CFD Activity for Future Winged Space Transport System

Yukimitsu Yamamoto (National Aerospace Laboratory yukimitu@nal.go.jp)
Takuji Kurotaki (National Aerospace Laboratory kurotaki@nal.go.jp)

Numerical Simulation for aerodynamic study of atmospheric space transport vehicles has been conducted at NAL (National Aerospace Laboratory) as the cooperative research work with NASDA (National Space Development Agency). CFD technology enlarges its application area with the progress of the computer hardware systems and now CFD has been developed as the strong aerodynamic design tool which covers flight range of space transport vehicle from the launching to the re-entry phase. Here, recent progress of numerical simulations at NAL are introduced.

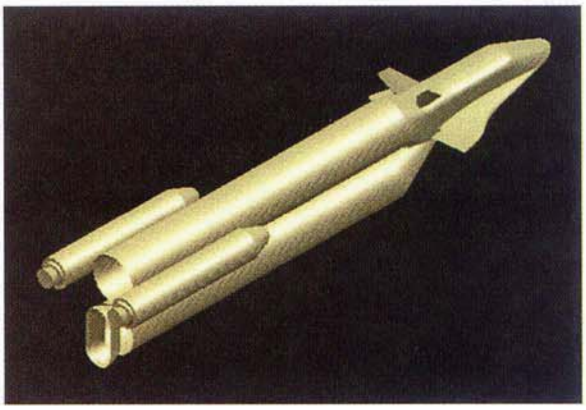


Fig.1 HOPE-X with H-IIA Rocket

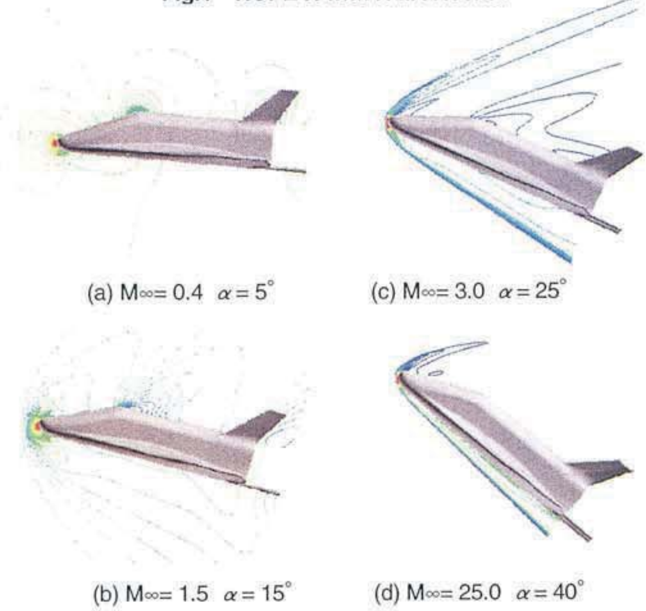
Parametric Design Study of New HOPE-X Configuration

HOPE-X (H-II Orbiting Plane Experiment) is scheduled for launch in 2004. For several years, systematic aerodynamic study has been conducted by the wind tunnel experiments and CFD simulations from low to hypersonic speeds.

Recently, in order to reduce vehicle weight and get high performance, configuration of HOPE-X is changed from tip-fin type to two-vertical fin type geometry.

For this new aerodynamic configuration design, CFD has been mainly used and Navier-Stokes calculations have been performed from low($M\infty$=0.2) to hypersonic speed ($M\infty$=26), using NAL NWT(Numerical Wind Tunnel) system. Totally, 250 computations were conducted for 10 candidate configurations of new HOPE-X and the geometry and control surface effects on aerodynamic characteristics were investigated in detail. In Fig.2, pressure contours around HOPE-X improved baseline configuration are plotted in symmetry plane at typical Mach numbers. In Fig.3, oil flow patterns of HOPE-X improved model are shown at $M\infty$=3.0 and an angle of attack α=25°, The control efficiency of The vertical fins with 10 deg deflection are investigated.

Through these systematic CFD applications, the demand from the design side is satisfied in time schedule as well as in number of test cases needed for design process. Also, through the rapid response to the design work, which is one of useful advantages of CFD, the concept of numerical wind tunnel is almost realized. In Fig.4, are shown the new HOPE-X basic design model which is determined by parametric CFD studies. In Fig.5, are shown a series of geometries investigated for the design of HOPE-X basic design model in Fig.4.



(a) $M\infty = 0.4$ $\alpha = 5°$ (c) $M\infty = 3.0$ $\alpha = 25°$

(b) $M\infty = 1.5$ $\alpha = 15°$ (d) $M\infty = 25.0$ $\alpha = 40°$

Fig.2 Pressure Contours around Two Vertical Fin Model at Various Mach Numbers

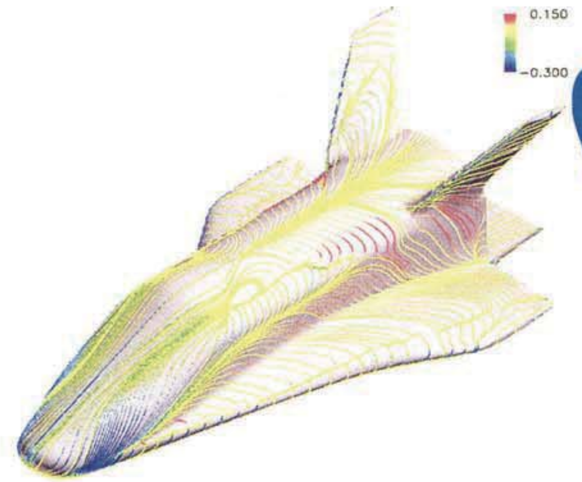


Fig.3 Oil Flow Patterns of Two Vertical Fin Model (M_∞=3.0, α=25°)

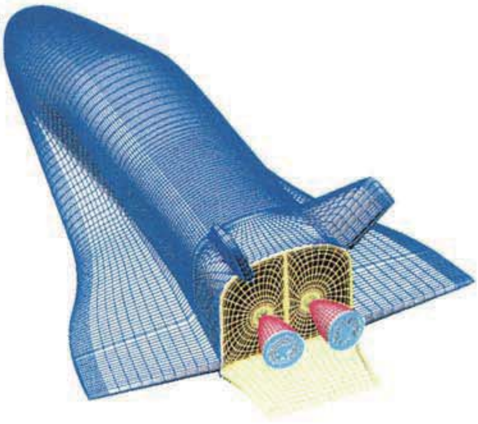
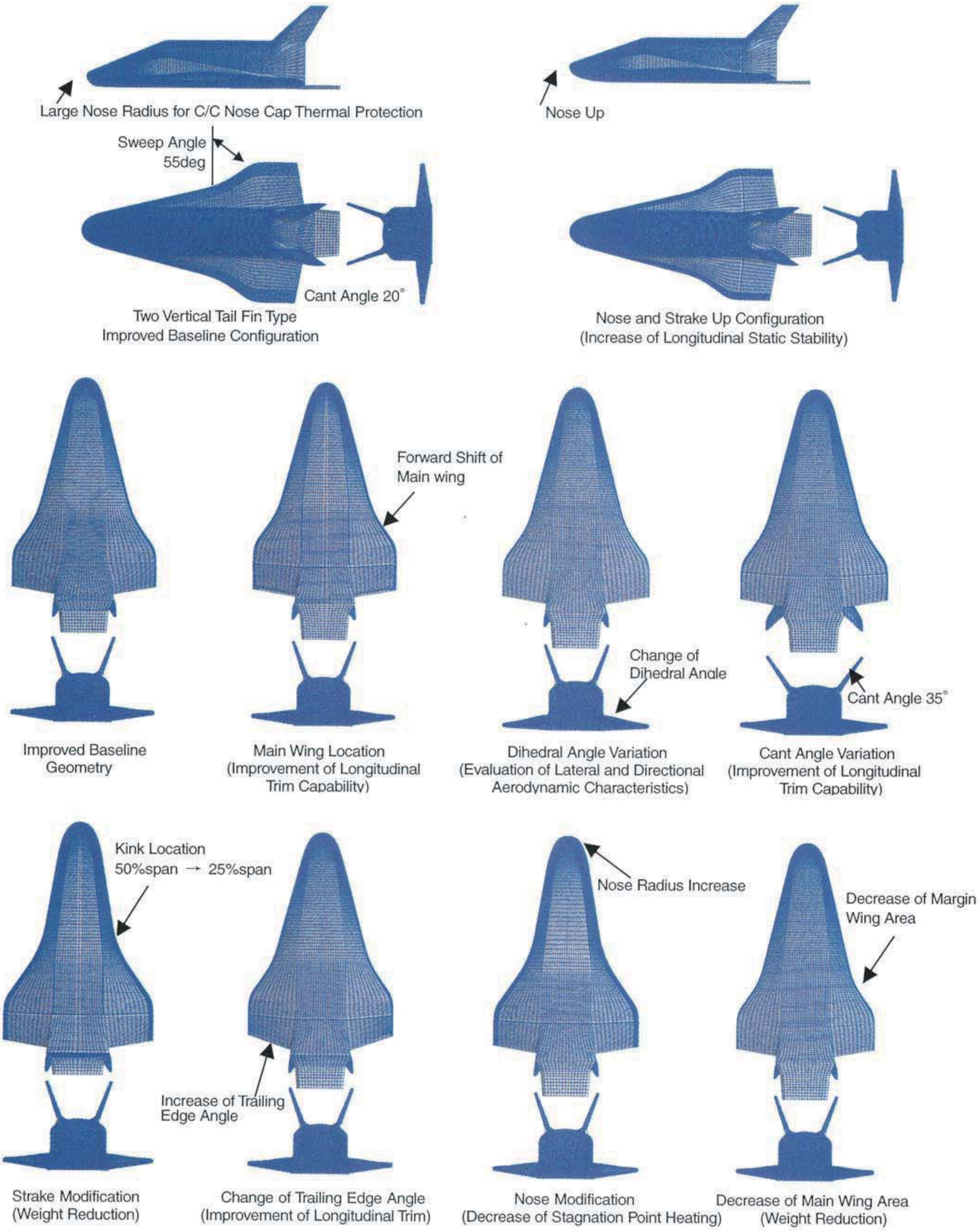


Fig.4 Surface Grid of New HOPE-X Basic Design Model



Two Vertical Tail Fin Type
Improved Baseline Configuration

Nose and Strake Up Configuration
(Increase of Longitudinal Static Stability)

Improved Baseline
Geometry

Main Wing Location
(Improvement of Longitudinal
Trim Capability)

Dihedral Angle Variation
(Evaluation of Lateral and Directional
Aerodynamic Characteristics)

Cant Angle Variation
(Improvement of Longitudinal
Trim Capability)

Strake Modification
(Weight Reduction)

Change of Trailing Edge Angle
(Improvement of Longitudinal Trim)

Nose Modification
(Decrease of Stagnation Point Heating)

Decrease of Main Wing Area
(Weight Reduction)

Fig.5 New HOPE-X Configurations

Transonic Flow Analysis for Detailed Design Phase of HOPE-X

CFD analysis using multi-block grids is developed to get more precise aerodynamic data, such as hinge moments of deflected flaps or base flow effects. In Fig.6, are shown multi-block surface grid of HOPE07 model with 20 deg elevon deflections. In order to investigate gap flows between the wing and elevon, 42 multi-block grids are generated and total number of grid points are 350 millions. Figure 7 shows pressure contours at M_∞= 0.7 and α = 0° . Total Reynolds number is 1 x 10^8 based on the body length. By this CFD analysis, hinge moment of deflected control surfaces can be evaluated more exactly.

On the other hand, evaluation of base flow effects on aerodynamic characteristics are very important and data correlation between wind tunnel results and flight is studied. Figure 8 shows of pressure contours of new HOPE-X configuration containing base flow regions by using the multi-block grids. CFD now plays an important role for the detailed design phase and complicated flow can be evaluated exactly.

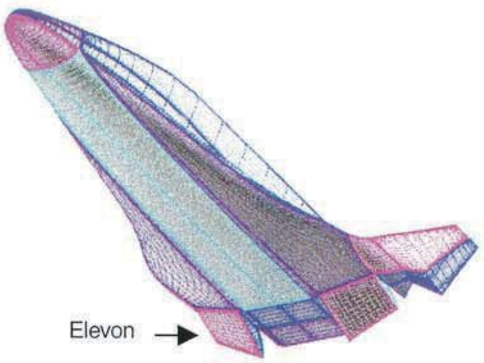


Fig.6 Multiblock Surface Grid for Deflected Elevon Analysis

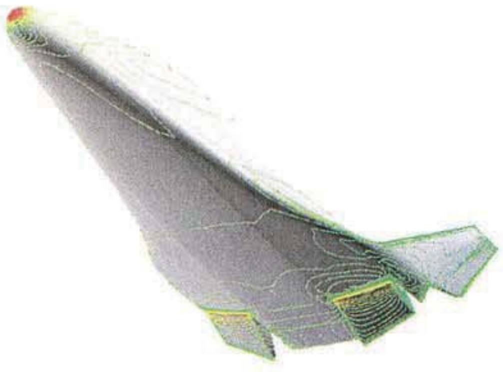


Fig.7 Surface Pressure Contours
M_∞= 0.7 α = 0° , Re_∞=1x10^8

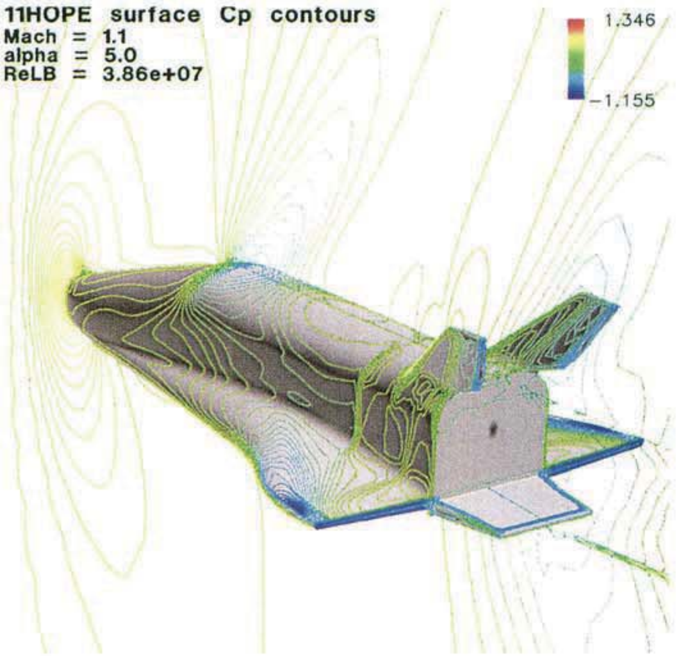
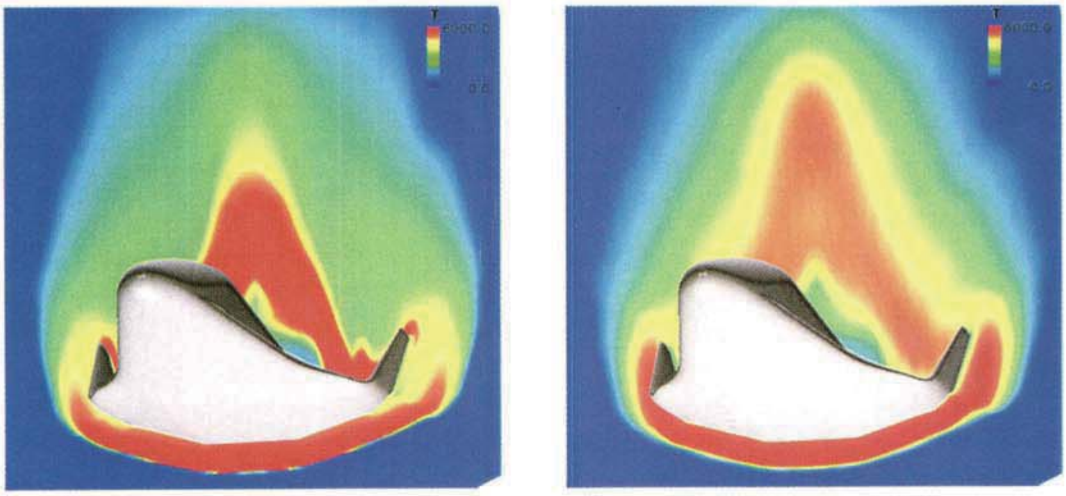


Fig.8 Pressure Contours around New HOPE-X Including Base Flows M_∞=1.1, α = 5° Re_∞=3.86 x 10^7

HOPE-X High Temperature Real Gas Analysis

In high temperature environments of re-entry flight of HOPE-X, chemical reactions, in which air molecules are dissociated and ionized, must be solved. Figure 9 shows cross sectional temperature contours around HOPE07 model at $M\infty$=26, α =40° . Flow analysis is made by using two-temperature chemically non-equilibrium Navier-Stokes code. In the figure, translational / rotational temperature are shown with the vibrational one. At present, finite rate catalycity effects are investigated by using flight data and developments of catalycity models are underway. Validation of the real gas CFD code are progressed by the comparisons of experimental data from NAL HIEST and ONERA F4 shock tunnels.



(a) Translational and Rotational Temperature

(b) Vibrational Temperature

Fig. 9 HOPE07 Real Gas Flow Analysis M_∞=26, α =40°

Aero-Thermal Structural Analysis and Application of Multi-Disciplinary Simulations to the Future Space Transport Systems

HYFLEX flight experiment was conducted in Feb. 1996. In this hypersonic flight, aerodynamic data and temperature data on TPS materials and internal structure parts were obtained. Figure 10 shows surface temperature history of HYFLEX flight, simulated by CFD-FEM coupling analysis. This type of multi-disciplinary simulation demonstrated its usefulness for high quality evaluation of aerothermal environments of re-entry flight.

In order to develop the present method to the total thermal-structural analysis, FEM mesh is generated including internal structures, such as frames and stringers as shown in Fig.11. In Fig.12, are shown typical temperature contours of TPS and internal aruminium frames. Flow-thermal analysis are made to the flight time of 300 sec and good agreements with flight data are obtained for temperature increase of HYFLEX vehicle.

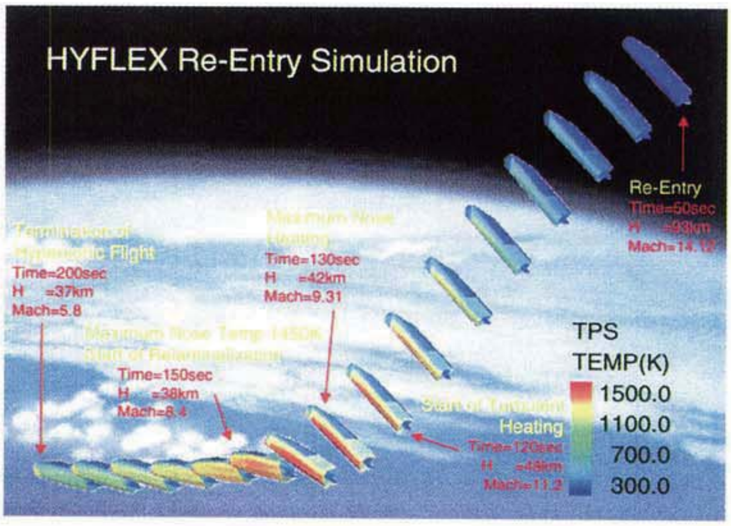


Fig.10 HYFLEX Re-Entry Flight History and TPS Surface Temperature Simulation

Finally, in Fig.13, pressure contours and oil flow patterns are shown on the X-33 type reusable rocket. In the developments of the future space transport systems, re-usability and weight reductions are the most important technological breakthrough points and CFD oriented multi-disciplinary simulations are needed for these optimum and precise design process. This study is progressed at NAL.

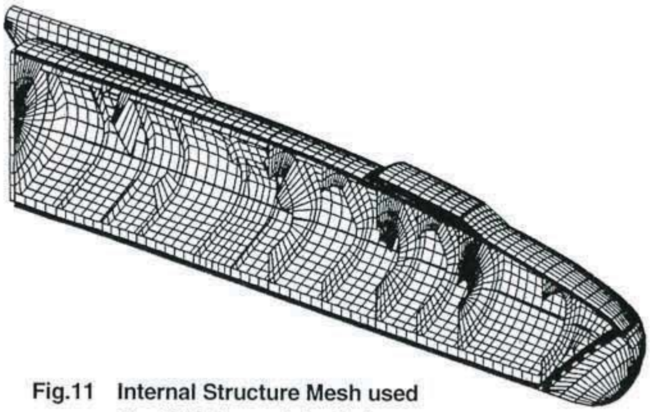
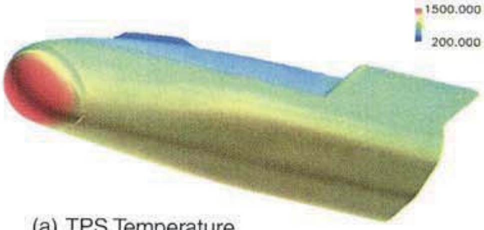
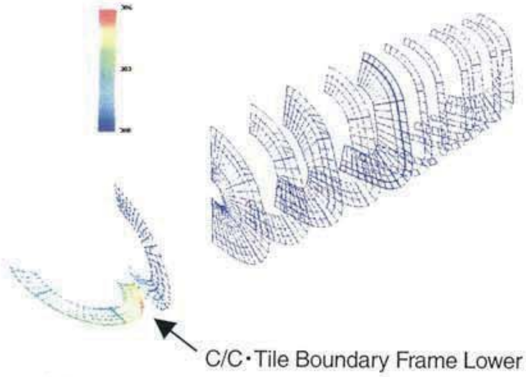


Fig.11 Internal Structure Mesh used for FEM Thermal Analysis



(a) TPS Temperature Contours at Flight Time150 sec.



(b) Temerature Contours of Internal Frame Structure at Flight Time 300sec

Fig.12 Aero-Thermal Analysis of HYFLEX Flight by CFD/FEM Coupling

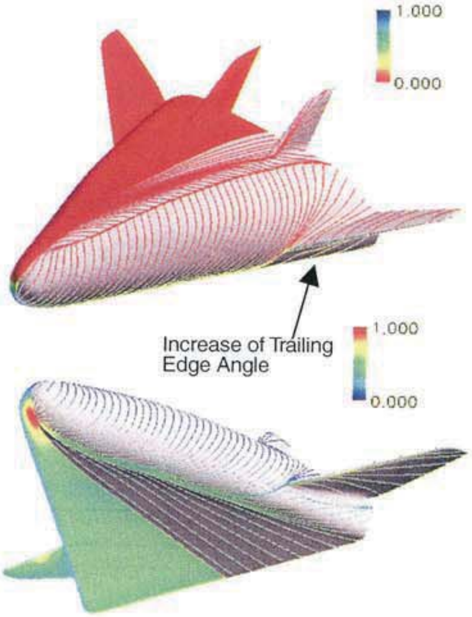


Fig.13 Pressure and Oil Flow Patterns around Future Re-usable X-33 type Vehicle M∞= 20.0 α=40°

Computational Aerodynamics Research Group

E-mail : aerocarg@nal.go.jp