

稠度对弯叶片压气机叶栅特性的影响^{*}

哈尔滨工业大学 钟兢军^{**} 王 苇 苏杰先 王仲奇

【摘要】 对压气机叶栅中采用弯曲叶片后,叶栅气动特性与叶栅稠度的关系进行了实验研究。结果表明,随叶栅稠度的增加,叶栅损失在大部分冲角下呈增高的趋势。在正弯曲叶栅中,对应最小损失的冲角出现在正冲角下;在反弯曲叶栅中,对应最小损失的冲角为负冲角。为了降低损失,正弯曲叶栅的稠度应低于直叶栅,而反弯曲叶栅在与直叶栅相当的稠度下,具有良好的冲角特性且叶栅损失降低。

主题词: 压气机 叶栅流动 稠度

分类号: V 231.3

1 前言

在影响叶栅二次流的因素中,长度项是一个重要的方面,即展弦比和叶栅稠度^[1]。展弦比不仅影响压气机的几何尺寸,而且与压气机的喘振裕度密切相关,其值的大小与叶栅二次流损失有着直接的关系,现代航空发动机发展趋势之一就是采用低的展弦比^[2]。叶栅稠度不仅影响叶片的气动负荷,而且还与强度、结构等因素和指标有关。而弯曲叶片作为降低压气机叶栅二次流损失、改善叶栅气动性能的一种方法和措施,已得到许多实验的证实^[3-9]。在本文中,针对不同的弯曲叶片型式,即常规直叶栅、叶片压力面两端皆与端壁成25°角的正弯曲叶栅和叶片吸力面两端皆与端壁成25°角的反弯曲叶栅,在4种叶栅稠度下进行了叶栅出口流场的研究,讨论了稠度对叶栅损失和二次流等叶栅特性的影响。这3种叶栅均是由7片NACA 65-24A 10-10系列叶型的叶片组成,其几何参数和进口条件为:叶片弦长160 mm,叶片弯曲角59.5°,安装角20°,弦长 b /节距 t 为2,展弦比 h/b 为1,叶片几何进气角49.75°,叶片几何出气角-9.75°,进口弦长雷诺数 6.5×10^5 ,进口马赫数0.2,进口附面层厚度20 mm,实验冲角 i 为0°,±5°,+10°,叶栅稠度 $\sigma = b/t$ 为1.33, 2.00, 2.67, 3.33。

实验是在哈工大低速风洞自动测试系统上完成的,利用五孔束状探针测量了3种叶栅出口气流的总压、静压以及出口气流角沿节距和叶高的变化,测点布置等与文献[3, 4]一致。

2 实验结果分析

当叶栅的几何进气角和几何出气角确定后,为保证压气机叶栅有效地工作(叶栅的扩压因子 $D \leq 0.6$),叶栅的稠度不能随意地选取。对我们采用的直叶栅几何参数,在叶栅中不可压气流流动情况下(本论文实验可以认为是不可压的),可以利用下式^[10]来计算可用的叶栅稠度范围,以便于满足 $D \leq 0.6$ 的要求:

1996年4月收稿,1996年5月收到修改稿。 * 本文系国家自然科学基金资助项目,编号:59376271

^{**} 哈尔滨工业大学二系241教研室 150001

$$D = \left[1 - \frac{\cos \alpha_{lp}}{\cos \alpha_{lp}} \right] + \frac{\cos \alpha_{lp}}{2(b/t)} (\tan \alpha_{lp} - \tan \alpha_{lp}) \quad (1)$$

其叶栅稠度 $\sigma = b/t$ 应该大于 1.71。文献[1]曾指出,稠度的一般范围在 0.5-2.5 之间,对弯曲角较大的叶栅应取较大的稠度值。以直叶栅的实验结果看(见图 1),叶栅稠度应取在 2.0-2.5 的数值范围内,这样叶栅的低损失区域增宽,损失系数较小,具有良好的冲角特性。为了分析 3 种叶栅之间的损失值变化情况,图 1 给出了不同叶栅在不同冲角下叶栅出口按质量平均的总损失系数 $\bar{\xi}_m$ 随 σ 的变化关系。在零冲角和负冲角下[图 1(a)和(b)],正弯曲叶栅在所有稠度下的叶栅损失值均高于直叶栅,表明在这样的冲角范围内,不宜采用正弯曲叶片来改善叶栅

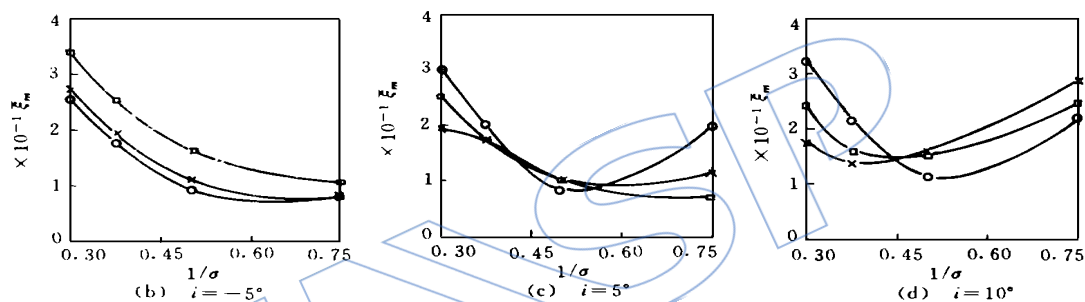
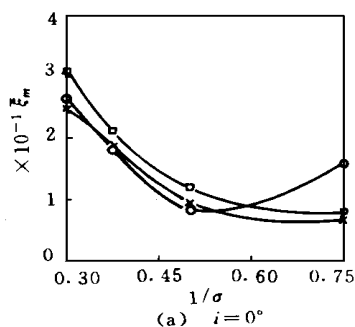


图 1 不同叶栅在不同冲角下损失与稠度的关系(* 直叶栅, O 正弯曲叶栅, □ 反弯曲叶栅)

的气动特性。但在正冲角及较低稠度下($1.33 < \sigma < 2.00$) [见图 1(c)和(d)],采用正弯曲叶片可以降低叶栅的损失,改善叶栅的气动性能,这说明对一定的具体情况,正弯曲叶片仍不失为降低叶栅二次流损失,改善叶栅气动特性的一种手段。反弯曲叶栅在 4 个冲角下均有损失系数值低于直叶栅的情况,表明只要几何参数选择得合理,反弯曲叶片可以降低叶栅的总损失。在 -5° 冲角时除 $\sigma = 1.33$ 时损失值与直叶栅相近外,其余 σ 下的损失均有所降低;零冲角时和 $+5^\circ$ 冲角时总损失比直叶栅有所降低的稠度范围有所减小;而在较大正冲角和合适的 σ 时,反弯曲叶栅的损失值降低的相当大,最大可达 29.03%^[4]。从图 1 的 4 个图中可见最明显的是反弯曲叶栅在 $\sigma = 2.00$ 时,所有冲角下的损失值对比直叶栅来讲均得到了降低,这表明在中等稠度下采用反弯曲叶栅是降低叶栅损失的有力手段。由图 1 可知,在本实验叶栅参数下,反弯曲叶栅合适的稠度范围大致为 $1.8 < \sigma < 2.7$ 。

稠度改变时,叶栅出口二次流速度矢量也发生了变化。对直叶栅(图 2 中 PS—压力面, SS—吸力面),稠度增加时,由于叶栅流道当量扩张角的减小,通道涡没有充分的发展,气流分离现象减弱,因此由于气流分离所造成的损失较小,但由于稠度的提高,气流的摩擦损失所占的比例大大提高,因此总损失提高。叶片正弯曲后,叶栅中的三维流动状况在叶栅出口处可见有所加强。在同一稠度下,与直叶栅相对比,正弯曲叶栅的通道涡更强(见图 3)。通道涡的卷吸和由两端指向叶栅中部负压梯度的共同作用,使两端区低能流体向叶栅中部迁移,两端区高损失核心更加靠近叶栅中部。低能流体迁移的结果使正弯曲叶栅中部的损失增加,这与文献[3,

6]的分析是一致的。低稠度时,正弯曲叶栅出口二维流动区域扩大(对应叶栅出口的低损失区),该区域的扩大使叶栅的总损失降低。在较大冲角下($i \geq +5^\circ$),虽然叶片吸力面附面层增厚和低能流体向叶栅中部的集聚导致了叶栅中部损失的增加,但由于正弯曲叶栅内三维压力分布的改善^[7,8],正弯曲叶栅的总损失系数相比于直叶栅得到降低。

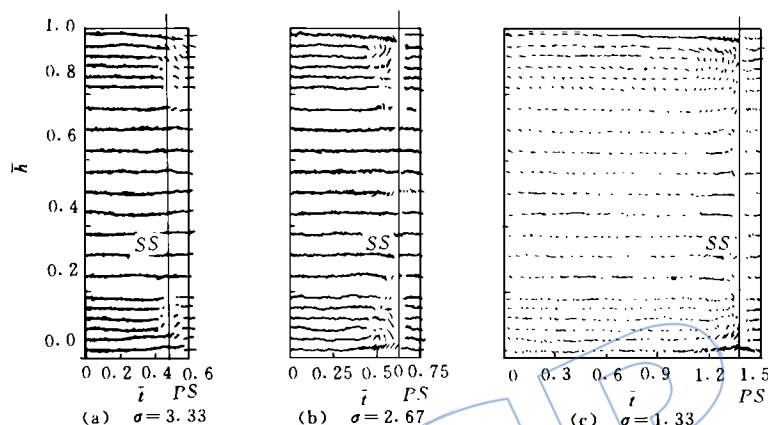


图 2 不同稠度时直叶栅出口二次流速度矢量图

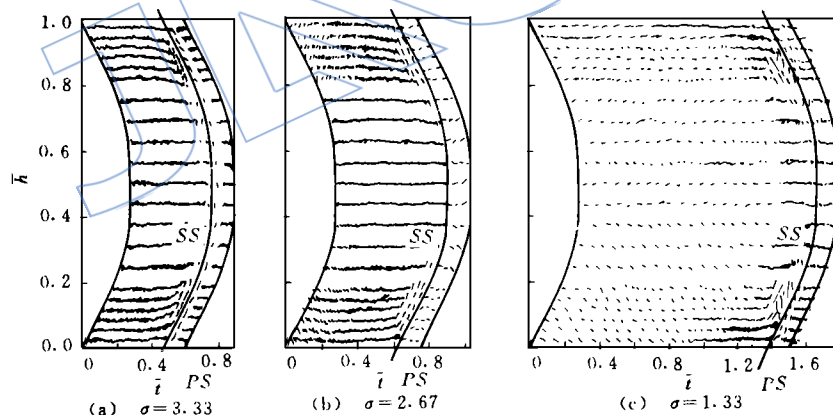


图 3 不同稠度时正弯曲叶栅出口二次流速度矢量图

反弯曲叶栅在高稠度时($\sigma = 3.33$ 和 $\sigma = 2.67$) [图 4(a) 和 (b)], 叶栅中部的势流区与直叶栅相比变化不大, 而两端角区流动状况更为复杂, 这两个稠度下的流场未见明显的改善。在稠度降低时, 根区损失有所降低, 角区三维流动状况比 $\sigma = 3.33$ 时减弱, 但叶栅中部有所加强, $\sigma = 2.00$ 时的叶栅损失在所有冲角下均得到降低(图 1), 吸力面上气流分离程度在 σ 再降低时又有所加强[图 4(c)], 因此叶栅总损失提高, 叶栅出口二次流速度矢量也更为复杂。这是由于在小稠度下($\sigma = 1.33$), 通道涡向吸力面/端壁角区输送低能流体的能力减弱, 所以增厚的端壁附面层沿节距方向扩展, 高损失区增宽, 角区分离有所加剧, 低能流体的大量积聚, 导致了损失

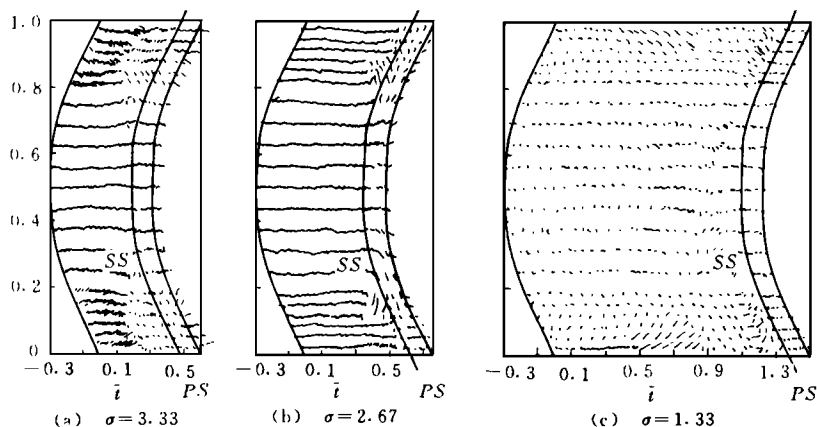


图 4 不同稠度时反弯曲叶栅出口二次流速度矢量图

的急剧升高。

3 结 论

(1) 随叶栅稠度的提高, 叶栅损失在大部分冲角下表现为增高的趋势。直叶栅和正弯曲叶栅随稠度的增加, 最小损失冲角向正冲角变化。反弯曲叶栅随稠度的增加, 最小损失冲角出现在负冲角下, 较小的稠度对反弯曲叶栅不适宜。(2) 合理的稠度选择, 可以改善叶栅的流场结构, 降低叶栅二次流损失。在本实验条件和叶栅参数下, 直叶栅的稠度应取在 2.0~2.5 范围内, 反弯曲叶栅在稠度为 1.8~2.5 时具有良好的冲角特性且叶栅损失比直叶栅降低; 正弯曲叶栅为降低损失, 较合理的稠度应在 1.33~2.0 之间。(3) 存在叶片弯曲型式、稠度的选择等叶栅几何参数的合理匹配问题。匹配合理, 采用弯曲叶片可有效控制叶栅内的二次流发展, 降低叶栅的二次流损失。

参 考 文 献

- 1 李根深, 陈乃兴, 强国芳. 船用燃气轮机轴流式叶轮机械气动热力学(原理、设计与实验研究)(下册). 北京: 国防工业出版社, 1985
- 2 陈矛章. 航空叶轮机的气动热力学. 气动热力学发展战略研讨会专题报告汇编, 北京: 1989
- 3 钟兢军, 苏杰先, 王仲奇, 陈莹. 叶片倾斜和弯曲对扩压叶栅出口流场的影响. 工程热物理学报, 1995, 16(1)
- 4 钟兢军, 王仲奇, 苏杰先, 王萼. 反弯曲扩压叶栅降低二次流损失实验研究. 中国工程热物理学会热机气动热力学学术会议: 宜昌, 1995
- 5 Breugelmans, F A E. Influence of Incidence Angle on the Secondary Flow in Compressor Cascade with Different Dihedral Distribution. ISABE paper 85-7078, Beijing, China, 1985
- 6 钟兢军, 苏杰先, 王仲奇, 吴继权. 不同冲角下弯曲扩压叶栅出口流场的实验研究. 工程热物理学报, 1995, 16(3)
- 7 钟兢军, 苏杰先, 王仲奇, 陈莹. 弯叶片扩压叶栅壁面静压分布的实验研究. 航空动力学报, 1994, 9(3)
- 8 钟兢军, 苏杰先, 王仲奇. 不同冲角下弯叶片扩压叶栅壁面静压的实验研究. 航空动力学报, 1995, 10(4)
- 9 Bogod A B. Direct and Inverted Calculation of 2D Axisymmetric and 3D Flows in Axial Compressor Blade Rows. T S A G I, Moscow, Russia, Nov., 1992
- 10 王仲奇, 秦仁. 透平机械原理. 机械工业出版社, 1985

(责任编辑 魏星禄)

Key Words Neural networks Failure detection Modeling

M U L T I P L E - S E C T O R P A R A L L E L C O M P R E S S O R M O D E L F O R F L O W F I E L D C A L C U L A T I O N U N D E R D I S T O R T I O N

Song Wenyan, Li Ming, Fan Feida

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

ABSTRACT This paper describes in some detail the method for computing an axial-flow compressor flow field under distortion. Based on the multiple-sector parallel compressor model and stream line curvature method, the stage distributions of the parameters of distorted flow field, such as total pressure, total temperature, static pressure, static temperature, air velocity and swirl angle, are obtained by solving the radial-equilibrium equation, then the details of compressor flow field under the given inlet distortion can be reached. As an example the flow field computation for a single stage compressor has been completed under the inlet circumferential distortion. The results demonstrate the practical value of the method.

Key Words Parallel compressors Distortion factors Flow field simulation

A N E X P E R I M E N T A L O B S E R V A T I O N O N S T A L L O S C I L L A T I O N A T L O W F L O W R A T E I N A C E N T R I F U G A L I M P E L L E R

Dai Jiant Chen Jingyi

(Institute of Engineering Thermophysics, CAS, Beijing 100080)

ABSTRACT An experiment was conducted to investigate the self-excited flow oscillation in a centrifugal fan impeller. The unsteady dynamic pressures were measured around the periphery of the impeller outlet with 8 high-response transducers. The experimental observation shows the existence of a lower frequency, nonrotating oscillation in the low flow rate conditions. The measurements also indicate that the oscillation is unstable at a short duration and at a local position, and the peripheral distribution of oscillation strength is non-uniform. The analysis shows that there may be some relationship between the self-excited flow oscillation and rotating stall, especially during inception behavior of rotating stall.

Key Words Centrifugal type Impeller Dynamic tests

E F F E C T S O F S O L I D I T Y O N C H A R A C T E R I S T I C S O F C O M P R E S S O R C A S C A D E W I T H C U R V E D - B L A D E

Zhong Jingjun, Wang Wei, Su Jiexian, Wang Zhongqi

(Energy Science and Engineering College, Harbin Institute of Technology, Harbin 150001)

ABSTRACT The relationship between solidity and the characteristics of compressor cascades with curved blades is investigated. The experimental results show that with the increase of cascade solidity, the cascade loss increases at most incidences. The minimum loss incidence of the straight and positive curved blade cascade takes place at positive incidences, while the negative curved blade cascade appears at negative incidence. In order to reduce the loss, the solidity for the positive curved blade should be less than that of the straight blade. In the case of the same solidity, the secondary loss can be reduced and good incidence characteristics can be obtained with the negative curved blade, compared with the

straight blade cascades

Key Words Compressors Cascade flow Solidity

THREE- D IMENSIONAL FLOW FIELD INSIDE A COMPRESSOR ROTOR TIP REGION

Ma Hongwei and Jiang Haokang

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

ABSTRACT An experimental study of the three- dimensional flow field inside the tip region of a low- speed, large- scale axial- flow compressor rotor has been completed. With the help of a rotating four- coordinate probe traverse mechanism, the measurements were conducted with a five- hole conical probe in both design condition and near stall condition. The results indicate that the tip leakage vortex plays a decisive role in the tip region flow in the design condition, leading to substantial flow loss and blockage in the passage. In near stall condition, flow separation occurs on the blade suction surface while radial migration of the boundary layer on the suction surface is strong, which makes more low- energy fluid accumulate at the corner between the casing annular wall and the suction surface. A corner vortex is induced by the tip leakage flow, and the low- energy fluid migrates to the pressure side due to the effect of the leakage vortex. Strong flow mixing among the corner vortex, the leakage vortex, the boundary layer on the casing wall and the main flow occurs at median part of the passage, which leads to substantial deficiencies of relative total pressure and relative velocity.

Key Words Compressors Rotors Three- dimensional boundary layer

SWEEP FINITE ELEMENT SOLVER FOR 3- D VISCOUS COMPRESSIBLE FLOW IN RECTANGULAR NOZZLE

Zhao Guiping and Shen Huili

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

ABSTRACT A numerical analysis method of three- dimensional compressible viscous flow has been developed for a rectangular exhaust nozzle in which the shape of the circular- to- rectangular duct changes abruptly and its flow field is very complex. Therefore, the sweep finite element method proposed by the second author in 1983 is employed since the governing equations are directly used to establish the element equation. In this paper, a hexahedral element of 27 nodes is adopted for spacial discretization, and applied to a C- D nozzle flow field and several transition duct and nozzle flow fields. The results are in good agreement with the experimental data, and indicate the effectiveness and accuracy of the method.

Key Words Finite element methods Jet nozzles Three- dimensional flows

EXPERIMENTAL STUDY ON PIN FIN ARRAY FLOW FIELD

Su Hongzhen, Liu Songling, Chen Fuqun, Xu Duchun

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

ABSTRACT An experimental study was performed to investigate the staggered pin fin array flow field in the specially constructed small wind tunnel. The experimental result gives the detailed distributions of the flow field parameters. The analysis reveals that the