temperature distribution.

The present numerical investigation is used in engineering design of a missile launching pad by the Department of military equipment, Singapore Navy, successfully. The stream-upwind technique used in present computation is powerfully in smoothing the oscillation and the finite element method can be extended in other aerodynamic drag calculation. The computational results obtained are in good agreement with experiment observation and other valid numerical computations.

REFERENCES

1 Mikhail A G, Hankey W L and Shang J S. Computation of a Supersonic Flow Past an Axisymmetric Nozzle Boattail with Jet Exhaust. *AIAA Journal*, 1980, 18:869～875

2 Sullins G A, Anderson J D Jr, and Drummond J P, Numerical Investigation of Supersonic Base Flow with Parallel Injection. AIAA Paper 82-1001, 1982.

3 Zheng Min, Zhang Hanxin. Application of Non-Oscillatory and Non-free-parameters Dissipative Finite Difference Scheme to the Calculation of Free-Jet Flows. Acta Aerodynamic Sinica (in Chinese), 1989, 7(3): 273～281

具有后挡板的喷流流场的数值研究

王发民　赵　烈　卞荫贵　于　欣

（中国科学院力学研究所）

摘要　本文对发射台底面与后挡板之间的喷流流场进行了数值研究。采用迎风的 Petrov-Galerkin 有限元方法求解欧拉方程。描述了喷流出口处及挡板附近的流场细节。计算结果已成功地应用于导弹发射台的设计。

关键词　喷流/外流相互作用，鼓形激波，有限元方法，迎风方法。

第13卷第1期 空气动力学学报 Vol, 13, No.1
1995年3月 ACTA AERODYNAMICA SINICA Mar.,1995

A Numerical Study of the Propulsive Jet Flow Against an Obstacle

Wang Famin Zhao Lie Bian Yingui Yu Xin
(Institute of Mechanics, Chinese Academy of Science)

Abstract A numerical study of a propulsive jet flow confined by bottom plane and against a flat plate is made. The Euler equations are solved using a Petrov-Galerkin finite element approach with a stream-upwind technique. The flow field details are described in both the jet exit base region and the near region of the opposite plane. The computation results are applied on a missile launching pad successfully.

Key words exhaust/external interaction, barrel shock, finite element method, stream-upwind technique.

INTRODUCTION

The propulsive jet flow confined by an obstacle located in a downstream position and a parallel plane fixed at the bottom, is studied numerically. For propulsive units, such as jet engines and rocket motor, the flow is influenced by the interaction of the exhaust and external flows. For engineering design calculations extensive use is made of component modeling methods called "plume simulator" that provide quick estimates of base pressure and base region slipstreams. For a more comprehensive description of the flow field details, a straight forward numerical calculation in full region approach is necessary.

Previous work of related flow using full field approach was made by Mikhail et al.[1], Sullins et al.[2], Zheng Min and Zhang Hanxin[3], Peery and Forester and Mace et al., for various kinds of aerodynamic drag associated with a centered propulsive jet. The differences of present research are that the jet flow is not only influenced by interaction of the exhaust and external flow but also influenced by these two planes. Here, the research is concerned with the base region of the propulsive jet and

The paper was received on Apl, 26, 1993,

the region near the opposite plane. This project is employed in engineering design of a missile launching pad which is armed in the guided missile destroyer.

This paper describes a computational study of the influence of exhaust pressure and the temperature on the character of the propulsive jet with the obstacle and the bottom plane. The time dependent Euler equations are solved numerically using a finite element method with an upstream technique. The jet exit Mach number equals 1 and the jet-exit pressure is 40 times the freestream pressure. Both the external flow and the exhaust jet (which is at a given temperature), are ideal air.

The geometric configuration is a missile launching pad which fixed in the bottom plane and associated with a plane shaped obstacle located downstream. The height of the launching pad is 1 meter and the distance between the jet exit plane and the obstacle is 1.2 meters.

I GOVERNING EQUATIONS

The axisymmetric, two- dimensional Euler equations in general cylindrical coordinate are

$$\frac{\partial U}{\partial t}+\frac{\partial F}{\partial x}+\frac{\partial G}{\partial r}+\frac{\widetilde{H}}{r}=0 \tag{1}$$

where $U=[\rho,\ \rho u,\ \rho v,\ e]^T$

The velocity components (u, v), corresponding to x and r direction respectively, are non-dimensionalized with sound speed a_∞, for density ρ referenced to ρ_∞. The non-dimensional total energy is defined as

$$e=\frac{p}{\gamma-1}+\frac{\rho}{2}(u^2+v^2) \tag{2}$$

where γ is the ratio of specific heat, for the ideal air $\gamma=1.4$, and pressure p is referenced to $\rho_\infty a_\infty^2$.

The fluxion F, G and $\widetilde{H}$ are

$$F=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ (e+p)u\end{bmatrix},\qquad G=\begin{bmatrix}\rho v\\ \rho uv\\ \rho v^2+p\\ (e+p)v\end{bmatrix},\qquad \widetilde{H}=\begin{bmatrix}\rho v\\ \rho uv\\ \rho v^2\\ v(e+p)\end{bmatrix} \tag{3}$$

II FINITE ELEMENT APPROACH

The mathematical model in section 2 consists of four equations, the boundary condition of which can be written in general form

$$L(U)=\alpha U+\beta\left[\frac{\partial U}{\partial x_i}+\frac{1}{r}\frac{\partial}{\partial r}(rU)\right]\cdot\widetilde{n}+\gamma=0, \qquad U\in\partial\Omega \tag{4}$$

here α and β are specified coefficient diagonal matrices, γ is a constant vector, and n is the outward unit normal vector. The solution domain of (1) is $\Omega(=R^2\times[t(0),t])$ and its boundary is $\partial\Omega$ $(=\partial R^2\times[t(0),t])$. To start the calculation, an initial distribution for U on $\Omega_0(=R^2\times t(0))$ is required.

Let $U^h\in W^h\subset H_0^1(\Omega)$ denote the finite element approximation solutions. The weak form of equation (1) subject to the boundary conditions (5) on Galerkin principle is

$$m(U^h,W_i)+a(U^h,W_i)+b(U^h,W_i)+\widetilde{h}(U^h,W_i)+e(U^h,W_i)=0 \qquad \forall W_i\in W^h \tag{5}$$

where

$$m(U^h,W_i)=\int_\Omega W_i\frac{\partial U^h}{\partial t}r\,\mathrm{d}x\mathrm{d}r \tag{6}$$

$$a(U^h,W_i)=\int_\Omega\frac{\partial W_i}{\partial x}F(U^h)\,r\,\mathrm{d}x\mathrm{d}r \tag{7}$$

$$b(U^h,W_i)=\int_\Omega\left(\frac{\partial W_i}{\partial r}r+W_i\right)G(U^h)\,\mathrm{d}x\mathrm{d}r \tag{8}$$

$$\widetilde{h}(U^h,W_i)=\int_\Omega W_i\frac{\widetilde{H}(U^h)}{r}r\,\mathrm{d}x\mathrm{d}r \tag{9}$$

$$e(U^h,W_i)=\oint_{\partial\Omega}W_iL(U^h)\,\mathrm{d}\sigma \tag{10}$$

Here $[W_i(x,r)]^T$ are the weighting functions which are given below. $H_0^1(\Omega)$ denotes the Hibert space of all functions possessing square integrable first derivatives and satisfying boundary conditions. $\mathrm{d}\sigma$ is a line integral element.

It is well known the classical Galerkin finite element method applied to convection dominated flow problems gives oscillatory solution. To preclude such oscillations, the streamline-upwind/petrov Galerkin method is employed. Where instead of using shape function as the weighting function, a modified weighting function is created as

$$W_i(x,r)=N_i(x,r)+\alpha_ip_i(x,r) \tag{11}$$

$N_i(x,r)$ is the shape function constructed to form a cardinal basis, and

$$p_i(x,r)=\frac{h}{2V}\left(u\frac{\partial N_i}{\partial x}+v\frac{\partial N_i}{\partial r}\right) \tag{12}$$

where $V = |\sqrt{u^2+v^2}|$ is the absolute value of local velocity. h is defined as

$$h = \frac{1}{V}[|u|\Delta x + |v|\Delta r] \tag{13}$$

and α_i is a parameter to be chosen.

The boundary conditions consist of (1) impervious and zero normal pressure gradient on the solid walls, (2) uniform freestream at the far-field lateral boundary (3) extrapolation at the downstream boundary, (4) inviscid conical flow at the jet-exit plane. When the entire flow is computed from an impulsive start, the upstream boundary condition is a uniform freestream.

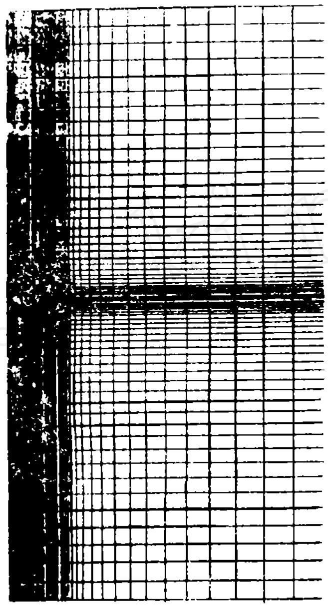

Fig. 1 Element distribution

In the present work, the rectangular elements utilizing linear basis function are used and the integral in equation (5) is carried out using a Gauss routine. The element distribution is shown in Fig.1, where 70×70 elements are used. The convergence solutions are obtained with 3000 times iteration.

Ⅲ COMPUTATIONAL RESULTS

Fig. 2 shows the computed density contours in the region near the jet exit plane. The experimental results for comparison are schlieren photographs taken by Dr. Henry Sun in Singapore Institute of Standards and Industrial Research. It shows the key flow feature at base region. A slip line emanates from the lip and defines the exhaust plume/external flow boundary. Inside the exhaust plume there is a barrel shock that close just 8 times the radius of the plume downstream. The agreement between

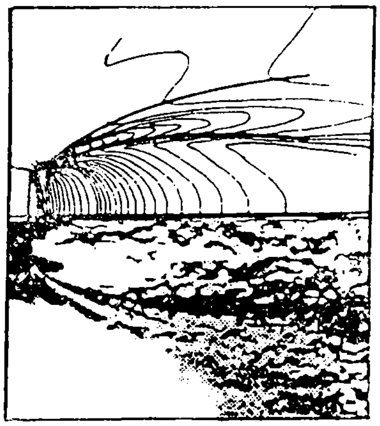

Fig. 2 Contours of density

computed and experimental observed flow features is good in every respect.

The contours of streamwise velocity and the contour lines of Mach number are plotted in Fig.3 and Fig.4, respectively. In the region near the

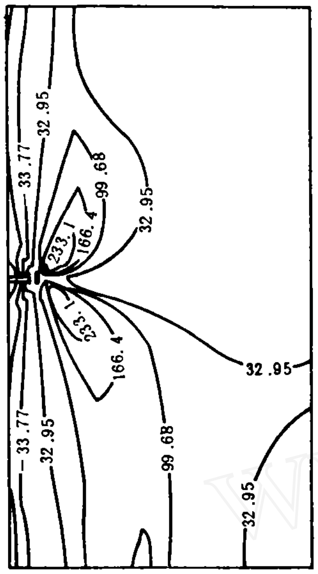

Fig. 3 Contours of streamwise velocity

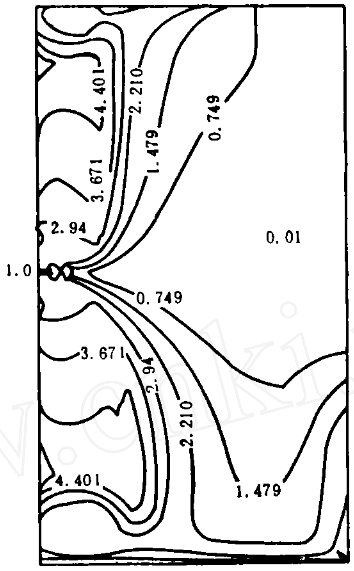

Fig. 4 Contours of Mach number

obstacle wall, there is an inverse velocity generated by the big inverse pressure gradient. The jet exhaust moves upwards and downwards along the edge of the obstacle with the inverse velocity at the downstream station. The interaction of the exhaust jet and the inverse velocity play a role of reducing energy (see also temperature contours). The maximum absolute value of the inverse velocity is 100m /s, which occurs near the edgy of the plume exit section.

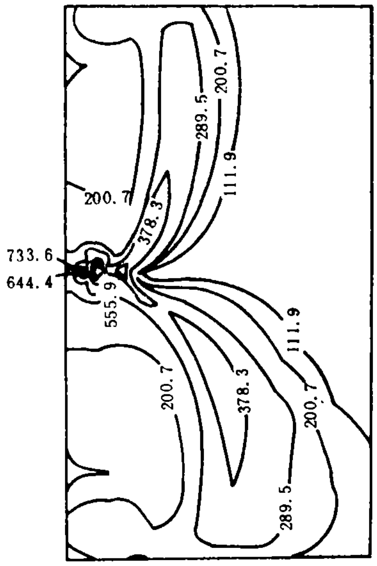

Fig. 5 Contours of temperature

Fig. 5 plots the contour lines of the temperature. The main physical features can be seen, first, near the jet exit plane, there is a deep temperature decrease region, second, the discontinuous line occurs at the location 5 times the diameter of the plume because of the dissipation energy of barrel shock wave, and third, the temperature in the lower region is higher than upper region as the plane at bottom influences the temperature distribution.

The present numerical investigation is used in engineering design of a missile launching pad by the Department of military equipment, Singapore Navy, successfully. The stream-upwind technique used in present computation is powerfully in smoothing the oscillation and the finite element method can be extended in other aerodynamic drag calculation. The computational results obtained are in good agreement with experiment observation and other valid numerical computations.

REFERENCES

1 Mikhail A G, Hankey W L and Shang J S. Computation of a Supersonic Flow Past an Axisymmetric Nozzle Boattail with Jet Exhaust. *AIAA Journal*, 1980, 18:869～875

2 Sullins G A, Anderson J D Jr, and Drummond J P, Numerical Investigation of Supersonic Base Flow with Parallel Injection. AIAA Paper 82-1001, 1982.

3 Zheng Min, Zhang Hanxin. Application of Non-Oscillatory and Non-free-parameters Dissipative Finite Difference Scheme to the Calculation of Free-Jet Flows. Acta Aerodynamic Sinica (in Chinese), 1989, 7(3): 273～281

具有后挡板的喷流流场的数值研究

王发民　赵　烈　卞荫贵　于　欣

（中国科学院力学研究所）

摘要　本文对发射台底面与后挡板之间的喷流流场进行了数值研究。采用迎风的 Petrov-Galerkin 有限元方法求解欧拉方程。描述了喷流出口处及挡板附近的流场细节。计算结果已成功地应用于导弹发射台的设计。

关键词　喷流/外流相互作用，鼓形激波，有限元方法，迎风方法。