第 7 卷　第 2 期　　航空动力学报　　Vol.7　No.2
1992 年　4 月　　Journal of Aerospace Power　　Apr.　1992

具有弯扭叶片的燃气透平正问题计算方法

哈尔滨工业大学　　王仲奇**　杨　弘　刘凤君　黄影虹　冯国泰

【摘要】　本文开发了一个实用的多级燃气透平 S_2 流面跨音正问题的计算程序，用它对一台燃气透平的改型设计进行了计算。通过计算研究了弯扭叶片在透平改型设计中应用的可能性；分析了不同的叶片弯曲方法给透平性能带来的影响；提出了对弯扭叶片的气动设计具有指导意义的意见。

一、前　　言

虽然目前国外已将弯扭叶片应用于航空发动机和小型涡喷发动机中，但设计方法很少公开发表。为了使我国的燃气透平尽快采用弯扭叶片，迫切需要开发一种适用于这类叶片的设计方法。在进行弯扭叶片的设计时，需要解决以下几个问题：

1. 建立一个适用于弯扭叶片设计的正问题计算方法。采用弯扭叶片主要是靠流面形状来控制压力的分布，控制边界层的发展及分离，从而达到减少能量损失的目的[1,2]。如采用反问题计算方法则很难保证流面形状，特别是合理的弯扭叶片侧型面的形状。因而目前迫切需要开发一个适用于弯扭叶片设计的正问题计算方法。

2. 解决跨音速流场的求解问题。当进行正问题计算时，只要 S_2 流面上的马赫数大于 1，就得考虑由于跨音速流动产生的混合型方程、激波和密度双值问题。

3. 解决 S_2 流面正问题计算和叶片造型一体化问题。因为 S_2 流面正问题计算本质上是一种校核计算，在设计中 S_2 流面的选择和叶片造型必须交叉进行，这样才能形成适用于弯扭叶片的以正问题计算为主的设计方法。

本文重点讨论了前两个问题（第三个问题另文[3]讨论），并通过一台燃气透平的改型设计，探讨了不同的叶片弯曲方法对气流参数沿叶高分布的影响。

二、S_2 流面跨音速正问题的计算方法

1. 主方程　在绝热定常的条件下非正交曲线坐标系（如图 1）中的 S_2 流面流函数方程为[4]：

$$A_1\partial^2\psi/\partial(x^2)^2 + A_2\partial^2\psi/\partial x^1\partial x^2 + A_3\partial^2\psi/\partial(x^1)^2 + A_4\partial\psi/\partial x^2 + A_5\partial\psi/\partial x^1 = A_6 + F_r \quad (1)$$

式中系数 $A_1 \sim A_5$ 参见文献 [4]，而 A_6 和 F_r 分别表示为：

$$A_6 = \sqrt{a_{11}}/W^1 \cdot [(-W^\varphi/r)\cdot\partial(V_u r)/\partial x^2 + \partial I/\partial x^2 - T\partial s/\partial x^2] \quad (2)$$

$$F_r = [\partial(V_u r)/\partial x^1 + (\sqrt{a_{11}}/\sqrt{a_{22}})(W^2/W^1)\partial(V_u r)/\partial x^2]\cdot\partial\psi/\partial x^2 \quad (3)$$

本文于 1991 年 3 月收到。　** 哈尔滨工业大学动力工程系　150006

对正问题,方程(1)中的环量 $V_u r = (\omega + W_u) r$,它需根据流面形状进行迭代计算。W_u 由关系式 $r\partial\varphi/\partial z = W_u/W_z$ 求出,其中 $W_z = W^1\cos\theta_1 - W^2\sin\theta_1$。

采用三点微分公式将主方程(1)离散,用线松驰迭代法求解所得的差分方程[4]。

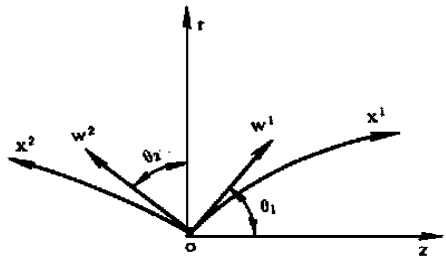

图 1　坐标系速度分量和 r、z 坐标的关系

2. 人工可压缩性及速度积分　为了保证超音区的计算稳定收敛,引入人工可压缩性概念[5],对计算网格 (j, k) 处的密度进行人工修正:

$$\tilde{\rho}_{j,k} = \rho_{j,k} - \mu_{j,k}\Big[(W^1/W)_{j,k}(\rho_{j,k} - \rho_{j-1,k}) + (W^2/W)_{j,k}(\rho_{j,k} - \rho_{j,k-1})(\sqrt{a_{22}}/\sqrt{a_{11}})_{j,k}\Big] \quad (\text{当 } W^2 > 0 \text{ 时}) \tag{4}$$

$$\tilde{\rho}_{j,k} = \rho_{j,k} - \mu_{j,k}\Big[(W^1/W)_{j,k}(\rho_{j,k} - \rho_{j-1,k}) - (W^2/W)_{j,k}(\rho_{j,k} - \rho_{j,k+1})(\sqrt{a_{22}}/\sqrt{a_{11}})_{j,k}\Big] \quad (\text{当 } W^2 < 0 \text{ 时}) \tag{5}$$

其中人工粘性系数 $\mu_{j,k} = C\cdot\max[0, (1 - 1/(Ma_{j,k})^2)]$。

为避免跨音速计算中出现的密度双值问题,使用速度积分方法求速度。将主方程(1)稍加变形后沿 x^2 线积分得到:

$$(W^1)_{j,k} = \frac{(\sqrt{a_{11}})_{j,k-1}}{(\sqrt{a_{11}})_{j,k}}(W^1)_{j,k-1} + \frac{1}{(\sqrt{a_{11}})_{j,k}}\int_{(x^2)_{j,k-1}}^{(x^2)_{j,k}}\left\{-A_3\frac{\partial^2\psi}{\partial(x^1)^2} - A_2\frac{\partial^2\psi}{\partial x^1\partial x^2} - A_5\frac{\partial\psi}{\partial x^1} + \frac{\partial}{\partial x^1}\left[\frac{\sin(\theta_1 - \theta_2)}{\rho f}\right]\frac{\partial\psi}{\partial x^2} + A_6 + F_r\right\}dx^2 \tag{6}$$

式中 f 代表平均 S_2 流面的厚度,采用 SIMPSON 数值积分法求积分,积分起始线选在亚音区(叶片根部或顶部)。然后由从流函数 ψ 的定义式导出的关系式求出 W^2:

$$(W^2)_{j,k} = -\Big[(\sqrt{a_{22}}/\sqrt{a_{11}})(\partial\psi/\partial x^2)/(\partial\psi/\partial x^1)\Big]_{j,k}(W^1)_{j,k} \tag{7}$$

最后得到

$$(W)^2 = (W^1)^2 + (W^2)^2 + 2\sin(\theta_1 - \theta_2)W^1W^2 + (W_u)^2 \tag{8}$$

$$\rho = \rho_i(T/T_i)e^{-(s-s_i)} = \{\rho_i[1 + (1/2)\omega^2r^2 - (1/2)(W)^2]/H_i\}\cdot e^{-(s-s_i)} \tag{9}$$

式(9)是沿流线积分吉伯斯恒等式而得出的。这样就确定了整个流场的 W 和 ρ 的分布。

3. S_2 流面的确定　本文使用的是平均 S_2 流面,叶片段内平均 S_2 流面形状用叶片中弧面代替。若以 $y_{j,k}$ 表示任一高度截面上某一计算站处的叶型中弧线的周向坐标,则表示流面形状的 φ 值可按 $\varphi_{j,k} = \mathrm{arctg}(y_{j,k}/h_{j,k})$ 来确定,其中 $h_{j,k}$ 为该截面的高度值。

为使 S_2 流面的形状合理,由上述方法确定的 φ 值需要修正。首先应使叶片尾缘附近的流面形状与给定的有效几何出气角保持一致。为此,可修改尾部附近的中弧面坐标然后据出口 Ma 数再对流面形状进行考虑落后角或偏转角的修正。

4. 计算网格及边界条件　沿径向网格点数取为 21,叶片槽道内沿轴向分为 10 站,轴向间隙内取 5 站。第一级静叶的前面和末级动叶的后面取尽可能多的计算站数。主方程(1)的边界条件取为:在流道内外壳处的流函数值分别为零和入口流量值;入口截面处流函数沿径向按均匀来流分布或由前一级出口状态确定;出口截面上的流函数满足简单径向平衡方程。

三、算例及分析

根据上述的计算方法，我们编制了多级燃气透平 S_2 流面跨音速正问题的计算程序。应用该程序对某型单级燃气透平（常规扭曲叶片，由常规设计方法设计）进行了验算，并在其他条件不变的前提下将静叶沿叶高进行了不同方案的弯曲。

目前，需从大量动态实验结果中归纳出的弯扭叶片的精确损失模型和落后角模型还没有建立起来（至少是没有公开发表）。本文使用的损失模型是建立在经邓纳梅修正的安利损失模型的基础上的。对常规叶片而言，用邓纳梅修正的安利损失模型求出的损失偏低。而弯扭叶片具有降低损失的优点，所以以这个损失模型为基础计算弯扭叶片的损失具有一定的合理性。然而弯扭叶片的损失分布又有其自身的特点，本文参照现有的弯曲叶片实验结果[1, 2]，适当修改了由该模型算出的二次流损失的大小和沿径分布规律。文献 [2] 等的实验结果表明，直叶片变弯后气流角沿整个叶高更趋于叶栅的几何出气角。因而本文计算中的弯扭叶片的出口气流角仍以喉道宽度与节距比来确定。

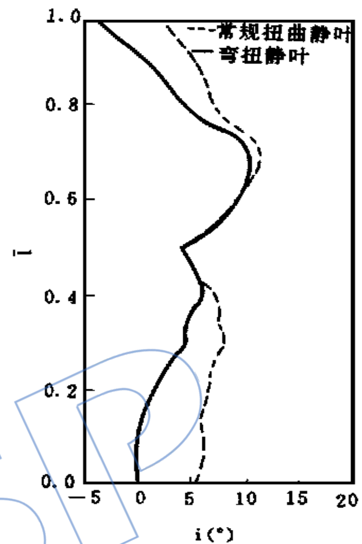

图 2　动叶冲角沿径向分布

图 2 示出了计算的动叶入口冲角沿叶高的分布曲线，虚线对应常规静叶片；实线对应弯曲静叶片（弯叶片倾角：根部 20°，顶部 10°）。造成图中所示的较大的冲角的原因是由于按常规计算方法得到的叶片入口气流角与正问题计算结果不一致。这个“不一致”主要是由于气流在实际叶片流道中的环量分布与反问题计算时

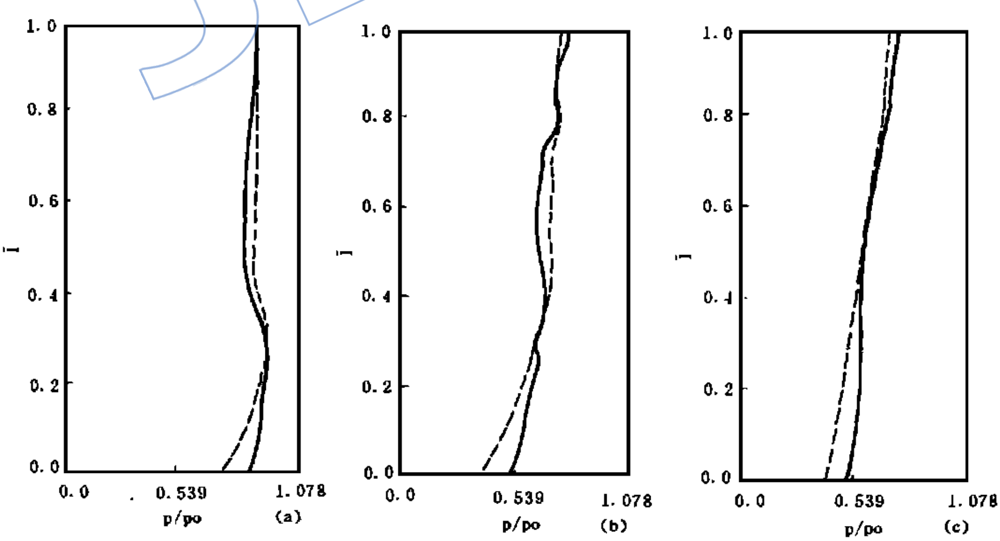

图 3　静叶流道内静压沿叶高的分布（—弯叶；—常规扭曲 a. 25%）

所选择的环量分布的差异引起的。为了减少动叶入口冲角，应在动叶不改动的条件下应用正问题计算程序，适当修改静叶片，尽可能使动叶入口气流角与反问题计算值相一致。这个计算结果说明，在进行透平设计时，用正问题计算方法进行校核计算是必要的。如果发现与反问题计算结果差别较大，应进行改型设计，以求设计结果能真实反映实际流动。

图 3 给出了在相同条件下常规扭曲静叶与弯扭静叶(弯叶片倾角：根部 20°，顶部 10°) 流道中的静压沿叶高的分布曲线，(*a*)、(*b*) 分别是 25%、90% 轴向张的，(*c*) 为静叶出口。可以看出，弯扭静叶片根部压力较常规扭曲静叶片的根部压力有所提高，相应地根部压力梯度也有所减小。这样的压力分布对改善根部流动状况、减少损失是有利的，但对降低流道内能量损失并不很理想，主要是因为原设计子午流道形状和 α_1 角沿叶高的变化与叶片弯曲不匹配。

图 4 示出了改变静叶子午通道型线和 α_1 角沿叶高的变化规律 (即采用可控子午型面和受控涡设计方法) 后的弯扭静叶出口静压分布曲线[3]。图中所示的根部负压力梯度和顶部正压力梯度，对于改善静叶根、顶部流动状况是大有裨益的[2]。为了改善叶片根部流动状况，弯曲叶片在根部应向压力面倾斜。如果叶片向吸力面倾斜 (即反弯) 的话，就会恶化根部流动状况 (如图 4 所示，并且计算得到的根部反力度为负值)。

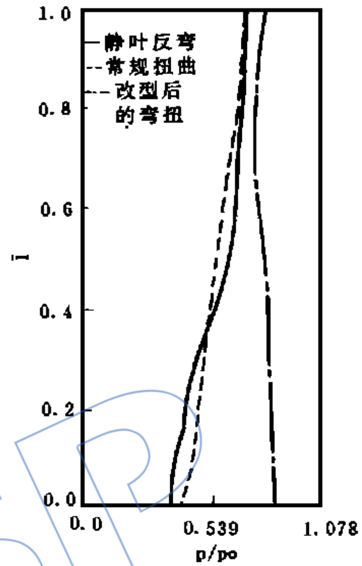

图 4　反弯、常规扭曲及改型后的弯扭静叶出口静压沿叶高的分布

四、结　　论

1. 对于采用弯扭叶片的燃气透平设计问题应当用以 S_2 流面正问题计算为主的设计方法。

2. 采用弯扭叶片可以比较有效地控制流道内的压力沿叶高的分布，因而较好地组织叶片流道内的流动。但叶片的弯、扭与子午面型线三者必须匹配，单使叶片弯曲可能得不到好的流型。

3. 本文的算例说明，采用常规方法设计的燃气透平在设计工况下运行就可能存在各参数不匹配的问题。因此，用正问题计算方法进行校核计算是必要的。

参　考　文　献

[1]　Hourmouziadis J. and Hübner N., "3-D Design of Turbine Airfoils", ASME Paper 85-GT-188, 1985。

[2]　王仲奇等，"在低展弦比透平静叶栅中叶片的弯曲作用"，工程热物理学报，Vol. 4，No. 3，1990。

[3]　冯国泰等，"获得综合优化流型的叶片成型方法和设计方法"，中国工程热物理学会第七届年会，No. 902032，1990。

[4]　朱荣国，"使用非正交曲线坐标与速度分量 S_2 流面反问题流场线松驰解"，工程热物理学报，Vol. 1，No. 1，1980。

[5]　赵晓路，"使用非正交曲线坐标与速度分量 S_2 流面跨音速流函数解"，工程热物理学报，Vol. 6，No. 4，1985。

distortion, the vortex size must be measured. It is defined classically by velocity crelation and more directly than by total pressure. Some typical results are shown in a table. The vortex size is increased by the flow velocity. At 1D section the vortex size of torus with 20 degree slanted cut is about two times larger than the size of 90 degree torus quadrant. But at 3D section the former decreases and the later increases slightly.

EXERGY ANALYSIS OF AEROENGINE AND ITS SIGNIFICANCE

Li Ruhui and Qiu Xiaomei

(*Beijing University of Aeronautics and Astronautics*, *Beijing* 100083)

ABSTRACT An exergy analysis method of an aeroengine is presented and compared with the traditional energy analysis method. The significance of the exergy analysis for the aeroengine engineering is considered. It must be underlined that the main part of the energy loss in the aeroengine comes out of the combustor. Therefore, to enhance of the exergy efficiency of the combustor is an essential measure to improve aeroengine performance.

STATISICAL ANALYSIS AND PREDICTION OF AEROENGINE DETERIORATION

Li Jia and Tao Zengyuan

(*Air Force Engineering College*, *Xian* 710038)

ABSTRACT An aeroengine deteriorates in the course of its service. Although its perfomance may be restored to some extent after overhaul, it goes down gradually in general. Based on the statistical information analysis of the test-bed performance decay of 500 engines after 0 to 3 overhauls, a concept of an overall performance index is proposed to reflect the overall performance of the engine. According to this concept, a mathematical model of the engine deterioration is presented which is established by means of the exponential smoothing theory. The prediction of the engine deterioration has been made whcih is quite consistent with the practice. The final analysis shows that it is necessary and also feasible to adopt the method of statistical and synthetic analysis for the overall engine performance decay of both randomness and regularity and, what is more, with many parameters.

A DIRECT-PROBLEM CALCULATION METHOD FOR GAS TURBINES WITH BOWED AND TWISTED BLADES

Wang Zhongqi, Yang Hong, Liu Fengjun, Huang Yinghong, Feng Guotai

(*Harbin Institute of Technology*, *Harbin* 150006)

ABSTRACT The necessity of the direct-problem calculation in aerodynamic design of gas turbines with bowed and twisted blades is discussed and a practical direct-problem calculation program is developed for the S_2 stream surface transonic flow in a multi-stage gas turbine. In the calculation, the shape of the mean S_2 stream surface is taken as that of the blade mean camber surface which is modified properly to guarantee a reasonable S_2 stream surface shape, and the circulation is specified by the stream surface shape and the axial velocity. In the supersonic region, the density is corrected by means of the artificial compressibility treatment to make the calculation stable and the velocity is solved by density.

This calculating method is applied to modification of a gas turbine. Meanwhile, the effects of different blade-bowing methods on the radial distribution of gas parameters are analyzed. The calculation results demonstrate the improvement on the flow pattern in the turbine cascade with bowed and twisted blades and the necessity of the proof calculation with the direct-problem method. Finally, several instructive opinions about the aerodynamic design of the bowed and twisted turbine blades are proposed.

A CIRCUMFERENTIAL NON-UNIFORM EFFECT MODEL FOR MULTISTAGE AXIAL-FLOW COMPRESSOR THROUGHFLOW

Li Shiming and Chen Maozhang

(*Beijing University of Aeronautics and Astronautics*, *Beijing* 100083)

ABSTRACT The effects of momentum and energy transportation caused by circumferential non-uniformities of the flow in a multistage axial compressor are studied. A model has been developed to represent the circumferential non-uniform stresses and heat flux. A preliminary comparison between experimental results and the model demonstrates that this model is feasible. This study comes to the conclusions that: 1. The momentum transport effects of circumferential non-uniformities are not distinct, but the energy transport effects of the non-uniformities are quite strong and have the same order of magnitude as the turbulent diffusion; 2. The relative importances of the turbulent diffusion and the non-uniformities depend on compressor configuration, loadings and spanwise positions in the compressor; 3. Compared with the effects of their main stream counterparts, the circumferential non-uniform stresses can be sometimes neglected, according to the compressor loadings, configuration and the calculation tolerences as well.

VIBRATION OF GEAR SYSTEM WITH WEB STRUCTURE

Fang Zongde, He Dawei, Shen Yunwen

(*Northwestern Polytechnical University*, *Xian* 710072)

ABSTRACT The web structure has been widely used in design of aviation gears, and its structure vibration has attracted much attention. But the function of web as a torsion spring in the dynamic behaviour of gear system needs a thorough investigation still. In this paper, a typical gear transmission is taken for example, its dynamic model has been built up to analyze the vibration mode of the system. The dynamic responses of this system at various speeds have been obtained with Fourier method by considering the transmission error of the pair of gears as excitation. Further, the influences of web flexibility and attached damping on the dynamic responses have been studied, the dynamic loads on the teeth when the web thickness is changed, and the damping, both attached and unattached, have been calculated respectively. Some useful conclusions have been drawn which show that the combination of flexible web and attached damping can lower the vibration level and decrease the dynamic loads obviously.