

压气机“修型”叶栅的实验研究

北京航空航天大学 吴国华* 彭泽琰 严 明 任丽芸

【摘要】 文中通过压气机叶片“修型”叶栅和常规叶栅的对比实验,研究了“修型”叶栅后三维流动特征。试验结果表明,对常规压气机叶片端部尾缘进行局部修型(即改变叶片的几何形状,但与“端弯”方式不同),在叶栅损失系数基本不变或略有下降的前提下,可以有效地改善和控制栅后出口气流角沿叶高的分布,以满足下游动叶进口气流方向的要求。平面叶栅试验结果还表明,尽管对常规叶栅端部尾缘实施局部修型,减小了端区的叶片出口构造角,但对整个气流转折角影响不是太大。同时,叶栅自身的流通能力基本不受影响,甚至有所改善。此技术已成功地应用于多级压气机和喘振裕度的改善、效率的提高以及压气机不稳定脉动压强(叶片激振动)和乱分离的抑制。

关键词: 叶栅 修型 压气机 端壁流

随着压气机不断朝着低展弦比、高级负荷方向发展,使得压气机内部流动完全三维化。压气机端部区损失成为主要损失来源之一^[1]。由于人们对压气机端壁流动的有限认识,使得按当前压气机设计理论体系设计的压气机,不能细致地控制压气机端壁流动,常常面临着喘振裕度偏小,稳定工作范围偏窄的困难。这一困难成了拖延新型压气机研制周期的主要原因之一。为了解决这一困难,除了进一步研究和理解压气机端壁流动的细致结构及特征之外,还应寻找、发现一些为工程适用的、提高压气机喘振裕度、扩大稳定工作范围的技术,便于新机性能调试和老机性能改进。本文报告了一种可用于此目的、由北航发明的全新而且简便的技术,即所谓的压气机叶片端壁“修型”技术^[2]的平面叶栅风洞的试验结果。所谓“修型”技术,就是指在原始叶片修改叶片端区出口的几何构造角,以减小下游叶片排端区的攻角,从而达到扩大压气机的喘振裕度,改善其非稳态工作性能和提高效率的目的。作者对此技术的各种修型量进行了大量的平面叶栅试验。试验结果表明,该技术的确减小压气机叶栅端区的能量损失,改善栅后出口气流角沿叶高分布。同时作者还成功地将此技术应用于提高新机的喘振裕度和效率及抑制压气机不稳定压强脉动和乱分离现象^[3]。本文通过介绍一个典型压气机“修型”叶栅(简称为叶栅 G1-C)同常规叶栅(“无修型”叶栅,简称为叶栅 G1-O)的对比性试验。描述了“修型”叶栅后三维流场的特征,尤其是栅后出口气流角、叶栅总压损失系数及流通系数等参数沿叶栅展向分布。平面叶栅风洞的试验结果对于工程上实际应用此技术有一定的参考和指导意义。

1 试验设备、试件及数据处理

试验工作是在北航低速平面叶栅风洞中完成。风洞来流紊流度为 0.5%,来流端壁附面层厚度为 20mm,试验的叶片弦长雷诺数为 $Re = 2.0 \times 10^5$ 。叶栅前设置一个常规的圆柱三孔探针,栅后设置一个束管式五孔探针。栅后五孔探针由风洞配备的 TMD-2 型三自由度数控位移

机构控制。位移机构的直线位移分辨率为 0.1mm, 角度位移分辨率为 0.1°。测量数据由一组 Y YT-200B 斜管压力计读出。在叶栅后沿展向共布置 10 个测站, 其距端壁的距离分别为 2.0、4.0、6.0、9.0、13.0、19.0、27.0、37.0、49.0、62.0mm。沿栅距方向安排 14 个测站, 且均匀分布。叶栅 G1-C 和叶栅 G1-O 的基本参数是相同的, 即叶型是 BC-6, 叶型转折角为 49.9°, 叶片弦长为 80mm、进口几何构造角为 46.6°、安装角为 76°、展弦比为 1.55, 叶栅稠度为 1.54, 叶片数目为 8。两套叶栅的差别在于, 叶栅 G1-C 是在常规叶栅 G1-O 的叶片端区尾缘处, 局部改变了叶片叶型的几何形状。实施修型的位置, 相对于叶片中间高度而言是对称的。

各测点的总压损失系数 ω 、栅距方向质量平均总压损失系数 $\bar{\omega}$ 及通道质量平均总压损失系数 $\bar{\bar{\omega}}$ 分别按下列形式定义:

$$\omega = \frac{p_{1m}^* - p_2^*}{(p_1^* - p_1)_m}, \quad \bar{\omega} = \frac{\int \omega \rho V_{2a} dx}{\int \rho V_{2a} dx}, \quad \bar{\bar{\omega}} = \frac{\int \int \omega \rho V_{2a} dx dy}{\int \int \rho V_{2a} dx dy}$$

其中, m 表示叶片中间高度位置; t 为一个栅距; h 为叶片高度; x 表示栅距方向, 其积分上、下限为 $0 \sim t$; y 表示叶片展向, 其积分上、下限为 $0 \sim h/2$; ρ 为气体密度; V_{2a} 为栅后出口气流的轴向分速。其它参数的栅距方向质量平均及通道平均分别类似 $\bar{\omega}$ 和 $\bar{\bar{\omega}}$ 的定义式。沿叶片展向各站的扩压比定义为: $\bar{\pi} = \bar{p}_2 / \bar{p}_{1m}$, 流通系数定义为: $\bar{Q} = V_{2a} / V_{1m0}$ 。

2 实验结果及讨论

2.1 栅后出口气流角 β_2 对比

图 1 为名义 0 攻角下, 叶栅 G1-C 与叶栅 G1-O 栅后出口气流角沿叶展分布的对比图。由图可见从距端壁 20.0% h 展向位置到叶片中间高度这一展向范围内, 两套叶栅的出口气流角是一样的。在距端壁 0.0~10.0% h 的展向区域内, 叶栅 G1-C 的出口气流角小于叶栅 G1-O。从整个展向分布看, 叶栅 G1-C 的栅后出口气流角较叶栅 G1-O 变化幅度小, 即叶栅 G1-C 的栅后出口气流角分布较均匀。在大、中攻角下的情形也是如此。这表明叶栅 G1-C 具有改善和控制栅后出口气流角的能力。虽然在各名义攻角下, 叶栅 G1-C 在端区的出口气流角均小于

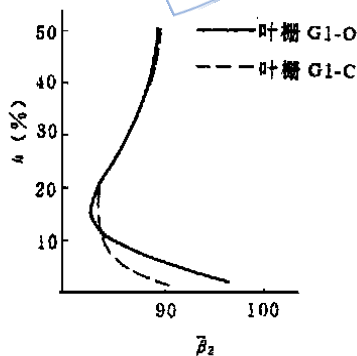


图 1 出口气流角沿叶展分布(测量位置 0.5b)

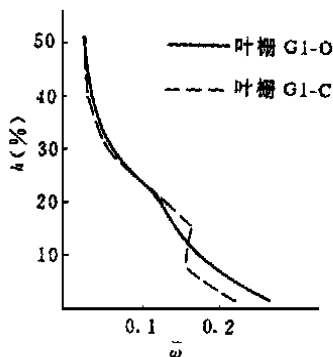


图 2 总压损失系数沿叶展分布(测量位置 0.5b)

叶栅 G1-O 的, 但按通道质量平均的气流转折角, 较叶栅 G1-O 减小不多, 分别减小 0.42°、0.24°、1.03°。应该强调的是“修型”叶栅端区出气角减小不是其缺点, 而是优点。正如“端部”减弯技术成功地应用于 WP7 高压压气机一样。此外, 对于设计不当的老机和新机, 其动叶攻角过

大时, 人们需要叶片出气角减小, 不仅是端区, 而且是全叶高范围, 当然量要合适, 恰当。

2.2 总压损失系数 $\bar{\omega}$ 对比

图 2 表示在名义 0 攻角下, G1-C 与 G1-O 的总压损失系数沿展向分布对比。由图可以看出, 在距端壁 0.0~12.0%h 的展向区域, G1-C 的总压损失系数小于 G1-O 的。这一现象同叶片端区修型后, 端区附面层由压力面流向吸力面的横向流动减弱有关。在距端壁 15.0%h 的展向位置, G1-C 的总压损失系数曲线有一凸包。这与叶片修型后, 吸力面角区的流场结构发生变化有关, 即具有较高能量的二次流, 由压力面跨越修型表面流向端区; 而低能量二次流则沿吸

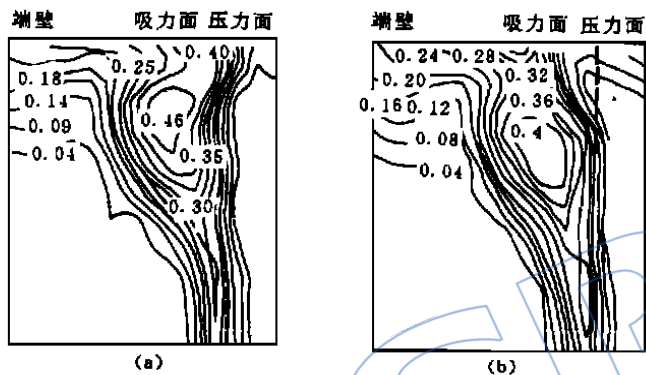


图 3 叶栅总损失系数等高线(a) G1-O (b) G1-C (名义攻角 0°; 测量位置 0.5h)

力面向叶片展向中部流动; 这种流动倾向改善了端区流场结构。从叶栅 G1-C 和叶栅 G1-O 的总压损失系数的等值线图, 即图 3 可以证实这一现象。图 3 表示 G1-C 的高损失区较 G1-O 远离端壁, 而且高损失区的面积较大, 但损失值较小。这表明在叶栅 G1-C 中, 低能量流体可以在流向速度较大的区域中被带走, 而不象常规叶栅那样堆积在流向速度较低的吸力面角区。因而, G1-C 的高损失区较叶栅 G1-O 远离端壁。当然, 由于较强烈的二次流展向流动, 使 G1-C 的高损失区面积较大。由于叶栅附加总能量损失是由低能量不堆积所带来的好处与附加的二次流展向流动所增加的损失共同决定, 因而按通道质量平均的总压损失值, G1-C

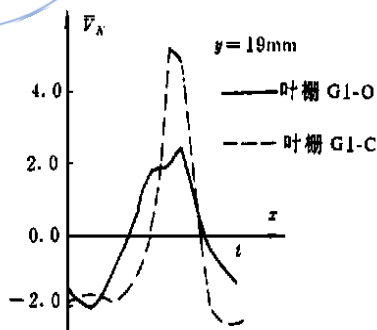


图 4 展向二次流速沿栅距分布
(名义攻角 0°; 测量截面 0.5h)

略小于 G1-O 或基本不变。在攻角较大情况下, G1-C 的二次流展向流动趋势更为显著。这可由图 4 看出, 在 $y = 19\text{mm}$ 处, 即修型表面与常规表面的过渡区, G1-C 的二次流展向分速较 G1-O 大, 而且栅距方向的宽度窄。

2.3 流通系数 $\bar{Q}$ 及扩压比 $\bar{\pi}_t$ 对比

在低速流动中, 反映叶栅流通能力的密流比, 可以用栅后无量纲的轴向速度代替。由于流通能力是同时受叶栅出口气流角及总压损失系数左右, 且这二者的变化趋势对流通能力的影响是相左的。按通道质量平均的流通系数表明, 在 3 个名义攻角下, 叶栅 G1-C 较叶栅 G1-O 分

别提高- 0.9%、0.4%、3.4%。叶栅扩压能力同总压损失系数、气流转折角及流管收缩比有关,前二者的变化趋势对叶栅扩压能力的影响是相反的。按照通道质量平均的扩压比数值来看,G1-C 和 G1-O 的扩压能力基本一样(参见表 1)。这是因为端区“修型”后,端区流动损失减少,因而出口气流总压较高。在同样速度下,静压升高,弥补了“修型”导致的端区扩压面积比下降。对我们所研究的基准叶栅而言,其端区出气角大于 90°,因此出气角下降会更有好处。

表 1 通道质量平均性能参数对比

攻角	原 始				修 型			
	ω	$\triangle\beta$	Q	π_k	ω	$\triangle\beta$	Q	π_k
0°	0.09216	40.63	0.686	1.00323	0.0926	40.22	0.680	1.00320
5°	0.17067	43.28	0.629	1.00359	0.1465	43.04	0.654	1.00389
10°	0.27660	49.79	0.558	1.00410	0.2684	48.75	0.577	1.00408

2.4 应用实例

将“修型”技术应用于某多级轴流压气机的静叶端区,多级压气机试验台上的性能实验结果为,在工作转速 $\bar{n}=0.8、0.9$ 及 1.0 下,压气机喘振裕度的提高值分别为 4.0%、2.0%、1.0%。压气机效率在 $\bar{n}=0.8$ 时,提高 1.0%。其它转速下,效率与原型机相同。实验结果表明,修型量选定是正确的,保证了喘振裕度和效率同时提高,压比和流量基本不下降。另一多级压气机实验结果表明,通过修型某排静叶端区,消除了该叶片下游转子叶尖的乱分离。将“修型”技术应用于另一多级压气机的静叶,使压气机不稳定压强脉动幅值下降 1 倍以上。

结论 (1)平面叶栅实验研究表明,压气机叶片端部“修型”技术,在总压损失系数不变或略有下降的前提下,可以有效地控制和改善栅后出口气流角沿叶展的分布。(2)在平面叶栅风洞中,压气机“修型”叶栅的流通能力在大、中攻角下略有提高,扩压能力与常规叶栅相同。(3)工程实际应用成功的实例表明,压气机叶片端“修型”技术是一种简便、实用、提高压气机喘振裕度和效率、改善压气机稳定性的有效手段,尤其是在新机性能调试或老机性能改进中,更显示出其简便、实用的优点。

参 考 文 献

1 Lakshminarayana B, Pouagare M, Davino R. Three-Dimensional Flow Field in the Tip Region of a Compressor Rotor Passage. Part 1: Mean Velocity Profiles and Annulus Wall Boundary Layer. ASME 82-GT-11, 1982

2 彭泽琰, 吴国华, 严明, 任丽芸. 一种用于改进压气机喘振裕度和效率的技术. 中国专利号 89 102542. 1, 1989

3 Peng Z, Wu G, Yan M, Ren L. An Experimental Investigation of Technologies of Endwall Flow Control in a Compressor Plane Cascade. AIAA-91-2005

(责任编辑 魏星祿)

RESEARCH ON FUNCTION OF 8-SHAPED VORTEX IN OSCILLATORY PROPULSION

Wu Jing and Gao Ge

(Beijing University of Aeronautics and Astronautics)

ABSTRACT

A wave-vortex principle of oscillating propulsion is presented, which is based upon the observation and experimental study of fish's oscillating propulsion. The theoretical analysis shows, that the occurrence and interaction of 8-shaped vortex is the reason for creation of thrust. The experiments accomplished with oscillating stiff blades give a strong heuristic evidence. It is that the discovered 8-shaped vortex possesses behaviour of energy storage. This new principle is of great significance for analysing fish's swimming and designing high efficiency propeller.

PREDICTION OF PROPELLER PRESSURE DISTRIBUTION AND PERFORMANCE BY LIFTING SURFACE THEORY

Chen Zemin and Pan Jieyan

(Beijing University of Aeronautics and Astronautics)

ABSTRACT

A numerical method is presented for prediction of propeller performance and pressure distribution on blade surface according to lifting surface theory. The lifting surface model consists of two layers of constant strength quadrilateral vortex - ring elements covering both surfaces of the blades. The velocity jump across vortex sheet is included in the computation of the pressure distribution. The effect of blade drag has been included in the calculation of propeller performance. Numerical results agree well with experimental data for the considered two propellers.

A TURBULENCE MODEL FOR FILM COOLING RECOVERY REGION

Jiang Tao, Liu Songling, Zhu Huiren

(Northwestern Polytechnical University)

ABSTRACT

It is observed that the normal $k-\epsilon$ turbulence model, which is very popular in computational fluid dynamics, is inapplicable to predicting the film cooling recovery region flow successfully. The reason might be the fact that the profiles of flow parameters are significantly different from those in normal boundary layer due to the blowing and the distance needed for flow returning to the normal boundary layer is too large. A new turbulence model based on Lam-Bremhorst low Reynolds number $k-\epsilon$ turbulence model has been proposed for prediction of the film cooling recovery region. It was found that the constant c_μ and c_1 in the original model should be modified to be functions of coordinates x and y , and the low Reynolds number functions f_μ , f_1 and f_2 should be varied with x and y too. The modified expressions feature that they should be approaching to nonblowing boundary layer when the distance from initial section becomes large enough. Three numerical examples demonstrate the advantages of this new model over the original one for the blowing ratio ranging from 0.4 to 1.0.

EXPERIMENTAL INVESTIGATION OF AN END-MODIFIED COMPRESSOR CASCADE

Wu Gouhua, Peng Zeyan, Yan Ming, Ren Liyin

(Beijing University of Aeronautics and Astronautics)

ABSTRACT

A compressor end-modified cascade and its original cascade are compared experimentally to study the three-dimensional flow characteristics at the exit plane of

the cascade. The results show that the modification of the end part at the trailing edge of the original blade (which is different from end-bend) would effectively improve the exit air angle spanwise distribution and meet the need of the inlet air angle of next row blades, on condition that the total pressure loss coefficient does not change or decrease. The test results also indicate that although the camber angle of the blades in the end part decreases due to the modification, the passage-mass-averaged value of the camber angle changes few. Otherwise, the air flow through the cascade is the same to the original or even better. This technique has been successfully applied to improving both stall margin and efficiency of axial multistage compressors and to depressing the unsteady pressure disturbance.

A NEW METHOD FOR SOLVING LOCAL FLOW NEAR LEADING OR TRAILING EDGE OF AN AIRFOIL

Liu Gaolian and He Hong

(Shanghai Institute of Mechanical Engineering)

ABSTRACT

a new algorithm coupled with the variational finite element method (FEM) is presented for solving the local flow near the leading or trailing edge of an airfoil or cascade. The solution of the local flow is considered as a combination of two parts. One of them is the approximate analytical solution of the flow near the stagnation point, it absorbs the greatly changing part in the original solution and makes the rest a more smooth distribution with better analytical nature which can be computed more accurately. Taking advantage of the variational principles, it is possible to locate the stagnation point more precisely and automatically. The numerical results show that the method described is effective to solve the subsonic flow accurately, especially in conditions with non-zero angle of attack. The theory can be coupled with the variational difference method as well as the variational FEM.

3-D ANALYSIS OF CIRCUMFERENTIAL GROOVED CASING TREATMENT

Du Zhaohui (*Shanghai Jiaotong University*)

Liu Zhiwei (*Northwestern Polytechnical University*)

ABSTRACT

The 3-D incompressible steady time-averaged N-S equation, continuity equation and $k-\epsilon$ turbulence model equations are applied to calculating the flowfield in the cascade passage with circumferential grooved casing treatment. The mechanism of extending stability of the circumferential grooved casing treatment is elaborated by means of a 3-D viscous numerical simulation. It is revealed that a pressure difference between suction side and pressure side induces a strong gas flow which is absorbed in the pressure side and poured out from the suction side. This flow can blow off the low energy boundary layer efficiently to retard the stall emergence. The physical model and the flow pattern of the flowfields affected by inductive velocities are described. This is a new approach to analysis and design of the compressor casing treatment.

A BIFURCATION PHENOMENON IN COMBUSTOR WITH A NON-SLOT CONICAL SWIRLER

Jiao Shujian, Lee Jin, Liu Jing, Chen Liru

(Tsinghua University)

ABSTRACT

A bifurcation phenomenon was discovered by experiments in a gas turbine combustor with a non-slot conical swirler. With different kind of perturbation, two different flow patterns may be found in the liner of this gas turbine combustor separately. The characteristics of these two flow patterns are quite different as described in great detail in this