

某型压气机轴流级改型静叶设计

北京航空航天大学 曹人靖* * 陶德平

【摘要】 通过对某原型轴流级静叶的分析, 并对轴流压气机设计中采用的端部处理技术进行分析的基础上, 提出了静叶改型设计以及端部处理方案, 并进行了通流计算和造型。对改型设计中的双排静叶参数进行了优化、采用端部处理技术以及任意叶片造型, 提高了轴流级效率。

主题词: 轴流式压缩机 静叶片 通流能力 叶片造型
分类号: TH453

文献[1, 2]对静叶基元特性作了大量的试验研究, 得出了静叶设计有关参数选择的优化准则, 这些为轴流压气机静叶设计提供了有用的参考。本文针对某原型轴流级静叶的特点, 重新进行气动参数优化和叶片造型, 并对叶根和叶尖进行端部处理, 使得所设计压气机在承受高负荷的同时具有高的效率。

1 原型轴流级静叶分析

某原型压气机轴流级静叶设计采用了双排串列叶片和静叶叶尖前缘增弯, 而原型轴流级静叶的“端弯”量过大。由表1可看出原型端部效率很低, 这说明了静叶改型设计的关键之一是提高端部效率。此原型压气机轴流级静叶叶根、叶尖总稠度分别为3.4、2.6。前后排弯度比叶尖为0.922, 而叶根已达4.94。前后排间隙值为3.1毫米, 间隙值与前排弦长之比在0.120~0.175的范围内, 前后排展弦比分别为1.52、1.40。

文献[1, 2]对此原型压气机轴流级静叶串列叶栅进行了系统的试验研究, 结果表明双排静叶损失系数偏高。其原因是: (1) 总稠度为3.07, 而文献[1, 2]推荐值为2.0左右; (2) 双排静叶间的周向偏距太大, 周向偏距与栅距的比值, 叶根、叶尖分别达到0.863、0.909, 文献[1, 2]推荐值为0.80左右。周向偏距大意味着后排叶片吸力面在前排叶背之尾流中, 这样损失大是可以想象的; (3) 双排静叶的弦长比, 叶根为1.037, 叶尖为

表1 双排静叶原型气动计算结果

半 径	叶 根	中 径	叶 尖
流经前排 η^*	0.7509	0.8932	0.6763
流经后排 η^*	0.7308	0.8725	0.6588

1.154, 弯度比叶根、叶尖分别为0.922和4.940, 文献[1, 2]推荐值弦长比应小于1.0, 弯度比应大于1.0, 但不应太大。与文献[1, 2]推荐值相比偏高, 特别是叶尖弯度比太大; (4) 根据文献[2]提供的优化公式计算的轴向偏距比比原型静叶设计值小, 这说明原型静叶轴向偏距过大。

要提高原型轴流级静叶效率, 减少损失, 必须针对上述问题给予合理的解决。

2 压气机轴流级改型双排静叶设计

2.1 双排静叶改型设计措施

由于改型级压比提高幅度大, 转子叶根转角大, 转子叶根出口绝对马赫数偏高, 因此, 静叶根部所受负荷很大; 另外, 静叶所承受的气流转角也很大。本次改型设计中静叶所采取的主要措施有如下几点: (1) 采用双排串列叶片; (2) 将叶片数减少, 从而减少叶片稠度; (3) 前排静叶根部前掠, 满足文献[3]中“后掠”准则, 即前缘叶根法向马赫数小于或等于 0.4; 提高根区效率; (4) 前排静叶尖部前掠加端弯; (5) 采用任意叶片造型, 前排积迭轴为叶片尾缘, 后排积迭轴为叶片前缘; (6) 减少双排静叶展弦比。由此看出改型设计采用了“端弯”和“前掠”技术。大量试验结果表明这两种端部处理技术能有效地提高根区效率、减少损失。

2.2 静叶气动设计及结果分析

轴流级前排静叶根部和尖部实施了“前掠”处理技术。前后排静叶 $\Delta(C_u R)$ 沿流向分布文献[3]中 HTFC 静子的给定。通过调整单排叶片 $\Delta(C_u R)$ 沿流向分布, 得到一定的静压沿流向分布规律以保证叶片尾缘满足 Kutta 条件; 前后排静叶负荷分布满足“前轻后重”的原则。静叶设计所必须检查的气动极限负荷参数为: (1) 轮毂 D 因子; (2) 轮毂进口马赫数。双排静叶通流设计结果中叶根和叶尖 D 因子、进出口气流转角和叶片排效率分布见表 2。由 D 因子分布可看出叶尖、叶根 D 因子大于中径, 从总的数值来看没有超出气动极限负荷的要求。效率分布呈现端部较小中径较大的趋势, 这是因为端壁损失较大的原因, 与表 1 所列数据对比可看出改型后静叶效率比原型要高。

根据气动设计结果, 可以计算叶根和叶尖的后掠角及法向马赫数。由几何关系可得: $\tan(90^\circ - \gamma_{sw}) = \cos\alpha \cdot \tan\beta$ 。式中 γ_{sw} 为叶根或叶尖后掠角; α 为叶根、叶尖前缘子午投影切线与轴线夹角; β 为叶根或叶尖处进口气流角。由上式求出叶根和叶尖后掠角及法向马赫数。叶根: $\gamma_{sw} = 64.7^\circ$; $Ma_n = 0.38$; 叶尖: $\gamma_{sw} = 49.6^\circ$; $Ma_n = 0.39$ 。由此可见, 改型静叶端部前掠满足文献[3]的来流马赫数的法向分量准则。

2.3 双排静叶造型结果

改型静叶造型采用任意叶片造型, 同多圆弧叶片造型相比, 其优点是可以使得所造叶片最大限度地与气动设计结果相匹配。与原型相比, 前排静叶稠度比原型低, 这是考虑到原型静叶稠度太大而在改型中将叶片数减少的结果。吸力面攻角除叶根较大外都在零附近, 根部前掠而使稠度较大, 因此根部吸力面攻角大于零以使得根部堵塞在合理的范围内。由于后排静叶位于前排尾缘, 所以, 许多参数的选取考虑到了后排静叶的工作特点, 攻角的选取参考叶栅试验结果, 以使得后排静叶在承受较大负荷的同时保证气流在叶背和叶盆不发生分离。经过造型得到了双排静叶弦长比、弯度比、轴向间隙与前排弦长之比和周向偏距与栅距之比, 与原型相比改型后的弦长比更小, 这与文献[1, 2]中提供的经验数值基本一致。由表 3 可看出, 弯度比与原型相比小很多, 原型双排静叶的弯度比太大。 $(h/t)_{opt}$ 是由文献[2]中总结出来的优化公式计算的结果。 a/C_f 和 $(h/t)_{opt}$ 是决定前后排间隙的重要参数, 本次改型设计是在参考文献[1, 2]提供的经验参数基础上进行的。

表 2 双排静叶改型气动设计结果

前 排				后 排			
R	D	$\triangle\alpha$	η^*	R	D	$\triangle\alpha$	η^*
叶根	0. 5630	18. 74	0. 7575	0. 0415	0. 5393	39. 33	0. 7339
中径	0. 4330	18. 67	0. 8923	0. 0594	0. 2976	25. 52	0. 8871
叶尖	0. 4847	18. 59	0. 8323	0. 0743	0. 4095	23. 12	0. 8188

表 3 双排静叶改型参数分布

$R\text{ (m)}$	C_r/C_f	Q_l/Q	a/C_f	$(h/t)_{opt}$
叶根	0. 7780	1. 7542	0. 059	0. 81
中径	1. 0832	1. 4880	0. 072	0. 83
叶尖	0. 9018	1. 2960	0. 056	0. 81

端弯叶片造型是通过改变叶尖流面的吸力面攻角在造型中直接实现,而不是在造型完后局部修型。前排静叶经过端弯处理后的造型结果,与末端弯结果相比,在叶尖区吸力面攻角减小、安装角增大。

参 考 文 献

1 苗厚武等. 串列叶片的应用研究. 航空动力学报, 1991, 6(3)

2 吴国钊等. 双圆弧串列叶栅流动性能的试验研究. 航空动力学报, 1987, 2(2)

3 Wennerstrom A J, Frost G R. Design of a 1500ft/sec, Transonic, High- Through- Flow, Single- Stage Axial- Flow Compressor with Low Hub/Tip Ratio. AFARL- TR- 76- 59, Aerospace Research Laboratories, Wright- Patterson Airforce Base, 1976.

(责任编辑 魏星禄)

MODIFICATION OF STATOR IN AN AXIAL FLOW COMPRESSOR— STAGE

Cao Renjing and Tao Deping

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

ABSTRACT

According to the detail analysis of the experimental results for the stator of an axial flow stage, the main factors of the stator losses have been found. The end treatment technique— hub swept technique, which can enhance the efficiency in stator hub under high blade loading, has been used in the modification. The redesigning scheme for the stator and its end treatment are presented, and the detailed through-flow calculation and blade profiling have been carried out. Optimization of stator parameters has been made in the modification. The results of the modification show that the end treatment technique and blade profiling help improve the efficiency in end-wall region.

Key words Axial flow compressors Stator blades Discharge current capability
Blade shape construction

NUMERICAL SIMULATION OF WAVES CONSTRUCTION IN TRANSONIC NOZZLE FLOW

Jiang Xinghong and Wu Xingping

(College of Aerospace, Northwestern polytechnic University, Xi'an 710072)

Chen Fulian

(Mechanical Engineering Dept., Zhejiang University of Technology, Hangzhou)

ABSTRACT

The Split—Coefficient Matrix (SCM) method is manipulated here for the numerical computation of Euler equations so as to simulate the two-dimensional axisymmetric transonic nozzle flow. The coefficient matrix is splitted directly by a set of simplified formula. A second-order accurate explicit scheme is constructed to compute inner grid points. The scheme is free of artificial viscosity and is economical in terms of computer memory. 5151 grid points are embedded into the nozzle flowfield and the computational results are in good agreement with experiments. Once the flow field is known, the Mach waves characteristic relations are numerically solved by Euler method. Based on the graphic output of the Mach number and Mach waves in nozzle, it is possible to demonstrate not only the formation and reflection of an oblique shock within a tested nozzle geometry but also its peculiar zonal distribution of expansion and compression waves. As a result excellent consistency between parameter distribution and waves construction is revealed.

Key words Transonic nozzles Numerical simulation Expansion waves Compression waves

NUMERICAL SIMULATION OF FLOW AND TEMPERATURE FIELDS IN DOUBLE CONVERGENT NOZZLES

Xia Chunlin and Liu Dezhang

(2nd Dept., Nanjing University of Aeronautics and Astronautics, Nanjing 210016)

ABSTRACT

A numerical simulation was employed to study the flow and temperature fields of hot exhaust gas mixing with cold air in exhaust ejector. In the calculation the algebraic stress model (ASM) with the effect of buoyancy forces as turbulence model was adopted. Four heat radiation flux model was taken as heat radiation model. Due to complicated geometry of the nozzles the boundary-fitted curvilinear coordinates were employed and curvilinear mesh was generated with the help of TTM method, then the blocks were put together under the given boundary conditions. A new general numerical method is developed on the basis of SIMPLE (Semi-Implicit Method for Pressure Linked Equations), and a general software was compiled in the boundary-fit-