ISSN 0389-4010
UDC 629.7.036.22

TECHNICAL REPORT OF NATIONAL AEROSPACE LABORATORY

TR-1187T

A Study on Heat Transfer in a Scramjet Leading Edge Model

Shuichi UEDA, Noboru SAKURANAKA,
Toshihito SAITO, Katsuhiro ITOH,
Yoshio WAKAMATSU, and Kiwamu IMAI

December 1992

NATIONAL AEROSPACE LABORATORY

CHŌFU, TOKYO, JAPAN

A Study on Heat Transfer in a Scramjet Leading Edge Model*

Shuichi UEDA*1, Noboru SAKURANAKA*1, Toshihito SAITO*1
Katsuhiro ITOH*1, Yoshio WAKAMATSU*1, and Kiwamu IMAI*2

ABSTRACT

Heating tests and analysis of a cooling panel that simulates the leading edge of the scramjet engine inlet or strut were performed using crossflow type water-cooled panel with a circular leading edge. The cooling panel was fabricated by nickel electroforming and heated by supersonic hot gas. An NTO/MMH rectangular chamber was used as the gas generator. The free stream Mach number at the nozzle exit was about 2.67. Water was used as the coolant and heat flux distribution was measured based on the temperature increase. Thermal analysis were performed using a two-dimensional CFD code and a finite element code. The results of the analysis were compared with the experimental data.

Key Words: Scramjet engine, Supersonic flow, Cooling Structure, Heat Transfer.

概　要

スクラムジェットエンジンのインレットやストラット前縁部を模擬した冷却構造パネルの加熱試験および解析を行なった。円弧型の前縁部を持つニッケル電鋳製の冷却パネルを超音速の高温ガス流中で加熱した。ガス発生器として，NTO／MMH を酸化剤及び燃料とする矩形燃焼器を使用した。ガス発生器出口の一様流マッハ数は約 2.67 である。冷却材として水を使用し，冷却水温の上昇から熱流束分布を測定した。二次元の CFD コードおよび有限要素法コードによる解析を行ない，実験結果と比較した。

1. Introduction

Airframe integrated scramjet engines are expected to be used for the propulsion system of a single-stage-to-orbit space plane[1),2)]. Scramjet engines are exposed to severe thermal environments during flight. The leading edges of scramjet inlets and struts must minimize distortions of the flow field due to excessive blunting and thermal warping of the compression surface to achieve the required high inlet performance. With minimization of the leading edge curvature, aerodynamic heating becomes severe[3)]. Shock-wave interference heating is also a critical problem in the structural design of the thermal protection system of the space plane. Extremely high heat flux can occur in highly localized regions where the interference pattern impinges on the surface[4),5)]. Because of this extremely high heat flux, the radiation equilibrium tem-

---

* receieved 8 July 1992
*1 Kakuda Research Center
*2 Ishikawajima-Harima Heavy Industries, Co. Ltd.

perature exceeds the material limitations and some means of active cooling for thermal protection must be provided[6].

One of the more attractive means of active cooling for this application is regenerative cooling (use of the fuel as a coolant). This technique appears especially attractive because cryogenic hydrogen, which is to be used as a fuel, is an excellent coolant. Conservation of coolant and minimization of weight become paramount, because the scramjet engine has large areas which require active cooling. Three basic types of cooling of the leading edge can be considered: impingement, parallel-flow and cross flow cooling. Impingement can be directed parallel to the hot gas flow or normal to the sweep line. With parallel-flow cooling, the flow turns nearly 180 deg, producing a near-impingement cooling effect. However, these two types of cooling are difficult to achieve in practice. In crossflow cooling, the coolant flows in a channel just behind and parallel to the leading edge. The coolant heat transfer coefficient of crossflow cooling is lower than that of the other two types and the pressure drop is higher. However, this type of cooling is easier to achieve than the other two[7].

In this study, a cooling panel that simulates the leading edge of a scramjet engine inlet or strut was subjected to heating tests and results were analyzed. The main purpose of these tests was to determine the design capability of the cooling structure and to obtain fundamental data for designing an actual thermal protection system. A crossflow water-cooled panel that has a circular leading edge was selected. The cooling panel was fabricated by nickel electroforming and heated by high temperature supersonic flow. An NTO/MMH rectangular chamber was used as the gas generator[8]. Mean heat flux distribution and wall temperatures were measured. Two-dimensional heat transfer analyses were performed by using the viscous CFD code and the FEM code.

2. Test Components

2.1 Hot Gas Test Facility

All tests were conducted at the High Altitude Test Facility of the National Aerospace Laboratory at Kakuda[9]. This facility is capable of obtaining an environmental pressure of about 0.1 kPa.

An NTO/MMH rectangular chamber with a 4:1 area ratio nozzle was used as the gas generator. The shape of the nozzle exit is rectangular, 147.3 mm by 32 mm. This chamber operated at a combustion chamber pressure of 1MPa and a mixture ratio of 2.0. Temperature of the combustion gas was estimated about 3170 K by the One-Dimensional Chemical Equilibrium Program[10]. The composites of this combustion gas are primary nitrogen and water vapor. Static pressure at the nozzle exit was about 50 kPa, the Mach number was 2.67 and the specific heat ratio wae 1.26. The uniformity of the nozzle exit flow was measured by total pressure probe. Distribution of total pressure was within ±2% in all portions except the boundary layer.[8] The fuel and oxidizer flow rates to the engine were measured by turbine flowmeters. The volume flow rate, temperature, pressure in the supply line and the engine combustion pressure were sufficient to calculate ideal gas stream characteristics.

Water was used as coolant for the cooling panel instead of liquid hydrogen which will be used as the actual coolant of the space plane's cooling system. A water flow rate of 1000 cm^3/s (1 liter/s) at a panel exit pressure up to 2 MPa is available.

Figure 1 is a schematic representation of the

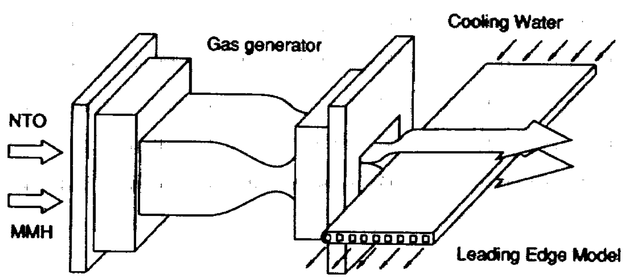


Figure 1. Schematic Representation of the Heating Test

heating test. The cooling panel was set in front of the gas generator nozzle. Hot gas from the gas generator nozzle heated the leading edge model cooled by water.

2.2 Test Panel

Various types of cooling were considered; impingement, parallel-flow and crossflow. The main purposes of this test were to validate the design techniques of the cooling structure and obtain fundamental data on leading edge heating for application to the design of an advanced cooling structure. To achieve this objective, the shape of the test panel must meet certain criteria.

1) The shape must be simple enough to enable to analyses of its thermal characteristics employing a simple geometrical model for which many experimental and analytical data are available.
2) The test panel must be sufficiently durable to endure various severe heating test conditions.
3) The test panel must enable measurement of local heat flux distribution along the streamline.

A crossflow type cooling panel was selected because of its simplicity and because it enables

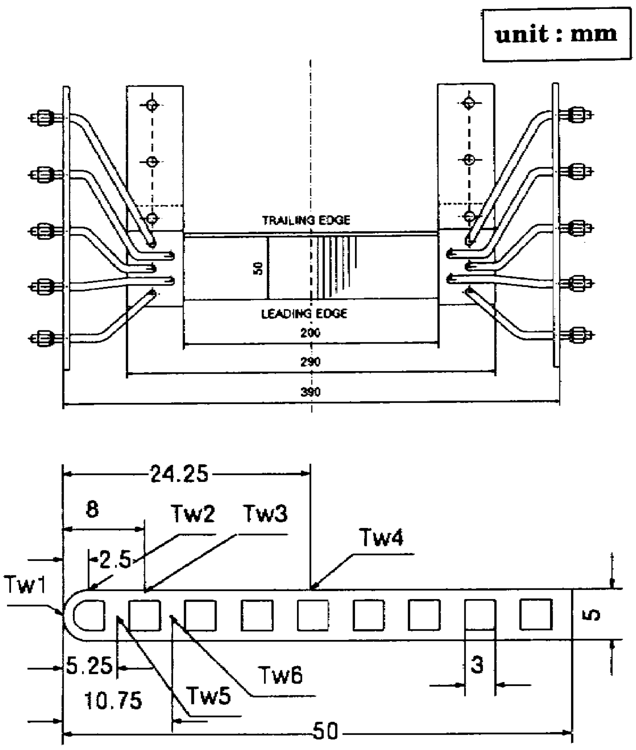


Figure 2. Crossflow Cooling Leading Edge Model

measurement of heat flux distribution. A detailed schematic of the crossflow panel is shown in Fig. 2. The test panel is 50 mm wide and 200 mm long, and has a circular leading edge with a diameter of 5 mm. It has 1 mm nickel walls, nine 3 mm square cooling passages and 2.5 mm partitions. All heating regions of the cooling panel are made of nickel. Nickel was chosen because of its high thermal conductivity and high melting temperature. The panel was made by electroforming and connected with fitting equipment by wax jointing.

The cycle life of the designed cooling panel were estimated by using the NASTRAN and the MARC finite element code. The required cycle life of this test panel is greater than 10 cycles. This durability of this test panel is sufficient, because the actual load to the panel is a half of that of the maximum condition.

Local heat flux distribution was measured by eight 3 mm square cooling passages. Coolant inlet and exit temperature and pressure as well as the mass flow rate of each cooling passage were measured. Wall temperature of the test panel was measured by type-K thermocouples imbedded in lateral grooves machined in the surface. The diameter of the thermocouples is 0.5 mm. One thermocouple (Tw1) is located at the leading edge, three (Tw2, Tw3, Tw4) are located in the surface of the panel, and two (Tw5, Tw6) are located inside of the test panel. A ninth cooling passage is used to protect the thermocouples from hot gas, and cannot, therefore, be used for heat flux measurement.

3. Numerical Analysis

Design of the leading edge model was observed assuming uniform flow conditions. However, because of the difference between static pressure of the gas generator exit and the environmental pressure, expansion or compression waves generate from the nozzle edge. Combustion pressure of the gas generator is constant at 1 MPa, and nozzle exit static pressure is 50 kPa. In the high altitude test, environmental pressure is about 2.5 kPa and expansion waves generate from the nozzle edge. In the sea-level

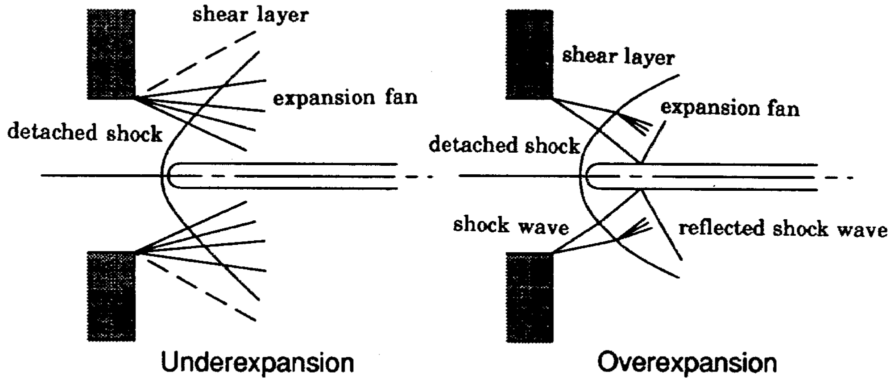


Figure 3. Interaction between the Test Panel and the Waves

test, environmental pressure is 100 kPa (standard atmospheric pressure) and compression waves generate from the nozzle edge. Figure 3 represents the interaction between the cooling panel and the waves from the nozzle edge. In the underexpansion conditions that occur in high altitude tests, expansion waves incident to the panel accelerate the boundary layer and decrease the pressure on and heat flux to the panel. In the overexpansion conditions that occur in sea-level tests, shock waves are incident to the panel, and extremely high heat flux can occur in highly localized regions. This high heat flux decreases due to interference between the shear layer and the panel.

The purpose of numerical analysis of this test is to validate equations used for structure design and to estimate the influence of various waves generated from the nozzle edge on heat flux to the panel. Two-dimensional analysis is sufficient because of a large aspect ratio of 9.2 (147.3/16) at the nozzle exit. The effect of waves from the side edge must be small. Steady-state heat transfer analysis requires that heat transfer coefficients be assigned to both the external surface and cooling channel surface elements of the panel. Two-dimensional analysis of the Navier-Stokes equation is conducted to estimate the heat transfer coefficient on the gas side. The Harten-Yee type TVD code[11] with the k-ϵ turbulent model[12] are used. Figure 4 shows the 201 by 41 numerical grid used for this analysis. In the analysis, only half of the grid was used when the symmetrical boundary condition was employed. Using such results as heating side boundary conditions, the model was analyzed with the ADINA finite element code.

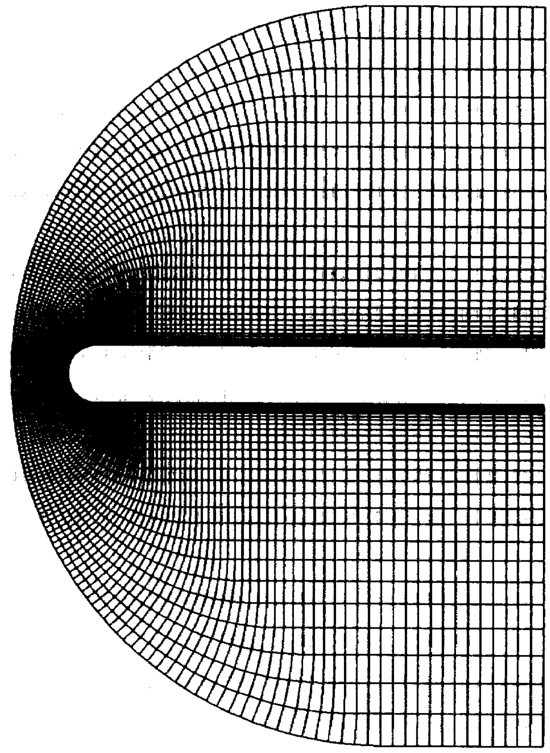


Figure 4. Numerical Grid around the Leading Edge Model

The cooling side heat transfer coefficient is calculated using the empirical relation[13]. ADINA is used to analyze the temperature distribution inside the panel and the heat flux to the cooling water. The two-dimensional model of the cooling panel made up of 668 nodes and 534 elements is shown in Fig. 5.

Figure 6 shows an example of flow field analysis. Uniform flow condition is used. This figure shows the Mach number contours of the

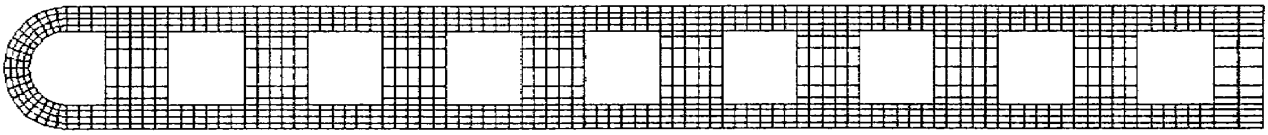


Figure 5. Two-dimensional Finite Element Model

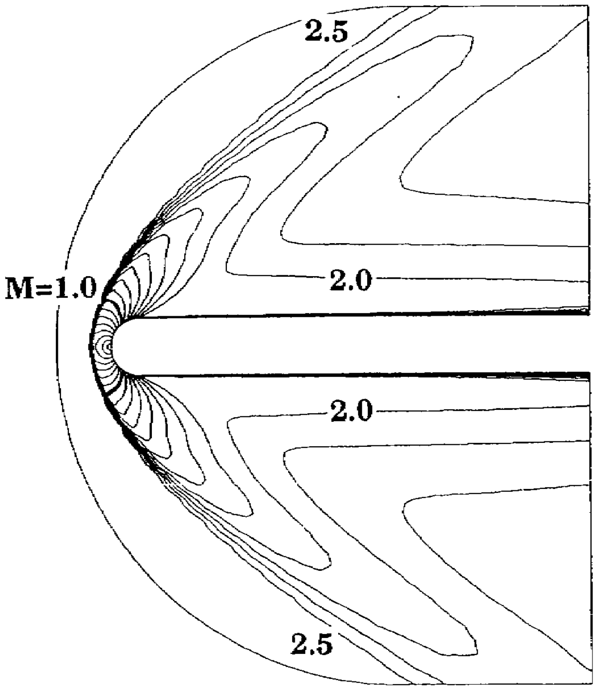


Figure 6. Mach Contour (Uniform flow)

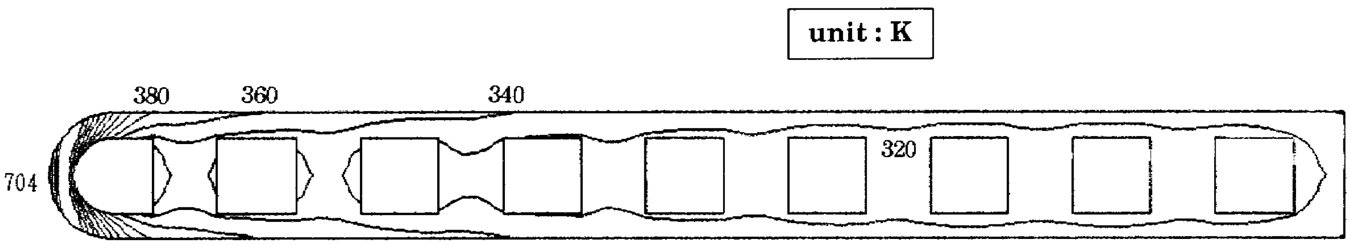


Figure 7. Temperature Contour (Uniform flow)

flow field around the cooling panel. Figure 7 shows the temperature distribution inside the model calculated by the ADINA using flow field analysis results as heating side boundary conditions. Temperature is low and almost constant in the parallel portion of the cooling panel.

4. Results and Discussion

The experimental procedure was to fire the gas generator for 10 sec and record the transient response of the instrumentation. The actual engine firing takes places about 0.3 sec after the start of the sequence. Figure 8 shows time variation of the coolant outlet temperature. TCWI is inlet temperature and TCO1~TCO5 are outlet temperatures of cooling channel 1~5. The wall temperature of the test panel rises rapidly and approaches steady state at about 1.5 sec into the sequence. The propellants are shut off at 10 sec, after which the temperature decays rapidly. The coolant flows during the entire firing sequence. The coolant flow rate of cooling channels two to eight are almost same, about 100 cm^3/s, and the flow rate of the leading edge cooling channel is about twice that of the others. Neither remarkable cooling flow rate nor cooling channel pressure fluctuation was observed during the test. This indicates that notable boiling heat transfer did not occur inside the cooling channel. After the heating test, no damage to the test panel was observed except some stains caused by the reactant. The heating area used to evaluate mean heat flux is the width of gas generator nozzle (147.3 mm) by the cooling channel interval (5.5 mm).

4.1 Underexpansion Tests

Underexpansion tests were conducted using the High Altitude Test Facility. The nozzle exit pressure is about 50 kPa and environmental pressure is about 2.5 kPa. The minimum distance between the nozzle edge and the leading edge is 7.5 mm. Two setting were conducted, 7.5 mm and 13 mm.

Figure 9 shows Mach number contours in the calculated region. This calculation simulate the heating test set the leading edge model 7.5 mm downstream of the nozzle edge. Constant temperature of 800 K was employed as the wall boundary condition. Expansion waves from the nozzle edge bend the bow shock and accelerate the boundary layer. The computed wall pressure and heat flux distributions over the leading edge segment are shown in Fig. 10. This figure shows the characteristically high stagnation pressure and heat flux level, which drops rapidly around the cylinder to the level in the downstream portion of the test specimen. Heat flux at the leading edge was estimated about 15 MW/m^2. Results of uniform flow calculation are also shown in this figure. The influence of expansion

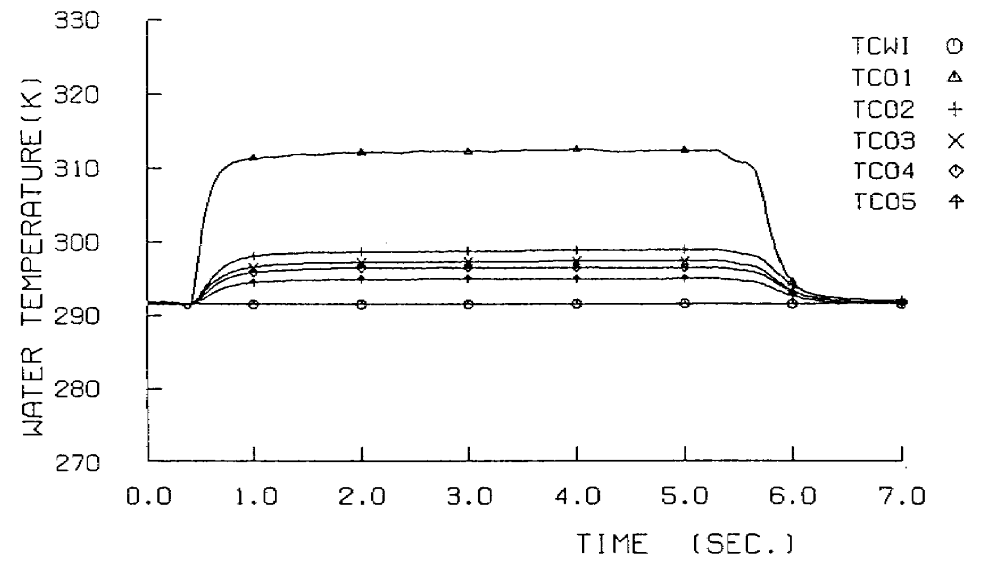


Figure 8. Time Variation of Coolant Outlet Temperature

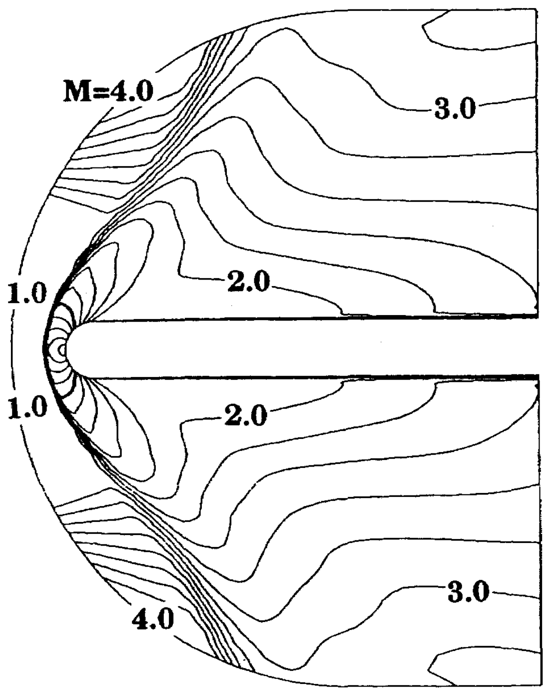


Figure 9. Mach Contour (Underexpansion)

waves appears in the rear part of the panel but is small. Figure 11 shows the temperature distribution inside the cooling panel. The Maximum temperature at the leading edge was estimated about 700 K. These temperature contours are almost the same as those under uniform flow conditions except for a slight difference at the rear part of the panel.

Figure 12 shows comparison of mean heat flux distribution between experiment and analysis. A very high heat flux of 7 MW/m^2 was observed at the leading edge portion. Good agreement between experiment and analysis is observed. No difference caused by changing the distance between the nozzle exit and the leading edge was observed. This result indicates the design method of the cooling structure is sufficiently reliable, that two-dimensional analysis of this test is suitable and that the underexpansion test can substitute for the uniform flow test at least in the front part of the panel.

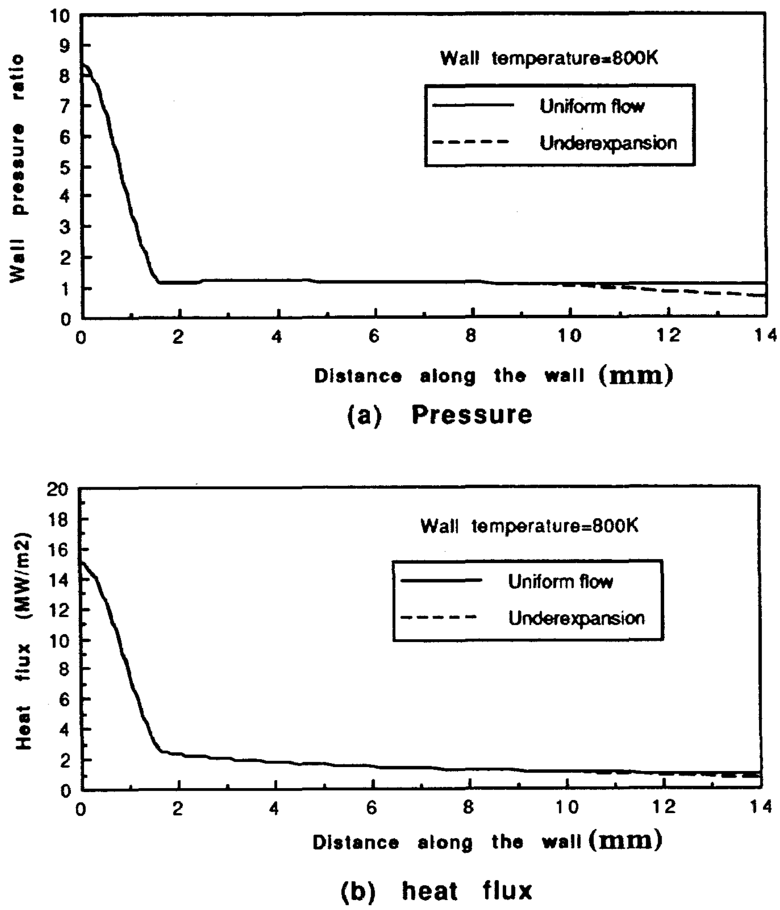


Figure 10. Wall Pressure and Heat Flux Dimtribution (Underexpansion)

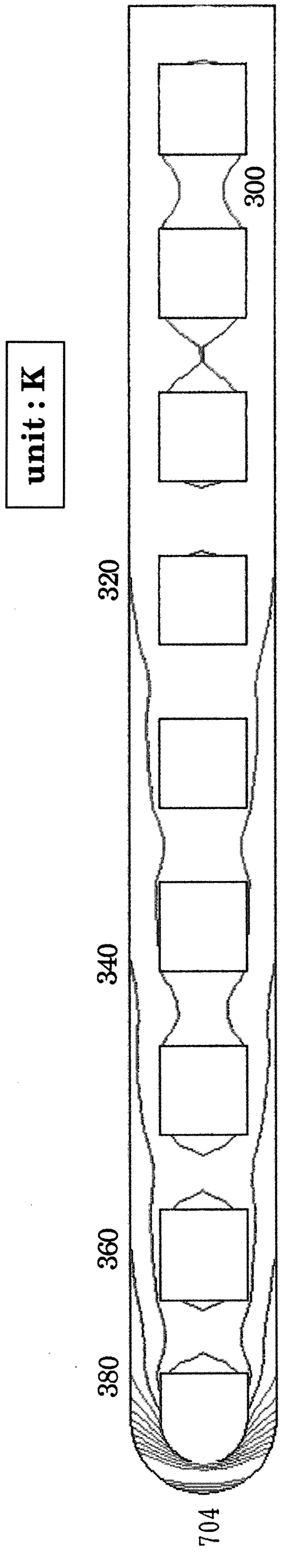


Figure 11. Temperature Contour
(Underexpansion)

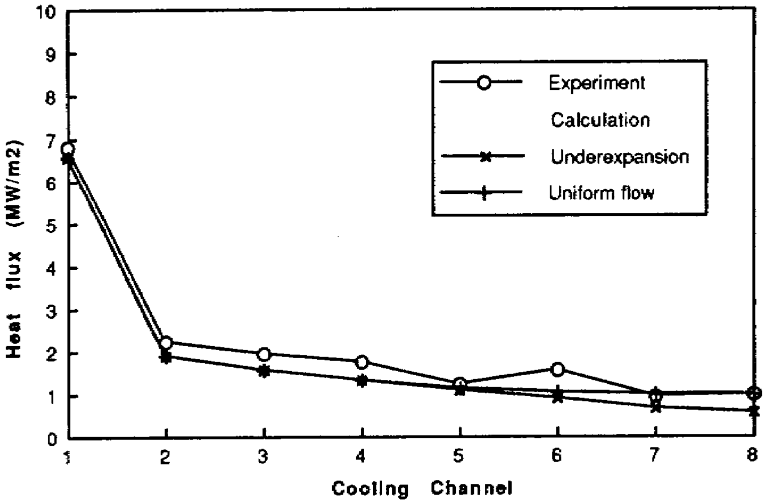


Figure 12. Mean Heat Flux Distribution (Underexpansion test)

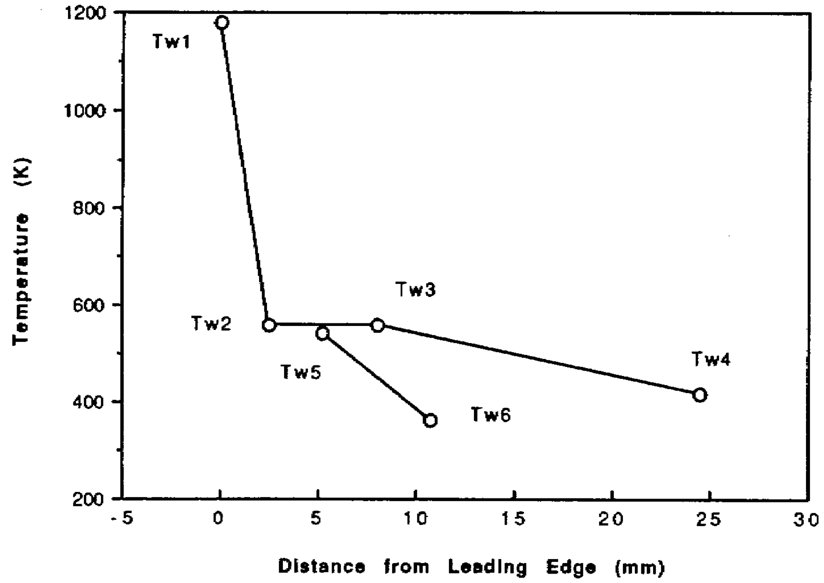


Figure 13. Measured Wall Temperature (Underexpansion test)

Figure 13 shows the measured temperatures of the cooling panel. These temperatures are higher than the predicted values in Fig. 11. This difference are caused by the difference of thermal conductivity between the thermocouple and the panel and by thermal resistance where these two components meet.

4.2 Overexpansion Tests

Overexpansion tests were also conducted at the High Altitude Test Facility but did not employ its exhaust system. Thus, environmental pressure is the same as atmospheric pressure. Other test conditions are the same as those of the underexpansion tests.

Figure 14 shows Mach number contours of the flow field around the test panel. This calculation simulate the heating test set the leading edge model 7.5 mm downstream of the nozzle edge. Boundary layer separation occurs due to incident shock waves. High heat flux caused by reattachment of the separated boundary layer and the boundary layer transition to the turbulence. The computed wall pressure and heat flux distribution over the leading edge segment are shown in Fig. 15. This figure shows the char-

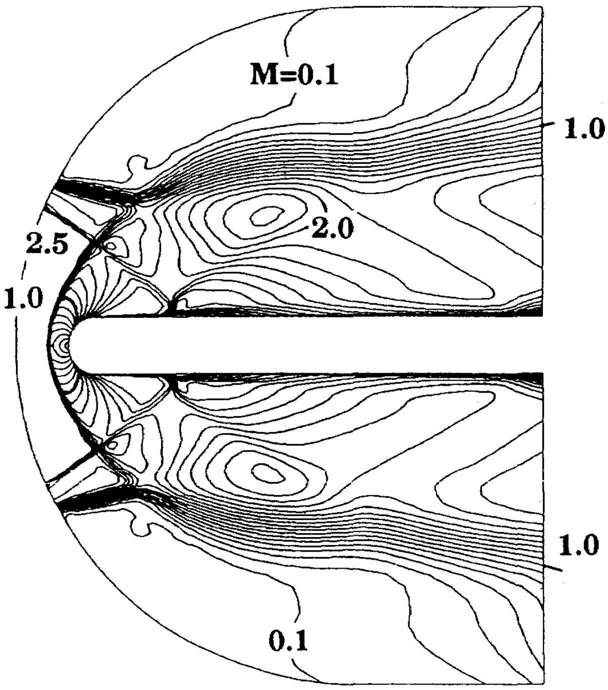
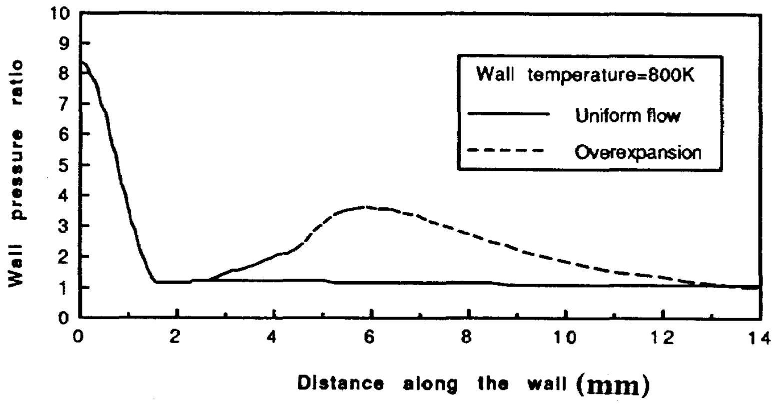


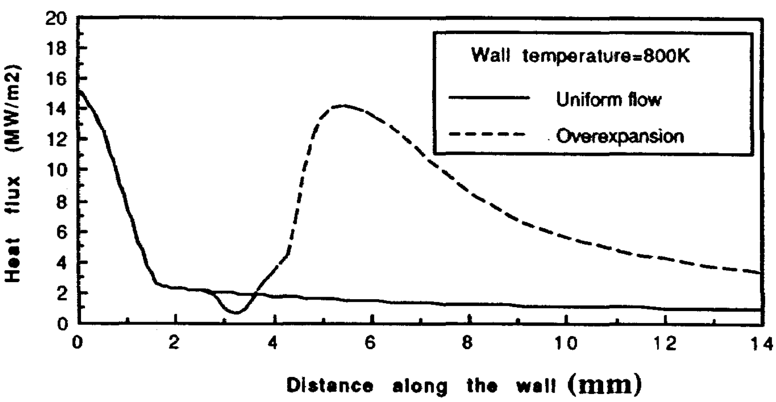
Figure 14. Mach Contour (Overexpansion)

acteristically high stagnation pressure and heat flux level, which drop rapidly around the cylinder to the level in the downstream portion of the test specimen but increase once more due to boundary layer separation and transition to the turbulence. Results of uniform flow calculations are also shown in this figure. The influence of shock waves appears to be remarkable. Peak heat flux of the interaction region was estimated to be about 15 MW/m^2 by CFD calculation. This heat flux is as high as that at the leading edge point. Figure 16 shows the temperature distribution inside the cooling panel with a very high wall temperature at the third cooling passage. Maximum temperature in this portion is as high as that of the leading edge stagnation point.

Figure 17 shows a comparison of mean heat flux distribution between experiment and analysis. Contrary to the underexpansion test, high heat flux is observed in cooling passages two



(a) Pressure



(b) heat flux

Figure 15. Wall Pressure and Heat Flux Distribution (Overexpansion)

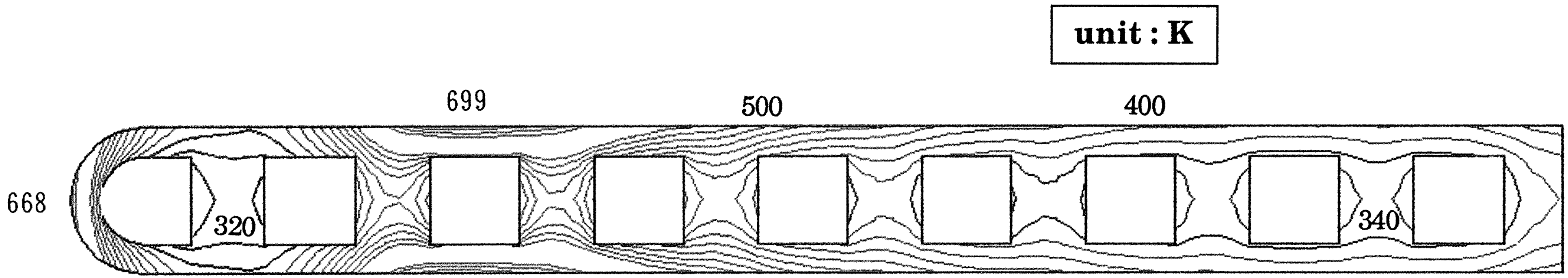


Figure 16. Temperature Contour
(Overexpansion)

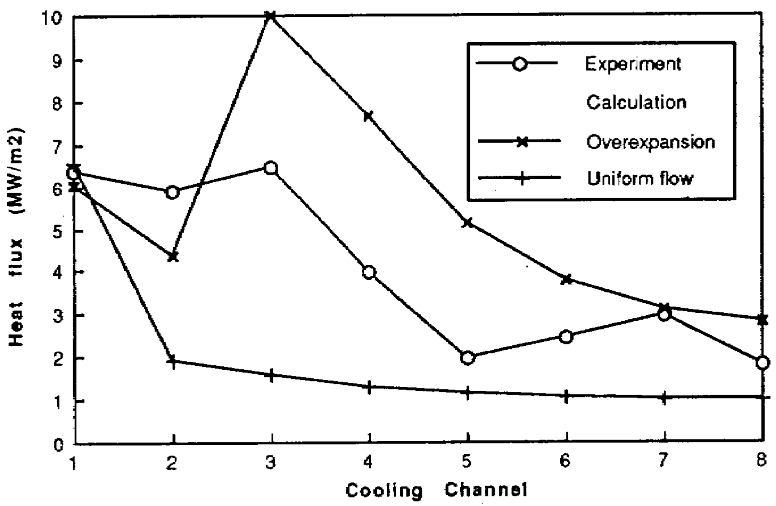


Figure 17. Mean Heat Flux Distribution (Overexpansion test)

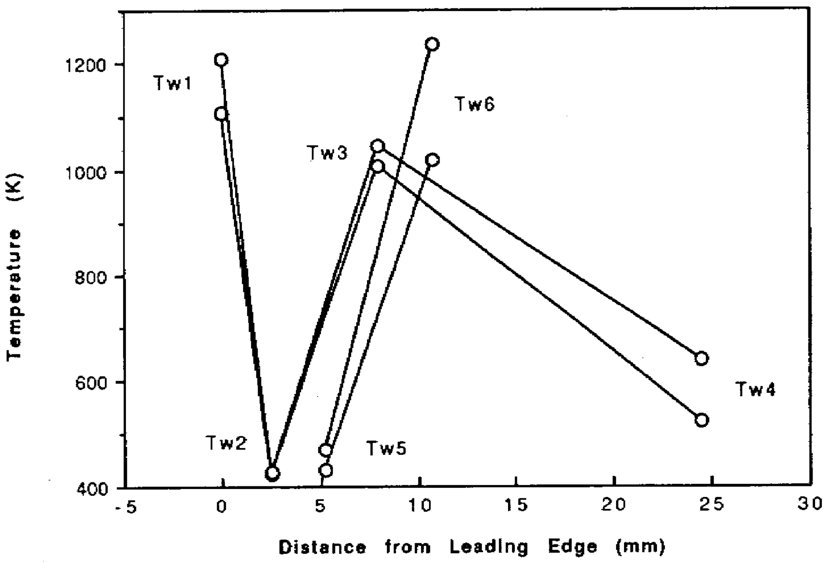


Figure 18. Measured Wall Temperature (Overexpansion test)

and three when the panel is set 7.5 mm downstream of the nozzle edge. Calculated results qualitatively agree with experimental data but underestimate the heat flux in the second cooling passage and overestimate heat flux in cooling passages 3~6. There are several possible reasons for this difference.

1) The test panel was set slightly off-center in the gas generator nozzle.
2) The boundary layer of the gas generator nozzle was not taken into consideration.
3) The three-dimensional effect is larger than in the underexpansion case.
4) It is difficult to calculate boundary translation and separation exactly by use of the current turbulence model.

Figure 18 shows temperature measured by thermocouples. It can be seen that very high temperatures exist not only at the leading edge but also between cooling channels two and three. This is induced by shock waves.

5. Concluding Remarks

A crossflow type water-cooled panel with a circular leading edge was heated by supersonic hot gas. An NTO/MMH rectangular chamber was used as the gas generator. Stagnation pressure and temperature of the gas generator were 1 MPa and 3170 K. The free stream Mach number at the nozzle exit was about 2.67. The results of heating test can be summarized as follows.

1) Heat flux and wall temperature at the leading edge were estimated about 15 MW/m^2 and 700 K. The predicted results of the crossflow model are compared with the experimental data and found to be in good agreement in the underexpansion test conditions.
2) The influence of expansion waves is small. Thus, the underexpansion test can substitute for the uniform flow test at least in the front part of the panel and the two-dimensional analysis of this test is suitable.
3) The shock waves from the nozzle edge are incident to the cooling panel and cause high heat flux. Peak heat flux of the interaction region was estimated to be about 15 MW/m^2 by CFD calculation. This heat flux is as high as that at the leading edge.
4) Some improvement is necessary to adequately analyze heat flux to the cooling panel when shock waves are incident to the panel.

This study is a first step in research on the cooling structure for use with the scramjet engine. We are now beginning to fabricate a more realistic cooling structure shaped almost the same shape as the actual leading edge of the experimental scramjet engine. A test facility for testing the cooling structure in supersonic airflow that simulates real circumstances of the scramjet engine is near completion.

Acknowledgement

The authors express their gratitude to Messrs. N. Niino, K. Kusaka, T. Kumagai, K. Kisara, T. Sudo (NAL) and M. Sayama, (IHI Co. Ltd) for their active participation in the test series.

References

1. T.A. Heppenheimer, "The National Aerospace Plane," Pasha Market Intelligence, 1987.
2. R.A. Jones and P.W. Huber, "Toward Scramjet Aircraft," J. Astronautics and Aeronautics, February 1978, pp. 38-48.
3. R.D. Neumann and J.R. Hayes, "Introduction to Aerodynamic Heating Analysis of Supersonic Missiles," in Tactical Missile Aerodynamics, M.J. Hemsch and J.N. Nielsen eds., Progress in Astronautics and Aeronautics, Vol. 104, 1986., pp. 421-481, AIAA, New York.
4. A.R. Wieting and M.S. Holding, "Experimental Shock-Wave Interference Heating on a Cylinder at Mach 6 and 8," AIAA Journal, Vol. 27, No. 11, November 1989, pp. 1557-1565.
5. G.H. Klopfer and H.C. Yee, "Viscous Hypersonic Shock-on-Shock Interaction on Blunt Cowl Lips," AIAA 88-0233, January 1988.
6. O.A. Buchmann, "Thermal-Structural Design Study of an Airframe-Integrated Scramjet," NASA CR-3141, 1979.
7. H.J. Gladden and M.E. Melis, "A High Heat Flux Experiment for Verification of Thermostructural Analysis," NASA TM-100931, 1989.
8. H. Miyajima, et al., "Study on SCRAMJET Nozzles (1) Performance of Two Dimensional Nozzles," NAL TR-1149, 1992.
9. National Aerospace Laboratory, "High Altitude Test Facility for Rocket Engine at NAL," NAL TR-454, April 1976.
10. K. Nakahashi, H. Miyajima, K. Kisara, A. Moro, "Pridiction Method of Rocket Nozzle Performance," NAL TR-771, July 1983.
11. H.C. Yee and A. Harten, "Implicit TVD Schemes for Hyperbolic Conservation Laws in Curvilinear Cordinates," AIAA Journal, Vol. 25, No. 2, February 1987.
12. R.H. Nichols, "A Two-Equation Model for Compressible Flows" AIAA 90-0494, January 1990.
13. S. Omori, K. W. Gross and A. Krebsbach, "Wall Temperature Distribution Calculation for a Rocket Nozzle Contour," NASA TN D-6825, July 1972.

TECHNICAL REPORT OF NATIONAL AEROSPACE LABORATORY
TR-1187T

航空宇宙技術研究所報告1187T号（欧文）

平成 4 年12月発行

発行所　航空宇宙技術研究所
東京都調布市深大寺東町 7－44－1
電話　三鷹(0422) 47－5911 (大代表)　〒182
印刷所　株式会社　共進
東京都杉並区久我山 5－6－17

Published by
NATIONAL AEROSPACE LABORATORY
7-44-1 Jindaiji higashi, Chōfu, Tokyo 182
JAPAN

Printed in Japan