

具有无源控制空腔时正激波/ 湍流 附面层干扰的数值模拟

南京航空航天大学 梁德旺^{**}
德国卡尔斯鲁厄大学 R. 包宁

【摘要】 采用雷诺平均 $N-S$ 方程和 B/L 代数湍流模型计算了具有无源控制空腔时正激波/ 湍流附面层干扰流场。计算与实验结果的比较表明, 本文方法可较准确地预测激波结构、激波与附面层干扰区流动基本特征及波后流动分离状态、激波位置、波前马赫数等参数。

主题词: 激波/ 干扰 控制问题 湍流边界层 数值模拟

分类号: O357.42

1 引言

激波前后压力的突然变化, 可能使得超临界机翼的阻力增加, 升力下降, 超音速飞行器进气道出口流场畸变增加, 总压恢复系数减小。Bahi^[1]首先通过实验证实了在激波下端开一空腔, 引波后的高压气流吹除波前附面层, 从而减弱激波, 减小波阻的无源控制空腔技术的有效性。关于激波/ 附面层干扰的数值模拟, 已有两种方法: 一是粘性/ 无粘耦合方法。近年来, 又进一步发展成三层模型方法。另一方法是 Chen^[2]等人提出的, 他们采用雷诺平均的薄层 $N-S$ 方程和修正的 B/L 代数湍流模型成功地计算了 NACA 翼型上有和无空腔控制时正激波/ 湍流附面层干扰流场。本文作者也曾用雷诺平均全 $N-S$ 方程成功地计算了管内正激波/ 湍流附面层干扰流场^[3]。本文则采用了雷诺平均全 $N-S$ 方程和 B/L 湍流模型, 计算了二维拉伐尔喷管内具有无源空腔控制时正激波/ 湍流附面层干扰流场。

2 基本方程与数值方法

为了计算方便, 本文采用了曲线坐标系下的雷诺平均 $N-S$ 方程和标准 B/L 代数湍流模型^[4]。

图 1 给出了本文计算域的几何形状和计算网格。实际上, 它是德国卡尔斯鲁厄大学流体及流体机械实验室的一个具有无源控制空腔的正激波/ 湍流附面层干扰实验段, 其喉道高度为 0.1 m (位于 $X = 0.3113$ m 处), 实验段在垂直于纸面方向的宽度为 0.05 m, 在 $X = 0.3719 - 0.4418$ m (边长 0.07 m) 的下壁为渗透直壁, 渗透壁下为一腔体, 渗透壁的渗透率 ϵ 可以根据需要通过更换渗透直壁而得到。计算网格为 65×289 (图示为 33×145), 网格生成采用了网格压缩技术, 近壁网格很密, 其中第一条网格线距壁面之最小值约为 10^{-5} m, 其对应的无量纲参数 $y^+ < 1.0$, 沿流动方向的网格在可能出现激波的区域也很密, 其 Δx 约为 0.5 mm。

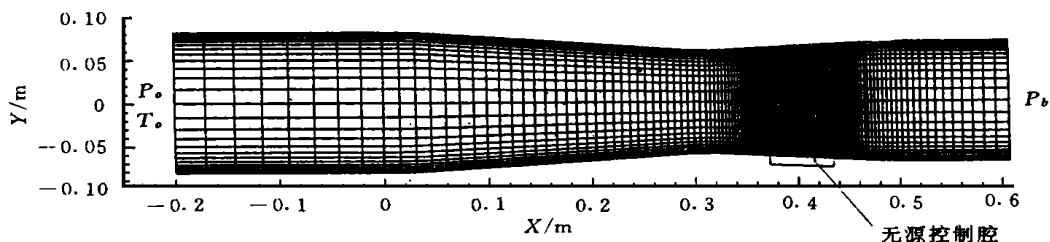


图 1 计算域及计算网格

由于喉道处音速截面的存在,很难直接给出来流速度或流量,因此来流物理条件为给定总温 T_0 、总压 P_0 。出流为亚音,其物理条件选定为静压 P_b 。因渗透壁上的小孔非常小($\varnothing 0.125$ mm)而且很密,直接用 N-S 方程求解所需网格数及机时是现有计算机所难承受的。文献[2]的计算是基于 Darcy 定律,假定速度分布规律,根据流量平衡求得垂直于壁面的渗透速度的,因而有一定的近似性。本文计算则是根据文献[5]给出的通过大量实验数据归纳出的经验公式。根据文献[5],渗透壁的法向气流速度

$$v_m = f(P, P_{inter}, T, \epsilon) \quad (1)$$

其中, P 和 T 分别为流场中渗透壁上流体的压力和温度; P_{inter} 为渗透壁下空腔内流体的静压力; ϵ 为壁面渗透率。计算中,先根据流进、流出腔体的流体质量相等迭代求出空腔内压力,然后根据式(1)求出壁面法向速度。本文算例中,计算所得空腔内压力和壁面法向速度与实验测得的结果基本一致。

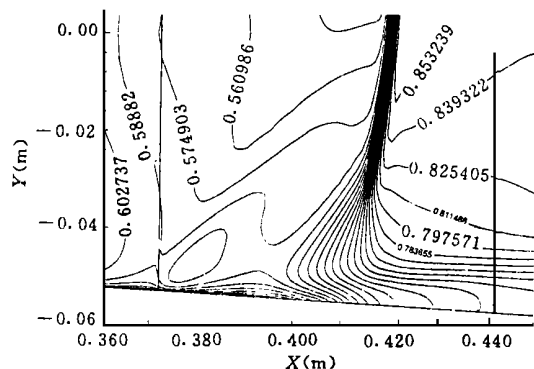
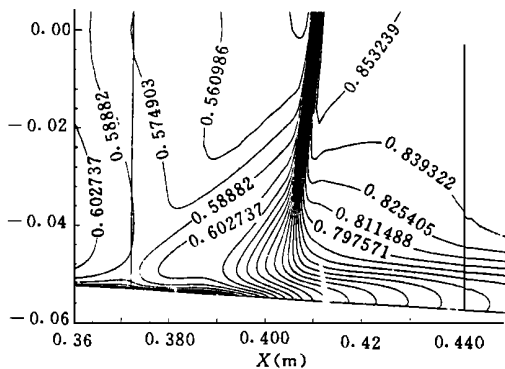
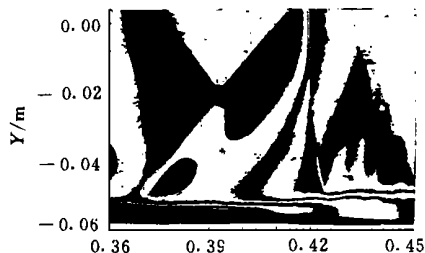
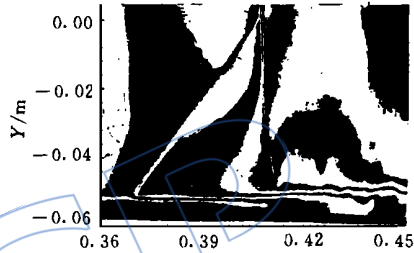
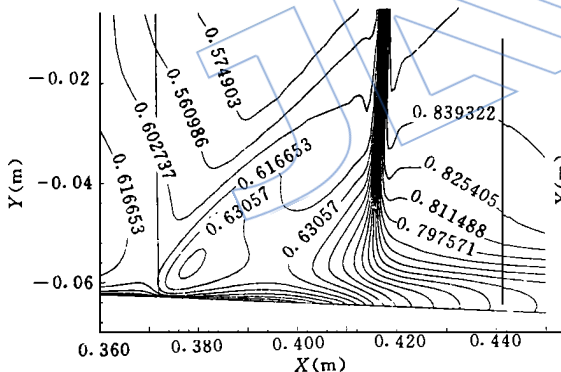
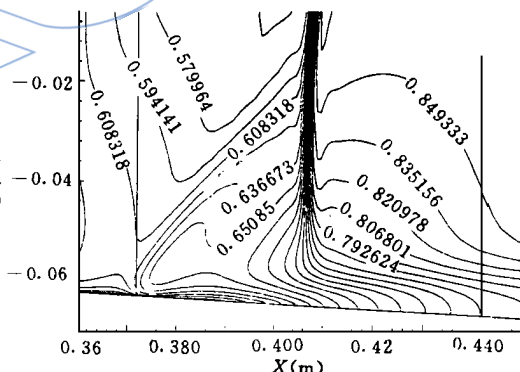
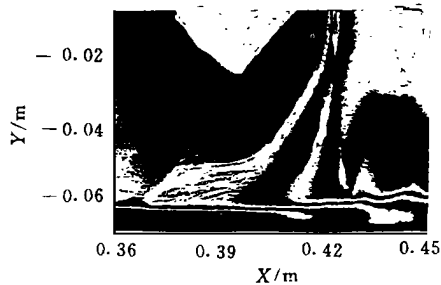
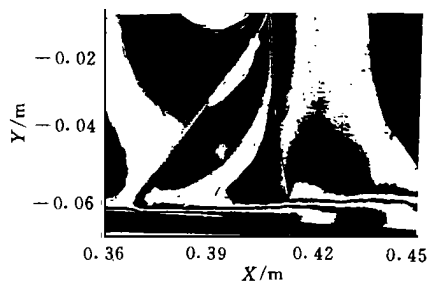
本文的计算采用时间相关法求解定常流动,其中空间采用格心格式的有限体积法,而时间采用五步龙格-库塔法,其 CFL 数为 5.0。粘性项的计算采用中心格式,对流项则采用二阶精度的 Jameson 方法^[6]。为了加速收敛,计算中同时采用了当地时间步长,网格加密,多层网格,隐式残余平均和焓阻尼等加速收敛技术。

3 计算结果与分析

图 2 和图 3 给出了壁面渗透率 $\epsilon = 0.053$ 和 0.040 情况下激波分别位于近壁处激波的位置 $X_w = 0.4069$ 和 0.4169 m 时,计算所得等密度线图与实验测得的纹影图谱的比较。由图可知,计算与实验所得图谱基本相似,激波形状也基本一致,“ λ ”波结构清晰可见,表明本文的关于渗透壁边界条件,用雷诺平均全 N-S 方程对激波/附面层干扰的计算是成功的。

此外,计算中还比较了激波前后速度分布,干扰区壁面压力分布。结果显示计算所得波前速度大小与分布、附面层厚度、壁面压力等参数与实验吻合很好,但波后速度、壁面压力等误差较大。造成这一误差的主要原因有:(1)因网格不够密,计算格式的精度造成计算所得激波厚度远大于其物理厚度;(2)所用 B/L 湍流模型没有压力梯度,物面法向速度等的影响,因而不能准确地估计干扰区内附面层的发展;(3)实验具有明显的三维效应,尤其是波后侧壁附面层的迅速加厚较大地影响了核心流速度,而计算是纯二维的,没有计及侧壁附面层。

由图 2 和图 3 还可以看出,无源控制空腔的存在,产生了很强的“ λ ”形波,使近壁区内形成

计算结果 ($P_b/P_o = 0.676$)计算结果 ($P_b/P_o = 0.690$)(a) $X_w = 0.4169$ m(b) $X_w = 0.4069$ m图2 计算等密度线与实验纹影图谱的比较 ($\epsilon = 0.040$)计算结果 ($P_b/P_o = 0.676$)计算结果 ($P_b/P_o = 0.690$)(a) $X_w = 0.4169$ m(b) $X_w = 0.4069$ m图3 计算等密度线与实验纹影图谱的比较 ($\epsilon = 0.053$)

了一系列的压缩波,从而弱化了激波,减小了波阻。

图 4 给出了有和无无源控制空腔及不同渗透率下反压变化对激波位置的影响曲线。图中横坐标为反应 P_b 与来流总压 P_o 之比,纵坐标则为中心区激波位置 X_s 。由图可知,在反压和来流总压比不变条件下,随着壁面渗透率 ϵ 增加(认为无空腔控制时 ϵ 为零),激波将前移,因而波前马赫数下降,波后总压增加。

图 2 和图 3 还表明,激波位置不变情况下,壁面渗透率增加,“ λ ”波的前支波加强,而扇形压缩区强度将弱,波阻减小。

4 结束语

本文用雷诺平均全 N-S 方程、B/L 湍流模型和渗透壁经验模型成功地计算了具有无源控制空腔时正激波/湍流附面层干扰流场,计算与实验结果吻合较好,从而进一步表明本文方法可较准确地预测有、无空腔控制措施时激波结构、激波/湍流附面层干扰区流动基本特征。

参 考 文 献

- 1 Bahi L, Ross J M, Nagamatsu H T. Passive Shock-Wave/ Boundary Layer Control for Transonic Airfoil Drag Reduction. AIAA 83- 0137
- 2 Chen C L, Chow C Y, William R Vandalsem, Terry L Holst. Computation of Viscous Transonic Flow over Porous Airfoils. AIAA 87- 0359
- 3 梁德旺,包宁. 管内正激波/湍流附面层干扰的数值模拟. 空气动力学学报(待发表)
- 4 Baldwin B S, Lomax H. Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows. AIAA 78- 257
- 5 Jameson A, Schmidt W, Turkel E. Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time Stepping Schemes. AIAA 81- 1259

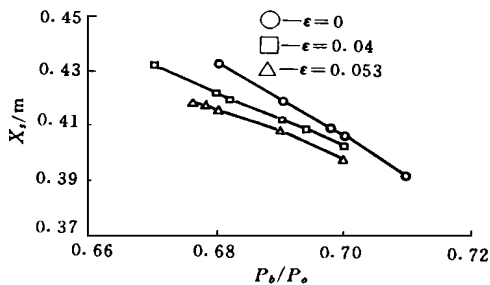


图 4 反压与激波位置之间的关系曲线

(责任编辑 魏星禄)

algorithm for the full Navier–Stokes solver in generalized curvilinear coordinates is applied to calculate the viscous flow in the IPS. At last, the particle trajectories in the above viscous flowfield and separator efficiency are obtained. Good agreement between the experimental data and predicted results indicates that the presented procedure and its program contribute to development and design of future high performance inlet particle separator.

Key Words Flow fields Separators Numerical Simulation

AN IMPROVEMENT OF $k-\omega$ TURBULENT MODEL

Shen Huili

(7th Dept. Northwestern Polytechnical University, Xi'an 710072)

Zheng Xiaoqing and Liu Feng

(University of California U. S. A.)

ABSTRACT

The $k-\omega$ turbulence model for predicting turbulence transition at a local Reynolds number lower than that observed in experiments. Wilcox suggested an approach to predict transition through modifying the viscosity in the ω equation with five more closure coefficients, but the approach made the model too complicated. Here we introduce only one simple parameter to improve the model. An inequality is proposed to set a lower limit for the specific dissipation. In order to make the Navier–Stokes equations and $k-\omega$ equations strongly coupled, a staggered finite volume scheme for discretizing the N–S equations and the $k-\omega$ model equations has been developed. The algorithm was tested for a flat plate flow. Computational results show that the present improvement of the $k-\omega$ turbulence model to control the transition position is effective.

Key Words Turbulent flows Mathematical models Transition points Boundary layer

NUMERICAL SIMULATION OF NORMAL SHOCK WAVE/TURBULENT BOUNDARY LAYER INTERACTION WITH PASSIVE CAVITY

Liang Dewang

(2nd Dept. Nanjing University of Aeronautics and Astronautics, Nanjing 210016)

R. Bohning

(University of Karlsruhe Germany)

ABSTRACT

A full Navier–Stokes procedure, coupling with a Baldwin/Lomax turbulence model, is used to study the normal shock wave/turbulent boundary layer interaction with passive cavity. In addition, the ventilation velocity profile on the porous plate is calculated by an empirical model formulated from experiments. The total pressure and total temperature boundary conditions for inflow are given due to the existence of sonic section in the channel flow and the static pressure boundary condition for outflow is given as well. The comparison between numerical and experimental results shows that the presented procedure can predict the shape of the shock, the Mach number in front of the shock, the pressure ratio on the wall and position of the shock quite well.

Key Words Turbulent boundary layer Shock wave interference Control problems

Numerical Simulation

PERFORMANCE PREDICTION OF BOWED VANE AXIAL- FLOW TURBINES

Yu Qing

(4th Dept. Beijing University of Aeronautics and Astronautics, Beijing 210016)

ABSTRACT

A method for the performance prediction of axial- flow turbines with bowed and conventional vanes is presented, which uses flow function as control equation on S2 surface. Profile loss is calculated by numerical method on several S1 flow surfaces. Secondary loss and tip- leadage loss are empirically estimated with pitch- wide average loss- es along the cascade blades. The effect of the blade bowed angle on the secondary loss is tak- en into account. The comparison of the predicted results with experimental data confirms the feasibility of the method.

Key Words Performance analysis Prediction techniques Turbines

MEASUREMENT AND DETERMINATION OF AEROENGINE THRUST

Du Heling

(Chinese Gas Turbine Establishment, Jiangyou 610500)

ABSTRACT

It is difficult to measure the flight thrust of Aeroengine directly. It is determined now by calculation with mathematical relationships of the flight thrust to the physical quantities which can be measured directly. According to the definition and the expression of the thrust, as well as the conditions derived from them, a method of aeroengine thrust determination and its program are provided on the basis of engine part tests, model tests, engine ground test, altitude- simulation tests, and flight tests. At the same time, the important function of the altitude- simulation facility is also analyzed in respect of correct flight thrust determination.

Key Words Aircraft engines Thrust Measurement method

VALIDATION TESTS WITH A SCALE- MODEL FOR SURFACE SOUND PRESSURE PREDICTION OF PROPELLER AIRCRAFT

Wang Tongqing

(4th Dept. Beijing University of Aeronautics and Astronautics, Beijing 100083)

ABSTRACT

An experimental research with a propeller aircraft scale model has been completed to obtain reasonably comprehensive data for validating its improved sound field model. Instead of the tests in anechoic windtunnel as abroad, the scale- model tests was accomplished in outdoor conditions. In the predicton of propeller noise in free field with the improved sound field model, the load data on the propeller blade surface were obtained from calculation of inverse problem. The measured results of sound pressure on the scale- model have been compared with that predicted by the improved sound field model in a reasonable a-