文章编号: 1001－2060(2007)05－0484－06

导叶预旋角对叶轮/扩压器相干的影响

周 莉[1], 席 光[2], 蔡元虎[1]
(1. 西北工业大学 动力与能源学院, 陕西 西安 710072; 2. 西安交通大学能源与动力工程学院, 陕西 西安 710049)

摘 要: 在不同进口导叶预旋角度下, 采用非定常的方法对进口导叶/叶轮/扩压器 3 部件之间非定常相干进行了数值模拟, 并与实验结果作比较。研究了进口导叶预旋角度对叶轮及扩压器内部流动及非定常性的影响, 同时探讨了 3 部件之间动静相干的机理。结果表明, 计算结果同实验结果吻合很好。在同等流量下, 进口导叶的尾流和大尺寸涡团的非定常作用使得叶轮和扩压器上的非定常性减小, 其变化弧度仅在原来的 1/4 左右。进口导叶为负预旋时, 由进口导叶尾流所带来的非定常影响比进口导叶正预旋角度下小的多, 但叶轮尾流及扩压器的势反冲效应所引起的非定常效应却较正预旋角度下有所增大。

关 键 词: 叶轮机械; 进口导叶; 叶轮; 扩压器; 非定常相干; 预旋角度; 非定常性

中图分类号: TK474.8 文献标识码: A

引 言

叶轮机械内部, 动/静叶片排之间相对运动所引起的非定常性会影响机器的效率、工况范围和运行稳定性等气动性能。近些年来, 研究者们对叶轮机械内部的非定常流动进行了大量的研究, 而且至今还是研究的热点。叶轮机械动/静相干非定常流动的研究工作较多的集中在轴流式叶轮机械方面[1~4]。径流式叶轮机械方面的研究工作相对较少, 而且主要研究叶轮与扩压器叶片之间的相干非定常流动[5~6]。而已有进口可调导叶方面的工作主要是采用进口可变导叶改变来流预旋条件, 以提高机器的总体气动性能[7~8]。目前有关进口导叶尾流尤其是大尺寸分离涡团与叶轮及扩压器相干非定常作用机制的认识还很浮浅, 开展离心压气机进口导叶与叶轮及其叶片扩压器之间的相干非定常作用机理的研究, 掌握导叶尾迹及大尺寸分离涡团对叶轮及扩压器叶片产生非定常激振的规律, 可为提高大型离心压缩机的设计和运行可靠性提供理论基础。本文基于此原因, 在不同进口导叶预旋角度下, 对进口导叶/叶轮/扩压器 3 部件之间非定常相干进行了数值模拟, 研究了进口导叶预旋角度对叶轮及扩压器内部流动及非定常性的影响。

1 数值计算方法

1.1 基本控制方程及数值算法

数值计算采用商用 Numeca 软件包, 求解三维非定常 Navier-Stokes 方程组; 计算中采用了 Baldwin-Lomax 的混合长度理论模型来求得紊流粘性系数; 空间差分采用中心差分格式; 时间项采用四阶 Runge-Kutta 法迭代求解。计算中使用全多重网格方法, 结合隐式残差光顺方法及当地时间步长法, 以获得最快的收敛速度[9]。

1.2 边界条件

计算中给定了整个级的进出口条件。在进口导叶进口给定总温、总压及绝对气流角; 在扩压器出口给定出口静压。上下游延伸部分的周向边界采用周期性边界条件。对于固体壁面, 取不可渗透、无滑移及绝热壁面边界条件。定常计算中采用混合平面直接在动静部件的交界面上对参数进行周向平均, 非定常流计算则利用滑移界面方法在界面上插值以进行参数的传递。

2 计算模型

以某一离心压缩机级(进口导叶＋叶轮＋叶片扩压器)为研究对象, 对其级环境下的流动进行了数值模拟。离心压缩机级的主要几何参数如表 1 所示。设计转速为 3 000 r/min, 设计进口质量流量为 2.75 kg/s。为了使计算所用各叶片排的通道数尽量少, 把进口导叶的叶片数由 15 调整到 14, 叶轮的叶

收稿日期: 2006－11－10; 修订日期: 2007－01－29
基金项目: 国家自然科学基金资助项目(50576073); 中国博士后科研基金资助项目(20070411139)
作者简介: 周 莉(1978－), 女, 河南信阳人, 西北工业大学讲师, 博士后。

片数由 20 调整到 21, 扩压器的叶片数由 13 改变至 14。这样根据区域缩放的原则, 计算在 2 个进口导叶通道、3 个叶轮通道和 2 个扩压器通道内进行。每个通道的网格数均为 41× 33× 81, 其中沿流向有 81 个计算站, 周向有 33 个计算站, 展向有 41 个。交界面位于动静部件中间的位置上。把叶轮通过两个扩压器通道所用的时间作为一个周期, 在进行非定常计算时, 每个周期内设定 90 个物理时间步, 每个物理时间步下进行 30 次的虚拟时间步的内迭代。非定常计算在 P4 1. 6G 的微型计算机上进行, 非定常计算以定常计算的收敛结果为初始解, 计算在进行 25 个周期后, 所监控点的压力开始呈现出周期性的分布, 此时认为计算结果收敛。计算所采用的三维网格如图 1 所示。

表 1　离心压缩机级的主要几何参数

	进口导叶	叶轮	扩压器
进口直径/mm	222	222	940
进口叶片高度/mm	—	103. 91	43. 8
出口直径/mm	422	796	1 250
出口叶片高度/mm	—	43. 8	43. 8
叶片数	15	20	13

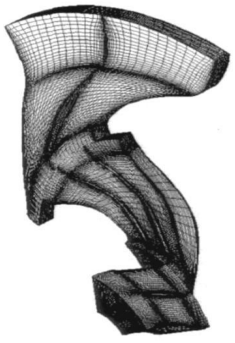


图 1　计算网格图

3　计算结果及分析

图 2 给出了非定常计算收敛后在各部件叶片附近所监控点的压力值在一个周期的变化。由图可知, 在每个周期内, 进口导叶尾缘附近和扩压器叶片前缘附近的压力值经历了 3 次明显的周期性变化, 这是因为在一个周期内有 3 个叶轮叶片尾迹通过的缘故。同样, 对于叶轮叶片的前缘和尾缘附近的压力, 在每个周期内则经历了两次显著的周期性变化。因而, 在本文分析中, 对于进口导叶和扩压器内变量随时间步变化的分布图, 时间步的取值范围为 t= 1 ~45; 而叶轮中时间步的范围为 t= 1 ~30。

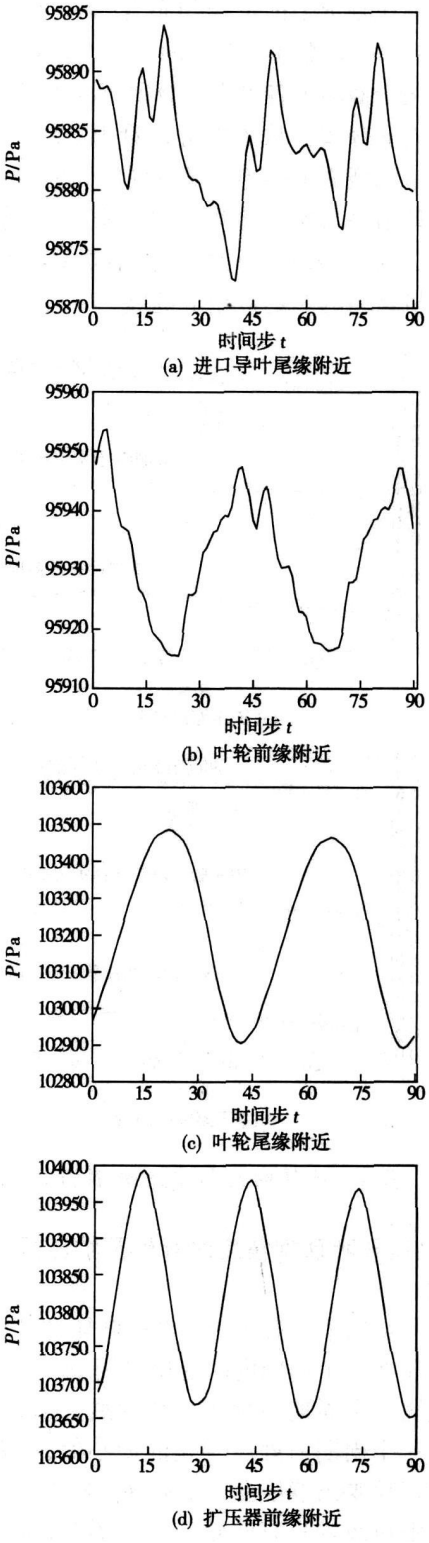
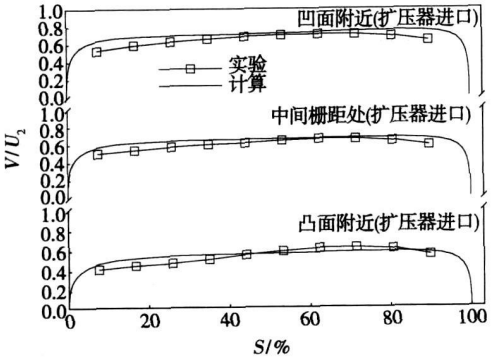


图 2　压力监控图

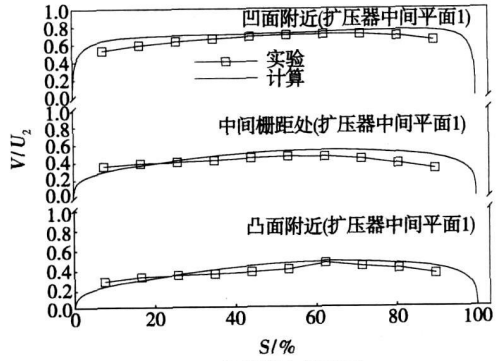
3.1　计算与实验结果的比较分析

为了考核计算结果的可靠性, 把非定常的计算结果同使用热线风速仪测量得到的实验数据进行了

比较。在扩压器进口和中间两个截面上，对两个扩压器通道的速度进行时间平均和空间平均的处理。各个测点在测量平面上均匀布置，第一个测点靠近凹面，最后一个测点靠近凸面。图 3 定量地与实验比较了两个截面上各测点计算得到的无量纲时均速度沿展向的分布。从图中可以看出，在扩压器进口截面和中间截面上的绝大部分周向和轴向测量位置处的计算结果与实验值吻合得非常好，两条曲线几乎重合。因而，采用的计算方法能很好地预示出离心压缩机级内的非定常相干机理。



(a) 扩压器进口平面

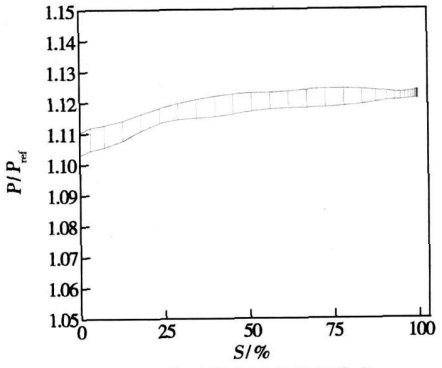


(b) 扩压器中间平面

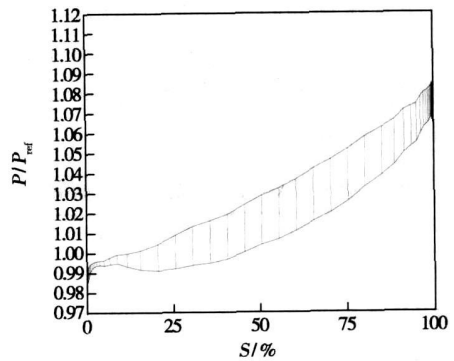
图 3 计算结果与实验数据的对比

3.2 进口导叶预旋角度对叶轮及扩压器非定常性的影响

图 4 给出了没有进口导叶时叶轮和扩压器叶片上的静压在一个周期内的变化包线图。其中上面的曲线代表一个周期内静压变化的最大值，下面的曲线代表一个周期内静压变化的最小值，二者之间的包线可以反映出静压在一个周期内的变化幅度。当离心压缩机级内只有叶轮和扩压器时，从图中可以看出叶轮上静压在叶轮叶片的后半部分变化不大，在前半部分的变化幅度从叶轮中部到前缘不断减小，即扩压器对叶轮的非定常势反冲效应不断减小，而叶轮尾流对扩压器的非定常效应也沿着流向不断减小。



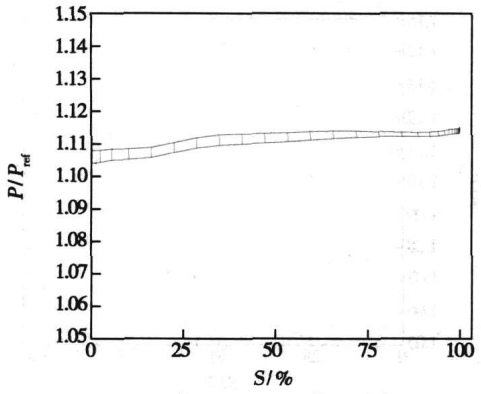
(a) 扩压器凸面的静压分布



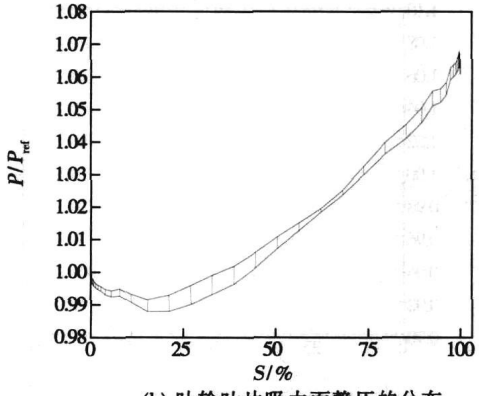
(b) 叶轮叶片吸力面静压的分布

图 4 扩压器和叶轮相干时叶片上的静压分布

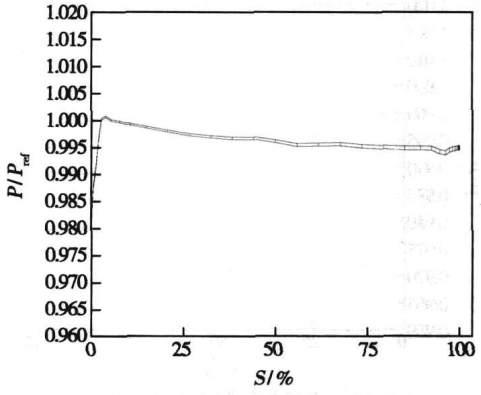
在较大的进口导叶预旋角度下，导叶后面产生了大尺寸的分离涡团，为了研究进口导叶尾流对叶轮及扩压器非定常性的影响，图 5 给出了在相同质量流量下，进口导叶正预旋 30°下 3 部件相干时，静压在一个周期内的变化包线图。由图可见，进口导叶叶片上的静压受下游部件的势反冲效应的影响，其从导叶尾缘到前缘变化的幅度不断减小，在前缘处压力几乎不发生变化。当进口导叶出现较大的预旋角度时，叶轮进口的流动出现攻角，其进口流动的不均匀化程度加重，在叶轮叶片上诱发出周期性变化的冲角。进口导叶的尾流作用使得叶轮叶片上的静压出现从尾缘到前缘不断增加的波动，而扩压器的势反冲作用则使静压的变化幅度从叶轮尾缘到前缘不断减小。由于粘性尾流的作用更加强烈，此时对于叶轮来说进口导叶/叶轮之间的非定常相干占主导作用。进口导叶尾流效应同扩压器势反冲效应的相互综合，使得叶轮上静压的非定常变化幅度减小，进而引起了扩压器进口气流角的波动减小，导致扩压器内静压的振荡幅度减小，因而同没带进口导叶的叶轮和扩压器叶片上静压的非定常变化相比，此时叶轮和扩压器叶片上静压在一个周期内的变化幅度显著减小，其变化弧度仅在原来的 1/4 左右。叶轮由于受进口导叶尾流及分离涡团的非定常影响，



(a) 扩压器凸面的静压分布



(b) 叶轮叶片吸力面静压的分布



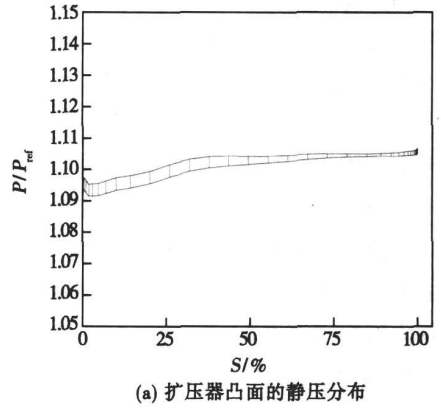
(c) 进口导叶凹面的静压分布

图 5　进口导叶正预旋 30°下 3 部件相干时叶片上的静压分布

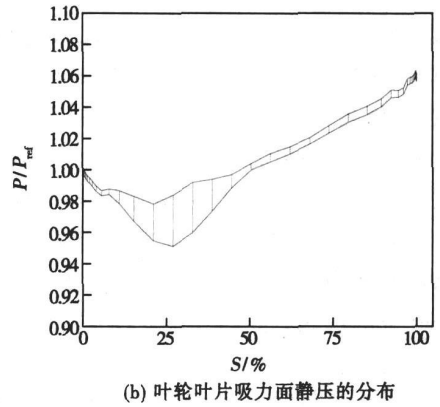
从叶轮前缘到距前缘 50%弧长处, 叶轮叶片上的静压有较大的变化。在叶轮叶片的后半部分, 由于扩压器对叶轮势反冲效应的影响, 叶轮叶片上的静压在一个周期内的变化幅值沿流向不断增大, 在尾缘附近达到最大值。此时叶轮叶片上最小的压力振荡出现在叶片中间附近, 其在一个周期内的变化幅值仅为 107 Pa。同时从叶轮吸力面上静压分布图上还可看出进口导叶尾流对叶轮叶片的非定常影响大于扩压器对叶轮的势反冲效应。在扩压器内, 扩压器叶片也不像仅有叶轮尾流对其非定常的影响那样, 其变化幅值大大减小, 在扩压器叶片上, 静压沿流向的大部分区域都在 380 Pa 左右进行非定常的变化。

3.3　进口导叶正负预旋角度对非定常性的影响程度

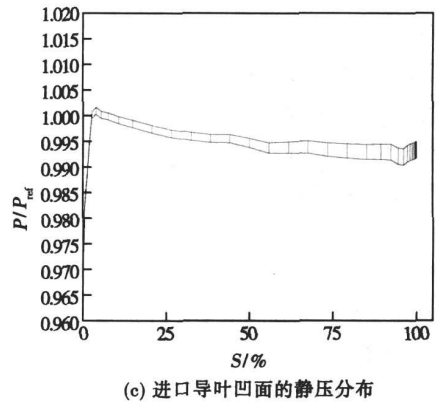
本文还计算了进口导叶 30°正预旋角度下, 离心压缩机在质量流量为 3.2 kg/s 时的非定常流场, 图 6 给出了各个部件叶片上的静压在一个周期内的变化包线图。从图中可以看出, 当质量流量增加时, 即计算工况偏离了设计工况, 非设计工况下流动的脉



(a) 扩压器凸面的静压分布



(b) 叶轮叶片吸力面静压的分布



(c) 进口导叶凹面的静压分布

图 6　质量流量为 3.2 kg/s 时进口导叶正预旋 30°下 3 部件相干时叶片上的静压分布

动增大，进口导叶尾流的非定常性大大增加。进口导叶上静压的变化包线仍然显示出从前缘到尾缘不断增大的趋势，但其最大的静压变化幅度为352 Pa，其值是质量流量为 2.75 kg/s 时压力振荡幅度的 5 倍。叶轮叶片上的静压变化幅度显示出从叶片前缘到尾缘不断减小的趋势，在叶轮前半部分静压有着非常大的变化尺度，其吸力面静压的变化幅度达到上千帕。在叶轮叶片尾缘处的静压的变化幅值也能达到 475 Pa。同质量流量为 2.75 kg/s 时不同的是，在扩压器内静压沿流向不断减小的变化趋势更为显著，在扩压器凸面上静压在距前缘 75%弧长处静压的变化已经非常不明显。由于此时进口导叶/叶轮的非定常相干增大，在进口导叶尾流效应和扩压器势反冲效应的共同作用下，叶轮出口处的非定常波动减小，使得扩压器进口气流角的振荡幅度减小，较质量流量为2.75 kg/s时的分布，叶轮尾流和扩压器势反冲效应的影响略有减小。

为了比较进口导叶正负预旋角度在进口导叶/叶轮/扩压器 3 部件相干中所起的作用，在同一质量流量下，对进口导叶正负预旋 30°时离心压缩机级的非定常相干流场进行了计算。图 7 给出了进口导叶 30°负预旋角度下，离心压缩机在质量流量为 3.2 kg/s时各个部件叶片上的静压在一个周期内的变化包线图。由图可见，当进口导叶为负预旋 30°时，进口导叶叶片后面出现的漩涡方向同叶轮旋转方向一致，其相当于减小了叶轮进口气流的脉动，而正预旋30°下的漩涡方向同叶轮旋转方向相反，同正预旋 30°相比，进口导叶/叶轮的非定常相干作用减小，从静压的变化包线上可以看出，进口导叶上静压的变化幅度大大减小，此时进口导叶叶片上静压在一个周期内变化的最大幅度仅为 67 Pa，而且在其叶片上沿流向的变化规律已不是十分明显。同时导叶尾流对叶轮叶片上静压的非定常影响也减小，在进口导叶尾流效应和扩压器势反冲效应的共同作用下，叶轮出口处的非定常波动增大，使得扩压器进口气流角的振荡幅度增大。在扩压器叶片上，同进口导叶正预旋 30°下的分布相比，负预旋 30°下叶轮尾流对扩压器的非定常影响增大，最大的静压变化幅度比正预旋30°下大 491 Pa，最小的静压变化幅度也相应的大 171 Pa。在叶轮叶片上，负预旋 30°下进口导叶尾流对叶轮的非定常影响大大减小，但扩压器对叶轮的势反冲效应却较正预旋 30°下有所提高，这从叶轮叶片后半部分静压的变化幅值上明显体现出来。

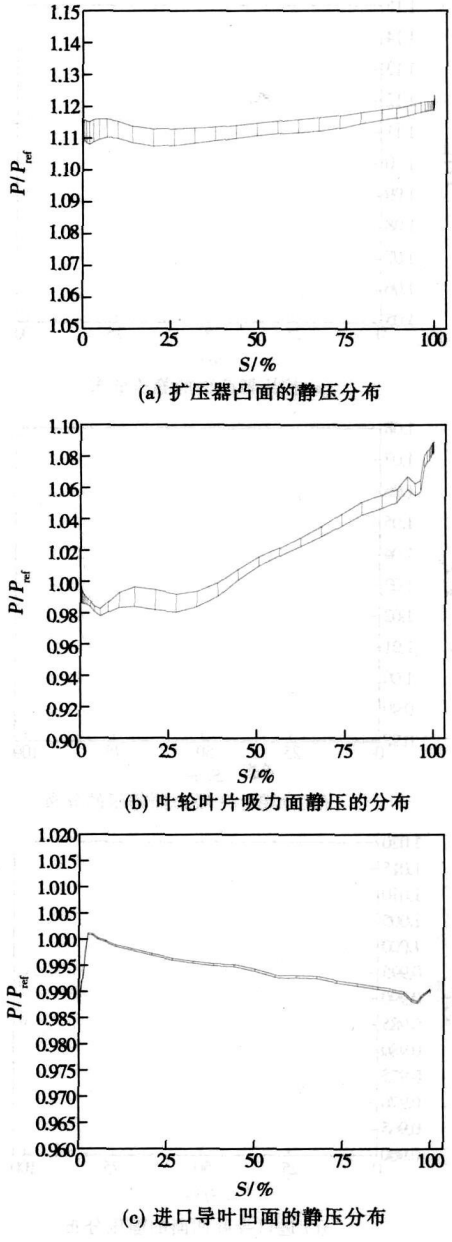


图 7 质量流量为 3.2 kg/s 时进口导叶负预旋 30°下 3 部件相干时叶片上的静压分布

4 结 论

在不同进口导叶预旋角度下，采用非定常的方法对进口导叶/叶轮/扩压器 3 部件之间非定常相干进行了数值模拟，研究其动静相干的机理，得出主要结论：

（1）在非定常计算中，对速度进行时间平均，并同实验比较，发现计算值和实验值的吻合很好。

（2）在同等流量下，当离心压缩机带有进口导叶时，叶轮和扩压器上静压的非定常变化趋势和不带进口导叶时显著不同，且其在一个周期内的变化

幅度显著减小。进口导叶的尾流和大尺寸涡团的非定常作用使得叶轮和扩压器上的非定常性减小，其变化弧度仅在原来的 1/4 左右。

（3）当流量增加时，进口导叶尾流的非定常影响增大，此时在进口导叶叶片和叶轮前半部分的体现极其显著。进口导叶上最大的静压变化幅度能达到小质量流量下压力振荡幅度的 5 倍之大。

（4）在同等流量下，进口导叶为负预旋时，由进口导叶尾流所带来的非定常影响比进口导叶正预旋角度下小的多，但叶轮尾流及扩压器的势反冲效应所引起的非定常效应却较正预旋角度下有所增大。

参考文献：

[1] TREBINJAC I, CHARBONNIER D, LEBOEUF F. Unsteady rotor-stator interaction in high-speed compressor and turbine stages[J]. Journal of Thermal Science, 2005, 14(4): 289—297.

[2] PAYNE C, STEPHEN J, AINSWORTH R W. Unsteady loss in a high-pressure turbine stage: interaction effects[J]. Journal of Heat and Fluid Flow, 2005, 26(5): 695—708.

[3] URBASSIK R M, WOLFF J M, POLANKA M D. Unsteady aerodynamics and interactions between a high-pressure turbine vane and rotor[J]. Journal of Turbomachinery, 2006, 128(1): 35—42.

[4] KULISA P, DANO C. Numerical simulation of unsteady blade row interactions induced by passing wakes[J]. European Journal of Mechanics, B/Fluids, 2006, 25(3): 379—392.

[5] ZHOU LI, XI GUANG, GAO LIMIN. Numerical study on the unsteady interaction flow in a centrifugal compressor stage[J]. Chinese Journal of Mechanical Engineering, 2004, 17(4): 609—613.

[6] ZHU BAOSHAN, KAMEMOTO KYOJI. Numerical simulation of unsteady interaction of centrifugal impeller with its diffuser using Lagrangian discrete vortex method[J]. Acta Mechanica Sinica, 2005, 21(1): 40—46.

[7] WHITFIELD A, ABDULLAH A H. Performance of a centrifugal compressor with high inlet prewhirl[J]. Journal of Turbomachinery, 1998, 120(2): 487—493.

[8] RODGERS C. Centrifugal compressor inlet guide vanes for increased surge margin[R]. ASME Paper, 90—GT—158, 1990.

[9] SHU C W, OSHER S. Efficient implementation of essentially non-oscillatory shock-capturing schemes[J]. Journal of Computational Physics, 1989, 83(1): 32—87.

（编辑　伟）

新设计

空气冷却的 H 技术

据《Gas Turbine World》2006 年年度手册报道，Siemens 公司正在试验一台新的 50 Hz 高温空气冷却燃气轮机，其性能与蒸汽冷却的 H 技术相同。

以简单循环方式运行，用天然气作为燃料，SGT5—8000H 燃气轮机的额定输出功率为 340 MW，热效率为 39%。燃气轮机（不包括发电机）重量为 444 t，外廓尺寸为长 13.2 m，高 5 m 和宽 5.2 m。

1×1 单轴联合循环装置的额定输出功率约为 530 MW，热效率超过 60%。

8000 H 是 Siemens 和 Westinghouse 公司合并后研制的一种全新机器。除了其它许多先进的性能以外，它将安装有新的热障涂层在线监视系统。

在蒸汽一侧，SCC5—8000H 联合循环具有一台有着双流低压部分的双缸汽轮机，低压部分将连接到水冷发电机的一端。

SCC5—8000H 也有一台用于循环和高效率负载的先进的高压直流式余热锅炉。装置的设计性能包括用于中间负荷的快速启动和基本负荷运行。

空气冷却燃气轮机设计的特色是新一代具有能够经受住非常高的燃气初温和排气温度的材料和涂层、装有先进叶片装置的压气机以及用于冷却空气系统的漏泄极低的密封。

（吉桂明　供稿）

高温气体辐射特性计算模型＝**A Survey of Models for the Calculation of Radiation Characteristics of High-temperature Gases**[刊, 汉] / YIN Xue-mei, LIU Lin-hua (College of Energy Science and Power Engineering, Harbin Institute of Technology, Harbin, China, Post Code: 150001) // Journal of Engineering for Thermal Energy & Power. — 2007, 22(5). —473 ~479

An accurate calculation of radiation characteristics of high-temperature gases is of major significance in such engineering applications as combustion and infrared detection etc. The current research results of gas radiation characteristics both at home and abroad are described. The fundamental theory and main features of the methods for calculating the radiation characteristics of three kinds of gases are analyzed with the emphasis on the newly developed full-spectrum k-distribution (FSK) model. Summarized and tabulated are the applicable conditions, calculation accuracy and speed of various models for the calculation of radiation characteristics. The authors have calculated the wall surface heat flux in the steam and carbon dioxide mixed gas layer between two parallel plates and presented a chart comparing the relative-error difference between the calculation results of various models and those of a line-by-line calculation. The selection of proper methods for the calculation of radiation characteristics of gases under different conditions is proposed and in the light of the shortcomings of the current models, the future research trend also forecasted. **Key words**: gas radiation characteristics, computational model, k-distribution model

缝隙位置对空心静叶去水性能影响的试验研究＝**Experimental Study of the Effect of Suction Slot Location on Water Removal Performance of Hollow Stationary Blades**[刊, 汉] / WANG Xin-jun, LU Cheng (National Key Laboratory on Multi-phase Flow in Power Engineering, Turbomachinery Research Institute, Xi' an Jiaotong University, Xi' an, China, Post Code: 710049), LIU Jian-cheng, ZHANG Jun-bo (Harbin No. 703 Research Institute, Harbin, China, Post Code: 150036) // Journal of Engineering for Thermal Energy & Power. — 2007, 22(5). —480 ~483

On the slot suction test rig of a humid-air plane-cascade, an experimental study was conducted of the water removal performance of the suction slots on hollow stationary blades of a steam turbine. The test conditions are given as follows: the air humidity at the cascade inlet, 7.94%; water droplet diameter, between 1.5 to 150 μm, airflow velocity at the cascade outlet, 170 m/s; width of the suction slot, 1.0 mm, angle of the suction slot 45° with the suction slots being located at the suction and pressure side of the stationary blades respectively. The test results show that with an increase in relative location of the suction slots, the water quantity sucked through the slots will increase and under a same suction pressure difference, the water quantity sucked through a unit length of a slot on the concave surface of a stationary blade is greater than that on the convex surface. The suction slots close to the water outlet on the concave side of a stationary blade enjoy an optimum water-removal performance. Furthermore, with an increase of the suction pressure difference, the water quantity sucked through the slots will also increase accordingly. **Key words**: steam turbine blade, hollow stationary blade, suction slot location, suction pressure difference, water quantity sucked

导叶预旋角对叶轮/扩压器相干的影响＝**A Study of the Effect of Inlet Guide Vane (IGV) Prewhirl Angles on Impeller/Diffuser Interaction**[刊, 汉] / ZHOU Li, CAI Yuan-hu (College of Power and Energy Source, Northwestern Polytechnical University, Xi' an, China, Post Code: 710072), XI Guang (College of Energy Source and Power Engineering, Xi' an Jiaotong University, Xi' an, China, Post Code: 710049) // Journal of Engineering for Thermal Energy & Power. — 2007, 22(5). —484 ~489

Under different inlet-guide-vane (IGV) prewhirl angles and by using an unsteady approach, a numerical simulation was conducted of the unsteady interactions among the following three components: IGV, impeller and diffuser, moreover, a comparison with the test results was performed. Studied was the effect of IGV prewhirl angles on the flow and unsteadiness inside the impeller and diffuser. Meanwhile, the mechanism of interactions among the above-mentioned three moving and stationary components was also explored. The results show that the calculation results are in good agreement with the test ones. At a same flow rate, the wake flow of the IGV and the unsteady function of a large-sized vortex cluster will

weaken the unsteadiness of the impeller and diffuser with the changing radian being only about 1/4 of the original magnitude. When the IGV assume a negative prewhirl angle, the unsteady influence brought about by the wake flow of the IGV is much smaller than that when the IGV take on a positive prewhirl angle. The unsteady effect arising from the wake flow of the impeller and potential-repercussion action of the diffuser, however, will be somewhat bigger than the case when the IGV have a positive prewhirl angle. **Key words**: inlet guided vane, impeller, diffuser, unsteady interaction, prewhirl angle, unsteadiness

某型压气机叶片防护层耐蚀性研究= **A Study of the Corrosion-resistant Characteristics of the Blades of a Compressor and Their Protective Coatings**[刊, 汉] / LIU Zheng-fa, XU Zhe, ZHANG Chun-mei (Compressor Design Department, Harbin No. 703 Research Institute, Harbin, China, Post Code: 150036)// Journal of Engineering for Thermal Energy & Power. — 2007, 22(5). —490 ~ 494

By using magnetically-controlled sputter technology, a TiN layer was sputtered onto compressor blades to serve as a protective coating of the blades. Through a combination of salt-mist tests and an electrochemical method, the corrosion-resistant performance of the material of compressor blades was studied when the blades have been sputtered with a TiN layer. With the help of a weight-loss method, calculated was the corrosion rate of the test piece after the salt-mist test. From the curve, it can be seen that the corrosion-resistant performance of the test piece covered with a TiN layer is better than that of the base material. In a 3.5% (by weight) NaCl solution, an electrochemical dynamic potential scan was conducted of the test pieces sampled at different times during the salt-mist test. Based on the data of polarization curves, the authors have analyzed the causes leading to changes in corrosion potential of the test pieces undergoing different salt-mist corrosion durations, thus verifying the salt-mist test results. **Key words**: compressor blade, TiN coating, salt-mist-caused corrosion, blade corrosion

变几何多级轴流压气机全工况性能预测模型= **A Model for the Prediction of the Full-load-operation Performance of Variable-geometry Multi-stage Axial Compressors**[刊, 汉] /CUI Ning, WANG Bing-shu, LI Bin, et al (Automation Department, North China Electric Power University, Baoding, China, Post Code: 071003)// Journal of Engineering for Thermal Energy & Power. — 2007, 22(5). —495 ~ 499

Based on the comprehensive characteristic curves of compressor stages and by using a stage-by-stage superposition method, the authors have developed a model for the prediction of the full-load-operation performance of a variable geometry multistage axial compressor incorporating adjustable stationary blades. During the calculation different stage characteristic curves at a low rotating speed were used for each compressor stage to establish the performance calculation module of each stage. On the basis of rational assumptions, derived was the influence of the change in adjustable inlet guide vane (IGV) angles of the compressor on its performance based on the speed triangles of moving blades. The introduction of the aerodynamic functions and specific-heat-variable calculation formulae has simplified the calculation process, enhancing the calculation accuracy of the model. The calculation examples show that the model lends itself to practical use to a certain extent, basically reflecting the full-load-operation characteristics of the compressor and exhibiting more or less accurately the effect of IGV regulation on the performance of the whole compressor. As a result, the foregoing can well provide reliable data for the performance calculation of compressors during the development of a dynamic simulation model for modern large-sized gas turbines. **Key words**: variable geometry, axial-flow compressor, stage characteristics, model, simulation

电站给水泵汽轮机头部流场的数值计算与结构改进= **Numerical Calculation of the Flow Fields in the Head Portion of a Steam Turbine Destined for Feedwater Pumps in Power Plants and Related Structural Improvements** [刊, 汉] /JI Chun-jun, ZHOU Zi-yun, (College of Energy Source and Power, Dalian University of Technology, Dalian, China, Post Code: 116023)// Journal of Engineering for Thermal Energy & Power. — 2007, 22(5). —500 ~ 503