

超声速主流中横向喷流干扰 流场数值模拟*

北京空气动力研究所 周伟江** 马汉东 李 锋

【摘要】 本文用二阶迎风 TVD 格式求解层流 N-S 方程, 数值模拟了超声速主流与横向喷流相互作用复杂干扰流场, 得到了合理的涡系结构, 计算波系结构及沿物面压力分布也与实验结果一致。计算还表明, TVD 格式不但具有较强的捕捉激波的能力, 而且用来模拟复杂涡系结构也是有效的。

关键词: 喷流 数值模拟 分离流动

1 前 言

随着宇航事业和空间技术的发展, 超燃冲压发动机的研究引起了越来越多的兴趣。在冲压发动机推进系统中, 燃料通过平行和垂直于主流两个不同方向注入到发动机中, 目的是使燃料与主流尽快混合^[1]。在一定的 Ma 数范围, 太多的垂直燃料注入可能导致发动机内部的热堵塞等现象。这些现象的发生又与流场结构密切相关。因此清楚地了解燃料注入与主流之间干扰流场细节, 对冲压发动机的设计具有实际意义。

燃烧室前垂直燃料注入主流的流动可简化为平板上横向喷流与主流的相互干扰。从实验照片和理论分析可以给出如图 1 所示的流场结构。流场中既有超声速区, 又有亚声速区。因此必须求解全 N-S 方程, 并选择既有较强的捕捉激波的能力, 又能模拟旋涡的差分格式, 才能给出如此复杂的流场结构。

目前通过实验^[2]对这一流动的基本波系结构及物面压力已有较为清楚的认识。但由于实验的局限性, 对涡系结构及一些流场细节(如喷流滑移线等)认识较少。Drummond^[1]用 MacCormack 格式模拟了这一流动, 但波系和涡系结构都不很理想。张涵信教授^[3]用显式 NND 格式研究了这类流场。本文选用的格式为二阶迎风 TVD 格式, 计算结果与实验进行了比较。

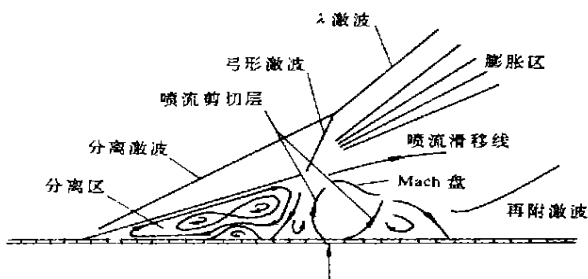


图1 流动模型

2 网格、微分方程、定解条件及差分方法

本文采用代数法生成变间距网格。在流动结构复杂或流动元变化较大的喷口及物面附近一定距离内, 均匀布置细网格, 然后按一定的代数关系逐渐增大, 但保持连接处有连续导数。这样可以比较合理地利用网格节点。记物理域与计算域之间存在关系式 $\xi = \xi(x, y)$, $\eta = \eta(x, y)$, $\tau = t$ 。则在计算坐标系 (τ, ξ, η) 下, N-S 方程无量纲形式为

$$\hat{\partial} \hat{U} / \partial \tau + \hat{\partial} \hat{E} / \partial \xi + \hat{\partial} \hat{F} / \partial \eta = \hat{Q} \quad (1)$$

其中 $\hat{U} = J^{-1} [\rho, \rho u, \rho v, e]^T$

$$\hat{E} = J^{-1} \begin{bmatrix} \rho \hat{u} \\ \rho u \hat{u} + \xi_x p \\ \rho v \hat{u} + \xi_y p \\ (e + p) u \end{bmatrix} \quad \hat{F} = J^{-1} \begin{bmatrix} \rho \hat{v} \\ \rho u \hat{v} + \eta_x p \\ \rho v \hat{v} + \eta_y p \\ (e + p) v \end{bmatrix}$$

J^{-1} 为坐标变换 Jacobian, $\hat{u}, \hat{v}$ 为反变速度。

$$\hat{Q} = \partial \hat{E} / \partial \xi + \partial \hat{F} / \partial \eta \quad (2)$$

$$\hat{E}_v = J^{-1} \begin{bmatrix} 0 \\ \xi_x \tau_x + \xi_y \tau_y \\ \xi_x \tau_y + \xi_y \tau_x \\ \xi_x \beta_x + \xi_y \beta_y \end{bmatrix} \quad \hat{F}_v = J^{-1} \begin{bmatrix} 0 \\ \eta_x \tau_x + \eta_y \tau_y \\ \eta_x \tau_y + \eta_y \tau_x \\ \eta_x \beta_x + \eta_y \beta_y \end{bmatrix}$$

$\tau_{xx} = \mu [(4/3) u_x - (2/3) v_y]$, $\tau_{xy} = \tau_{yx} = \mu (u_y + v_x)$, $\tau_{yy} = \mu [(4/3) v_y - (2/3) u_x]$
 $\beta_x = \gamma \mu p^{-1} \partial e / \partial x + u \tau_{xx} + v \tau_{xy}$, $\beta_y = \gamma \mu p^{-1} \partial e / \partial y + u \tau_{xy} + v \tau_{yy}$, $e = c^2 (\gamma - 1)$

求解(1) 需要给出初值条件和边界条件处理。本计算初场给定自由流值。上游边界在无扰动区, 给定 $\rho_\infty = 1.0, u_\infty = 1.0, v_\infty = 0, p_\infty = 1/\gamma Ma_\infty^2$; 下游边界认为足够远离喷口, 边界值采用沿 x 方向零阶外插; 外边界施加单波条件; 喷口给定喷口参数; 物面边界上 $u = v = 0, \partial \tau / \partial n = 0$, 并由法向动量方程求出压力 p_w 。

对一维方程组

$$\hat{\partial} \hat{U} / \partial \tau + \hat{\partial} \hat{E} / \partial \xi = 0 \quad (3)$$

求解格式为

$$A \delta_\tau U_{i-1} + B \delta_\tau U_i + C \delta_\tau U_{i+1} = D \quad (4)$$

$$A = - \frac{\lambda \theta}{2} \{ \hat{A}_{i-1} J_{i-1}^{-1} + R_{i-1/2} \text{diag} \mathcal{Y}'_{i-1/2} + \Psi(a_{i-1/2}^l + \mathcal{Y}'_{i-1/2}) \} R_{i-1/2}^{-1} J_{i-1/2}^{-1/2}$$

$$B = J_i^{-1} I + \frac{\lambda \theta}{2} \{ -R_{i+1/2} \text{diag} \mathcal{Y}'_{i+1/2} + \Psi(a_{i+1/2}^l + \mathcal{Y}'_{i+1/2}) \} R_{i+1/2}^{-1/2} J_{i+1/2}^{-1/2} \\ + R_{i-1/2} \text{diag} \mathcal{Y}'_{i-1/2} - \Psi(a_{i-1/2}^l + \mathcal{Y}'_{i-1/2}) \} R_{i-1/2}^{-1} J_{i-1/2}^{-1/2}$$

$$C = \frac{\lambda \theta}{2} \{ \hat{A}_{i+1} J_{i+1}^{-1} + R_{i+1/2} \text{diag} \mathcal{Y}'_{i+1/2} - \Psi(a_{i+1/2}^l + \mathcal{Y}'_{i+1/2}) \} R_{i+1/2}^{-1/2} J_{i+1/2}^{-1/2}$$

$$D = - (1/2) \lambda (\hat{E}_{i+1}^n - \hat{E}_{i-1}^n) + R_{i+1/2} \cdot \Phi_{i+1/2} - R_{i-1/2} \cdot \Phi_{i-1/2}$$

其中 $\lambda = \Delta \tau / \Delta \xi$, I 为单位矩阵, $l = 1, 4$, $\text{diag}(h^l)$ 为对角线元素为 h^l 的对角矩阵。

$$\Phi_{i+1/2} = [\Phi_{i+1/2}^1, \Phi_{i+1/2}^2, \Phi_{i+1/2}^3, \Phi_{i+1/2}^4]^T \quad (5)$$

$$\Phi_{i+1/2}^l = \sigma(a_{i+1/2}^l) \cdot (g_{i+1}^l + g_i^l) - \Psi(a_{i+1/2}^l + \mathcal{Y}'_{i+1/2}) \cdot \alpha_{i+1/2}^l$$

$$\alpha_{i+1/2}^l = (R^{-1} U)_{i+1}^l - (R^{-1} U)_i^l, \quad g_{i+1/2}^l = \min \text{mod}(\alpha_{i+1/2}^l, \alpha_{i+1/2}^l)$$

$$y'_{i+1/2} = \sigma(a'_{i+1/2}) \cdot \begin{cases} (g'_{i+1} - g'_i) / \alpha'_{i+1/2} & \alpha'_{i+1/2} \neq 0 \\ 0 & \alpha'_{i+1/2} = 0 \end{cases}$$

$$\sigma(a'_{i+1/2}) = (1/2) \psi(a'_{i+1/2}) + \lambda(\theta - 1/2)(a'_{i+1/2})^2$$

$$\psi(z) = \begin{cases} (z^2/\delta + \delta)/2 & |z| < \delta \\ |z| & |z| \geq \delta \end{cases}$$

δ 为一选定的小量。 $\theta = 0$ 时格式退化为显式, $\theta \neq 0$ 时为隐式格式。

在二维情况, 求解(1)式分以下两步

$$\begin{cases} A_{\xi} \delta \tau U_{i-1,j}^* + B_{\xi} \delta \tau U_{i,j}^* + C_{\xi} \delta \tau U_{i+1,j}^* = D_{\xi} + D_{\eta} \\ A_{\eta} \delta \tau U_{i,j-1} + B_{\eta} \delta \tau U_{i,j} + C_{\eta} \delta \tau U_{i,j+1} = \delta \tau U_{i,j}^* \end{cases} \quad (6)$$

(6)式中下标“ ξ ”和“ η ”表示 ξ 和 η 方向。

3 分析及结果

计算共分两部分。首先为验证程序和方法, 对超声速球锥绕流进行了计算, 给出等压线图和物面压力分布与文献结果的比较。接着对超声速主流与横向喷流干扰流场进行了模拟。暂时选主流和喷流同为空气, 不带化学反应, 与不带化学反应的实验结果进行了比较。

3.1 超声速钝体绕流

来流 Ma 数为 4.0, 外形为球锥, 长度为球头半径的 5 倍, 半锥角 $\theta = 10^\circ$; Re 数为 10^4 , 计算网格数为 41×37 。图 2 为计算流场等压线图。图中显示了清晰的头激波位置以及格式强的

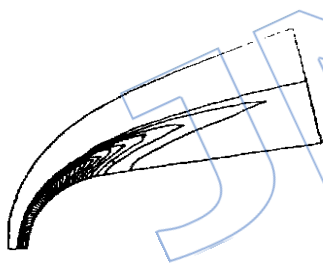


图 2 球锥等压线

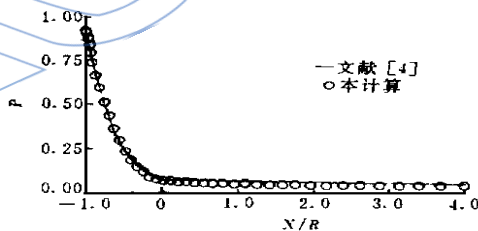


图 3 球锥物面压力分布

捕捉激波的能力。图 3 为计算物面压力分布有粘、无粘与文献[4]结果的比较。压力(p/p_{∞})差, 头部驻点附近在 0.01 以下, 锥体上为 0.005 以下。所以无论是定性上还是定量上都与文献[4]相符。

3.2 横向喷流流场

计算条件为来流 Ma 数为 2.5, 来流静温 $T_{\infty} = 130^\circ\text{K}$, 基于喷口宽度的 Re 数为 4×10^4 (相当于来流 Re 数为 5×10^5)。喷口条件为 $Ma_j = 1.0$, $T_j = 243\text{K}$, $p_j/p_{\infty} = 43.0$ 。网格数 121×85 。

图 4a 为等密度线图。从等值线图分析可知, 本计算得到了分离激波、弓形激波、 λ 激波、喷口前分离区与主流的剪切层、高速喷流与外流的混合剪切层、Mach 盘、扇形膨胀区、喷流滑移线和再附激波等流场理论分析应有的波系和接触间断结构。与实验照片^[2]相比, 发现两者相当符合。尤其从等密度线图中可以看到清晰的喷流滑移线及高速喷流与外流的混合剪切层, 这在以往的数值计算文献中较少见到。图 4b 为文献[1]计算的等密度线。从上述波系分析, 并与文

献[1]结果比较表明, 本文在捕捉激波及接触间断方面是成功的。

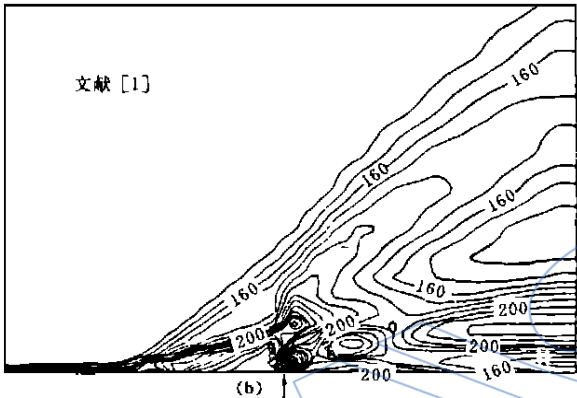
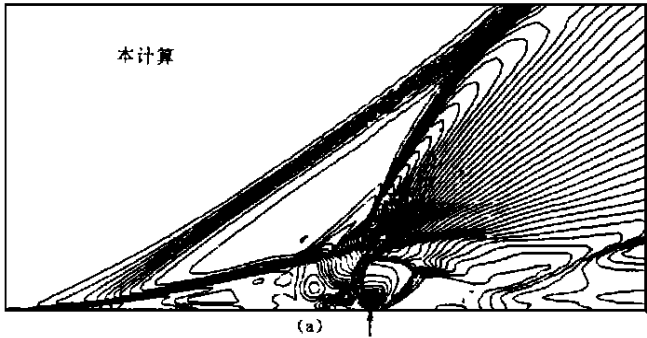


图 4 等密度线

图 5 为喷口前速度向量图, 图 6 给出了喷口前分离区中两个不同区域 $A-A$ 和 $B-B$ 的局部速度放大图。从图 5 和图 6 可以看出, 计算流场喷口前主分离涡分叉为二个同向的支涡, 支涡之间以鞍点相连接。鞍点的下方形成一个分离泡, 并有反向二次涡存在(图 6B-B)。喷口上游拐角处, 主涡与喷流相互作用, 产生另一个反向二次涡(图 6A-A)。

图 7 为喷口后流场速度向量图及其局部速度放大图。从图 7 中不但可以看到明显的喷口后主分离涡, 而且从主分离下方局部速度放大图中, 发现喷口后主分离流场在回流过程中再次分离, 在其下方形成一个分离泡和反向二次涡。造成回流再次分离的原因主要可能在于回流与高速喷流相互作用, 引起喷口后、再附点前物面压力波动形成的局部逆压梯度(见图 8a, 图中 S 点为近似回

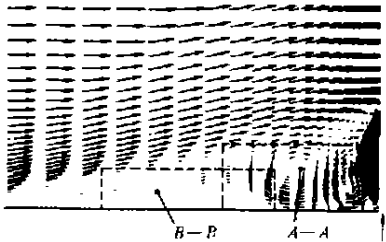


图 5 喷口前速度场

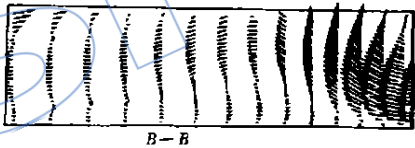
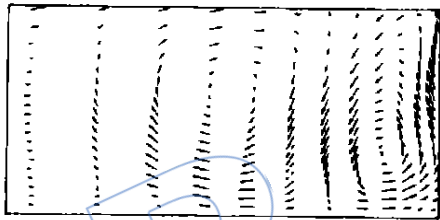


图 6 喷口前涡系速度放大图

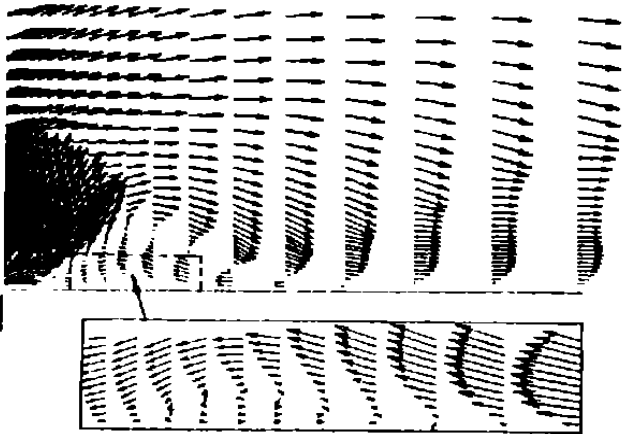


图 7 喷口后速度场

流分离点位置)。在文献[5]的同类流场计算中,也发现了喷口后压力分布具有与本文类似的局部逆压梯度区。

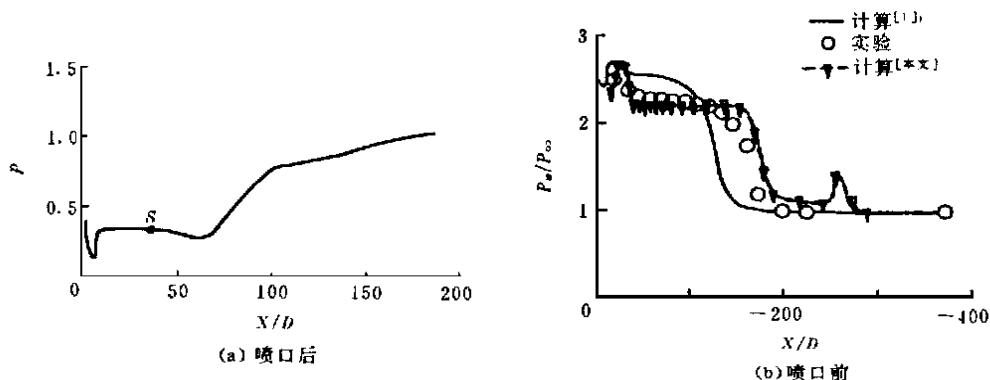


图 8 物面压力分布

图 8 为平板物面压力分布。图 8a 为喷口后压力分布,图 8b 为喷口前压力分布。横坐标为离喷口中心的距离(喷口宽度的倍数)。在图 8b 中还给出文献[1]计算结果和实验压力分布。由图可见,本文压力分布与实验值符合较好。只是分离点位置(190~195d)略大于实验结果(175~180d)。影响因素主要有两个:(1)本文采用的为层流模型;(2) Re 数低于实验值。因此分离点略远也是合理的。

4 结 论

从上面计算和分析表明:

(1) 本文成功地模拟了超声速主流与横向喷流复杂干扰流场,得到了较为细致的波系和涡系结构,流动图象及压力分布与实验结果一致。

(2) 计算表明 TVD 格式不但有较强的捕捉激波的能力,而且用于模拟复杂涡系结构也是有效的。这对今后实际应用中差分格式的选取具有实际意义。

参 考 文 献

- 1 Drummond J P, Weidner E H. Numerical Study of a Scramjet Engine Flow Field. AIAA 81-0186, 1981
- 2 Thayer W J. The Two-Dimensional Separated Flow Region Upstream of Inert and Chemically Reactive Transverse Jets. Ph. D. Thesis Department of Mechanical Engineering University of Washington, 1974
- 3 张涵信,刘君. 超声速主流与横向喷流场激波—旋涡结构的数值模型拟. 空气动力学学报, 1991, 1
- 4 Любимов А Н, Ру анов В В. Течения Таза Около Ту лых Те л (II). Мосца, 1970
- 5 Takahashi M, Hayashi A K. Numerical Study on Mixing and Combustion of Injecting Hydrogen Jet in a Supersonic Air Flow. AIAA 91-0574, 1991
- 6 Yee H C, Warming R F, Harten A. Implicit Total Variation Diminishing(TVD) Schemes for Steady-state Calculations. J of Computational Physics, 1985: 57

STUDY OF OPTICAL VISUALIZATION METHODS FOR EXHAUST JET OF ROCKET ENGINE

Yan Dapeng, He Anzhi, Miao Pengcheng, Wang Hailin
(*East-China Institute of technology*)

ABSTRACT

Two optical methods are studied which are used to visualize the flow field of exhaust free jet and jet impingement from a real solid rocket engine. One is equal-thickness-fringe F-P interferometry which is constructed with two mirrors about 200mm in diameter. It features that the optical layout is co-optical light path and separated form, the distance between two mirrors is more than 20m and the resistance to vibration distortion is strong. Another is moire deflectometry which has an aperture over 400mm in diameter. In this method a big-diameter collimated light beam is used to explore the flow field, then it is reduced and recollimated into a small-diameter recollimated light beam and detected by a two-gratings system to produce moire fringes. It is characterized by large aperture and simplicity of the method. The experimental results show that the equal-thickness-fringe F-P interferometry can be used to visualize the flow field of the exhaust jet from a real solid rocket, and the moire deflectometry can be even used to visualize the extensive flow field of rocket exhaust jet impingement besides the exhaust free jet.

NUMERICAL SIMULATION OF INTERACTION BETWEEN SUPERSONIC MAIN STREAM AND TRANSVERSE JET

Zhou Weijiang, Ma Handong, Li Feng
(*Beijing Institute of Aerodynamics*)

ABSTRACT

The laminar N-S equations have been solved for a supersonic flow with a transverse sonic jet. The numerical algorithm used here is a second-order windward TVD scheme. The conditions of supersonic main stream are: $Ma_\infty = 2.5$, $T_\infty = 130\text{K}$, $Re_\infty = 5 \times 10^5$. In sonic jet, $T_j = 243^\circ\text{K}$ ($T_j/T_\infty = 1.87$), $P_j/P_\infty = 43.0$. The grid nodes are 121×95 with compressed in both a transverse direction near the wall and a streamwise direction near the injector. The detail waves and vortex structures are obtained, the flow field pictures and pressure distribution along plate are compared with experiments in a good agreement. The calculation also demonstrates that the TVD scheme not only has high capability of shock capturing, but also can be applied to efficient simulation of the complex vortex structure.

COMPUTATION OF FLEXIBLE-WALL AIRFOIL FLOW USING N-S EQUATIONS

Ma Xia, Xiao Linkui, Jiang Jinggui, Chang Mianchun
(*Institute of Mechanics, Chinese Academy of Sciences*)

ABSTRACT

The 2-D Reynolds-averaged compressible unsteady full Navier-Stokes equations are solved by using the Beam-Warming scheme with Baldwin-Lomax turbulent model about the flexible-wall airfoil. A method is proposed which can treat the locally changing surface of airfoils. The adaptive surface in the shock region can influence the