

亚音速复合倾斜透平静叶栅 的理论实验研究

中国科学院工程热物理研究所
哈尔滨汽轮机厂

蒋洪德*
陆永良 周凡贞 王松君

【摘要】 采用三维全势函数多重网格数值方法求解亚音速透平静叶栅流场。以 300MW 汽轮机调节级静叶栅的设计为目标, 采用子午面上外环壁收缩和叶片周向复合倾斜两项措施, 以求大幅度地减少叶栅二次流损失。实验研究表明新设计叶栅的损失系数比直叶栅降低 42%, 测出的叶片表面静压分布与理论计算结果十分吻合。

关键词: 全势函数 多重网格 亚音速静叶栅 子午面顶部收缩 周向复合倾斜

1 前 言

大功率汽轮机调节级损失偏大是高压缸热效率较低的重要原因之一。众周所知, 调节级容积流量小而工作压差大, 由于强度和振动安全性的要求, 叶片都较宽, 因而相对叶高都很小。在这种情况下, 叶片通道内的二次流相当强烈甚至可充塞整个流道, 从而造成大的二次流损失。本文基于对哈尔滨汽轮机厂 300MW 汽轮机调节级静叶栅的理论实验研究, 给出了亚音速大折转透平静叶栅采用子午面顶部收缩和周向复合倾斜以大幅度地减少二次流损失的途径。实验研究已表明, 当环形叶栅的径高比和叶片型线确定后, 叶片的相对高度(即展弦比 l/b) 是影响叶栅损失的主要因素。对于长叶片, 二次流引起的高损失区分布在通道两端, 中部的叶型损失一般较小。当叶片相对高度减小时, 端部高损失区占的比例逐渐加大, 当相对高度小于某一数值时(通常为 $0.6 \sim 0.7^{[1]}$), 两端高损失区联成一体, 使环形叶栅的损失系数大幅度上升, 这时叶片型面损失在总损失中退居次要地位。300MW 汽轮机调节级静叶栅相对高度为 0.26, 二次流损失严重。如何降低其损失提高效率是 300MW 汽轮机优化设计的一项重要任务。

目前国内外汽轮机设计中为减少调节级静叶栅损失所采用的技术措施有: (1) 采用带加强筋结构的分流叶栅, 或增大叶型的断面抗弯模数, 以加大相对叶高; (2) 子午面成型; (3) 倾斜叶片。由于结构和强度方面的限制, 相对叶高增大的幅度是有限的, 本文采用后两条措施。采用子午面收缩(顶部端壁收缩)的造型可使叶栅流道内压力面与吸力面的横向压差减小, 减轻二次流的强度, 同时流道的收敛使吸力面上沿流向逆压梯度区缩短, 减少附面层粘性损失。文献[2、3]的实验研究表明, 适当选择子午面的收敛度和型线可使透平级效率提高约 1%。

减少透平静叶栅二次流损失更有效的措施是采用周向复合倾斜叶片^[4]。这一新概念近十年来在先进喷气发动机和汽轮机中逐步获得应用并取得了良好的效益^[5]。本文的设计思想是

同时采用子午面收缩和复合倾斜两项措施,以求大幅度地提高透平静叶栅的效率。这种设计在国内尚属首创,在国外已发表的文献中亦未见报道。显然,这种叶栅几何形状比较复杂,流场具有强的全三维特征。本文中 300M W 汽轮机调节级静叶栅的设计按如下方法进行: (1) 用三维全势函数多重网格计算方法对子午面造型和复合倾斜叶片进行优化设计。这一计算方法收敛迅速稳定,对于这类三维特征强、叶片相对高度小的亚音速叶栅流场的计算特别适用。但此计算中未计入粘性效应故无法求得损失系数及其分布。(2) 为了探索和掌握不同设计参数对叶栅损失的影响,在理论计算方案优选的基础上,设计制造不同子午面型线、不同栅距和不同倾斜程度的复合倾斜叶栅,通过吹风实验测得叶栅总损失系数、流量系数和叶片表面静压分布,得到损失沿叶高的分布规律、损失系数与主要设计参数的关系,并验证无粘三维计算的准确程度,从而为建立这类叶栅的经验设计方法打下基础。

2 理论计算方法与无粘三维设计准则

2.1 三维全势函数多重网格数值方法

三维全势函数叶栅流场数值解法的特点是未知函数简单和求解过程稳定,使用椭圆型偏微分方程的多重网格技术可大大加速收敛过程从而便于工程应用。近年来我们对这一方法的主要问题进行了较为深入的研究,要点如下: (1) 在固定或相对旋转的任意曲线坐标系中求解以守恒形式表示的三维全势函数基本方程^[6]; (2) 提出了对三维空间中网格畸变具有良好适应性的高精度差分格式^[6],提出了三维位流解析解^[7]作为校核差分格式和计算程序准确度的依据,保证了计算精度; (3) 基本方程采用三维线松弛和多重网格技术加速收敛^[8],当采用三层以上网格时,可节省机时 40% 以上,便于工程应用; (4) 采用人工密度概念处理局部超音区,使这一方法的适用范围扩大到低跨音流动(流场中最大 Ma 数不超过 1.3)。但由于无旋流动假设的限制,本方法主要适用于来流均匀(无旋)的短叶片三维流场计算。

2.2 无粘三维设计准则

三维全势函数数值解虽然可以方便地求得叶栅内的无粘流场,但无法求得损失及其分布,为此我们提出以下的准则对无粘三维流场进行分析,并据此进行不同设计方案的对比优选。

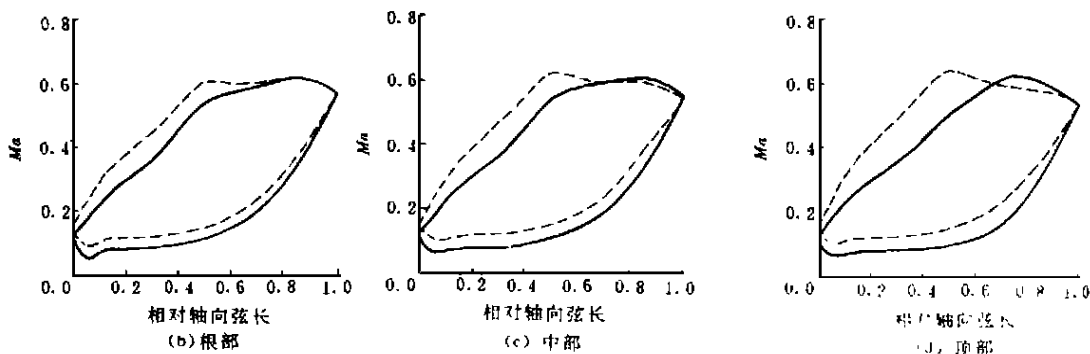
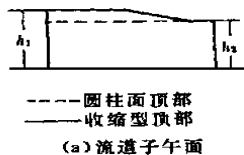


图 1 外环壁收缩对透平静叶栅流场的影响(a. 流道子午面 b、c、d 分别为根、中、顶部的 Ma 分布)

子午面上外环壁的渐缩造型对整个叶栅流道都有影响,但主要影响在流道顶部。选择外环壁的收敛度 $CR = (h_1 - h_2)/h_2$ (图 1a) 和收敛部型线的原则是,既要减小压力面与吸力面的静压差,又要使吸力面后部逆压梯度区的长度缩短且逆压梯度不致太大。

图 1 给出子午面收缩对直叶片环形叶栅性能的影响。对于轴向来流、气流转折角为 60° 以上的透平静叶栅,当子午面收敛度为 $15\sim 20\%$ 时,计算表明叶栅顶部截面的横向压差可降低 $10\sim 20\%$,沿叶高所有截面上吸力面后段逆压梯度区的长度与吸力面长度之比可由 75% 下降到 50% 左右,但逆压梯度的值有少许增加,尤其是在顶部。有关子午面外环壁收缩对三维叶栅流场影响的详细讨论可见文献[8]。

叶片周向复合倾斜(根部正倾斜、顶部负倾斜)对环形叶栅内气流参数的径向分布规律有较大影响,但主要影响表现在叶片根部的吸力面后段。选择复合倾斜型线的原则是,尽量使吸力面后段静压的径向梯度在根部为负值而顶部为正值,同时使流道内径向分速在根部为负值而顶部为正值。有关复合倾斜对三维叶栅流场影响可参见文献[9]。

图 2 给出了复合倾斜叶栅流场特征的示意图(V_r 为径向分速, P 为静压)。复合倾斜造成吸力面后部静压沿径向的“C”型分布使叶片两端高损失区的流体向中部迁移并被主流带走,而主流区径向分速的分布则使主流同时对内外环壁产生“挤压”作用从而抑制了环壁附面层的扩展。这一流动模型已为大量的实验研究所确认^[5],亦为本文的数值计算所证实。

在 300MW 汽轮机调节级静叶栅设计中,基本叶型采用红旗 4 型,进行数十个方案的对比计算以综合考虑上述两个因素的影响。在最终方案中,子午面外环壁采用三次曲线,为简化工艺,复合倾斜型线的根部、顶部为直线,中部为圆弧。

3 实验研究及结果分析

本文的实验研究包括若干叶栅的对比实验,它们是:

- (1) 圆柱面内外环壁直叶栅;
- (2) 外环壁收缩的直叶栅;
- (3) 外环壁收缩的复合倾斜叶栅。考虑到 300MW 汽轮机实际叶栅的径高比大于 30,故实验叶栅采用平面叶栅,吹风实验在哈尔滨汽轮机厂亚音速叶栅试验台上进行。在栅后用探针测量气流方向与总压损失,在叶片表面根、中、顶三个截面测静压,每个截面上吸力面与压力面各有 14 个测孔。有关实验和测量的详细内容可参阅文献[10、11],本文仅给出外环壁收缩情况下直叶栅与复合倾斜叶栅的实验结果。

图 3 给出了两种叶栅损失系数沿叶高的分布。可以看出,对于直叶栅,即使采用子午面收

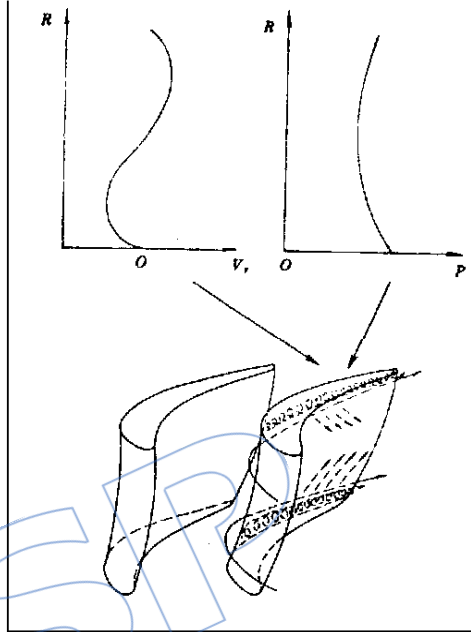


图 2 复合倾斜对透平静叶栅流场特征示意图

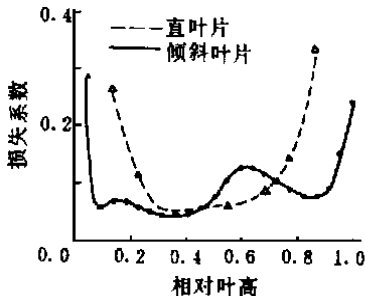


图 3 外环壁收缩直叶栅与复合倾斜叶栅损失系数实验结果

缩结构, 由于相对叶高很小, 损失系数大于 10% 的高损失区仍约占总叶高的 60% (根部约 20%, 顶部约 40%), 叶栅平均损失系数高达 17.5%。而采用复合倾斜设计后, 两端高损失区分别缩小为 4% 和 14%, 值得注意的是在 60% 相对叶高处复合倾斜叶栅出现了一个较高损失区 (最大损失系数 13%), 表明流道内存在某种径向掺混现象。即使如此, 其总损失系数仍比直叶栅下降 42%, 为 10.2%, 相应的流量系数为 96%, 比直叶栅有大幅度提高。

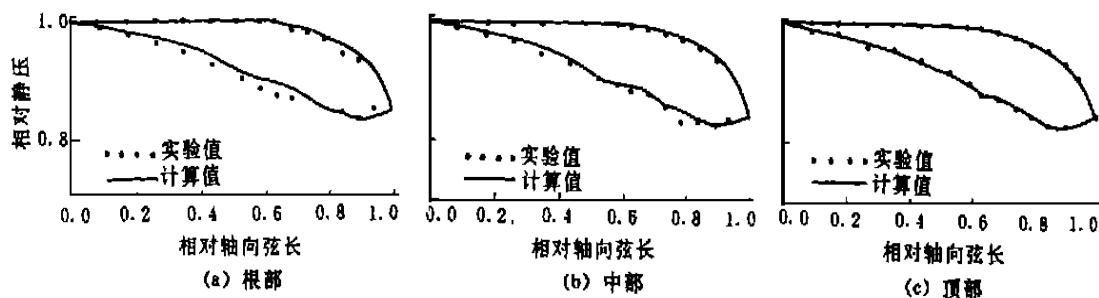


图 4 外环壁收缩直叶栅静压分布计算与实验结果的比较(a、b、c 分别为根部中部和顶部)

外环壁收缩直叶栅根、中、顶三个截面在平均截面出口 Ma 数为 0.5 时的静压分布测量值与计算结果的比较示于图 4。可以看出, 理论计算结果与实验值十分吻合, 从而表明了本计算方法的可靠性。复合倾斜叶片由于埋管困难, 暂未测量叶片表面静压分布。

致谢 本文的理论和实验研究工作是近年来中国科学院工程热物理研究所与哈尔滨汽轮机厂共同创建的热机气动热力学联合研究室从事的科研工作的一部分, 本文的工作始终得到所、厂领导的关心与支持, 任大康、张宏光、陈善作、黄钢、蔡睿贤、朱宇航和何咏梅同志在实验和理论研究方面也做了不少工作, 在此一并致谢。

参 考 文 献

- 1 蔡颐年. 蒸汽轮机. 西安交通大学出版社, 1988 年。
- 2 Dejc E. et al. Atlas of Axial Turbine Blade Characteristic. Machinostroenic Publishing House, Moscow, 1965(CEGB Inf. Services, CE Trans. 4563)
- 3 Boletis E. Effects of Tip Endwall Contouring on the Three Dimensional Flow Field in an Annular Turbine Nozzle Guide Vane, Part 1: Experimental Investigation. ASME 85-GT-71
- 4 Филипов Г. А. И Ван Чжун-цз. Влияние закрутки потока на характеристики оплывых решеток. Теплоэнергетика, 1964, (5)
- 5 王仲奇. 一代新型轴流式透平机械叶片—叶片的弯扭联气合动成型理论及其实验结果. 气动热力学发展战略研讨会专题报告汇编, 1989
- 6 Jiang Hongde (蒋洪德), Cai Ruixian (蔡睿贤). An Accurate Spatial Differencing Scheme with High Suitability to Curvilinear Grids for 3D Full Potential Flow Computation. AIAA Journal, 1986, 24(7)
- 7 蔡睿贤, 蒋洪德, 孙春林. 可用于校验叶轮机械三维数值解的某些位流解析解. 工程热物理学报, 1984, 5(4)
- 8 蒋洪德, 蔡睿贤. 三维势函数多重网格数值方法的研究与应用. 中国工程热物理学会第七届年会论文 902056, 1990
- 9 蒋洪德. 倾斜透平静叶栅的全三元流场分析. 工程热物理学报, 1989, 10(1)
- 10 周凡贞. 300MW 机组调节级喷嘴流量系数试验研究. 哈尔滨汽轮机厂设计所, 1989
- 11 周凡贞. 300MW 机组调节级喷嘴子午面型线研究报告. 哈尔滨汽轮机厂设计所, 1990

based on this approach. It gives satisfactory results of the flow field computation and is also very simple and practical. The subprogram can meet all requirements of aerodynamic design and analysis for an advanced axial-flow compressor.

EXPERIMENTAL STUDY ON UNSTABLE BEHAVIOR OF AXIAL COMPRESSION SYSTEM

Li Jun (*Air Force Engineering College*)

Liu Zhiwei (*Northwestern Polytechnical University*)

ABSTRACT

Unstable behavior of an axial-flow single-stage compressor has been researched successfully by use of a simple dynamic measuring system. The character and the influence factors of the unstable behavior have been analyzed from systematic view. The experimental results demonstrate that the unstable types of the compressor could convert each other in certain conditions. The compressor rotor speed, the construction and size of the exhaust parts are the main factors influencing the conversion. The surge occurs in the compressor on condition of high rotor speed or large exhaust volume, otherwise the rotating stall appears. The decrease of the exhaust volume leads to increment of the surge frequency. There is an obvious difference of the unstable behavior between the single-stage compressor and the isolated rotor compared. In the condition of high rotor speed ($\bar{n} > 0.85$), a stall phenomena of nonrecoverable stall appears. Its character is also considered.

EXPERIMENTAL INVESTIGATION ON WHIRL FLUTTER OF PROPELLER ROTOR SYSTEM

Li Zhiwei, Xu Ming, Ren Xingmin

(*Northwestern Polytechnical University*)

ABSTRACT

A whirl flutter of a propeller rotor system has been investigated experimentally in the wind tunnel. The design of whirl flutter experimental equipment and a dynamic stiffness coefficient measurement method for the supporting system are also studied. The experiments show that if the support stiffness of the propeller rotor system is reduced as a result of failure of some elements in the engine frame or mount, the threshold flight velocity of onset whirl flutter will be decreased. The results of its wind tunnel experiments exhibit the same trends as in its theoretical calculation.

STUDY OF A SUBSONIC COMBINED-LEANING TURBINE GUIDE VANE WITH TIP ENDWALL CONTOURING

Jiang Hongde

(*Institute of Engineering Thermophysics, Chinese Academy of Sciences*)

Lu Yongliang, Zhou Fanzen, Wang Songjun

(*Harbine Steam Turbine Work*)

ABSTRACT

A three-dimensional, multigrid potential flow computation method is developed to analyse subsonic flow in steam turbine nozzle guide vane. In order to reduce the large secondary losses in the first annular nozzle guide vane with very small aspect ratio of 0.26 in a 300MW steam turbine, both tip endwall contouring and combined-lean blades are used in the new design. Experimental investigation shows that the mass-weighted loss coefficient of the new nozzle guide vane is 42 percent less than traditional radial one. The agreement between the measured and calculated results of the pressure distribution on the

blade surface in the annular nozzle guide vane is very good. The present study is a success in developing fully 3D steam turbine nozzle guide vane with tip endwall contouring and combined-leaning blade.

SUPERSONIC UNSTALLED FLUTTER IN ARBITRARY MISTUNED CASCADES

Li Qianwu (*National University of Defence Technology*)

Hen Qun and Fan Feida (*Northwestern Polytechnical University*)

ABSTRACT

By the use of Hamilton's principle, a set of aeroelastic equations is formulated to describe the motion of an arbitrary mistuned rotating cascade with high aspect-ratio, flexible pretwisted, nonuniformed blades. A two-dimensional aerodynamic theory is used to calculate unsteady force and moment coefficients for vibrating aerofoil in supersonic unstalled cascade. This set of equations is of great value to a fan blade with the inertial, structural and aerodynamic coupling in bending-bending-torsion. Results obtained from the aeroelastic stability analysis of a real fan stage show that amplitude and different order of mistuning and other kinds of structural properties have an apparent effect on boundaries of supersonic unstalled flutter.

RESEARCH ON 3-D DISCHARGE FLOW IN A CENTRIFUGAL IMPELLER

Xi Guang and Wang Shangjin (*Xi'an Jiaotong University*)

ABSTRACT

A method for calculating 3-D discharge flow in a centrifugal impeller is developed by means of solving the incompressible steady time-averaged N-S equations, the continuity equation and the $k-\epsilon$ turbulence model equation. This method is accomplished with calculation grids generated by an algebraic transformation, and thus it can be easily applied to the design analysis of centrifugal impellers. The discharge flow of a three-dimensional shrouded centrifugal compressor impeller was measured for assessing the validity of the present calculation procedure. The comparison between the experimental data and the calculation results shows that the present method can predict the mean velocities.

NUMERICAL STUDY ON BLADE-TO-BLADE PASSAGE FLOW FIELD IN A CENTRIFUGAL CASCADE

Wu Xiaosong and Wu Guochuan

(*Nanjing Aeronautical Institute*)

ABSTRACT

Numerical simulations of the blade-to-blade flow field in a centrifugal impeller have been carried on by using 2-D mean steady incompressible Navier-Stokes equations and $k-\epsilon$ turbulent model in arbitrary boundary-fitted coordinate system. It is shown that the underrelaxation factors of line-by-line iteration are lower than those in non-rotational flow calculations and a more reasonable initial flow field is required due to the larger ratio of centrifugal forces in the source terms of momentum equations. The calculation results of the flow fields are in a good agreement with the LDA data with the exception of the near-wall region of the blade surfaces. Some of the discrepancies in the near-wall region are caused mainly by the assumption of 2-D flow and ignoring the rotational effects in the turbulence model. Additionally, the calculation precision is also affected, since inlet and outlet boundaries of the computational domain cannot be located so far away from the blades.