Numerical Research of Airframe/Engine Integrative Hypersonic Vehicle

He Yuanyuan Le Jialing Ni Hongli
China Aerodynamics Research and Development Center,
Mianyang Sichuan 621000, China

Abstract: Hypersonic airbreathing vehicles are characterized by highly-integrated airframe and propulsion system. Computational Fluid Dynamics (CFD) tools have been used extensively in the design and analysis phrase of hypersonic vehicles. In this paper, an engineering method and a finite volume method based on the center of grid are developed for preliminary research of interested integrative hypersonic vehicles with waverider configurations (cone-shaped or canard structure). Some longitudinal aerodynamic performance increments are obtained under conditions of shut-down engine or working engine. For the vehicle with canard structure, in order to study the lateral-directional stability characteristics, we also consider the influence of yaw angle on the flow field of fore-body.

1 Introduction

Hypersonic airbreathing vehicle configurations are characterized by highly-integrated airframe/engine system. Waverider configuration is a typical hypersonic vehicle shape, whose fore-body can compress the flow in advance, increase the lift, reduce the drag, and provide uniform air flow entering engine, and whose aft-body is used as an expansion surface of nozzle to increase the aft-body lift, reduce the drag and increase the thrust with the expanded exhaust. With the development of hypersonic technology, advanced experimental, analytical and computational methods are being exploited in the design of hypersonic configurations to obtain excellent aerodynamic characteristics[5]. Due to the limitation of test capabilities to model all the impossible flight conditions, Computational Fluid Dynamics (CFD) becomes a quite useful predictive methodology in the design and research phase of hypersonic vehicles. In this paper, an engineering method and a finite volume method based on the center of grid are developed for preliminary study of interested integrative hypersonic configurations (cone or canard structure). The numerical simulations are carried out under two conditions of engine (shut-down or working). Some longitudinal aerodynamic performance increments are obtained and presented. For the vehicle with canard structure, in order to study the lateral-directional stability characteristics, we also consider the influence of yaw angle on the flow field of fore-body.

2 Analysis tools

2.1 Engineering Method

A combination of modified Newton method and the method of tangent cones is used to preliminarily define some aerodynamic parameters[4]. On body surface pressure factor *Cp* can be determined by

$$Cp = Cp_0 \cos^2\varphi$$

Report Documentation Page

Report Date	Report Type	Dates Covered (from... to)
23 Aug 2002	N/A	-

Title and Subtitle Numerical Research of Airframe/Engine Integrative Hypersonic Vehicle	**Contract Number**
	Grant Number
	Program Element Number
Author(s)	**Project Number**
	Task Number
	Work Unit Number
Performing Organization Name(s) and Address(es) Institute of Theoretical and Applied Mechanics Institutskaya 4/1 Novosibirsk 530090 Russia	**Performing Organization Report Number**
Sponsoring/Monitoring Agency Name(s) and Address(es) EOARD PSC 802 Box 14 FPO 09499-0014	**Sponsor/Monitor's Acronym(s)**
	Sponsor/Monitor's Report Number(s)

Distribution/Availability Statement
Approved for public release, distribution unlimited

Supplementary Notes
See also ADM001433, Conference held International Conference on Methods of Aerophysical Research (11th) Held in Novosibirsk, Russia on 1-7 Jul 2002

Abstract

Subject Terms

Report Classification unclassified	**Classification of this page** unclassified
Classification of Abstract unclassified	**Limitation of Abstract** UU

Number of Pages
7

Cp	**pressure factor**		
Cx	**drag force coefficient**	***Cy***	**lift force coefficient**
Cmz	**picthing moment coefficient**	***k***	**lift-drag ratio**
Xp	**pressure center**	***M∞***	**free stream Mach number**
Re	**Reynolds number**	***α***	**attack angle**

where φ (radian) is the angle between the normal vector of the body and the velocity of airflow , Cp_0 is calculated by modified method. Additionally, correction for boundary layer and " shaded part" is employed to improve the approximate method.

2.2 Numerical Method

The 3D Euler or N-S equations are taken as control equations for numerical simulation. An implicit, finite volume discretization[1] [2] [3] based on the center of grid is employed, which can be expressed as below for 3D Euler equations:

$$\frac{Q_{i,j,k}^{n+1}-Q_{i,j,k}^{n}}{\Delta\tau}=-(\frac{F_{i+1/2,j,k}^{n+1}-F_{i-1/2,j,k}^{n+1}}{\Delta\xi}+$$
$$\frac{G_{i,j+1/2,k}^{n+1}-G_{i,j-1/2,k}^{n+1}}{\Delta\eta}+\frac{H_{i,j,k+1/2}^{n+1}-H_{i,j,k-1/2}^{n+1}}{\Delta\zeta})$$

The flux terms at cell faces are evaluated using MUSCL methods and Steger-Warming splitting. To increase the speed of computation and save memory, a kind of advancing method along diagonal faces instead of the general IJK advancing method is used. The boundary conditions are given as below:

1. Far Field Boundary

Characteristic variable boundary conditions are used.

2. Wall Boundary

For Euler equations, zero pressure gradient condition is used firstly, then characteristic variable boundary conditions are employed. For viscous computation, no slip conditions are used.

3. Symmetric Boundary

Using zero pressure gradient condition.

2.3 Grid Generation

An algebraic grid generation system is used to generate the computational grid. For complex configuration, the computational grid is divided into many blocks. Numerical simulation is carried out in each block, and the computation of whole flow field is completed according to the data transfer among the blocks.

2.4 Parallel Method

Because of the limit of speed and memory, an individual computer could hardly meet the requirement for computing complex configuration. Therefore a PVM parallel method is developed. The computation of multi-block is distributed averagely to many computers instead of one. It is proved that the speed and scale of computation could be improved greatly.

3 Result and Discussion

3.1 Hypersonic vehicle configuration (cone-shaped)

The flow field of a cone-shaped hypersonic vehicle with shut-down engine is numerically simulated by solving 3D Euler equations. The free stream conditions are given as below.

$$M_\infty = 6, \quad \alpha = 8^o$$

Fig1.1 shows the distribution of pressure contours on the surface of the vehicle. In order to increase the stabilitiy of airflow near aft-body, there are two laminae on both sides of the tail-nozzle. Pressure contours at different axial cross-sections are shown in Fig1.2 to Fig1.4. The air flow before the inlet wedge is close to uniform (Fig1.2). In Fig1.3 the typical pressure distribution near fuselage is presented. Here it is important that the wings separate the upwind part of the flow from the leeward part without creating additional disturbance. In Fig1.4, we can see the effect of the laminae, the distributions of pressure contours are almost uniform and the flowing under the tail-nozzle is stable because the laminae decrease the disturbance of outflow. Table.1 shows the comparison between the results by solving Euler Equations and those obtained using engineering method. The lift-drag ratio calculated by solving Euler Equations is 3.6.

Table.1 The comparison between numerical results and engineering results

(*L* is the total length of vehicle)

	Numerical	Engineering
Cx	0.015	0.013
Cy	0.055	0.041
K	3.61	3.16
Cmz	0.031	0.021
Xp	0.55*L*	0.51*L*

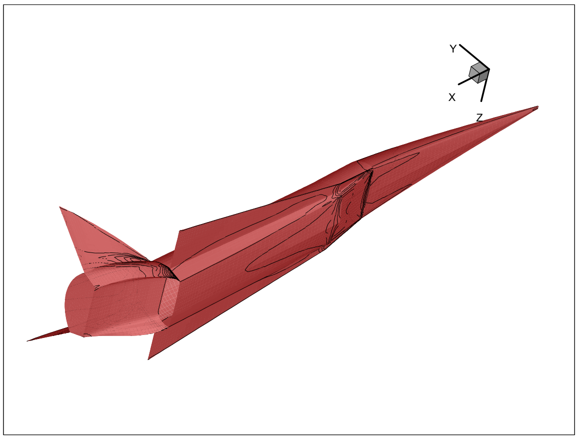


Fig1.1 Surface pressure contours of hypersonic vehicle

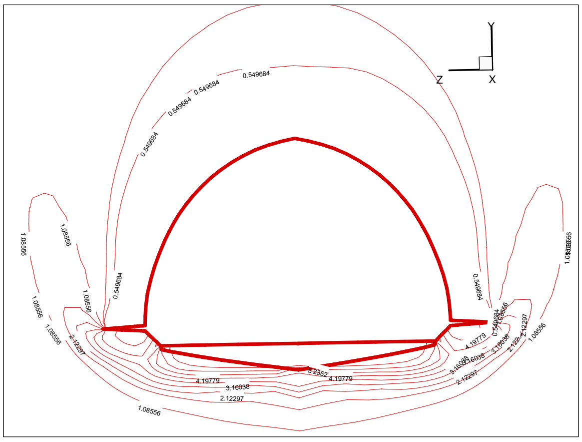


Fig1.2 Pressure contours near inlet

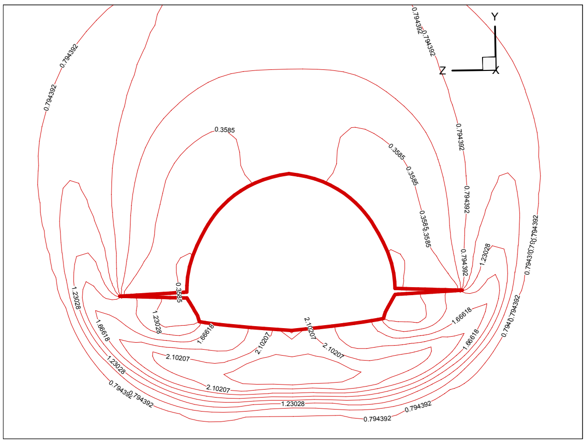


Fig1.3 Pressure contours near fuselage

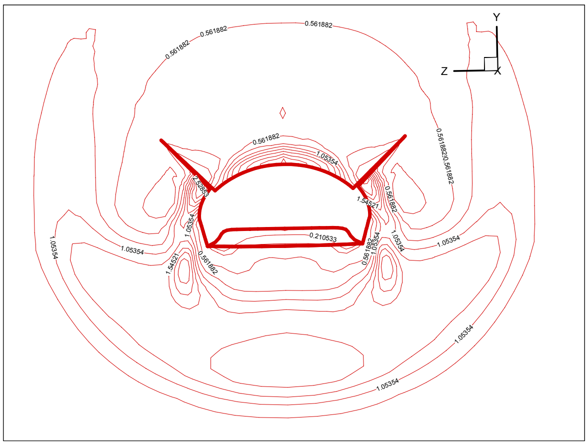


Fig1.4 Pressure contours near tail-nozzle

3.2 Hypersonic vehicle configuration (canard structure)

Numerical simulation is also carried out for a hypersonic vehicle with canard structure under two different conditions of engine, shut-down and working, using 3D N-S (Euler) equations. The free stream conditions are given as below.

$$M_\infty = 6,\ \ \alpha = 6^o,\ \mathrm{Re}=1.73\mathrm{e}6,\ \mathrm{H}=25\mathrm{km}$$

Fig2.1 shows the distribution of pressure contours on the vehicle's surface with shut-down engine. It can be seen that the pressure near the inlet wedge is higher due to the preliminary compression of fore-body. Table.2 shows the comparison between the results by solving N-S and Euler Equations with shut-down engine. The numerical results by solving Euler equations are quite consistent with results obtaibed from N-S equations of laminar flow. It proves the validity of Euler equations in simulating the hypersonic flow field. With working engine, the influence of exhaust is studied. The outflow conditions of engine are taken as the inflow conditions for the computation of aft-body. The conditions of exhaust may be given as

$$P_0=406170\mathrm{Pa},\ T_0=1791.381\mathrm{K},\ \rho_0=0.50039\mathrm{Kg/m^3},\ u_0=1255.758\mathrm{m/s}$$

These parameters are gained from the internal numerical simulation of the engine. In order to simplify the computation, we take the outflow as perfect gas. Table 3 presents the computational results for vehicle with working engine. The depiction of the computed performance increments is also shown in Table 3. When the engine is working, the exhaust of high velocity and high temperature expands along the external nozzle, and the surface pressure of aftbody is much higher than when the engine is shut-down. A decrease in drag force coefficient as well as the nose-down picthing moment increment is predicted for the vehicle because of the influence of high pressure on the external nozzle suface, just as the depiction presented in refenrence[5]. The pressure distributions on the surface of tail-nozzle under different working conditions of engine are shown in Fig2.3 to Fig2.4. Fig2.5 gives the Mach number distributions on the symmetric cross-section with working engine.

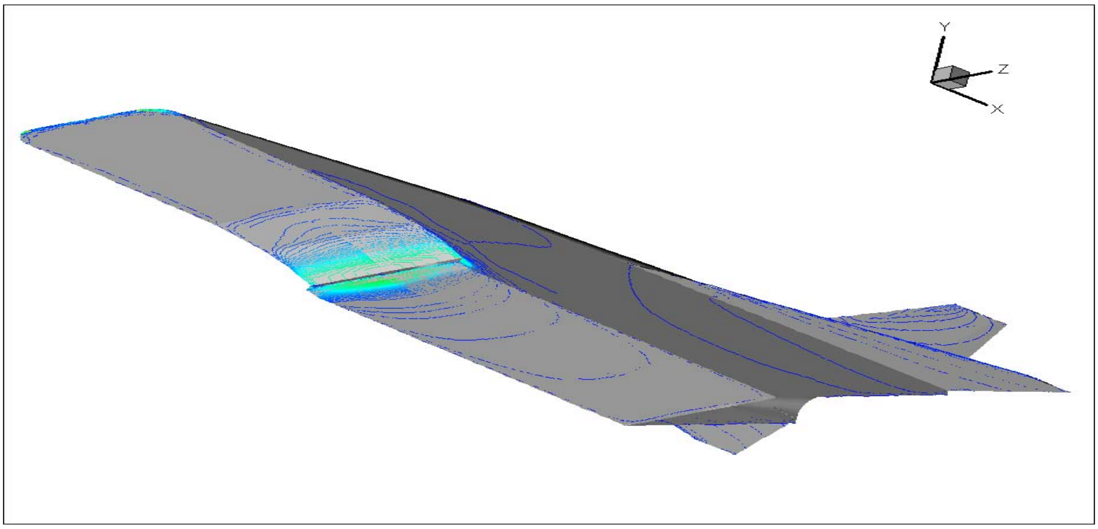


Fig2.1 Surface pressure contours of hypersonic vehicle with canard structure (shut-down engine)

Table.2 The comparison of numerically simulated results

	N-S	Euler
Cx	0.0093	0.0094
Cy	0.0295	0.0299
C_{mz}	0.2231	0.2243
K	3.16	3.18
Xp	0.43L	0.43L

Table.3 Numerical results under different conditions of engine

	N-S(shut-down engine)	N-S(working engine)	$\Delta(CFD_{working}-CFD_{shut\text{-}down})$
Cx	0.0093	-0.0005	-0.0098
Cy	0.0295	0.0328	0.0033
C_{mz}	0.2231	0.2541	0.0310

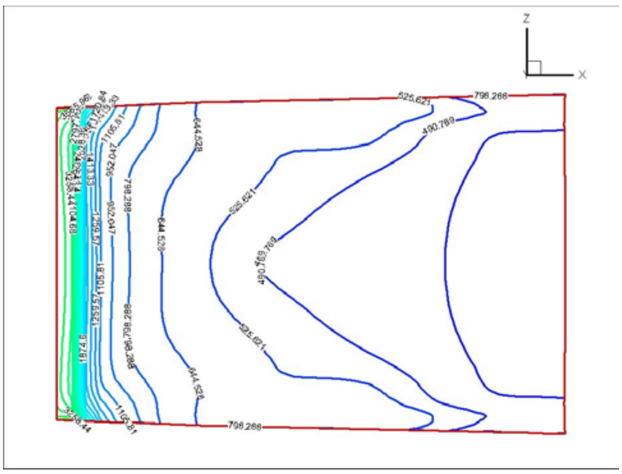


Fig2.2 Pressure contours on the surface of tail-nozzle (shut-down engine)

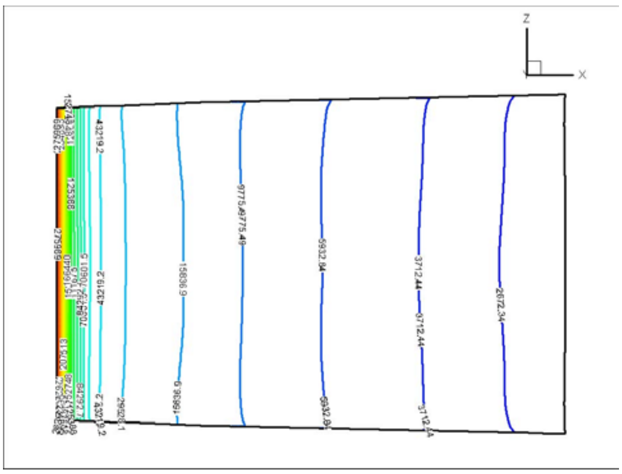


Fig2.3 Pressure contours on the surface of tail-nozzle (working engine)

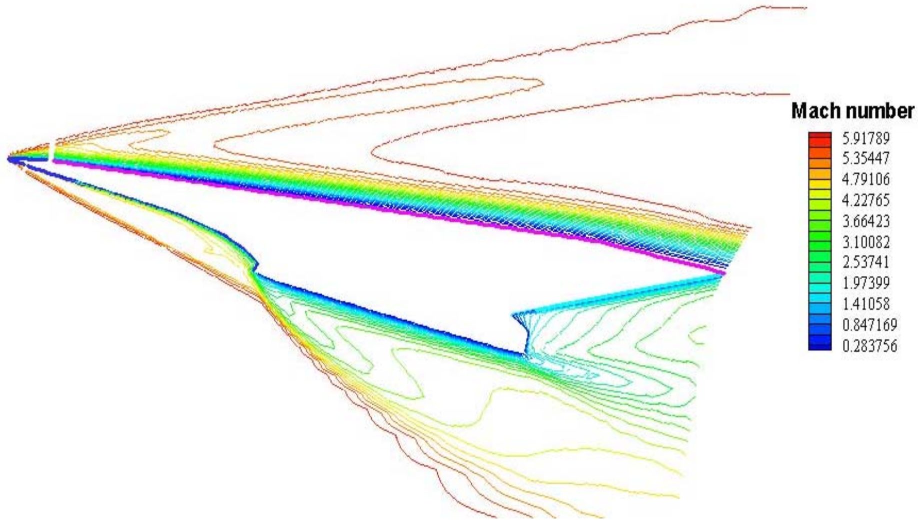


Fig2.4 Mach number contours on symmetric cross-section (working engine)

3.3 Study of fore-body

In order to study the influence of yaw angle on the uniformity of the flow near fore-body and the flux entering the engine, the fore-body is separated from the whole vehicle and studied solely. The free stream conditions are the same as the whole configuration. Fig3.1 shows the distribution of pressure contours on the surface of fore-body under the condition of no yaw angle. Fig 3.2 shows the pressure contours arround the inlet of the engine. We can see that the flowing near the inlet is close to uniform. This result is quite useful for the design of hypersonic vehicle, because whether the fore-body can provide uniform flow into the engine is very important. Fig 3.3-Fig 3.4 show the numerical results when the yaw angle is 3º. The influence of yaw angle can be seen here, there is a little departure of flow field from symmtry, and according to calculation, the flux entering the engine with no-zero slip(which is 10.653kg/s) is slightly smaller than that of zero slip (which is 10.868kg/s).

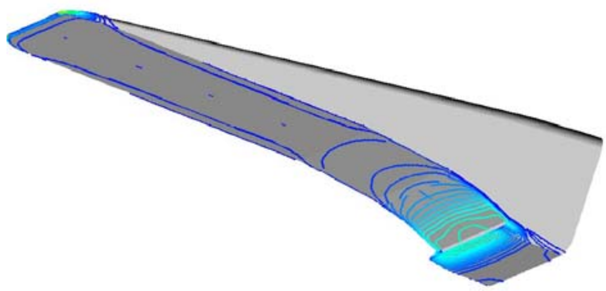


Fig3.1 Surface pressure contours of fore-body

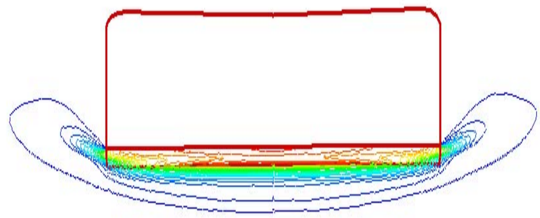
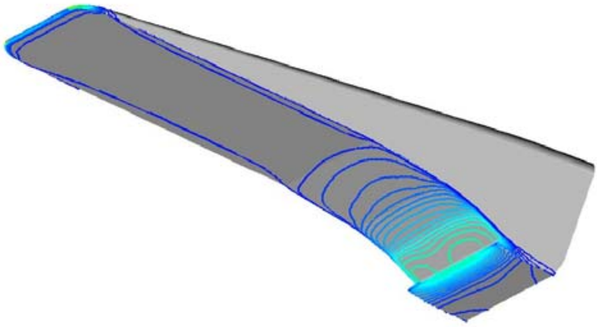


Fig3.2 Pressure contours near inlet of engine

(β=0°)



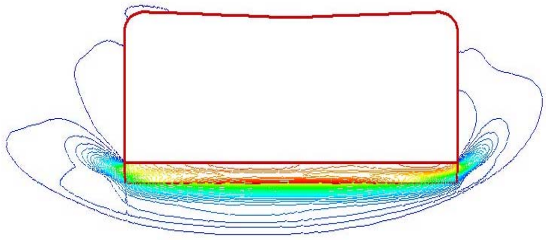


Fig3.3 Surface pressure contours of fore-body Fig3.4 Pressure contours near inlet of engine

(β=3°)

4 Conclusions

Computational Fluid Dynamics (CFD) have been used in the development and pre-flight analysis of hypersonic vehicles. The flow fields of two configurations (cone or canard structure) are numerically simulated and some longitudinal performances are obtained with the considerations of different situations of engine (shut-down or working). CFD are also used to qualitatively analyze the lateral-directional stability characteristics with the influence of yaw angle. According to the numerical simulations for complex configurations, it is proved that we have possessed the capabilities of generating complex multi-block computational grid, large-scale parallel computing and efficient numerical simulation by solving 3D N-S (Euler) equations.

5 References

[1] W.K.Anderson, J.L.Thomas, B.Van Leer, A comparison of finite volume flux vector splittings for the Euler equations, AIAA paper No.85-0122

[2] James N.Scott, Yang-Yao.Niu, Comparison of limiters in flux-split algorithms for Euler equations, AIAA paper No.93-0068

[3] Marcel.Vinokur, An analysis of Finite-difference and Finite-volume formulations of conservation laws, Journal of Computational Physics, Vol.81, 1989

[4] Institute of theoretical and applied mechanics, Siberian branch, Russian academy of sciences, Design aerodynamics scheme of vehicle

[5] Charles E. Cockrell, Jr., Walter C.Engelund, Integrated Aero-Propulsive CFD Methodology for the Hyper-X Fight Experiment, AIAA paper No.2000-4010