of the fluid-thermal-solid coupling strategy established in literature [17]. Verification was conducted using the hypersonic plate as a model, and contrast analysis was conducted with the traditional method.

Overall, the current research on the fluid-thermal-structure coupling problem of hypersonic vehicles has already developed into a mature and effective analysis method, and the effectiveness of the method has been verified, but continuous research is still needed in the aspect of engineering application.

In this study, in order to realize the two-way coupling of the fluid-thermal-structure in the study of the thermal and structural response characteristics of the hypersonic vehicle, and to improve the efficiency of the selection of the coupling variables and the coupling calculation, a method of coupling ANSYS and CFX by ANSYS' multi-field solver-multi-code combiner MFX is proposed. MFX solver is mainly used for fluid-structural coupling analysis, and combines the structural part of ANSYS Multi-physics (or Mechanical) solver and the fluid part of CFX solver. In this study, the structure field is composed of elements with thermal and structural coupling degrees of freedom to extend the fluid-structure coupling analysis function of MFX to the fluid-thermal-structural coupling analysis, and the effectiveness of this method is verified through example calculations.

II. METHODOLOGY AND MODELLING

In this section, firstly, the governing equation of aerodynamic flow in the fluid domain and the governing equations for modeling thermal and structural responses in the solid domain are briefly outlined for the Two-way Fluid-Thermal- Structure Interaction (FTSI) analysis. Next, the geometry of the computational model is presented, the coupling scenario is discussed in detail, and a detailed solver settings for fluid and solid solvers follow to be conducted.

2.1 Governing equations

The flowfield around a hypersonic vehicle is explained and modeled by the unsteady compressible flow and
RANS equation. Since this study includes the effect of an incidence angle and the thermo-structural response of the body, cell-centered finite volume approach should be applied to solve the Navier-Stokes equation of the unsteady compressible flow [18]. The integral form of governing equation in 3D Cartesian coordinates can be expressed as follows.

$$\frac{\partial}{\partial t} \int_{\Omega} Q d\Omega + \sum_{F_i} (F_i - F_i^c) dS = 0 \quad (2-1)$$

where:
- $\Omega$: control volume
- $\partial \Omega$: boundary surface of control volume
- $Q$: conservative variable
- $F_i$: convective flux
- $F_i^c$: viscous flux

For the calorically perfect gas, $\partial &$ of the governing equation is zero vector. The remaining physical quantities are expressed as follows.

$$Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{bmatrix}, \quad F_i = \begin{bmatrix} 0 \\ \rho u \nu + p \frac{\partial \nu}{\partial x} \\ \rho v \nu + p \frac{\partial \nu}{\partial y} \\ \rho w \nu + p \frac{\partial \nu}{\partial z} \\ \rho \frac{\partial \nu}{\partial t} \end{bmatrix}$$

$$F_i^c = \begin{bmatrix} n_x \tau_{xx} + n_y \tau_{xy} + n_z \tau_{xz} \\ n_x \tau_{yx} + n_y \tau_{yy} + n_z \tau_{yz} \\ n_x \tau_{zx} + n_y \tau_{zy} + n_z \tau_{zz} \\ n_x \Theta_x + n_y \Theta_y + n_z \Theta_z \end{bmatrix}$$

where:
- $\rho$: fluid density
- $u, v, w$: velocity component in $x, y, z$ directions
- $E$: total energy per unit mass
- $U$: normal velocity on the control surface
- $p$: pressure
- $H$: total enthalpy per unit mass
- $n_x, n_y, n_z$: 3 components of normal vector on the control surface
- $\tau_{ij}$: 9 components of stress tensor
- $\Theta_x, \Theta_y, \Theta_z$ are expressed as follows.

$$\Theta_x = u \tau_{xx} + v \tau_{xy} + w \tau_{xz} + \frac{\partial T}{\partial x}$$
$$\Theta_y = u \tau_{yx} + v \tau_{yy} + w \tau_{yz} + \frac{\partial T}{\partial y}$$
$$\Theta_z = u \tau_{zx} + v \tau_{zy} + w \tau_{zz} + \frac{\partial T}{\partial z}$$

The unsteady temperature field of structure caused by aeroheating is modeled by solving Fourier heat conduction [19]. The integral form can be written as follows.

$$\frac{\partial}{\partial t} \int_{\Omega} Edq_n dS + \sum_{S} (q - s \cdot n) dS = \int_{\Omega} S d\Omega \quad (2-3)$$

where:
- $\rho_s$: density
- $c_s$: specific heat
- $k$: thermal conductivity
- $S$: volumetric heat source
- $q$: heat flux imposed on the structure boundary
- $T_s$: structure temperature

Here, the effect of the wall’s radiation is neglected.

In the absence of deformation energy in the above, the heat source of the right part becomes zero. Under the influence of aerodynamic forces and heat fluxes, the governing equations for the thermo-elasticity of three dimensional body are the compatibility equations of strains, Cauchy’s relations between the tractions and stresses and equilibrium equations of stresses in Cartesian coordinates. The solid part is regarded as the isotropic linear thermoelastic body.

Newton’s second law is used to express the equilibrium equation of the elastic body as follows.

$$\sigma_{ji,j} + F_i = 0 \quad (i, j = 1, 2, 3) \quad (2-4)$$

where:
- $\sigma_{ji}$: components of stress
- $F_i$: components of body force per unit volume

$\sigma_{ji}$ are in the symmetrical relationships as

$$\sigma_{ji} = \sigma_{ij} \quad (i, j = 1, 2, 3)$$

The following equation shows Cauchy’s fundamental relations.

$$\sigma_{ji,n} = p_{ni} \quad (i, j = 1, 2, 3) \quad (2-5)$$
n: the direction cosines between the external normal and axis
The geometric equation showing the relationship between strains and displacements is as below.

\[ \varepsilon_{ij} = \frac{1}{2} (u_{i,j} + u_{j,i}) \quad (i, j = 1, 2, 3) \]  \hspace{1cm} (2-6)

\[ \varepsilon_{ij}, \quad u_{i,j} \] : the components of the strains and displacements

\[ \varepsilon_{ij} \] are in the symmetrical relationships.

\[ \varepsilon_{ji} = \varepsilon_{ij} \quad (i, j = 1, 2, 3) \]

The stresses and strains are explained by the generalized Hooke’s law of the homogeneous isotropic body.

\[ \sigma_{ij} = 2 \mu \varepsilon_{ij} + (\lambda + \beta \tau) \delta_{ij} \quad (i, j = 1, 2, 3) \] \hspace{1cm} (2-7)

\[ \tau: \quad \tau = T_s - T_0, \quad T_0: \text{reference temperature} \]

\[ \lambda, \mu: \text{Lamé elastic constants} \]

\[ \beta: \text{thermoelastic constant} \]

\[ \delta_{ij}: \text{Kronecker's symbol} \]

The following compatibility equations involves the stress components.

\[ \nabla^2 \sigma_{ij} + \frac{1}{1 + \nu} \sigma_{kk,j} + \alpha E \left( \frac{1}{1 - \nu} \nabla^2 \tau \delta_{ij} + \frac{1}{1 + \nu} \tau_{ij} \right) \]

\[ = - \left( \frac{\nu}{1 - \nu} F_{i,k} \delta_{i,j} + F_{i,j} + F_{j,i} \right) \]

\[ E: \text{elastic modulus} \]

ANSYS MAPDL is employed to solve the heat transfer equation and thermo-elasticity’s governing equations based on the finite element method [20].

### 2.2 Analysis of Coupling Relations of Multi-field Mathematical Equations

The N-S equations of flow dynamics and the basic equations of heat conduction and structural dynamics in the structural domain fully describe the relationship between the physical variables in the multi-physical field of the fluid-thermal-structure, but it is highly nonlinear to consider all of these equations in the entire coupled field. Theoretically speaking, it can directly solve the equation described in the right to obtain the solution of multi-physical field two-way coupling, but the direct solution is very difficult and it is difficult to realize even a slightly complicated problem. Therefore, in actual calculation, each field is separated and calculated, and then variables are transferred.

When applying this method, since arc coupling variables are transmitted at the boundary of the computation area between each field and the field, a certain boundary condition must be satisfied. Figure 2 shows the relationship of the mathematical modeling of the multi-physical field two-way coupling.

In Fig. 2, \( \Gamma \) denotes the coupling boundary between the fluid region and the structural region, and \( n \) denotes the normal direction of the boundary \( \Gamma \). In the coupling boundary \( \Gamma \), the equations of the fluid field and the structure field are combined through the velocity matching condition and the force equilibrium condition, which can be expressed as follows.

\[ \dot{u} = \nu \]

\[ \delta_{on} = -p_n + \delta_{Fn} \] \hspace{1cm} (2-8)

Where \( \nu \) is velocity of the fluid, \( p \) is the pressure in the flow field, \( \delta_s \) and \( \delta_F \) are the stress density of the structure and the stress density of the flow field, respectively. In addition, the conditions of temperature continuity and heat flow balance must be
satisfied at the coupling boundary. Expressed as an equation, it is as follows.

\[ T_s = T_F \]
\[ \lambda_s \nabla T_s = -\lambda_F \nabla T_F \]  

(2-9)

Where \( T_s \) and \( T_F \) are the temperature of the structure and the fluid, respectively, \( \lambda_s \) and \( \lambda_F \) are the coefficient of thermal conductivity of the structure and fluid, respectively. In the calculation, the coupling between grid deformation of the structure and fluid must be taken into account, and they are combined together through the relationship between the governing equation of the structure and the dynamic grid.

\[ x = u \]
\[ \hat{x} = \hat{u} \]  

(2-10)

Where \( x \) and \( u \) represent the displacement of the grid and the displacement of the structure, respectively. Based on the governing equation of each field and the corresponding boundary conditions, the solution of the entire coupling field can be obtained. In this study, a numerical solution method is applied to solve the governing equations and boundary conditions of each field. In the fluid-thermal-structure coupling field, the fluid and solid field are two different solving areas and their physical properties are different, so different numerical solution methods are applied accordingly.

In this study we introduce application programs that have already been developed and are widely used. The solution is obtained by applying the finite volume method of CFX for the flowfield, and the finite element method of ANSYS for the solid field. Next, based on the coupling conditions between the two fields discussed in over, the coupling variables are determined and data transfer is performed to complete the coupling calculation. The next section presents specific methods for this.

2.3 Calculation process of the Multi-physical field two-way coupling

Currently, the method to realize multi-physical field two-way coupling is mainly the MPCCI program developed by Deutsche Land and the System Coupling component module of ANSYS Workbench [24]. However, both of these methods have a series of problems. First of all, MPCCI program is a kind of data transmission program, and because it conducts coupling analysis by calling two calculation simulation programs (for example, Fluent and Abaqus) in the calculation process, the calculation efficiency is low and the simulation calculation of a relatively complex model cannot be realized. Moreover, two-way coupling for the three fields of the fluid-thermal-structure is not possible, only bidirectional coupling between the two fields of fluid-heat, fluid-structure, and structure-row is possible.

Next, since ANSYS 14.5 version, the newly added System Coupling module has solved problems such as subdivision of the coupling boundary and has undergone ceaseless improvement, and is already relatively mature. It is relatively practical in the field of calculation of the two-way coupling of a series of simple models.

However, the coupling variables that are passed in the actual using process cannot be freely selected, and the interpolation methods of the coupling variables are all preset. Although the convergence criterion and the number of step cannot be freely defined by the user in the solution process. Also, like the MPCCI program, two-way coupling for the three fields of fluid-thermal-structure is impossible.

From this, in the paper, in order to realize the two-way coupling of the three fields of the fluid-thermal-structure and to improve the selection of the coupling variables and the efficiency of the coupling calculation, we establish a method of coupling ANSYS and CFX by ANSYS’ MFX Multi-field Solver. MFX solver is mainly used for fluid-structural coupling analysis, and
combines the structural part of ANSYS Multi-physics (or Mechanical) solver and the fluid part of CFX solver. In the study, the structure field is composed of elements with thermal and structural coupling freedom, and the function of MFX fluid-structure coupling analysis is extended to fluid-thermal-structure coupling analysis.

2.3.1 Coupling of ANSYS and CFX by MFX Multi-field Solver

In order to use MFX solver, the following requirements must be satisfied.

1) Fluid and solid models for coupling analysis must be three-dimensional models.
2) The ANSYS model is a single field model, and the elements used for load transfer must be three-dimensional elements, and these must be structural or thermal degrees of freedom or their combined degrees of freedom (structure-thermal coupling degrees of freedom, for example solid 226, solid57, etc.).
3) Only the face load is transmitted. Effective face loads are displacement, temperature, force and force density and heat flux.
4) MFX can only couple two solvers. That is, ANSYS and CFX. A given coupling analysis can only proceed with a coupling between two field solvers, but can transfer multiple loads simultaneously.
5) Parallel processing is impossible for ANSYS solver, but parallel processing is possible for CFX solver. CFX solver using parallel processing are still single field solver.
6) ANSYS solver allows steady and transient analysis, but CFX solve only allows transient analysis.

Mechanical. Their coupling analysis procedures are as follows, respectively.

1) Coupling method 1
   ① In Transient Structural, define materials, grids, constraints, and fluid boundaries for structural analysis.
   ② Create an ANSYS Mechanical structure file in *.inp format.
   ③ In CFX-Pre’s Simulation Type, set External Solver Coupling to ANSYS MultiField and load the *.inp type ANSYS Mechanical structure file created in ②.
   ④ Set the boundary condition of CFX-Pre fluid analysis and set the Mesh Motion of the fluid solid coupling interface to ANSYS MultiField. Set the FSI data transfer parameters of CFX-Pre and ANSYS Mechanical (for example, displacement, force, temperature, heat). Other boundary conditions are set according to the requirements of CFX fluid analysis.
   ⑤ Set the unit through Solver/Solver Units of CFX-Pre and match the units of ANSYS and CFX-Pre.
   ⑥ Set the convergence condition of CFX-Pre in Basic Setting of Solver/Solver control of CFX-Pre, and in External Coupling, set the order of solving ANSYS and CFX-Pre solver, and perform a series of MFX settings.
   ⑦ After completing the setting, use the Write Solver Input file from the FILE list of CFX-Pre to create a *.def file.
   ⑧ Enter CFX-Solver Manager, load CFX-Pre file (*.def) and ANSYS Mechanical file (*.inp) created in Transient Structural, proceed with other settings, and click Run to execute.

2.3.2 Two-way coupling method by MFX solver

There are three types of ANSYS-CFX two-way coupling method by MFX solver. That is, CFX+Transient Structural (ANSYS Workbench), CFX+ANSYS Mechanical, and MFX+ANSYS Mechanical. Their coupling analysis procedures are as follows, respectively.
2) Coupling method 2

1. In ANSYS Mechanical, define materials, grids, constraints, and boundary condition for structural analysis.

2. Select MFX-ANSYS/CFX solve in Multi-field Setup of ANSYS Mechanical and proceed with a series of settings related to CFX. For example, time steps, solving type, and number of steps.

3. Using the Write Input of MFX-ANSYS/CFX, create an ANSYS Mechanical fluid solid combination file (*.dat).

4. As in step 3 of method 1, the *.dat file created in the over step is loaded from the ANSYS MultiField item in CFX-Pre.

5. Apply steps 4 to 6 of Method 1 and proceed with the relevant settings in CFX-Pre. Note that the unit of CFX-Pre must match the unit set by ANSYS. In addition, CFX-Pre sets CFX as priority solve by default, and ANSYS Mechanical sets ANSYS as priority solve by default, so care must be taken to match them.

6. After completing the setting, create a Write Solver Input file from the file list of CFX-Pre.

7. Open MFX-ANSYS/CFX environment setting in ANSYS Product Lancher and proceed with MFX-ANSYS/CFX setup. Here, the *.dat file created by ANSYS and the *.def file created by CFX are loaded into the structure and fluid file of MFX, respectively, and after completing the settings, press RUN directly to execute the solution.

Here, the second and third methods are suitable for users who are familiar with the ANSYS Mechanical environment. In this study, the first method is applied.

III. RESULTS AND DISCUSSION

In this section, we verify the accuracy of a fluid-thermal coupling model based on system coupling module by referring to the experimental result of Allan R Wieting\(^1\). This experiment was implemented in a NASA Langley 8-ft High Temperature Tunnel with a Mach 6.47 flow of a cylindrical leading-edge model. In this study, the freestream condition and the thermal constraints, and boundary condition for structural analysis.
physical properties and geometric dimensions of structural material for validation calculation are set with reference to the literature [22].

![Figure 4](image_url)

**Figure 4.** Grids and boundary condition of the cylindrical leading edge: (a) Grids of the fluid and solid domain; (b) Boundary conditions of Fluid domain.

Each parameter of freestream in the experiment is shown in Table 1. Each thermo-physical properties of the cylindrical tube material is shown in Table 2. The cylindrical tube material is stainless steel.

<table>
<thead>
<tr>
<th>Parameters of freestream</th>
<th>Pressure (Pa)</th>
<th>Temperature (K)</th>
<th>Mach number</th>
<th>Reynolds number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>648.1</td>
<td>241.5</td>
<td>6.47</td>
<td>1.31×10⁶</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material</th>
<th>Density (kg·m⁻³)</th>
<th>Young’s modulus (GPa)</th>
<th>Poisson’s ratio</th>
<th>Specific heat (J/kg·K)</th>
<th>Heat transfer coefficient (W/m·K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1Cr18Ni9Ti</td>
<td>8030</td>
<td>206</td>
<td>0.3</td>
<td>502.48</td>
<td>6.47</td>
</tr>
</tbody>
</table>

The outer diameter of the cylindrical tube is 38.1mm and the inner diameter is 25.4mm. The inner wall of the stainless steel tube is an isothermal wall, and the temperature is 294.4K. The grid of the flowfield is created using the GAMBIT program, and the grid of the solid field is created using the Mesh tool of the Transient structural module of Ansys Workbench. In addition, a 3-D 1/4 cylindrical model is produced in order to save computing resources. In order to accurately predict the shock wave and aerodynamic heating, the mesh near the wall and the location of the shock wave formation is made densely. Engineering calculation methods of hemisphere’s shock separation distance have been introduced in several literatures [23].

The ratio of the hemisphere diameter to the shock separation distance is calculated as

\[
\frac{\Delta}{D} = \frac{2\varepsilon}{1 + \sqrt[3]{8\varepsilon}}
\]  

(2-11)

Where \(\varepsilon\) is the density ratio passing through the vertical shock wave, it can be calculated by the following equation.

\[
\varepsilon = \frac{(\gamma - 1)M_\infty^2 + 2}{(\gamma + 1)M_\infty^2}
\]  

(2-12)

From this, it can be seen that the shock separation distance \(\Delta\) is related to the specific heat ratio of the gas and the Mach number of freestream. For example, if the freestream Mach number \(M_\infty = 6.47\), the specific heat ratio of the rear gas \(\gamma = 1.4\), and the diameter of the hemisphere \(D = 38.4\)mm, the impact flyover distance \(\Delta = 54.5\)mm. Its geometry and computational grids are shown in Figure 7. In fluid domain, the total number of grids is 1,000,000. Moreover, the grids are densely arranged near the shock and wall for an accurate prediction of aeroheating. In addition, the smallest height of boundary-layer grid on the cylinder surface is \(2\times10^{-3}\)m.
In solid domain, the total number of grids is 100,000, and the smallest length of grid is 0.2mm. Figure 5 show the distributions of pressure and temperature along the center line of flowfield. From this Figure, the predicted position of shock is $x=-0.0547m$, which is almost consistent with the value from empirical formula, $x=-0.0545m$. Moreover, the predicted value of shock is a little larger than its experimental value not because of the effect of high-temperature air, but because of ideal gas ($\gamma=1.4$) considered in this study.

The maximum pressure and temperature of flowfield predicted behind shock are 35240Pa and 2263K, respectively. This is well-consistent with the experimental result of Allan R Wieting.

In solid domain, the total number of grids is 100,000, and the smallest length of grid is 0.2mm.

$\theta$ is the center angle of the cylindrical model. The pressure distributions in two cases are well-consistent with each other, and the difference between predicted and experimental values of pressure at stagnation-point is 7%.

The pressure distributions in two cases are well-consistent with each other, and the difference between predicted and experimental values of pressure at stagnation-point is 7%.

The maximum pressure and temperature of flowfield predicted behind shock are 35240Pa and 2263K, respectively. This is well-consistent with the experimental result of Allan R Wieting.

In Figure 6, $P_{w}/P_s$ is the ratio of wall pressure to wall pressure at stagnation point, and $\theta$ is the center angle of the cylindrical model. The pressure distributions in two cases are well-consistent with each other, and the difference between predicted and experimental values of pressure at stagnation-point is 7%.

Figure 7 shows the comparison between predicted and experimental values of cold wall heat flux along cylinder surface. In Figure 7, $q_w/q_s$ is the ratio of wall heat flux to wall heat flux at stagnation point. Here, the difference between predicted and experimental values of heat flux at stagnation-point is 16%.

Figure 8 shows the surface temperature distribution along cylinder surface. The surface temperature distribution predicted at t=5s agrees well with the experimental result. Here, the magnitude of coupling...
time step is Δtc=0.1s. Therefore, the fluid-thermal coupling strategy based on the system coupling module applied to this study can be introduced into the thermal analysis of vehicle.

IV. CONCLUSION

A new flow-thermal-structural coupling method for the hypersonic aerodynamically heated leading edge is developed in this paper. The Favre-averaged Navier-Stokes equations for the hypersonic compressible flow and the associated structure heat transfer equations are solved using the finite volume approach, and the structural equilibrium equations are solved with the finite element approach. The interaction between structure heat transfer and flow is taken into account in the coupling procedure, thus the variation of surface heating-rate and inner temperature distributions are obtained, and the aerodynamic heating process of structure can be simulated, and then structural response under aerodynamic and aero-thermal loads can also be studied. The accuracy, reliability and efficiency of this coupling method are validated by comparing with the experimental data for Mach Number 6.47 flow over a stainless steel cylinder. Good agreements of the numerical results with the experimental pressure, cold-wall heating-rate, hot-wall heating-rate and temperature distributions are achieved. Interaction among flow, structural heat transfer and deformation has been studied, which shows that the coupling effect caused by the cylinder deformation that could alter the flow field is negligible for this specific example as the structure has a high stiffness, but the flow-thermal coupling that could increase the cylinder temperature and reduce the aerodynamic heating is significant. The advantage of k – ε-R turbulence model for the hypersonic flow is also demonstrated. Studies with the grid resolution indicate that a near wall distance requirement of y+≤5 is necessary for this model to correctly predict the heating-rate.

The results presented in this paper suggest that an accurate prediction of aerodynamic and aero-thermal loads for hypersonic vehicles can be achieved with the developed method. With this work, the structural dynamic analysis can be introduced into the coupling procedure and the aero-thermo-elastic analysis of high-speed vehicles can thus be implemented, which will be considered in our future work.

V. REFERENCES


Cite this article as :
URL : http://ijsrmme.com/IJSRMME21514