Numerical Analysis of Scramjet Internal Flow by Hybrid Unstructured Grid Method*

Masatoshi KODERA**, Yasutaka IGARASHI***,
Kazuhiro NAKAHASHI****, Takeshi KANDA**,
Tetsuo HIRAIWA** and Tohru MITANI**

Computations of internal viscous flowfields of scramjet models were conducted at inflow Mach numbers of 3.4 and 5.45. A hybrid unstructured grid method was used to compute the scramjet models with and without a short strut. The numerical method to solve the Navier-Stokes equations on the hybrid unstructured grid was developed using a finite volume cell vertex scheme and the lower-upper symmetric Gauss-Seidel (LU-SGS) implicit time integration algorithm. The computational results revealed that a thick subsonic region did not exist in the combustor near the top wall at Mach number 5.45. It was a desirable feature to avoid unstating the engine. With the use of a strut, the relatively low velocity regions increased and the downward flow toward the cowl behind the step became strong. An overconcentration of fuel toward the top wall during weak combustion was observed in the experiment. The reason for this was that the airflow near the injector was turned to the top wall due to the small influence of combustion in the experiment.

Key Words: Numerical Analysis, Supersonic Flow, Internal Flow, Propulsion, Hybrid Unstructured Grid Method, Scramjet Engine

1. Introduction

The scramjet engine is considered to be as one of the most feasible candidates for space plane propulsion systems. Many studies have been conducted on each of its elements such as inlet, combustor and nozzle⁽¹⁾. However, to understand the detailed flow by experimental studies is very difficult because the flow is high speed and surrounded by walls. Therefore, numerical studies are indispensable tools for developing high-performance engines.

Most of the current Computational Fluid

* Received 22nd November, 1999. Japanese original: Trans. Jpn. Soc. Mech. Eng., Vol. 65, No. 633, B (1999), p. 9-15 (Received 30th October, 1998)

** Ramjet Propulsion Research Division, National Aerospace Laboratory, Kakuda Research Center, 1 Koganezawa, Kimigaya, Kakuda-shi, Miyagi 981-1525, Japan. E-mail: koderam@polestar.kakuda-splab.go.jp

*** Kawasaki Heavy Industries, Ltd., 1-1 Kawasaki-machi, Akashi-shi, Hyogo 673-8666, Japan

**** Department of Aeronautics and Space Engineering, Tohoku University, 01 Aoba, Aramaki, Aoba-ku, Sendai 980-8579, Japan

Dynamics (CFD) methods use structured grids to discretize the flow field and an efficient solution algorithm for the Navier-Stokes equations has been developed. However, the conventional CFD method often has difficulty in generating a structured grid with three-dimensional complex geometries. Although the scramjet configurations are relatively simple, the structured grid method requires a time-consuming zoning procedure for three-dimensional grid generations. Mesh singularities arising from structured grids for scramjet configurations sometimes cause numerical difficulties, especially in high Mach number conditions. For practical applications of the CFD method, more efficient and flexible solutions to treat complex configurations are demanded.

In recent years, use of unstructured grids is becoming more popular for computing the flows around complex geometrical structures because of their flexibility and grid refinement capabilities⁽²⁾. For high Reynolds number viscous flows, hybrid grids which are composed of prismatic grids for viscous regions and tetrahedral grids for other regions seem to be very appropriate for accurately and efficiently resolving the turbulent boundary layers⁽³⁾.

Fig. 1 Scramjet engine configuration

In the present study, the inside flow of a scramjet engine, not including chemical reactions, was numerically analyzed by applying the newly developed hybrid unstructured grid method⁽⁴⁾. The computed results were compared with the experiments⁽⁵⁾ that were carried out at National Aerospace Laboratory in Japan. In order to clarify the purpose of this study, the following facts observed during the experiments are mentioned in advance.

(a) The engine inlet easily unstated at Mach number 3.4^{(6),(7)}.

(b) Two combustion modes termed as “weak combustion” and “intensive combustion” were observed⁽⁵⁾. In the absence of a strut no intensive combustion mode was observed.

(c) An overconcentration of fuel was observed around the top wall at the nozzle exit, during the weak combustion mode during the experiments using the strut at Mach number 5.45⁽⁸⁾.

The flow structures responsible for these three facts determined by experiments are numerically investigated in detail.

2. The Engine Configuration

The engine model configuration investigated in this study has a side wall compression inlet and a side wall expansion nozzle, as shown in Fig. 1. The cross section of its flow passage is rectangular. This engine model is used in a series of experimental studies on scramjet engines at the Ramjet Engine Test Facility at National Aerospace Laboratory, Kakuda Research Center (NAL-KRC). In the experiments, a strut was integrated on the top wall inside the engine to attain intensive combustion. The height of the strut (short strut) used in this study was one-fifth of the inlet

Fig. 2 Surface grids and cut views of hybrid grid

entrance (H). The inlet has swept leading edges at the side walls angled at 45 degrees for better engine starting characteristic and its geometrical contraction ratio, which is the area of inlet entrance to that of inlet exit is approximately three without the strut. An isolator is set up between the inlet and the combustor to avoid the influence of pressure increase in the combustor on the inlet flow. Backward facing steps are used for flame holding at the side wall between the isolator and the combustor. The fuel is injected vertically to the side wall and the location of the injector on the side wall is 30 mm downstream from the step. The ignitor is installed on the top wall slightly upstream from the location of the step.

Figure 2 shows surface grids and sectional views of the hybrid grid for the model illustrated in Fig. 1. The generation of a single grid without grid partitioning is a simple task even for this complicated configuration with the strut. In the case of using the conventional structured grid, the partitioning procedure of the computational region would be necessary and mesh singularities, resulting in numerical instability, would arise from it. The grid generation also requires a time-consuming procedure for deciding partitioning locations or setting boundary conditions between partitioned regions.

Fig. 3 The shape of unstructured grid near the wall

3. Numerical Approach

3.1 Solution algorithm

The Navier-Stokes equations are solved as the governing equations and are written in an integral form as follows.

$$\frac{\partial}{\partial t} \int_{\Omega} \mathbf{Q} dV + \int_{\partial\Omega} (\mathbf{F}(\mathbf{Q}) - \mathbf{G}(\mathbf{Q})) \cdot \mathbf{n} dS = 0 \quad (1)$$

where $\mathbf{Q} = [\rho, \rho u, \rho v, \rho w, e]^T$ is the vector of conservative variables, ρ is the density, u, v and w are the velocity components in the x, y and z directions, respectively and e is the total energy. The vectors $\mathbf{F}(\mathbf{Q})$ and $\mathbf{G}(\mathbf{Q})$ represent the inviscid and viscous flux vectors and $\mathbf{n}$ is the outward normal of $\partial\Omega$ which is the boundary of the control volume Ω . This system of equations is closed by the perfect gas equation of state.

Equation (1) is solved by a finite volume cell-vertex scheme. The Harten-Lax-Van Leer-Einfeldt-Wada (HLLW) Riemann solver⁽⁹⁾ is used for the flux computation. As a limiter to achieve the scheme monotone and realize the second-order spatial accuracy, Venkatakrishnan's limiter⁽¹⁰⁾ is used because of its superior convergence properties. Implicit time integration is required for computing high Reynolds number flows because of severe CFL condition due to very fine grids near the wall boundary. Here the LU-SGS implicit method, improved for the unstructured grids⁽⁴⁾, is applied for time integration in Eq. (1).

For the grid, hybrid unstructured grids⁽³⁾ comprised of tetrahedra, prism and pyramids are generated. The prismatic semistructured grid is generated around viscous boundary surfaces and covers viscous regions while the tetrahedral grid covers the rest of the computational domain, as shown in Fig. 3.

For the turbulence model, Goldberg-Ramakrishnan's

Table 1 Mach number and configurations

	Mach No.	Short Strut
(a)	3.40	No
	5.45	
(b)	5.45	Yes
		No
(c)	5.45	Yes
		No

Table 2 Computational conditions

Mach No.	3.40	5.45
Reynolds No.	2.752E6	2.252E6
$\delta_{\infty}(z/H)$	0.125	0.23
T_{wall} (K)	300.	300.

one-equation turbulence model⁽¹¹⁾ is used. In the previous study, the solver including this turbulence model was applied to a shock/turbulent boundary layer interaction flow problems caused in a flow between symmetrical sharp fins on a flat plate⁽¹²⁾.

3.2 Numerical conditions

The information regarding the Mach number and the engine model configurations that are mainly used to investigate the three facts, (a), (b) and (c), mentioned in the introduction, are shown in Table 1. The details of numerical conditions at each Mach number are indicated in Table 2. These values are consistent with those of the experiment conducted at NAL-KRC. Here, the reference length of Reynolds number is the height of the inlet entrance. δ_{∞} is the inflow boundary layer thickness on the fuselage at the inlet entrance and it is normalized by the inlet height. It is thought that a scramjet engine is integrated on the bottom surface of the fuselage in order to capture the air compressed by the fuselage. In the experiment a thick boundary layer is introduced into the engine inlet in order to simulate the integration between the engine and the fuselage. T_{wall} is the wall temperature assuming an isothermal wall. It is assumed that the air is an ideal gas and the specific heat ratio is 1.4.

3.3 Boundary conditions

We are concerned with the inside of the engine and half the physical space divided by a symmetry plane. The boundary layer properties having the thickness of δ_{∞} are specified at the inlet entrance as an inflow boundary. The constant boundary layer properties in the span direction are also specified. To get the boundary layer properties at the inlet boundary, the two dimensional Navier-Stokes computations were carried out beforehand. As an outflow condition, extrapolation from upstream is imposed on the nozzle

Table 3 Details of grid

	A	B	C
G1 (Without Strut)	281	307194	2.E-5
G2 (Without Strut)	393	586568	2.E-5
G3 (With Strut)	281	318099	2.E-5

A: Spanwise points along side wall center line,
B: Total points, C: Minimum space

Fig. 4 Comparison of C_f distribution along the side wall center line

exit. All walls are assumed to be isothermal and possess nonslip conditions, in which case the velocity is zero. An outflow condition is also imposed at the open plane upstream of the cowl, where the inflow air in the inlet escapes from the inside of the engine.

3.4 Grid details

The details of three main types of grids generated in this study are indicated in Table 3. In this table, G1 shows the grid conditions for the configuration without the short strut. The number of grid points along the side wall center line in the main flow direction is 281. G2 has 1.4 times the number of points in the main direction larger than G1. The comparison of the results by G1 and G2 for the distributions of friction coefficient (C_f) along the side wall center line are shown in Fig. 4. It is found that G1 has sufficient number of grid points in the main flow direction, because there is no difference in the results between G1 and G2. Hence, the computational results obtained by using the G1 grid will be demonstrated next. G3 indicates a grid with the short strut. The distributions of surface grid points are almost same as those of G1.

For the grid point distributions in the vertical direction to the wall, the minimum spacing between the wall and the next grid point to the wall is about $2.5E-5$, normalized by the inlet height. Then, for Mach number 5.45 the maximum value of y^+ is about 2.5 and its average value is about 1.5 and for Mach number 3.4 the maximum value of y^+ is about 3.2 and

Fig. 5 Wall pressure distribution along the side wall center line

Fig. 6 Oil flow patterns on the side wall (with strut)

its average value is about 2.2.

3.5 Numerical accuracy and computational time

In Fig. 5, the computed pressure distributions in the case of the configuration without the short strut at Mach 5.45 are compared with the experiments⁽⁶⁾. This figure shows qualitative agreement between both the results. However, the computation underestimates the peak pressure behind the step (at the location of 850 mm on the horizontal axis), because the turbulence model can not accurately predict the separation behind the step. Except for this, the agreement between both the results is quantitatively good. The results of the case for Mach 3.4 showed good agreement with the experiment⁽¹³⁾. Figure 6 shows the oil flow patterns on the side wall for the engine model with the short strut. The computed oil flow pattern in Fig. 6(b) which is equivalent to the streamlines near the surface, predicts the experimental results qualitatively well, as shown in Fig. 6(a).

The time taken for grid generation was about 3 hr on SGI's Indigo 2 workstation, to generate triangle surface grids on the boundary faces and body faces in the case of G1 grid. It took about 1 hr to generate hybrid unstructured grids comprised of tetrahedra, prism and pyramids automatically, hence the total

Fig. 7 Flow on the symmetry plane at the inlet

time taken was about 4 hr.

Regarding the computational time, it took about 14 sec per iteration on NEC SX-4 supercomputer in the case of the G1 grid. The number of iterations until convergence was about 10 000, since it was impossible to specify a large CFL number at the early stage of computations and it took large number of iterations to obtain invariable physical properties in boundary layers after the residual was constant.

4. Results and Discussion of Internal Flow Analysis

Internal flow analyses are applied to the three items mentioned in the introduction, as (a), (b) and (c), observed in the experiment, assuming a nonreacting perfect gas. Numerical results and discussions of these items are described below.

4.1 Item (a) — On starting characteristic —

Figure 7 shows the pressure contour on a symmetry plane without a strut at Mach numbers 3.4 and 5.45. The streamlines from the upstream of the inlet and the line of Mach number 1 ($M=1$) near the top wall are also shown in this figure. Compared with the results of Mach number 3.4, the angle of streamlines to the top wall at Mach number 5.45 is smaller. The subsonic regions near the top wall, which are regions between the line of Mach number 1 and the top wall are also thinner. On the other hand, at Mach number 3.4, a thick subsonic layer exists near the top wall from the inlet to the entrance of the combustor through the isolator. This rapid increase of the boundary layer at Mach 3.4 is mainly caused by side wall shock reflections.

In the experiments, the engine easily unstarted at Mach number 3.4^{(6),(7)}. It was assumed that pressure increase in the combustor during combustion affected the inlet flow through the thick subsonic regions near

(a) Upstream of the step (b) Downstream of the step

Fig. 8 Mach number contours on cross-flow planes parallel to the side wall leading edge

the top wall. This is one of the reasons for causing the unstart easily for M 3.4 case. In computations, the thick subsonic region did not exist for the M 5.45 case in the combustor near the top wall and thus, the engine unstart seldom occurred. These results support the findings gained from the experiment.

4.2 Item (b) — On insertion of a strut —

Figure 8 shows Mach number contours on the cross sections parallel to the sweep angle of the side wall with/without the short strut on the left and right half of the figure respectively. Figure 8(a) illustrates the distance which is 4 mm upstream from the step and Fig. 8(b) illustrates the location behind the step including the injector plane, which is at the distance of 30 mm downstream from the step.

The low Mach number regions near the top wall were larger with the strut than those without the strut, as shown in both Figs. 8(a) and 8(b). In Fig. 8(a), the computed result with the strut showed a somewhat thick subsonic region at the root of the strut. Even the area of lower Mach numbers near the side wall at the center of flow passage was larger compared to the case with no strut. The cut plane shown in Fig. 8(b) is located at the positions of reattachment of the separation generated behind the step. The flow was affected by expansion waves from the corner of the step. This figure shows an increase of the high Mach number region compared with Fig. 8(a).

Fig. 9 Separation regions behind the step on the side wall

In the experiments it was reported that two combustion modes were observed that were termed the weak combustion and the intensive combustion⁽⁵⁾. There was no intensive combustion without the strut. In computations, the larger low Mach number regions were obtained near the step with the strut. Hence, it is thought that the fuel flow from the injector in the downstream region from the step might interact easily with the separation region behind the step. This interaction might be one of the reasons for the intensive combustion.

Figure 9 shows the approximated reattachment lines behind the step with/without the short strut. These lines are considered to be due to a velocity, which is normal to the backward face of the step and on a plane that is at a close distance of $2.E-5 H$ from the side wall and it is zero ($v_n=0$, as shown in this figure). This figure also includes pressure contours on the side wall with the strut and the locations of the injector.

These separation lines were located at almost the same position from the top wall to the middle area between the top wall and the cowl for both the cases. Near the cowl, the separation region was extended to the downstream region and reached the injector. This occurred because the region behind the step was affected by the pressure increase of the shock from the cowl leading edge without the strut and the shock from the cowl and the strut leading edge with it.

Figure 10 shows the velocity vectors with/without the short strut on the cross sections which include the injector and are parallel to the sweep angle of the side wall. These figures show the magnified images at the corner of the side wall and the cowl. The height here is about one-tenth the inlet height, including an injector closest to the cowl.

As shown in these figures, the flow toward the cowl (downward in this figure) near the side wall in the case with the strut was stronger than the one without the strut. This downward flow was caused by the shock from the strut leading edge. The vortex

(a) With strut

(b) Without strut

Fig. 10 Flow at the corner of the side wall and the cowl behind the step (velocity vector)

generated at the corner of the side wall and the cowl was also larger. For this reason, the low Mach number region at the corner of the side wall and the cowl shown in Fig. 8(b) was larger with the strut than in the case without it. Hence, it seems that a radical near the top wall might be brought by this downward flow into the cowl, since an ignition is caused by an ignitor on the top wall near the step, in this experiment. The pressure is higher in the region near the cowl. Thus a combustion might be promoted in the entire step region.

4.3 Item (c) — On overconcentration of fuel at the nozzle exit —

In order to observe flow patterns from the region near the injector toward downstream, the streamlines departed from positions at several close distances from the injector were simulated near the side wall at the combustor and the nozzle. Among them, the streamlines released at a normal distance of 2 mm apart from the injector are shown in Fig. 11(a) for the cases with/without a strut. Incidentally, the results at other distances to the injector are almost similar. The streamlines from the region in the separation vortex behind the step are shown in Fig. 11(b).

As shown in Fig. 11(a), the streamlines moved toward the top wall in both the cases because of the expansion at the divergence section. The streamlines shown in Fig. 11(b) also tended to move toward the top wall, similar to those of Fig. 11(a). However, in Fig. 11(b), it is observed that the streamlines from the area between the center of the flow passage and the cowl flow through the separation area behind the step toward the cowl and go downstream along the

Fig. 11 Streamlines downstream of the step

cowl wall after reaching the cowl.

In experiments with the strut, it was reported that there was an overconcentration of the fuel during the weak combustion around the top wall at the nozzle exit⁽⁸⁾. This is the reason why the flow from the injector behaved as shown in Fig. 11(a) in the experiment, since the airflow was hardly affected by the combustion in the weak combustion mode. It was also unlikely that the fuel flow could interact with the separation region behind the step during the weak combustion because the fuel did not exist around the cowl at the exit of nozzle in the experiment and the situation was different from that in Fig. 11(b). If the fuel flow behaved as that shown in Fig. 11(b), it would have mixed with the flow in the separation region behind the step in the case of the strut, would have reached the high-pressure region near the cowl, and the combustion would have been accelerated.

5. Conclusion

The hybrid unstructured grid method to compute the Navier-Stokes equations was applied to internal flow fields of a scramjet engine at Mach numbers 3.4 and 5.45. The following results were obtained.

(1) A thick subsonic region appeared at Mach 3.4 in the combustor near the top wall, but did not appear at the Mach number 5.45. Thus, a higher Mach number is safer to prevent engine unstart.

(2) In the presence of a strut, the downward flow toward the cowl behind the step became strong.

(3) The overconcentration of fuel toward the top wall during the weak combustion observed in the experiments occurred because the airflow near the injector was turned to the top wall.

References

- (1) Kumar, A., Singh, D.J. and Trexler, C.A., Numerical Study of the Effects of Reverse Sweep on Scramjet Inlet Performance, *J. of Prop. and Power*, Vol. 8, No. 3 (1992), p. 714-719.
- (2) Lee, R.A., Hosangadi, A., York, B.J., Sinha, N. and Dash, S.M., Applications of an Unstructured Solver to Reactive, Multiphase Plume/Propulsive Flowfields, *AIAA Paper 96-2955*, (1996).
- (3) Sharov, D. and Nakahashi, K., Hybrid Prismatic/Tetrahedral Grid Generation for Viscous Flow Applications, *AIAA J.*, Vol. 36, No. 2 (1998), p. 157-162.
- (4) Sharov, D. and Nakahashi, K., Reordering of 3-D Hybrid Unstructured Grids for Lower-Upper Symmetric Gauss-Seidel Computations, *AIAA J.*, Vol. 36, No. 3 (1998), p. 484-486.
- (5) Kanda, T., Hiraiwa, T., Mitani, S., Tomioka, S. and Chinzei, N., Mach 6 Testing of a Scramjet Engine Model, *J. of Prop. and Power*, Vol. 13, No. 4 (1997), p. 543-551.
- (6) Yatsuyanagi, N., Chinzei, N. and Miki, Y., Initial Tests of a Sub-Scale Scramjet Engine, *Proc. of the 12th Int. Symp. on Air Breathing Engines*, Melbourne, Australia, AIAA, Washington, D. C., (1995-9), p. 1330-1337.
- (7) Sunami, T., Sakuranaka, N., Tani, K., Hiraiwa, T. and Shimura, T., Mach 4 Tests of a Scramjet Engine-Effects of Isolator, *Proc. of the 13th ISABE*, Chattanooga, TN, (1997-9), p. 615-625.
- (8) Sato, S., Izumikawa, M., Tomioka, S. and Mitani, T., Scramjet Engine Test at Mach 6 Flight Condition, *AIAA Paper 97-3021*, (1997).
- (9) Obayashi, S. and Guruswamy, G.P., Convergence Acceleration of an Aeroelastic Navier-Stokes Solver, *AIAA Paper 94-2268*, (1994).
- (10) Venkatakrishnan, V., On the Accuracy of Limiters and Convergence to Steady State Solutions, *AIAA Paper 93-0880*, (1993).
- (11) Goldberg, U.C. and Ramakrishnan, S.V., A Pointwise Version of Baldwin-Barth Turbulence Model, *Comp. Fluid Dyn.*, Vol. 1 (1993), p. 321-338.
- (12) Koder, M., Nakahashi, K. and Sharov, D., Simulation of 3D Shock Wave/Turbulent Boundary Layer Interactions by Hybrid Grid Method, *Trans. Jpn. Soc. Mech. Eng.*, (in Japanese), Vol. 64, No. 627, B (1998), p. 143-148.
- (13) Igarashi, Y., Koder, M., Nakahashi, K., Mitani, T. and Shimura, T., Numerical and Experimental Analysis of Scramjet Internal Flows — Comparative Studies of Engine Drag —, *AIAA Paper 98-1512*, (1998).