文章编号: 1001-2060(2009)03-0286-05

叶片的周向前弯角度对低压轴流风扇叶顶泄漏流场的影响

李 杨[1], 卢纪富[2]
(1. 青岛科技大学 机电工程学院, 山东 青岛 266061; 2. 郑州大学 化学工程学院, 河南 郑州 450001)

摘 要: 基于 4 个具有不同周向前弯角度叶片(前弯 1.27°, 6.1°, 8.3°, 12°)的低压轴流风扇, 对叶顶泄漏流场进行了实验和数值模拟。利用雷诺平均 $N-S$ 方程组加 $S-A$ 一方程湍流模型对叶轮在稳定工况点处进行了三维粘性流场的数值计算, 分析了叶顶处叶片表面压力的轴向分布, 计算结果表明: 随着叶片周向前弯角度的增加, 叶顶泄漏涡的初始位置逐渐向叶片后缘移动。利用粒子图像测速仪(PIV)系统对叶轮的叶顶泄漏流场进行了实验测量, 清晰地展示叶顶泄漏涡的发展过程, 结果显示: 随着叶片周向前弯角度的增加, 叶顶泄漏涡的轴向位移"先减小后增大", 周向位移"先增大后减小"。

关 键 词: 轴流风扇; 周向前弯叶片; 叶顶泄漏流动; 粒子图像测速仪

中图分类号: TH432.1 文献标识码: A

引 言

近些年来, 弯掠技术在叶轮机械领域逐渐显示出良好的应用前景。经过大量的实践证明, 采用叶片弯掠技术可有效地实现减小流动损失、提高气动效率、降低气动噪声以及扩大稳定工作范围的目的。Beiler 等人对低速轴流风扇进行了研究[1], 结果表明, 前掠动叶能够提高风扇的气动和声学性能, 应用潜力巨大, 而后掠叶片会导致风扇的气动性能降低。Benaissa 等人分别对带有径向、弯以及掠转子叶片的轴流风扇进行了试验和理论研究[2], 结果显示, 弯掠叶片能够提高超过 3% 的气动效率, 同时风扇的稳定工作范围拓宽了 20%; Corsini 等人的研究显示[3], 前掠叶片能够进一步延迟亚音速轴流风机旋转失速的发生。

由于叶顶流场对轴流风扇/压气机的气动效率、气动噪声等产生很大的影响[4~6], 因此, 近年来对弯掠叶片叶顶泄漏流动的研究逐渐成为了热点。Lee 等人利用三维 $N-S$ 方程研究了强烈前掠轴流风机在设计工况下的叶顶间隙流动[7~9], 结果表明, 由于旋涡沿流动方向的速度快速减小, 泄漏涡与机匣壁面边界层和主流的相互作用导致叶顶泄漏涡在叶道内衰减得非常快, 在叶片后缘没有明显的泄漏涡出现。另外, 利用计算结果, 给出了该风机在峰值效率工况下叶顶泄漏流动结构, 泄漏流动的卷起最初发生在最大静压差附近。Decker 和 Mcnulty 等人对叶顶部带有前掠转子的多级轴流压气机在两种不同的配置下(一个是带有强烈的叶顶间隙流; 另一个是适中的叶顶间隙流)进行了研究[10~11], 与传统的直叶片相比, 前掠叶片展示了在失速裕度、效率和间隙敏感度上的改善。Yoon 等人利用 PIV 系统测量了一带有 5 个前掠叶片的轴流风扇的出口流场[12], 研究发现, 在叶顶涡附近区域出口位置的周向平均速度和湍流度明显增大。

本文对周向弯曲叶片的叶顶流场开展了理论和实验研究, 揭示了在低压轴流风扇中不同周向弯曲方向的叶片对叶顶泄漏流场的影响。利用数值方法对叶片表面的压力分布进行了分析, 以展示不同周向前弯角度叶片对叶顶泄漏流在初始位置的影响。另外, 利用 PIV 系统对不同周向前弯角度叶片的顶部间隙流场进行了实验测量, 研究了不同周向前弯角度叶片对叶顶泄漏流在发展阶段的影响。

1 研究对象

本研究对象是基于当前广泛采用的低压轴流通风机 T35 系列的改进叶轮, 其叶片重心积迭线基本是直线, 属于径向叶轮, 叶顶和机匣壁面之间的间隙高度为 1.5% 倍的叶高, 在这里称之为原始或径向叶轮。保持该叶片几何参数(叶片数、弦长和进口安装角等), 取消该叶片前弯角后将叶片沿周向弯曲分别得到其它 3 个前弯叶片—前弯 6.1° 叶轮、前弯

收稿日期: 2008-05-26; 修订日期: 2008-12-19
作者简介: 李 杨(1973-), 男, 辽宁沈阳人, 青岛科技大学副教授.

8.3°叶轮和前弯 12°叶轮[13]。叶轮的主要设计参数如表 1 所示。其它参数见文献[13]。

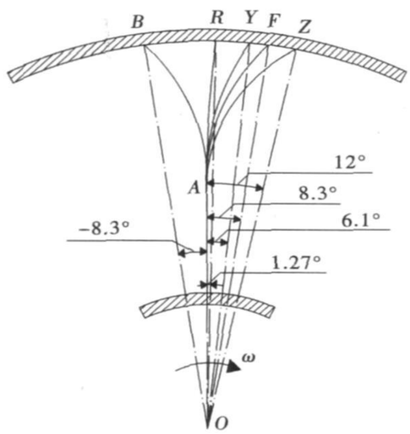


图 1　叶片重心积迭线和周向前弯角度

表 1　研究叶轮的主要设计参数

	原始叶轮	前弯 6.1°叶轮	前弯 8.3°叶轮	前弯 12°叶轮
额定转速/ r·min^{-1}	1 440			
轮毂比	0.35			
叶片数	5			
叶轮外径/ mm	500			
进口安装角/(°)	25			
重心积迭线	直线+圆弧			
叶顶间隙高度/叶片高度	1.5			
叶型种类	等厚平板圆弧叶型			
叶轮与电机连接方式	直联			
周向弯曲方向	顺叶片旋转方向			
周向弯曲角度/(°)	1.27	6.1	8.3	12

2　网格划分和计算方法

计算区域由叶片主流区和顶部间隙区两部分组成, 均采用结构化网格, 如图 2 所示。其中, 叶片通道主流区采用 H 型网格, 网格点数分布为: 主流方向×叶展方向×跨叶片方向=129×73×65, 顶部间隙区采用 H 型网格加 O 型网格, 网格点数为 65×13×13, 即在叶顶间隙高度方向和叶片厚度方向分别取 13 个点。

计算流场为三维不可压缩粘性流动, 采用时间相关法求解雷诺平均 Navier-Stokes 控制方程组加湍流方程。由于流动中热量交换很小可以忽略不计, 因此只应用连续性方程和动量方程组, 其表达式为:

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u_j)}{\partial x_j}=0 \tag{1}$$

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_j u_i)}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial\tau_{ij}}{\partial x_j} \tag{2}$$

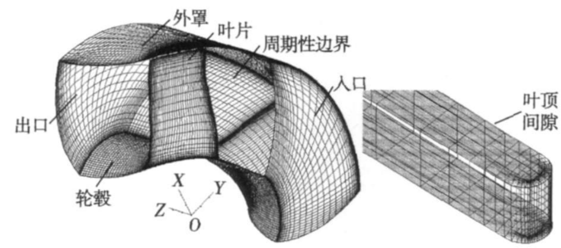


图 2　计算网格

选用 Spalart-Allmaras 一方程湍流模型来封闭上述方程组, 采用守恒形式的有限体积法, 中心差分格式进行空间离散, 时间推进采用四步 Runge-Kutta 法。为节约计算时间, 利用多重网格和隐式残差均化对流动实施加速收敛。

边界条件: 进口给定绝对总压、总温; 出口给定质量流量以及参考静压; 叶轮进出口延伸区给周期性条件, 叶片和轮毂设定为相对静止壁面, 机匣设定为绝对静止壁面。

计算结果的收敛标准为:

(1) 全局残差达到 1×10^{-5};

(2) 进、出口质量流量误差小于或等于 5×10^{-4};

(3) 计算模型进、出口全压比与试验测量结果之间的误差小于 5%;

(4) 计算模型的全压效率值与试验测量结果之间的误差小于 5%。

3　实验设计

3.1　PIV 测量系统

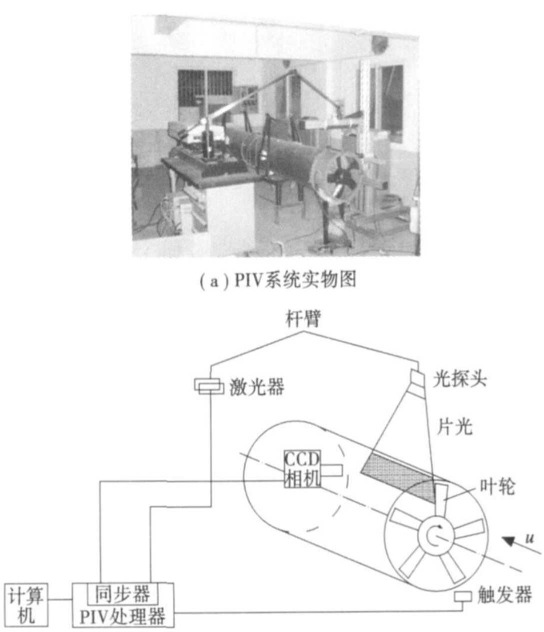
粒子图像测速技术是近些年从流场显示技术基础上发展起来的一种崭新的流速测量技术。随着计算机图像处理与光学技术等的发展, 粒子图像测速技术可在同一时刻记录下整个测量平面的相关信息, 从而可以获得流动的瞬时平面速度场。PIV 就属于这项测速技术中的一种, 本文采用 PIV 系统对轴流风扇的叶顶流场进行了测量。

PIV 测速系统的工作原理是在流场中添加适当的示踪粒子(Seeding Particles), 这些粒子随着实验流

体一起进入测试区域,此时用脉冲激光片光源照射所测流场区域,利用电荷耦合装置(Charge Coupled Device,简称CCD)相机通过成像记录系统摄取两次曝光的粒子图像,形成PIV 底片,再利用图像处理技术处理 PIV 底片,获得示踪粒子图像的平均位移,从而获得测量区域内的速度场信息。用速度的定义式可以表示为:

$$\begin{cases} u=\lim\limits_{\Delta t}\dfrac{x_2-x_1}{\Delta t} \\ v=\lim\limits_{\Delta t}\dfrac{y_2-y_1}{\Delta t} \end{cases} \tag{3}$$

式中:u,v—粒子速度在 x 方向和 y 方向分量;Δt—两次曝光的时间间隔;x_1,x_2,y_1 和 y_2—Δt 时刻前后的粒子坐标位置。从数学的角度讲,当 Δt 选择合适时,被测量粒子的位移足够小,粒子轨迹接近于直线,使得测量得到的速度极限值能够很好地逼近该处的速度。图 3 给出了所采用的 PIV 测量系统。



(a) PIV系统实物图

(b) PIV系统示意图

图 3　PIV 测量系统

3.2　实验设计

图4 为 PIV 实验装置的示意图。在 PIV 激光测量中,激光面的布置需有效照亮被测叶轮的叶顶区域,以便详细测量叶顶泄漏涡流动。实验中选用1/4圆弧的有机玻璃罩镶嵌于风筒上,与其相邻的部分为铁皮板制成的风筒,两者结合处用法兰连接。转子叶轮直联电机,电机机体用筒内支架固定。在叶轮风筒前设置了加长段,以保证进气的均匀性。

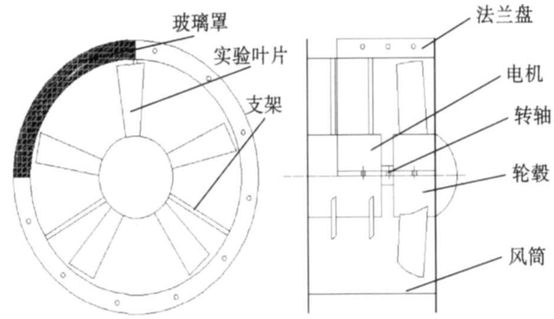


图4　PIV 实验装置示意图

根据实验分析和以往的使用经验,本次 PIV 实验采用超声波加湿器产生的纯水粒子,利用超声波高频振荡,将水雾化为 1~5 μm 的雾粒,利用导管引入到风扇进气边前 20 个叶顶弦长的位置处向流场添加粒子。利用 PIV 处理软件 Flowmap3.70 来控制和协调各组成部件间的工作以及得到最后的处理结果。

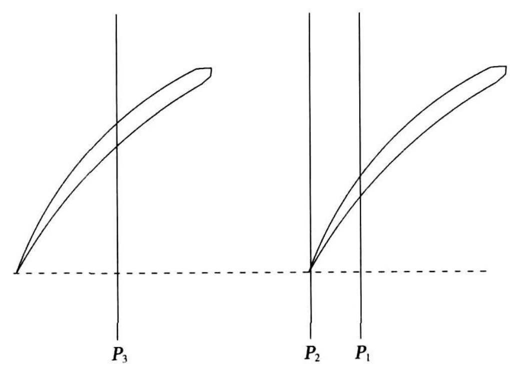


图 5　PIV 测量区域的周向位置示意图

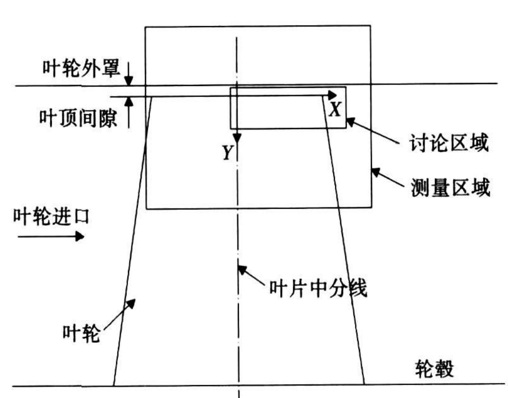


图 6　PIV 测量区域的轴向和径向位置示意图

图 5 给出了 PIV 拍摄面 P_1、P_2 和 P_3 的周向相对位置。逆叶轮旋转方向,依次经过 P_1、P_2、P_3;其

中 P_1 面与 P_2 面的周向夹角为9°,P_2 面与 P_3 面的周向夹角为36°,相对转子栅距分别为0.125和0.5,叶顶的实际周向距离分别为39和153 mm;P_1 面距离叶片吸力面侧叶顶翼型出气端的周向角度为5°,P_1 面在叶顶距离叶片吸力面出气端的周向距离为21.6 mm。

图6给出了PIV测量区域的轴向和径向位置。以下的实验研究与分析都是集中在图中所示的讨论区域(Discussion zone)内进行的。

4　结果分析

4.1　叶顶处叶片表面压力沿轴向分布

图7给出了4个周向前弯叶轮在叶顶位置(99%相对叶高)叶片表面静压系数沿轴向的变化情况。图中"0"代表叶片前缘位置,"1"代表叶片后缘位置。静压系数定义为:

$$C_p=\frac{2\cdot(p-p_0)}{\rho u_t^2} \tag{4}$$

式中:u_t—叶片尖部旋转速度,m/s;p_0—叶轮进口处的压力平均值,Pa;p—当地静压值,Pa;ρ—流体密度,kg/m³。

在99%相对叶高位置如图7所示,叶片吸力面压力分布出现非常明显的变化,同时这种变化对于不同周向前弯角度叶片产生不同的影响。首先,从叶片表面压力的轴向位置来看,随着叶片周向前弯角度的增加,叶片吸力面压力极小值点从叶片前缘逐渐向叶片后缘移动。由于叶顶处叶片表面压力的轴向分布特征能够比较准确地反映出叶顶泄漏流动的初始位置[4~6]。可以说,随着叶片前弯角度的增加,叶顶泄漏涡的起始位置逐渐向叶片后缘移动。具体表现为:原始叶轮叶顶泄漏涡起始于距叶片前缘约34%的轴向位置,前弯6.1°叶轮叶顶泄漏涡起始于距叶片前缘约41%的轴向位置,前弯8.3°叶轮起始于距叶片前缘约46%的位置,前弯12°叶轮起始于距叶片前缘约52%的位置。另一方面,从叶片表面的压力大小来看,随着叶片前弯角度的增加,叶片吸力面压力极小值逐渐增加,即原始叶轮吸力边压力下降得最为明显,前弯6.1°叶片居次,前弯12°叶片压降幅度最小。这说明叶顶泄漏流动对叶片顶部的载荷产生很大的影响,随着叶片前弯角度的增加,叶顶泄漏流动在靠近叶片吸力边处的强度逐渐减弱,即原始叶轮叶顶泄漏涡在起始位置涡强度最大,前弯6.1°叶轮居次,前弯12°叶轮最弱,这与涡特征分析结果是一致的[13]。

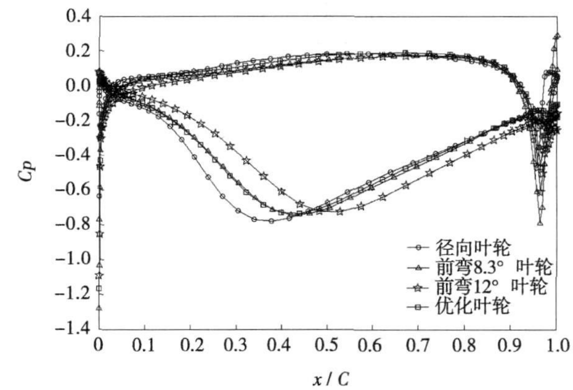


图7　周向前弯叶轮叶顶区域叶片表面静压系数沿轴向分布

4.2　叶顶泄漏流动的变化

图8~图11分别给出了4个叶轮叶顶流场的速度矢量分布,(a)—P_1 面,(b)—P_2 面,(c)—P_3 面,图8~图11为测量结果。

图8~图11的结果位于流场中包含旋转轴线在内的不同周向位置上的叶顶区域平面,在这些面上的速度信息为试验粒子的轴向速度分量和径向速度分量,不包括切向速度分量。

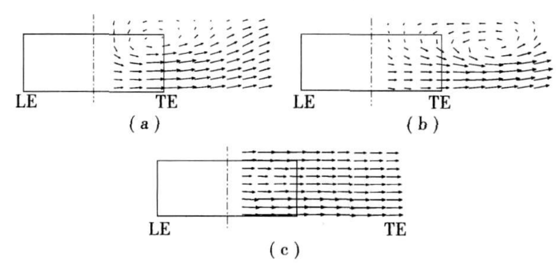


图8　径向叶轮叶顶区域相对速度分布

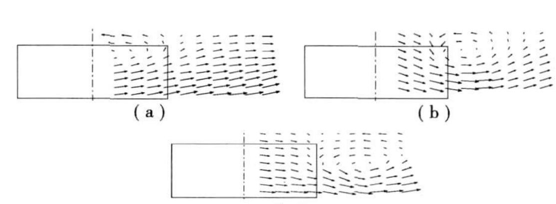


图9　前弯6.1°叶轮叶顶区域相对速度分布

研究区域的位置是:在径向90%相对叶高以上,在轴向主要集中在叶片中弦线以后出气边前、后的区域,如图6所示。

从 P_1 面的实验结果来看,原始叶轮和前弯12°

叶轮的叶顶泄漏涡已经完全被观察到, 而其它两个叶轮的叶顶泄漏涡仅仅是一部分被观察到。说明在此周向位置, 这两个叶轮的泄漏涡轴向位置更加靠近叶片后缘。4 个叶轮叶顶泄漏涡沿轴向分布依次为: 前弯 6.1°叶轮、前弯 8.3°叶轮、原始叶轮和前弯 12°叶轮。

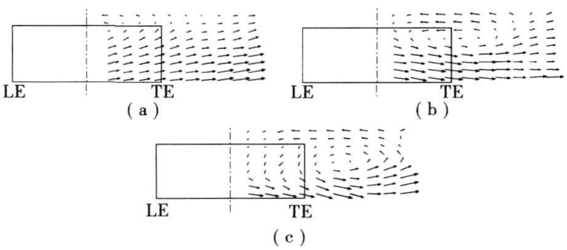


图 10　前弯 8.3°叶轮叶顶区域相对速度分布

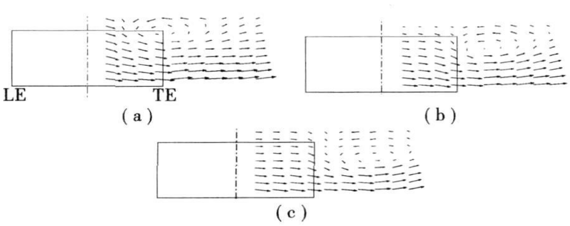


图 11　前弯 12°叶轮叶顶区域相对速度分布

从 P_2 面的实验结果来看, 这 4 个叶轮的叶顶泄漏涡都能被观察到。沿轴向分布依次为: 前弯 6.1°叶轮、前弯 8.3°叶轮、前弯 12°叶轮和原始叶轮。从 P_1 面到 P_2 面, 原始叶轮的叶顶泄漏涡沿轴向移动最快, 其它 3 个叶轮相差不大。

从 P_3 面的速度分布来看, 除了原始叶轮的泄漏涡已经无法观察到之外, 其它 3 个叶轮的叶顶泄漏涡都可以清晰可见。对比这 3 个叶轮, 可以看出, 随着叶片周向前弯角度的增加, 叶顶泄漏涡沿轴向逐渐后移。从 P_2 面到 P_3 面, 随着叶片周向前弯角度的增加, 叶顶泄漏涡的轴向位移出现"先减小后增加"的特点, 即前弯 8.3°叶轮的轴向位移最小。

5 结　论

(1) 随着叶片周向前弯角度的增加, 叶顶泄漏涡的初始位置从叶片前缘沿轴向逐渐向叶片后缘移动, 即原始叶轮叶顶泄漏涡起始于距叶片前缘约 34%的轴向位置、周向前弯 6.1°叶轮叶顶泄漏涡起始于距叶片前缘约 41%的轴向位置, 周向前弯 8.3°叶轮起始于距叶片前缘约 46%的位置, 周向前弯 12°叶轮起始于距叶片前缘约 52%的位置, 同时泄漏涡的强度逐渐减弱。

(2) 从叶顶泄漏涡的行进轨迹来看, 随着叶片周向前弯角度的增加, 叶顶泄漏涡的轴向位移呈现"先减小后增大"的变化特征, 而叶顶泄漏涡的周向位移呈现"先增大后减小"的变化规律。

参考文献:

[1] BEILER M G, CAROLUS T H. Computation and measurement of the flow in axial flow fans with skewed blades[J]. Journal of Turbomachinery, 1999, 121(1): 59—66.

[2] CAI NA, XU JIAN ZHONG, BENAISSA A. Aerodynamic and aeroacoustic performance of a skewed rotor, American Society of Mechanical Engineers, International Gas Turbine Institute[J]. Turbo Expo, 2003, 6: 497—504.

[3] CORSINI A, RISPOLI F. The role of forward sweep in subsonic axial fan rotor aerodynamics at design and off—design operating conditions [J]. ASME, International Gas Turbine Institute, Turbo Expo (Publication) IGTI, 2003, 6: 543—553.

[4] RAINS D A. Tip clearance flows in axial flow compressors and pumps [D]. Pasadena: California Institute of Technology, 1954.

[5] KOCH C C, SMITH L H JR. Loss sources and magnitudes in axial—flow compressor[J]. Journal of Engineering for Power ASME Series A, 1976, 98(3): 411—424.

[6] KAMEIER F, NEISE W. Experimental study of tip clearance losses and noise in axial turbomachines and their reduction[J]. Journal of Turbomachinery ASME, 1997, 119(3): 460—471.

[7] LEE GONG HEE, BAEK JE HYUN. A numerical study on the structure of tip clearance flow in a highly forward—swept axial—flow fan[J]. ASME Fluids Engineering, 2002, 257(2B): 799—806.

[8] LEE GONG HEE, BAEK JE HYUN, MYUNG HWAN-JOO. Structure of tip leakage flow in a forward—swept axial—flow fan[J]. Turbulence and Combustion, 2003, 70(1—4): 241—265.

[9] LEE GONG—HEE, BAEK JE—HYUN, MYUNG HWAN JOO. Effect of blade loading on the structure of tip leakage flow in a forward—swept axial—flow fan[J]. HVAC and R Research, 2005, 11(1): 95—117.

[10] DECKER J J, BEACHER B F, SCOTT MCNULTY G S, et al. The impact of forward swept rotors on tip—limited low—speed axial compressors, American society of mechanical engineers, international gas turbine institute[J]. Turbo Expo (Publication) IGTI, 2003, 6: 613—624.

[11] MCNULTY G S, DECKER J J, BEACHER B F, et al. The impact of forward swept rotors on tip clearance flows in subsonic axial compressors[J]. Journal of Turbomachinery, 2004, 126(4): 445—454.

[12] YOON JONG—HWAN, LEE SANG—JOON. Stereoscopic PIV measurements of flow behind an isolated low—speed axial—fan[J]. Experimental Thermal and Fluid Science, 2004, 28(8): 791—802.

[13] 李　杨. 低压轴流风扇周向弯曲动叶内部流动特征的实验和数值研究[D]. 上海: 上海交通大学, 2007.

(编辑　伟)

超临界循环流化床锅炉技术特点比较＝**A Comparison of Technical Features of Supercritical Circulating Fluidized Bed Boilers**[刊，汉] /ZHANG Man, BIE Ru-shan (College of Energy Science and Engineering, Harbin Institute of Technology, Harbin, China, Post Code: 150001), WANG Feng-jun, JIANG Xiao-guo (Harbin Boiler Works Co. Ltd., Harbin, China, Post Code: 150046)// Journal of Engineering for Thermal Energy & Power. — 2009, 24(3). —271 ~276

To summarize the technical features of supercritical CFB (circulating fluidized bed) boilers, analyzed were the parameter selection for supercritical boilers and the reasons why CFB boilers are more suitable than pulverized coal boilers for adopting supercritical parameters. The status quo of the study on supercritical CFB boilers both at home and abroad was described in detail. An analysis and comparison of the versions of a supercritical 600 MW CFB boiler for Baima Project proposed by three domestic boiler manufacturers shows that there exists no overriding technical barrier in the design of a supercritical CFB boiler. Moreover, the supercritical CFB combustion technology will become an important clean coal-based power generation technology for coal-fired power plants in China. However, with the in-depth development of research on supercritical CFB boilers, some relevant problems still merit further study and investigation. **Key words**: circulating fluidized bed boiler, supercritical, water wall, technical feature

圆形截面离心压缩机蜗壳内部三维流动的测量与分析＝**Measurement and Analysis of Three-dimensional Flows in the Volute of a Centrifugal Compressor with a Round Section**[刊，汉] / GAO Li-min, WANG Huan, LIU Bo (Key Laboratory on Airfoil and Cascade Aerodynamics, College of Power and Energy Source, Northwest Polytechnic University, Xi' an, China, Post Code: 710072), WANG Shang-jin (College of Energy Source and Power Engineering, Xi' an Jiaotong University, Xi' an, China, Post Code: 710049)//Journal of Engineering for Thermal Energy & Power. — 2009, 24(3). —277 ~280

By utilizing a five-hole probe, measured in detail were three-dimensional flows in the volute of a large-sized low-speed centrifugal compressor with a round section. As a result, a flow velocity distribution and a distribution chart of flow speed, total and static pressure along the radial and circumferential direction in the radial measurement section of the spiral flow passage portion of the volute were given, and the flow rates thus obtained, compared with those obtained from the one-dimensional calculation. It has been found that the measured flow rates in various sections assume an identical variation tendency with those obtained from the one-dimensional calculation. In step with increasing angles, the difference between the calculation results and test ones gradually decreases. In addition, the flows in the volute under test pertain to complex three-dimensional flows, and the flow speed distribution along the radial direction in various radial sections of the volute features a comparatively conspicuous difference from the momentum conservation law. The change in total pressures along the circumferential direction is not manifest. **Key words**: centrifugal compressor, volute, round section, flow measurement

悬臂转子系统振动特性分析＝**An Analysis of the Vibration Characteristics of a Cantilever Rotor System**[刊，汉] /AN Xue-li, ZHOU Jian-zhong, LI Chao-shun, LIU Li (College of Hydropower and Digital Engineering, Huazhong University of Science and Technology, Wuhan, China, Post Code: 430074) // Journal of Engineering for Thermal Energy & Power. — 2009, 24(3). —281 ~285

Derived and established was a kinetic equation for cantilever rotor systems with the role of rotor blades being taken into account. Through a numerical integration of the motion differential equation, the influence of various factors on the vibration characteristics of a cantilever rotor system was analyzed. The factors include mass eccentricity, rotor blade mass and bearing spacing etc. The numerical analytic results show that with a continuous increase of the mass eccentricity of wheel disk No. 2, the radial displacement of the rotor system will exhibit an approximately linear increase. In case different values are chosen for rotor blade mass and the spacing between bearings etc., the change of the radial displacement of the rotor system becomes relatively complicated. The radial displacement value of wheel disk No. 1 may be greater than, equivalent to or less than that of wheel disk No. 2. **Key words**: overhung rotor, blade quality, mass eccentricity, radial displacement, bearing spacing

叶片的周向前弯角度对低压轴流风扇叶顶泄漏流场的影响＝**Influence of the Circumferential Forward Skew**

Angle of Blades on the Blade Tip Leakage Flow Field of a Low-pressure Axial Flow Fan[刊, 汉] / LI Yang (College of Electromechanical Engineering, Qingdao University of Science and Technology, Qingdao, China, Post Code: 266061), LU Ji-fu (College of Chemical Engineering, Zhengzhou University, Zhengzhou, China, Post Code: 450001)// Journal of Engineering for Thermal Energy & Power. — 2009, 24(3). —286 ~ 290

An experimental and numerical study was performed of a low-pressure axial flow fan installed with blades of four different circumferential forward skew angles (1.27, 6.1, 8.3 and 12 degrees). By utilizing Reynolds Number averaged Navier-Stokes (N-S) equation group and a turbulent flow model of Spalart-Allmaras (S-A) No. 1 equation, a numerical calculation was conducted of the three-dimensional viscous flow field of the impeller at steady operation points. Under the precondition of the numerical calculation results being identical with the measured ones, analyzed was the axial distribution of blade surface pressures at the blade tip. The calculation results show that with an increase of the circumferential forward skew angle of the blades, the initial location of the blade tip leakage vortex will gradually shift toward the blade trailing edge. By employing a particle image velocimetry (PIV) system, the blade tip leakage flow field of the impeller was tested and measured, explicitly showing the evolution of the blade tip leakage vortex. It has been found that with an increase of the circumferential forward skew angle of the blades, the stability of the blade tip leakage vortex will "first become greater and then weaker", the axial displacement of the vortex in question will "first decrease and then increase" while its circumferential displacement will "first increase and then decrease". **Key words**: axial flow fan, circumferential forward-skewed blade, blade tip leakage flow, particle image velocimetry (PIV)

压缩机旋转失速发展传播的计算分析 = **Calculation and Analysis of the Rotating Stall Development and Dissemination in a Compressor**[刊, 汉] / JI Chun-jun, WANG Yang (College of Energy Source and Power, Dalian University of Science and Technology, Dalian, China, Post Code: 116023), JI Wen-hui (CSIC Harbin No. 703 Research Institute, Harbin, China, Post Code: 150036)// Journal of Engineering for Thermal Energy & Power. — 2009, 24(3). —291 ~ 295

In the light of a blade fracture problem occurring to the first impeller of a compressor during its operation, set up was an inner flow field numerical-analytic platform based on a centrifugal compressor with an entire and true three-dimensional geometrical structure by using CFD (Computational Fluid Dynamics) software Numeca. A calculation was performed by choosing practical operating conditions. It has been found that a definite problem existing in the design of the air inlet chamber and guide vanes leads to a distortion of the flow field before the first-stage impeller, triggering a stall of the impeller in the compressor. Through an unsteady flow calculation, the development and dissemination process of the rotating stall was dynamically simulated and a relatively true pressure fluctuation caused by the rotating stall, obtained. As a result, a breakthrough for the study of blade fracture problems of the compressor unit was attained, offering sufficient data and information for further investigation and analysis. **Key words**: unsteady flow, rotating stall, numerical simulation

某重型燃气轮机喷嘴组流量特性试验研究 = **Experimental Study of the Nozzle Group Flow Characteristics of a Heavy-duty Gas Turbine**[刊, 汉] / LIU Kai, ZHANG Bao-cheng, MA Hong-an (School of Power and Energy Engineering, Shenyang Institute of Aeronartical Engineering, Shenyang, China, Post Code: 110034)// Journal of Engineering for Thermal Energy & Power. — 2009, 24(3). —296 ~ 299

Briefly described were the results of an experimental study on the nozzle group oil-circuit flow characteristics of a heavy-duty gas turbine (National 863 Program Major and Special Item). Through tests performed on the components and nozzle groups, the dimensions of various flow circuits of the nozzle groups were determined. A quantitative relationship of flows in mutual interference in case of the first and second circuit of the spray nozzle groups jointly supplying oil was obtained. Moreover, a quantitative relationship of the decrease in flow rate caused by such factors as welding, etc. was also acquired. The spray nozzle groups designed based on the test data have passed the single-tube combustion test with various indexes satisfying the design standard. It has been proven that the test facilities and methods are practical, feasible and reliable. The test data can well provide a reliable basis for the retrofitted design of the nozzle groups. **Key words**: heavy-duty gas turbine, nozzle group, flow characteristics, test