

涡轮级三维粘性流场的数值模拟*

哈尔滨工业大学 袁 宁** 王松涛 张振家 王仲奇 冯国泰

【摘要】 本文给出了一种求解涡轮级三维定常粘性流场的计算方法,并对某型航空发动机的高压涡轮进行了数值模拟。该计算程序考虑到了涡轮级工作温度范围较大而引起的变比热问题。文中采用了三阶精度的 TVD 格式和 B-L 湍流模型。叶列间参数的传递采用了“混合平面”法。叶列间处理采用外推无反射边界条件来保证参数在交接面上的周向不均匀。同时在交接面上构造了 Riemann 解和使用松弛因子来提高程序在叶列间小间隙情况下的稳定性。

关键词: 涡轮 三维流动 数值模拟 计算方法

分类号: V 211.18

1 引言

涡轮级计算同单个叶栅排的计算存在着差别,首先涡轮级在很大的温度范围内工作,比热变化对流场气动参数的影响在计算中应该被考虑;其次叶列间参数的传递是否合理也将直接影响到计算结果的准确性。因此本文的计算采用了变比热计算方法;在叶列间处理上采用了“混合平面”法,混合模型着眼于流量、动量、总能量、焓和熵的守恒性。这种模型的计算量小,在工程计算中应用也最为广泛。由于在绝大多数涡轮级的结构中动、静叶间隙很小,所以气动参数在交接面上具有很强的不均匀性,这使得计算程序的稳定性很差。Saxer^[1]在他的文献中考虑了特征传播方向用来消除参数在交接面上的跳动,解决了定常计算中叶列间的小间隙问题。受此启发本文在交接面上构造了 Riemann 解并使用了松弛因子,大大提高了小间隙涡轮级计算程序的稳定性。另外涡轮级计算应尽量保持参数在交接面上的周向不均匀性,本文采用外推边界条件的方法保证了本列参数在交接面上的周向不均匀。

2 数值方法

计算忽略了动、静叶间流动的非定常性,认为各叶栅内部流动是定常的。各叶片排在同一个时间步各被看作是相对独立的系统,通过“混合平面”传递参数。同时考虑到变比热的情况。

基本方程:差分格式采用了三阶精度的 Goddave 格式,它具有 TVD 的性质。采用了隐式近似因式分解法加快程序的计算速度。湍流模型采用 B-L 代数模型^[2]。

2.1 变比热计算

北航崔济亚教授^[3]曾为变比热计算应用到气体流动中做了大量的工作。通常认为比热是与温度无关的常数,但当起始温度与终止温度的差异较大时,比热值的差异也较大,因此计

算的准确性也就降低了。在多级涡轮计算中, 尤其当级数越多时这种差异就越大。所以在涡轮级计算中采用变比热计算是十分必要的。本文所指的变比热计算是在求解气动热力参数时, 假定气体是比热只随温度变化的完全气体, 任意一个温度都有与其对应的热力参数。

等熵过程的求解必须满足 dS 为零的条件, 采用关系式: $T dS = c_p dt - R/p \cdot dp$ 来进行逼近求解。应该注意到控制方程中 e 与压力 p 的关系不应再表示为定比热的形式。认为在每一个时间步中的每一个控制单元体上比热比为定值, 保持了控制方程的特征矩阵在形式上同定比热时一致。

2.2 叶列间处理

本文采用的是“混合平面”法^[4,5]。选取 $\bar{\rho}$, ρ_u , ρ_v , ρ_w 和 ρ_p 作为平均处理量。根据文献[6]中的方法可分析出这种平均模型可保证流量、动量在混合前后保持守恒, 同时总能量在混合后减小的幅度和熵在混合后增加的幅度都很小。

由于涡轮中动、静叶在通常的情况下, 其间隙是比较小的。这样动、静叶的进出口边界就十分靠近前、后缘, 从而引起附加误差。采用无反射边界条件可以很好的解决这一问题。Giles^[7]曾提出了一个完善的无反射边界条件, 但使用起来比较复杂。本文采用的是Denton^[4]的一个简化的近似无反射边界条件—外推边界条件的思想。由于进出口边界与叶片的前后缘相靠很近, 在边界上存在着很强的周向不均匀, 为此本文采用了外推周向不均匀性的做法, 在混合平面上只保证其周向平均值等于上(下)游传递来的周向平均值。这样就可以使本列的周向不均匀性在进出口边界上得到体现。

在叶列间小间隙情况下, 由于在交接面上参数的不均匀性非常强, 如果在尾缘出现激波时这种不均匀性还会被加剧, 因此程序的稳定性非常差。本文采用在交接面构造Remainn解来解决在交接面上参数的跳动问题。首先Remainn问题考虑到了波特特征的传播方向, 同时Remainn问题的解可以是有间断的(需要满足熵条件), 也可以是无间断的。Remainn问题解的这些特点正好适合于处理叶列间参数的传递问题。本文以动叶进口为例介绍交接面上Remainn解的构造。上游静叶出口的平均参数记为: $\bar{\rho}_s$, $\bar{u}_s$, $\bar{v}_s$, $\bar{w}_s$ 和 $\bar{p}_s$, 下游动叶进口的平均参数记为: $\bar{\rho}_r$, $\bar{u}_r$, $\bar{v}_r$, $\bar{w}_r$, $\bar{p}_r$ 。考虑到动叶周向旋转速度 ω 则得到: $\Delta\rho = \bar{\rho}_r - \bar{\rho}_s$, $\Delta u = \bar{u}_r - \bar{u}_s$, $\Delta v = \bar{v}_r - \bar{v}_s$, $\Delta w = \bar{w}_r - \bar{w}_s + \omega r$, $\Delta p = \bar{p}_r - \bar{p}_s$ 。记动叶进口轴向第一站($I=1$)参数为: $\rho_{1,j,k}$, $u_{1,j,k}$, $v_{1,j,k}$, $w_{1,j,k}$ 和 $p_{1,j,k}$ 。构造 $I=-1$ 时, $\rho_{-1,j,k} = \rho_{1,j,k} - \Delta\rho$, $u_{-1,j,k} = u_{1,j,k} - \Delta u$, $v_{-1,j,k} = v_{1,j,k} - \Delta v$, $w_{-1,j,k} = w_{1,j,k} - \Delta w$ 和 $p_{-1,j,k} = p_{1,j,k} - \Delta p$, 这样就通过 $I=-1$, $I=1$ 在 $I=0$ 的交接面上构造了Remainn问题。虽然精度仅为二阶, 但作为进口边界条件的处理已经是足够的了。

2.3 边界条件

涡轮级的进口给定进口总压、总温、进口气流角在涡轮级的出口即动叶出口, 给定出口叶片中部的静压 p_{out} , 简单径平衡方程 $\bar{q}/\bar{a} = \rho_w^2/r$ 被应用到出口边界上。进出口边界上考虑到了特征波的传播方向。在静叶、动叶进出口的上下游应用了周期性边界条件, 在叶片内、背弧和上下端壁给定无滑移条件。

在每一时间步通过“混合平面”, 上游向下游传递平均总温、平均总压以及平均速度, 而下游应向上游反馈平均静压。这些平均参数在每一时间步都会发生变化, 直到收敛时为止。

3 算例结果与讨论

应用本文的计算方法对某一航空燃气涡轮的高压级进行了计算, 计算网格采用了H型网

格 (见图 1), 静子、动子流场的计算网格点数分别为 $60 \times 39 \times 39$, $62 \times 39 \times 39$ 。

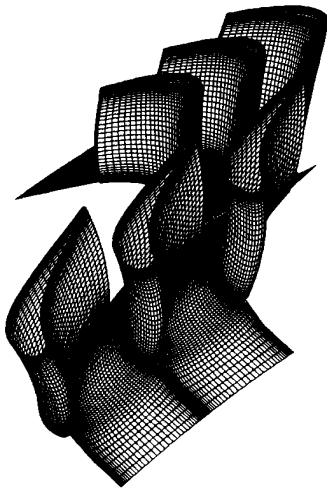


图 1 计算网格

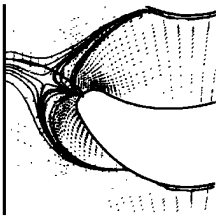


图 2 小间隙情况动叶轮毂处
流线速度与速度矢量

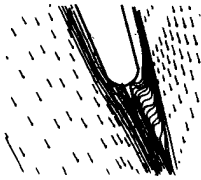


图 3 静叶尾缘流线
与速度矢量

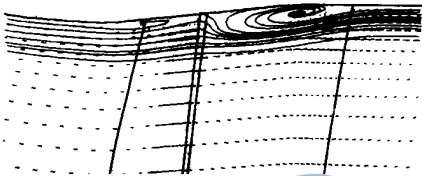


图 4 高压涡轮机匣部位动、静叶交接面处分离区

该计算方法还无法考虑涡轮级中流动的非定常性, 只是从定常的角度分析参数的分布。该方法叶列间处理保证了各叶列在小间隙情况下流动在周向的不均匀性, 并且流动的细节没有因此而破坏, 如图 2 和图 3 中可以清楚地看出粘流计算的高压动叶轮毂处叶片前缘的鞍点和马蹄涡生成线, 静叶中部的尾迹。甚至可以描述在动、静叶交接面上存在的分离现象 (见图 4)。

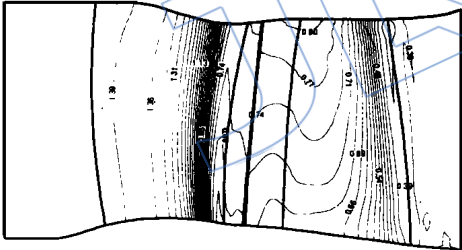


图 5 子午流道中部静压等值线分布

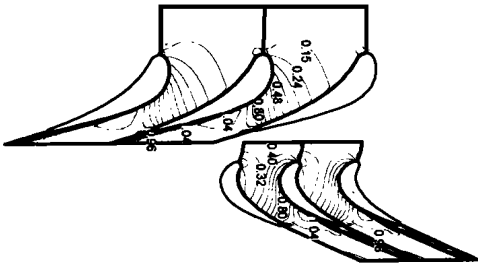


图 6 叶展中部回转面 M_a 等值线

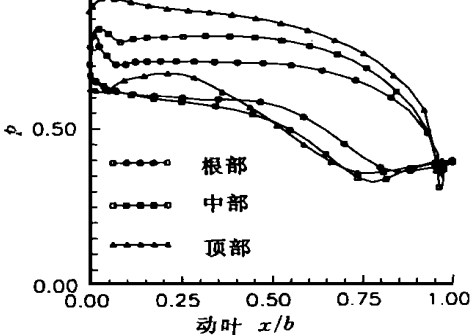
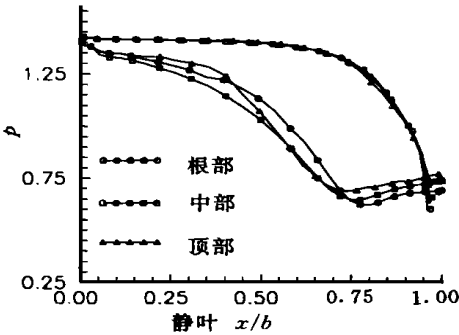


图 7 叶片表面静压分布

从图 5 中可看出由于高压涡轮静叶采用收敛子午型面和弯扭叶片, 所以在静叶的顶部流

道内形成了一个C型的静压分布。它会带来如下的结果: (1) 减小静叶顶部的横向二次流动; (2) 增加静叶顶部的密流, 有利于减小或消除静叶顶部的分离, 从而降低损失。从图6中可以看到清楚地看到附面层在叶片表面的分布, 尾迹以及气体的膨胀流动。从图7可以看出在静叶的根、中、顶部气流的膨胀流动平稳, 气流基本在后部加载; 动叶存在一些攻角, 属于均匀加载叶型。

高压涡轮静叶吸力面的极限流线反映出静叶吸力面附面层内部的径向压力梯度较大, 气流在后缘近机匣处发生分离后又再附着, 在轮毂处通道涡向上卷起形成一螺旋点 (见图8)。高压涡轮动叶吸力面的极限流线显示出机匣与轮毂处通道涡的分离线, 在动叶后部气体发生的分离现象也清晰可见 (见图9), 这是由于气体的过度膨胀造成的。

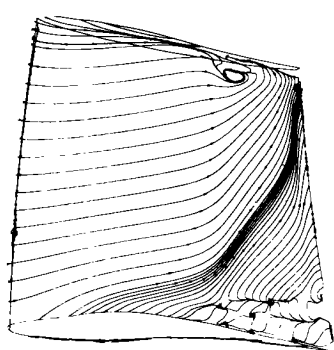


图8 高压涡轮静叶吸力面
极限流线

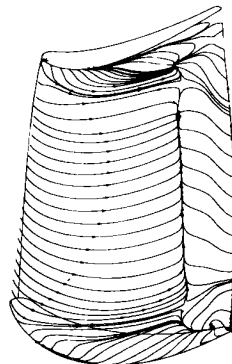


图9 高压涡轮动叶吸力面
极限流线

4 结束语

(1) 涡轮级的工作温度范围通常很大, 所以在计算中考虑变比热是十分必要的。本文的变比热计算仍然建立在完全气体的假设上, 在每一时间步每一控制单元体保持定比热, 这样的处理使得程序编制简单, 非常适合工程计算。(2) 在定常多级计算中, 动、静叶的相互干扰和数据传输的处理是一个十分重要的问题。本文使用“混合平面”法进行叶列间处理, 在叶片间的混合平面上采用简化的无反射外推边界条件完成计算域上下游信息的传递; 且外推周向不均匀性进出口边界条件。这些措施可以使程序更好地描述气体在交接面上的流动状态。(3) 涡轮级计算中叶列间小间隙情况的求解会引起参数的振荡而很难使程序得到收敛解。本文采用了在交接面上构造Riemann问题解和使用松弛因子的方法大大提高了程序的稳定性。(4) 算例表明本文的涡轮级三维粘性计算方法, 不但可以清晰地给出叶栅槽道中的流动状态, 而且在叶列间小间隙情况下, 交接面附近动、静叶前后缘的流动状态也可以很好的描述。

参 考 文 献

- 1 Saxer A P, Giles M B. Prediction of Three- Dimensional Steady and Unsteady Inviscid Transonic Stator/Rotor Interaction With Inlet Radial Temperature Nonuniformity. Journal of turbomachinery, Transactions of the ASME, 1994, 116 (3): 347
- 2 Baldwin B, et al. Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows. AIAA paper 78, 1978
- 3 崔济亚. 变比热气动函数式及计算精确解和近似解. 工程热物理学报, 1986, 7 (3): 201- 205
- 4 Dawes W N. Toward Improved Throughflow Capability: The Use of Three dimensional Viscous Flow Solvers in a Multistage Environment. ASME J. of Turbomachinery, 1992, 114: 8- 17
- 5 Denton. The Calculation of Three- dimensional Viscous Flows through Multistage Turbomachines. ASME J of Turbomachinery, 1992, 114: 18- 26
- 6 姚征, Ch Hirsch. 多级叶轮机三维流动计算的级间混合方法分析. 工程热物理学报, 1996, 17, (4): 417- 422
- 7 Giles M B. Nonreflecting Boundary Condition for Euler Equation Calculation. AIAA Journal, 1990, 28 (12): 2050- 2058

(责任编辑 杨再荣)

PIV STUDY OF AIRFOIL NEAR WAKE FLOW PART II —DYNAMIC MECHANISM

Liu Baojie, Wang Guanghua, Gao Ge

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

ABSTRACT Detailed 2D velocity and vorticity fields were measured to investigate the turbulent near wake flow of NACA 0012 airfoil with on-line cross-correlation PIV in a low-speed wind tunnel. The Reynolds number on chord length is 2.39×10^5 , the angles of attack are 0° and 4° . The experimental results indicate that ordered vortex streets exist in the turbulent near wake, the vortex streets created at the airfoil trailing edge region will fully develop and finally break up downstream, and the ordered wake flow turns into disordered. The near wake flow is dominated by the kinetic characteristics and dynamic mechanism of vortices. As the second part of the investigation, the paper mainly discusses the formation of the vortex streets, the dynamic mechanism of the near wake, and the fluid instability of the vortex and vortex streets.

Key Words Airfoils Wakes Turbulence measurements Fluid Instability

FEM CALCULATION AND EXPERIMENTAL STUDY OF THE 3-D FLOW IN VANE DIFFUSER

Liu Xiaomin, Xi Guang, Wang Shangjin

(School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an 710049)

ABSTRACT The viscous incompressible turbulent flow in a vane diffuser was analyzed numerically by using penalty finite element method, based on Galerkin variational finite-element method. During discretizing the controlled equations in arbitrary curvilinear coordinates, the consistent formulation of penalty pressure matrix was chosen, and connected with the streamline upwind Petrov-Galerkin weighed residual formulation to modify the weight function. In reference to the experience of the finite difference method, the streamline upwind term only along the flow direction was controlled by the upwinding parameters. This approach ensured the stability the solution of the pressure. Numerical results of the turbulent flow in the vane diffuser were satisfactorily consistent with the experimental measurements and indicated that the spurious pressure was eliminated while the false diffusion of other terms was simultaneously minimized.

Key Words Finite element Numerical analysis Turbulence Van diffuser

A NUMERICAL SIMULATION OF 3D VISCOUS STEADY FLOW IN A TURBINE STAGE

Yuan Ning, Wang Songtao, Zhang Zhenjia, Wang Zhongqi, Feng Guotai

(Power Engineering Dept Harbin Institute of Technology, Harbin 150001)

ABSTRACT A 3D numerical methodology is presented for simulating blade rows in a turbine stage by solving 3D time-dependent compressible Navier-Stokes equations. This numerical method is based on Godunov's finite volume scheme with third-order-accuracy and TVD property. Baldwin-Lomax algebraic turbulence model is employed to simulate largely separated flow. The principal assumptions are made as follows: the flow is

steady between stator and rotor and these rows can communicate via inter-row mixing plane. These reformative "mixing planes" introduce circumferential averaging of flow properties but preserve general radial variations. The circumferential disequilibrium in the same row is assured by using non-reflecting extrapolated boundary condition. In view of wide working temperature range in turbine stage, varying specific heat is applied in this method. The calculation instability resulted from inter-row mixing treatment is improved remarkably by using Riemann solver and lax-godunov in the case of small inter-row gap. The application of this method is illustrated for an axial high-pressure turbine stage in an aeroengine. The results show that this method not only is available but also reveals flow phenomenon in rows very well.

Key Words Turbines Three dimensional flow Numerical simulation
Computational methods

COUPLING OPTIMIZATION CALCULATION FOR AEROENGINE PERFORMANCE

Wang Xun, Zhang Shizheng, Cai Ruixian

(Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing 100080)

Du Heling, Guo Xin (Chinese Gas Turbine Establishment, Jiangyou 610500)

ABSTRACT Coupling-Optimization Algorithm is comprised of coupling methods used in SPEY aeroengines and optimization method. This new method can be used to simulate the engine performance accurately on the basis of very limited engine test results even if its component characteristics are unknown. In addition, the component characteristics can also be inferred in the same time. This method has been used to simulate the performance of a turbofan engine and to infer its unknown component characteristics. The comparison of the results with the engine test data confirms the feasibility of the proposed method.

Key Words Coupling Optimization Aeroengine Performance

AN EXTENDABLE OBJECT MODEL FOR GAS TURBINE SIMULATION

Xie Zhiwu

(3rd Dept. Naval Academy of Engineering, Wuhan 430033)

Su Ming and Weng Shilie

(School of Power and Energy, Shanghai Jiao Tong University, Shanghai)

ABSTRACT A generalized gas turbine simulation software must be not only capable of extending to new engine components and configuration, but also able to shift among different simulation algorithms. For this purpose the object-oriented analysis and design method is applied and the Unified Modeling Language (UML) is used. Different computational tasks of simulation are categorized and summarized into a 3-layer generalization: component calculation, gas path calculation, and system state calculation. Based on this generalization, a class framework is proposed and a Node-Connector-Component model is presented and analyzed. New components, gas path algorithms, and numerical integration methods can be extended from the appropriate base classes in the framework. New engine configura-