

计算压气机特性的基元叶片法

西北工业大学

唐狄毅**

四川发动机研究所

王永明

【摘要】 本文介绍一种轴流压气机非设计性能的预测方法。该法特点是,在流线曲率法解径向平衡方程基础上,引入环壁附面层损失的计算,考虑叶型存在对气流熵增和总焓变化的影响,并计入二次流/径向间隙流引起的损失,激波、尾迹损失等。结果与试验数据对比符合工程要求。

关键词: 压气机 非设计 性能 环壁附面层

1 前 言

常用的计算压气机特性的方法有 3 种:统计法、级叠加法和基元叶片法,本法属于第 3 类。由于内部流动复杂,各种损失源多,且相互干扰,因而很难建立一个完整的数学模型。本文试图通过环形流动,叶排流动和环壁附面层流动的迭代求解,把各种损失通过下列公式引入流动迭代的链环中

$$\text{熵增 } \Delta S = K_1 \frac{C_D}{2} \frac{c}{s} \frac{(\bar{V}_m^2 + \bar{W}_u^2)^{3/2}}{\bar{V}_m h} + \Delta \tilde{S} \quad (1)$$

$$\text{总焓变化 } \Delta H = U \cdot \Delta V_u + \Delta \tilde{H} \quad (2)$$

$$\text{转速 } \Delta V_u = K_2 (-C_L/2)(c/s) [\bar{V}_m^2 + \bar{W}_u^2]^{1/2} \quad (3)$$

式中, K_1 、 K_2 为反映级与叶排差别以及多级间干扰影响的系数,其值随来流相对马赫数而变(图 1); c/s 为稠度; V_m 为子午速度; W_u 为流体相对切向速度; U 为圆周速度; h 为静焓。上标“-”表示叶排平均值;上标“~”表示损失项。 C_L 为升力系数; C_D 为阻力系数。

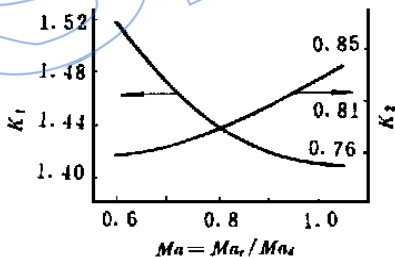


图 1 系数 K_1 、 K_2 随相对马赫数变化

求出各条流线的熵增和总焓变化后,根据下式就可求出出口总温 T_{t2} 和压比 π_k^*

$$\Delta H = C_P [\dot{T}_{t2} - T_{t1}] \quad , \quad \Delta S = -C_P \ln \pi_k^{*(k-1)/k} \cdot (T_{t1}/T_{t2}) \quad (4)$$

然后用质量平均求出出口截面上的 T_{t2} 和 π_k^* ,并由 T_{t2} 和 π_k^* 求得效率 η

2 环形流动与叶排流动

环形流动不计叶排和附面层的影响,并假设无粘、轴对称以及定常绝热。其基本方程为

$$\dot{m} = 2\pi \int_h \rho V_a r dr \quad , \quad (dV_a^2/dr) + FV_a^2 + G = 0 \quad (5)$$

$$\text{式中 } F = \frac{2}{V_a} \frac{\partial V_r}{\partial r} - \frac{1}{C_P} \frac{1}{\cos^2 \sigma} \frac{\partial}{\partial r} \quad , \quad G = 2 \frac{V_u}{r} \frac{\partial (r V_u)}{\partial r} - 2 \frac{\partial H}{\partial r} + \frac{1}{C_P} (2H - V_u^2) \frac{\partial}{\partial r} - \frac{2}{V_a} \frac{\partial V_r}{\partial r}$$

V_a 、 V_u 、 V_r 分别为轴向、切向和径向速度, 有关几何尺寸如图 2 所示, C_p 为等压比热。由公式 (5) 解得轴向速度 V_a , 而子午面速度 $V_m = V_a / \cos \sigma$ 。

式(1)、(3) 中的 C_L 和 C_D 是通过叶排流动求得的。计算思路是已知流动速度分布后, 根据叶片几何形状、安装角、稠度、攻角和雷诺数, 通过位流理论计算理论升力系数 C_{LPT} , 然后计入气流分离和失速的影响, 根据试验数据进行修正^[1]。求得的修正升、阻力系数为:

$$C_D^* = 0.144 R_N^{-0.2} + R_6 \cdot SLF^2 + R_7 \cdot |cb| \quad (6)$$

$$C_D = C_D^* + 0.01\{sb \hat{R}_8(ILF - I^+) + sb \hat{R}_8(I^- - ILF)\} \\ C_L/C_{LPT} = 1 - R_9\{sb \hat{R}_{10}(ILF - I^+) + sb \hat{R}_{10}(I^- - ILF)\} \quad (7)$$

式中, R_N 为雷诺数; SLF 为光滑负荷因子, ILF 为冲量负荷因子, 对于一个孤立翼型来说, SLF 和 ILF 分别代表由于弯度和攻角引起的升力系数分量; I^+ 为正失速处(攻角过大)对应的 ILF 值, I^- 为负失速处对应的 ILF 值; cb 为孤立翼型在设计攻角下的升力系数; sb 为失速位置函数, 规定它具有 $\exp(x)$ 的变化形式; $R_6 \cdots R_{10}$ 为定值, 不同系列翼型其值是不同的, 对于 65 系列其值可取为 0.004、0.002、0.7、0.1 和 2.0, 对其它系列可参考文献[1]。

3 环壁附面层流动

根据文献[2], 可压缩流体的环壁附面层动量积分方程为:

$$\frac{1}{\rho r} \frac{d}{dm} (r V_m^2 \theta_m) + H \theta_m V_m \frac{dV_m}{dm} = \frac{\bar{f}_m}{\rho} + \frac{\tau_m}{\rho}, \quad \frac{1}{\rho r} \frac{d}{dm} (r V_m W_u \theta_u) + H \theta_u V_m \frac{dW_u}{dm} = \frac{\bar{f}_u}{\rho} + \frac{\tau_u}{\rho} \quad (8)$$

式中, m 为流线方向, u 为切向, θ 为绝对动量厚度, $\bar{f}$ 为叶片亏损力, V 为绝对速度, W 为相对速度, τ 为壁面剪应力, H 为环壁附面层形状因子, ρ 为密度, 上标 “-” 为叶排平均值。

描述叶片亏损力的方向有方程

$$V_m F_m + (1 - \epsilon) W_u F_u = 0 \quad (9)$$

式中, $(W^2/2) F_m = \int (\bar{f}_m / \rho) dm$ 和 $(W^2/2) F_u = \int (\bar{f}_u / \rho) dm$, $W^2 = V_m^2 + W_u^2$ 为气体相对速度。 ϵ 为一不确定性参数, 由试验数据分析确定, 计算中取 0.5。积分限 s 为叶排的子午向间距。

另外, 根据叶片出口处附面层流动方向与主流方向平行的假定, 并考虑二次流可导出

$$V_{u2}(\theta_{u2} - \theta_{u1}) = K \delta_c / \cos \lambda \left| (s/c) \nabla (W_{u2} - W_{u1}) \right|^{1/2} \cdot \operatorname{sgn}(W_{u2} - W_{u1}) \quad (10)$$

式中 λ 是安装角, δ_c 为叶尖间隙, K 为间隙漏系数, ∇ 为叶片通道平均速度, 下标 1 和 2 指叶排的进、出口, 符号 sgn 表示取 $(\cdots)$ 的正负号。

表面摩擦力的定向方程为

$$\bar{\tau}_m / \bar{p} = (1/2) \bar{V}_m^2 + (W_u - U)^2)^{1/2} \cdot \nabla_m \cdot C_f \\ \bar{\tau}_u / \bar{p} = (1/2) \bar{V}_m^2 + (W_u - U)^2)^{1/2} (W_u - U) C_f \quad (11)$$

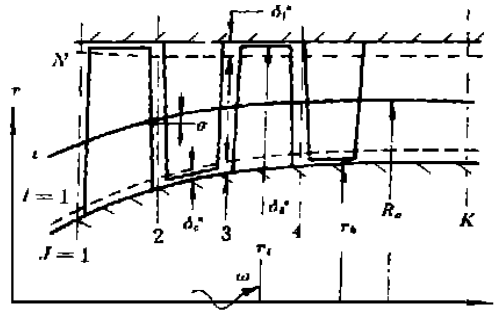


图 2 计算流线与计算站

式中, C_f 为壁面摩擦系数。

由 (8) ~ (11) 可解出 θ_{n2} 、 θ_{u2} 、 F_{m2} 和 F_{u2} 四个未知数, 再按下式确定附面层中的能量损失

$$\begin{aligned} \Delta \bar{H} &= W_{(1)} + W_{(2)} \\ W_{(1)} &= \frac{1}{m} \sum_{H,T} r \bar{p} U \left[\bar{V}_m \Delta V_u \delta^* - \frac{1}{2} (\bar{V}_m^2 + \bar{W}_u^2) F_u / \cos \sigma \right] \\ W_{(2)} &= - (1/2m) \{ r \bar{p} U \bar{V}_m^2 + (\bar{V}_u - U)^2 \}^{1/2} (\bar{V}_u - U) C_f \Delta \bar{m} \}_H \end{aligned} \quad (12)$$

式中, $W_{(1)}$ 的含义为, 附面层内转子叶片所作的功未贡献于滞止压升, 而最后变成熵增的部分。

$W_{(2)}$ 是克服根部附面层表面摩擦力的切向分量所作的泄漏功, 它最终也转变成熵增。

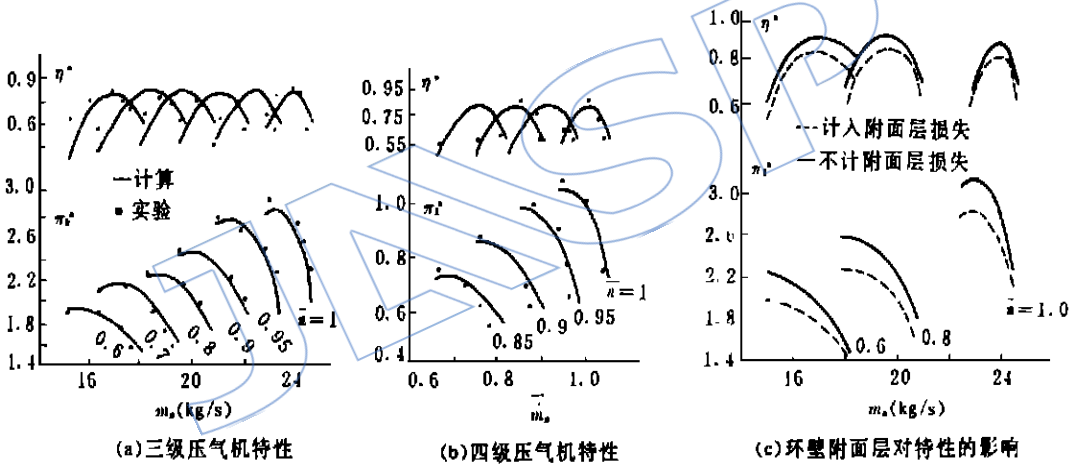
引起熵增的还有附面层内的动量损失, 故熵增的最终表达式为

$$\Delta \bar{S} = \left[\Delta \bar{H} + \sum_{H,T} r \Delta (\rho V_m k) / m \right] / \bar{h} \quad (13)$$

式中, $\kappa = V_m^2 \theta_m + V_u^2 \theta_u$, $\bar{h}$ 为静焓的通道平均值。

4 计算步骤、算例

(1) 计算站取叶排进出口, 如图 2 中的 $J = 1 \cdots K$ 各站所示。径向按流线分格, 如 $I = 1 \cdots N$



等。流线与轴向计算站组成网格。(2) 在给定进气角、总焓、熵、速度分布条件下, 通过解径向平衡方程求得新的 V_a 分布。(3) 从叶排流动计算中求得气流流过叶排后熵增以及切向速度的变化, 见 (1) 和 (3) 式。(4) 从环壁附面层流动的计算中求得由于附

面层存在产生的能量损失 $\Delta \bar{H}$ 和熵增 $\Delta \bar{S}$ 。(5) 计算激波、尾迹及二次流/间隙漏等项损失引起的熵增 $\Delta \bar{S}$ 。(6) 将环壁附面层以及二次流/间隙漏衍生的能量损失和熵增分配到各条流线上, 返回步骤 (2) 计算新的轴向速度分布。(7) 新流线的确定: 根据第 1 个计算站上各条流线位

图 3 算例示图

置不变的前提, 计算下游各站的流线位置, 以各流管间流量相等作为准则, 求解新的流线位置及其对应的半径 r , 并比较前后两次迭代所得的流线位置的半径差 Δr , 直至小于允许误差为止。(8) 考虑附面层堵塞影响:

$$\dot{m} = 2\pi \int_h^{\circ} r \rho V_a dr - \sum_{H,T} \rho r V_a \delta^* \quad (14)$$

式中, $\delta^* = \theta_m / H$ 为位移厚度, H 为形状因子。

计算了 3 台多级轴流压气机的特性。图 3(a) 为一、三级高压压气机的特性。图 3(b) 为以相对值给出的一台四级压气机特性。图 3(c) 示出环壁附面层对压气机特性的影响。图 3(d)、(e) 示出附面层动量厚度沿轴向的发展以及堵塞系数的变化。该图表明, 堵塞影响沿轴向并非单调下降, 而是在转子出口处减小, 静子出口处增大, 其原因在于叶片亏损力的影响。

参 考 文 献

- 1 Meller G L 等. A combined Theoretical and Empirical method of Axial Compressor Cascade Prediction. ASME Paper, 72-WA/GT-5 1972
- 2 Meller G L, Wood G M. An Axial Compressor End-wall Boundary Layer Theory. T. of ASME, J. of Basic Engineering, June, 1971

(责任编辑 魏星禄)

欢迎订阅中国科协报

《中国科协报》是中国科学技术协会主管和主办的报纸, 江泽民总书记题写了报名。《中国科协报》是中国科协指导工作、交流经验、研讨政策、传播信息、反映科技人员的意愿的新闻媒介, 其宗旨是: 发挥全国各级科技群众团体的优势, 依靠广大会员和科技工作者, 宣传“科学技术是第一生产力”的重要思想, 宣传尊重知识、尊重人才的社会风尚, 宣传科技社会团体的功能和作用, 促进科学技术人才的成长和提高, 为经济振兴、科技进步和社会发展作出贡献。

《中国科协报》的读者对象是: 各级科协和学会(协会、研究会)干部和从事科普工作的人员, 各级各类学会会员, 各个领域从事科研、生产的科技人员和管理人员, 高等和中等学校师生, 一切关心科技事业发展和科协工作的党政领导干部、行政管理干部和社会各界人士。

《中国科协报》对开四版, 采用计算机激光照排和胶版印刷, 每周四出版。全国公开发行, 国内邮发代号 1—179, 读者可及时到当地邮局订阅。如在当地邮局订阅有困难, 请直接汇款至本社经理部。本报月价 1.08 元, 季价 3.24 元, 全年 12.96 元。另请加订报款的 15% 为邮费。

本社地址: 北京海淀区学院南路 86 号

联系人: 冷艳玲

邮 编: 100081

电 话: 8318877—496

FULL- LENGTH PAPERS

AN EXPERIMENTAL STUDY ON BLADE NEGATIVE CURVING IN A TURBINE CASCADE WITH A LARGE TURNING ANGLE

Wang Zhongqi, Tan Chunqing, Han Wanjin, Zhou Mochun
(*Harbin Institute of Technology*)

ABSTRACT

Turbine rectangular cascades with conventional straight blades and negative curved ones at obtuse angles between their pressure surfaces and both endwalls were investigated in a low speed plane cascade wind-tunnel. The measuring planes with five-hole micro-spherical probes were arranged as 3 before, 6 in and 1 after the cascades. The experimental results show that the origination and development of horseshoe vortices and passage vortices as well as the meeting of the passage vortices in the midspan region almost dominate the whole flow field of conventional straight cascade. In comparison with those of conventional straight cascade, blade negative curving improves the flows around blade leading edges near both the endwalls, and reduces the intensities and viscous dissipation losses of leading edge vortices and passage vortices. As a result, the flux capability of the cascade increases by 3.2%, the mass flux averaged overall loss coefficient is cut down by 19.1%, at the same time the good inlet condition is provided for the rotor cascade.

A COMPUTER PROGRAM FOR MERIDIONAL FLOWS IN MULTISTAGE AXIAL FLOW COMPRESSORS WITH TURBULENCE AND MULTI- EFFECTS OF 3- D FLOWS

Li Shiming (*Tsinghua University*)

Chen Maozhang (*Beijing University of Aeronautics and Astronautics*)

ABSTRACT

It has been shown early by the authors that the spanwise mixing is caused by turbulent diffusion and different kinds of three-dimensional flow effects. This paper introduces a computer program based on this theory. A new kind of rotary enthalpy has been defined and its related equation has been derived to simplify the present problem. The details for modeling all the unknown correlation mixing terms involved in the computations and the numerical scheme have been also presented. The results obtained agree well with the experiments for five compressors. The results also show some improvement over the previous theories.

A BLADE ELEMENT METHOD FOR PREDICTING THE OFF- DESIGN PERFORMANCE OF COMPRESSORS

Tang Diyi (*Northwestern Polytechnical University*)

Wang Yongming (*Sichuan Aeroengine Research Institute*)

ABSTRACT

A method for predicting the off-design performance of an axial flow compressor is presented. The fundamental of the calculation is incorporation of three flow phenomena: the annulus flow, the cascade flow and the flow of annulus wall boundary layer. The calculation of the annulus flow is based on the radial equilibrium of mainstream

and neglects the existence of cascades and the effects of annulus wall boundary layers, so it aims at calculating the mainstream axial velocity distribution. The calculation of the cascade flow is intended to obtain lift and drag coefficients of cascade airfoil and to calculate the incremental of total enthalpy and entropy across cascades. The calculation of the annulus wall boundary layer provides the blockage effect and the losses caused by annulus wall boundary layer. For obtaining the whole losses of the compressors, the loss models of shoke wave, blade wake and secondary flow/clearance leakage should be coupled with the above mentioned calculations. As examples, the off-design performance curves of compressors have been calculated, one of them gives the effect of the annulus wall boundary layer on the compressor characteristics. Additionally, the axial development of the boundary layer momentum thickness and blockage coefficient are also given. The comparison of calculated curves with the rig test results shows that the errors will be allowable in engineering.

INVESTIGATION OF FLOWS IN A CONTROLLED DIFFUSION AIRFOIL CASCADE PASSAGE

Cheng Ronghui and Zhong Yongxing (*Gas Turbine Establishment*)

Yun Ruqun (*Northwestern Polytechnical University*)

ABSTRACT

This paper presents the flow measurement in the blade passage and downstream of a high turning controlled diffusion airfoil (CDA) cascade in details and the investigation of flow visualization on the endwall and blade surface. As a result, a thorough comprehension is obtained in respect of the flow in the blade passage especially in the endwall area, the vortex structure and of the effect of endwall boundary layer and secondary flow on the performance of the cascade. It is of great practical value in improving the flow conditions in the endwall area of a compressor and developing the second generation of the controlled diffusion airfoil.

VIBRATION CHARACTERISTICS OF ROTOR SYSTEMS WITH POROUS SQUEEZE FILM DAMPER (PSFD) SUPPORTS

Yan Litang, Zhang Shiping, Li Qihan

(*Beijing University of Aeronautics and Astronautics*)

ABSTRACT

A Porous Squeeze Film Damper (PSFD) had been developed by the authors based on the normal Squeeze Film Damper (SFD). The outer race of the PSFD is made of permeable metal material. Its permeability allows some of the oil in the clearance of the damper to seep in or out through the outer porous race for remarkable improvement of the squeeze film damping properties. In this paper, the steady state response and the transient response resulted from sudden unbalance in the case of blade loss are investigated for simple rigid rotor and a flexible Jeffcott rotor supported on concentric PSFD or SFD. The nonsynchronous response of the simple rigid rotor is analyzed. The investigations show that the nonlinear vibration characteristics with the cavitated SFD systems such as bistable jump phenomenon, nonsynchronous response and "lock up" at rotor pin-pin critical speeds under much high unbalance level are greatly improved or avoided with the PSFD systems. PSFD systems could be operated efficiently at higher shaft speeds and under greater unbalance loads especially under greater sudden loading of blade loss. As a new damper based on SFD, PSFD retains all the advantages of SFD and provides more effective vibration control for the rotor systems.