文章编号:1001-2060(2006)03-0259-05

# 总压畸变对小型风扇气动影响的数值模拟

孙 鹏[1],冯国泰[1],隗东伟[2]
(1. 哈尔滨工业大学 能源与动力工程学院,黑龙江 哈尔滨 150001;2. 哈尔滨职业技术学院,黑龙江 哈尔滨 150040)

摘 要:利用全三维非定常数值模拟的方法,求解某小型风扇在进口单畸变区方波型总压畸变情况下的流场参数。详细分析了总压畸变对风扇流场参数的影响,并从幅频分析的角度讨论了总压畸变经风扇衰减,以及动静叶间干扰的现象。计算和分析结果表明,该方法可以清晰描述流场参数沿周向分布情况;进口周向总压畸变会引起进口参数的重新分布,造成高低压区边界上有明显的周向测流,并且其在经过风扇衰减的同时会引起总温畸变。另外,进口畸变度相同时,随着畸变角度增加,畸变扰动和叶列间干扰明显增强。

关 键 词:非定常数值模拟;风扇;总压畸变

中图分类号:TK474.8,TH42 文献标识码:A

## 1 引 言

众多试验研究表明,压气机中上游畸变流场经叶栅流道传递到下游的强弱会显著改变该列叶栅的气动性能,使压气机性能下降,是所有降稳因子中影响最大的因子,是使压气机提前失速和喘振的主要因素[1],从而成为影响整个发动机性能的关键因素[2~4]。

几十年来,人们发展了许多数值方法来研究畸变问题。由于畸变流场和动静叶间相对运动存在非定常性,对叶轮机械进行准确的数值模拟需要大量计算时间,同时后处理也十分复杂。因此在以往的研究中,大多模型所采用的二维粘性方程和三维无粘方程无法准确描述非定常流动现象。虽然,文献[5]中使用三维 $N-S$ 方程求解流场,但该研究对叶片进行调整,减少叶栅流道数量以满足计算机运算能力的要求,从而降低了计算的准确性。

为了更好地模拟畸变对流场参数的影响,本文针对某风扇流场建立全周计算网格,用三维非定常方法求解进口总压畸变对风扇流场的影响。计算中考虑参数周向不均匀性和非定常性。

## 2 计算模型和边界条件给定

### 2.1 计算模型

本文研究的某型风扇由一列动叶和一列静叶组成。考虑到动叶扭转较大,因此在生成网格时选择非结构化网格,而静叶则采用结构化网格。在叶栅流道中采用较粗的网格分布,而在叶片表面网格局部加密。整级网格数约为880 000,其中动叶约450 000,静叶约430 000。具体网格示意图如图1所示。

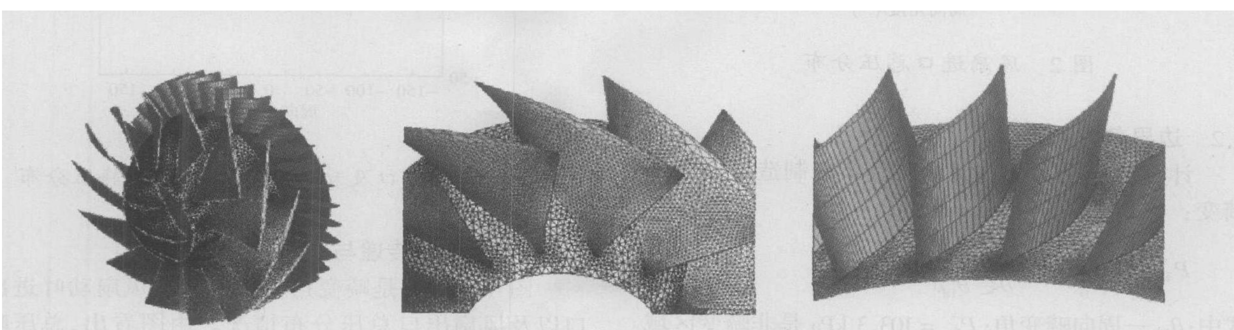

图1 三维全周网格示意图

收稿日期:2005-08-08; 修订日期:2006-03-08
基金项目:国家自然科学基金资助项目(50236020)
作者简介:孙 鹏(1979-),男,辽宁新宾人,哈尔滨工业大学博士研究生。

为了准确模拟总压畸变对流场的影响，本文使用 Fluent 软件包对流场进行非定常求解。求解过程可以分成两步：定常计算和非定常计算。其中定常计算结果作为非定常计算的初场，从而确保非定常计算可以较快地收敛到周期解。定常计算采用显式耦合求解方法，方程格式采用二阶迎风格式，并使用多重网格法进行加速计算；非定常计算采用双时间步法，该方法在时间上使用二阶精度的隐式求解，而作为虚拟时间步中的定常求解也使用了显式耦合法求解。定常计算采用混合平面法处理动静叶的交接面问题；非定常计算则使用滑移网格技术处理动静叶交接面。另外，出于节省计算时间的考虑，湍流模型选用标准 $k-\varepsilon$ 模型。

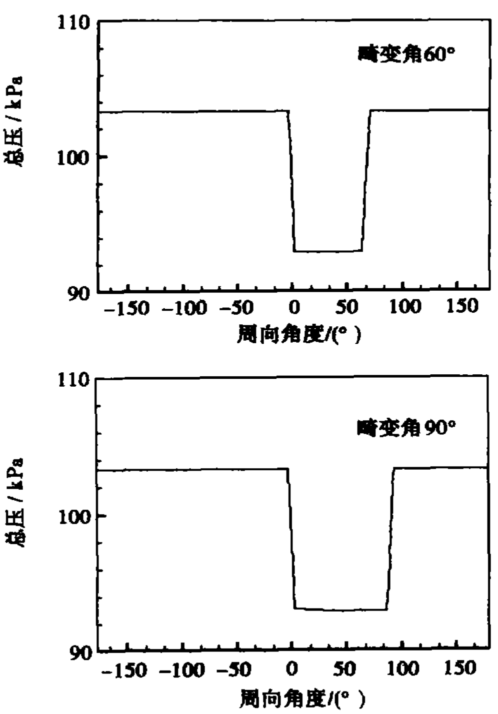


图 2 风扇进口总压分布

## 2.2 边界条件给定

计算通过给定远上游总压分布来制造进口总压畸变：

$$P_T^* = \begin{cases} P_0^* & \theta \notin \theta_{DP} \\ P_D^* & \theta \in \theta_{DP} \end{cases} \tag{1}$$

式中：$\theta_{DP}$—周向畸变角；$P_0^* = 103.3$ kPa 是非畸变区域总压；$P_D^*$—畸变区域总压。则总压畸变度 $DA_p$ 为：

$$DA_p = \frac{(P_0^* - P_D^*)}{P_0^*} \tag{2}$$

在畸变区域和非畸变区域内的总压是稳态、均匀的。计算中畸变角度 $\theta_{DP} = 60^\circ$、$90^\circ$，单畸变区扰，即动叶旋转一周遭遇一次畸变扰动，畸变强度 $DA_p = 0.1$。总压分布如图 2 所示。

远上游总温为 288 K，轴向进气，出口背压 169 kPa。计算转速选定风扇额定转速。

# 3 单畸变区周向总压畸变影响算例分析

## 3.1 周向总压畸变引起进口参数重新分布

图 3 是风扇进口和动叶栅前缘处畸变流场参数分布曲线。由图看出，由于风扇与上游非均匀流场相互作用的影响，风扇前缘处的参数分布不同于其进口处的参数分布。主要表现为：风扇前缘和进口边界的总压分布范围大致相同，但是进口处梯形分布的畸变区域在动叶前缘已不再是规则的梯形；由于与风扇的相互作用，风扇前缘处静压的不均匀度大于进口的不均匀度。

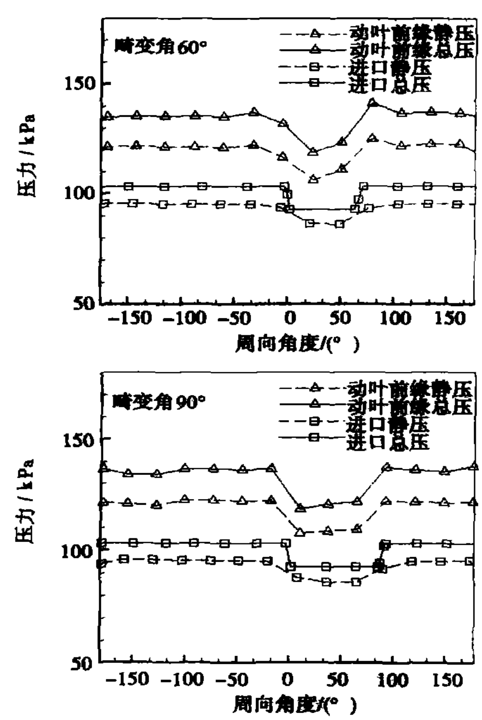


图 3 风扇进口及动叶前缘处总压和静压分布

## 3.2 总压畸变传递与总温畸变生成

图 4 和图 5 是畸变角 60°和 90°时风扇动叶进出口以及风扇出口总压分布情况。由图看出，总压畸变经风扇转子后高低压区域发生了变化，顺着转子的旋转方向，从高压区向低压区过渡处，总压有一个低谷值，随后是一个高峰值，而后又是一个低谷值。对应进口畸变区域位置，在动叶出口高压区占了较大范围；而在静叶出口，低压区所占范围较大(见图

6 和图 7）。但总体趋势都是两个低压区夹着一个高压区。动叶出口总压峰值有较大范围，基本与进口低压区范围一致，而在静叶出口，总压峰值范围减小，低谷值范围增加，该波动范围也基本与进口低压区范围一致，只是由于折转角的作用，畸变区域也相应偏转。

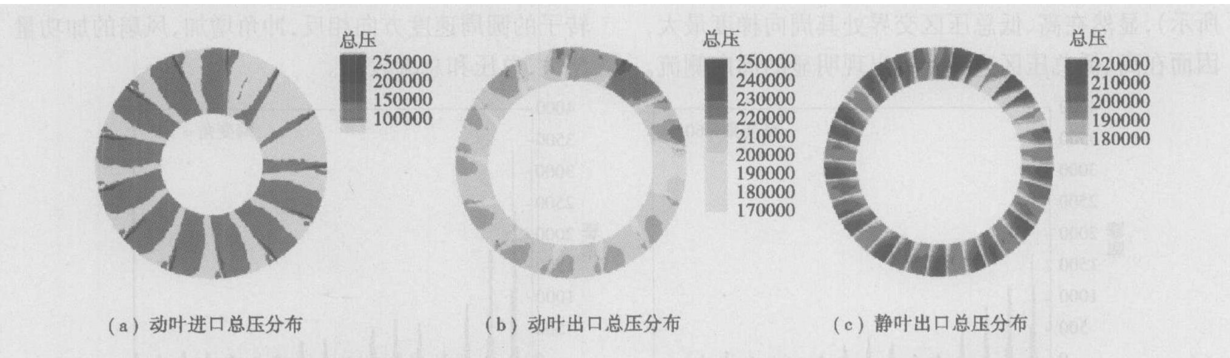


图 4　风扇进出口总压分布（$\theta_D=60^\circ$）

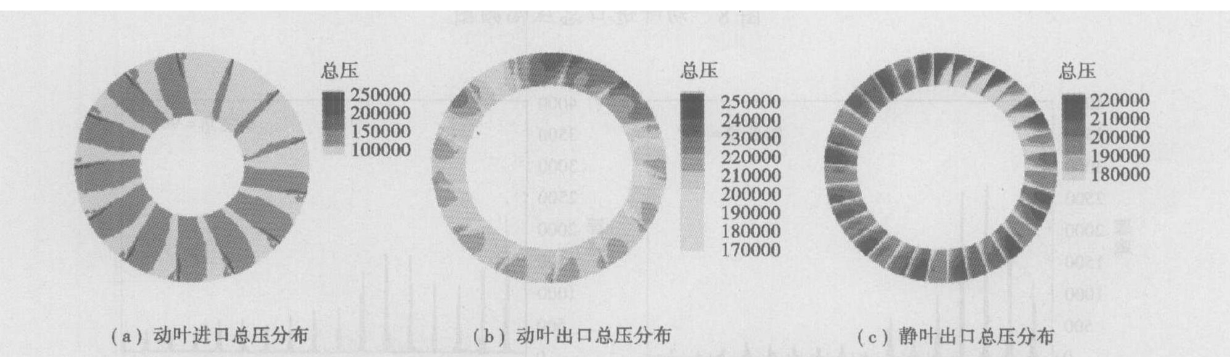


图 5　风扇进出口总压分布（$\theta_D=90^\circ$）

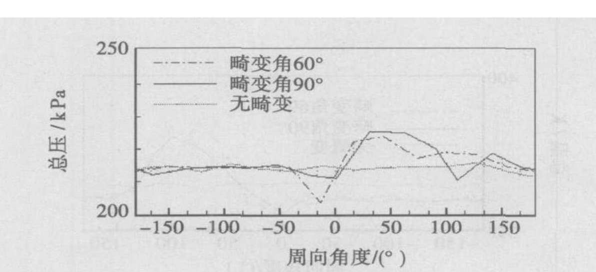


图 6　动叶出口总压分布

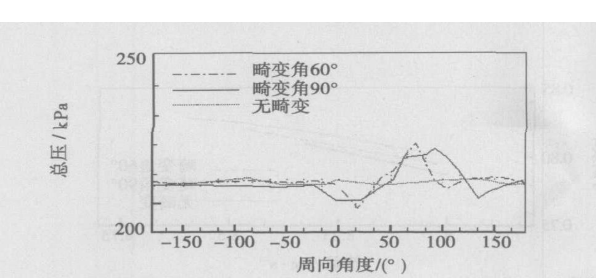


图 7　静叶出口总压分布

为了更清晰地表述畸变沿动叶的衰减，对转子进出口总压参数进行傅立叶展开，得到进出口总压的各阶扰动分量如图 8 和图 9 所示。进口处总压的一阶分量影响最大，这与初始给定的单畸变条件相吻合；二阶和三阶分量也影响较大，是由于上游气流经动叶后被动叶尾缘、静叶前缘反射回来，与进口流场相叠加的结果。在动叶出口，一阶分量明显减弱，说明总压畸变经过风扇动叶后明显衰减。但是动静叶栅相互干扰增强，使得二阶以后的高阶分量明显增强。另外，从图中可以看出，畸变角度 $\theta_D=90^\circ$时，进口处总压畸变引起的扰动较大，虽然经过风扇后有相对较大程度衰减，但其扰动影响仍然大于 $\theta_D=60^\circ$时的动叶出口。而叶列间的相互干扰也明显强于 $\theta_D=60^\circ$的情况。

虽然风扇进口处的边界条件给定总温是均匀的，但是由于存在总压畸变，导致转子叶栅沿周向的加功量分布不均匀，从而使风扇出口的总温分布也不均匀，如图 10 和图 11 所示。顺着转子的旋转方

向，从高压区向低压区过渡处，总压、总温均有一个低谷值，而从低压区向高压区过渡处均有一个高峰值。这是由于总压畸变造成静压周向畸变（如图 3 所示），显然在高、低总压区交界处其周向梯度最大，因而在高、低总压区边界上将出现明显的周向侧流。在转子叶片进入低压区处，周向侧流与转子的圆周速度方向相同，因而使得气流角（气流与周向的夹角）增加，冲角减小，风扇的加功量减小，总压和总温较低；反之，在转子叶片离开低压区处，周向侧流与转子的圆周速度方向相反，冲角增加，风扇的加功量增大，总压和总温较高。

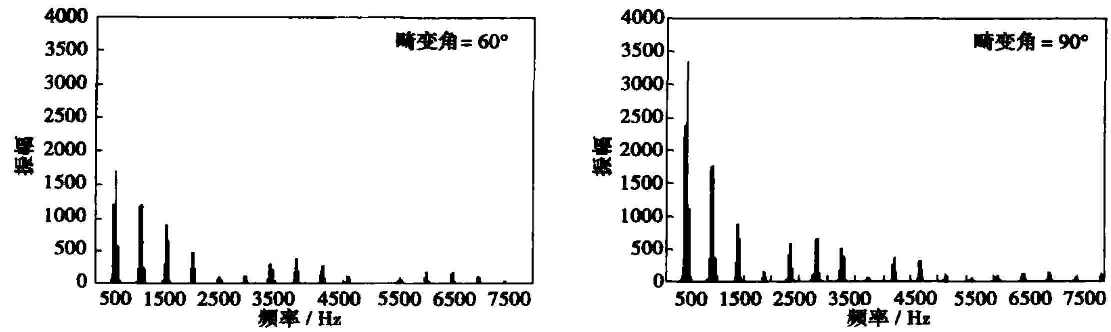


图 8　动叶进口总压幅频图

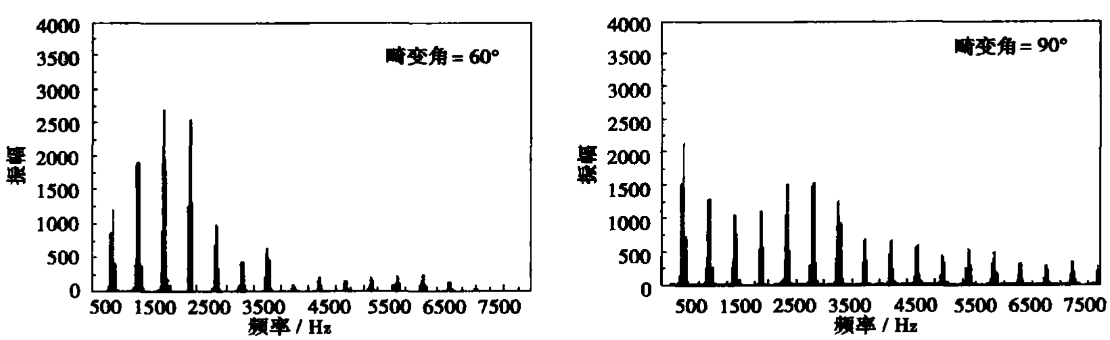


图 9　动叶出口总压幅频图

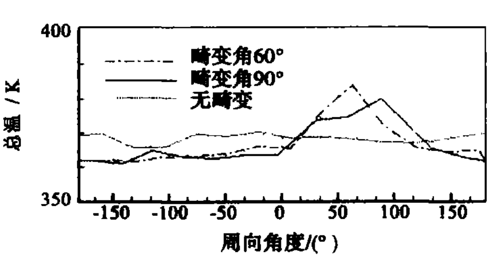


图 10　动叶出口总温分布

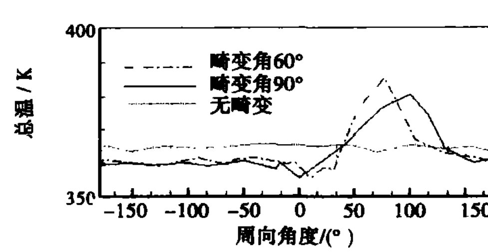


图 11　静叶出口总温分布

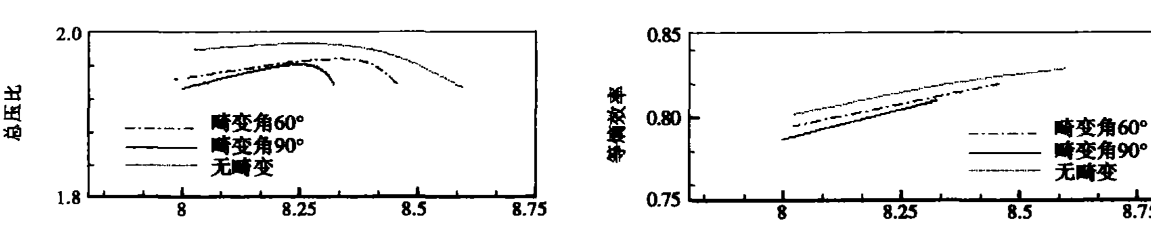


图 12　风扇特性线

### 3.3　周向总压畸变对风扇特性的影响

文中分别计算了均匀进气和畸变度 $DA_p=0.1$、畸变角 $\theta_{Dp}=60°$、$90°$ 时的风扇特性。如图 12 所示, 将均匀进气条件下与总压畸变条件下的风扇特性线进行比较, 可以看出, 进口流场周向畸变时, 效率线下移, 压比和效率降低, 风扇的稳定工作范围减小, 性能变差; 而且随着畸变角度增加, 曲线下移幅度越大, 风扇性能越差。

## 4　结　论

(1) 全周数值模拟能够清晰描述流场参数沿周向分布情况。

(2) 周向总压畸变引起进口流场参数的重新分布, 造成高低压区边界上有明显的周向测流。

(3) 对于单畸变区的扰动, 进口总压的一阶分量影响最大。总压畸变经动叶后明显衰减, 但是由于叶列间相互干扰, 使得动叶出口其它高阶分量增强。随着畸变角度增加, 畸变扰动和叶列间干扰明显增强。

(4) 进口总压畸变经过风扇传递的同时, 引起总温畸变。由于转子对高低压区的加功量不同, 顺着转子的旋转方向, 从高压区向低压区过渡处, 总压、总温均有一个低谷值, 而从低压区向高压区过渡处均有一个高峰值。

(5) 单畸变区进口总压畸变使风扇气动性能变差。并且在畸变度相同时, 畸变角越大, 对风扇的负面影响也越大。

参考文献:


[1]　刘大响, 叶培梁. 航空燃气涡轮发动机稳定性设计与评定技术[M]. 北京: 航空工业出版社, 2004.

[2]　CHERRETT M A, BRYCE J D, GINDER R B. Unsteady three-dimensional flow in a single-stage transonic fan: Part I-unsteady rotor exit flow field[J]. **ASME Journal of Turbomachinery**, 1995, **20**(3): 569－575.

[3]　GOVARDHAN M, VISWANATH K. Effect of circumferential Inlet flow distortion and swirl on the flow field of an axial flow fan stage[R]. ASME Paper, 96－GT－263, 1996.

[4]　COUSINS W T, GEORGES M J, REZAEI H. Inlet distortion testing and analysis of a high-bypass ratio turbofan engine[R]. ISABE Paper, 2003－1100, 2003.

[5]　DORNEY D J, SCHWAB J R. Unsteady numerical simulation of radial temperature profile redistribution in a single-stage turbine[R]. ASME Paper, 95－GT－178, 1995.


(渠　源　编辑)

(上接第 258 页)


[4]　KRAIN H. A CAD-method for centrifugal compressor impellers[J]. **ASME Journal of Engineering for Gas Turbines and Power**, 1984, **106**: 482－488.

[5]　HAH C, KRAIN H. Secondary flows and vortex motion in a high-efficiency backswept impeller at design and off-design conditions[J]. **ASME Journal of Turbomachinery**, 1990, **112**: 7－13.

[6]　KRAIN H. A study on centrifugal impeller and diffuser flow[J]. **ASME Journal of Engineering for Power**, 1981, **103**: 688－697.

[7]　KRAIN H. Swirling impeller flow[J]. **ASME Journal of Turbomachinery**, 1988, **119**: 122－128.

[8]　HATHAWAY M D, CHRISS R M. Experimental and computational investigation of the NASA low-speed centrifugal compressor flow field[J]. **ASME Journal of Turbomachinery**, 1993, **115**: 527－542.

[9]　HATHAWAY M D. Laser anemometer measurement of the three-dimensional rotor flow field in the NASA low-speed centrifugal compressor[R]. TP－3527, NASA, 1995.

[10]　HILLEWAERT K, VAN DEN BREMBUSSCHE R A. Numerical simulation of impeller-volute interaction in centrifugal compressors[J]. **ASME Journal of Turbomachinery**, 1999, **121**: 603－608.

[11]　杨　策, 索沂生, 朱　伟. 反弯角达 30°离心压气机叶轮内部流场的数值研究及改型设计[J]. 动力工程, 2000, **20**(1): 580－584.


(渠　源　编辑)

The thin-layer activation method is a kind of nuclear method for the dynamic and qualitative measurement of wear-and-tear of specified parts and components with no need for system disassembly. This method features high sensitivity, an ability to perform on-line detection, and low radioactivity etc. The working principles and relevant techniques of the thin-layer activation method are described. Its application to the on-line and off-line measurement of wear of turbine blades has made it possible to realize an off-line monitoring of cavitation-caused wear for three kinds of turbine blades and an on-line monitoring of same at specified locations for two kinds of turbine blades. A detailed test and detection procedure is given. Through tests lasting 53 hours, the worn-out mass actually measured of the blades is identical to that obtained by using a weighting method. The standard error of average measured values is ±0.2 μm. **Key words**: turbine blade, thin layer activation method, wear

对旋叶栅级间内流干涉的数值研究=**Numerical Study of Interference of Inter-stage Flows in a Counter-rotating Cascade**[刊,汉]/XIAO Peng, WANG Jun (College of Energy Sources and Power Engineering under the Central China University of Science and Technology, Wuhan, China, Post Code: 430074)// Journal of Engineering for Thermal Energy &Power. —2006, 21(3). —249~254

With a whole counter-rotating axial fan serving as an analytic model and through the use of software Fluent and the adoption of SIMPLE algorithm, conducted was a numerical simulation of interference flows between two stages of moving blades in the model. This has been accomplished after solving a full three-dimensional Reynolds time-averaged $N-S$ equation. By combining the numerical simulation results obtained from a steady flow analysis with the flow characteristics of the counter-rotating fan, the flow field distribution of stream surfaces in different circumferential planes S1 and different radial planes S3 in the two stage impeller of the counter-rotating axial fan was given and the interference phenomenon and mechanism of two stages of the counter-rotating fan has been revealed qualitatively and quantitatively. It has been found that in the inter-stage flow field between counter-rotating impellers, relatively speaking, the wake interference effect of the front-stage impeller is stronger than that of the rear stage impeller under the action of potential energy. **Key words**: counter rotating, cascade, numerical simulation, interference

后置蜗壳斜流叶轮内部射流—尾迹数值研究=**A Numerical Study of the Jet-flow Wake in the Oblique Flow Impeller of a Rear-mounted Volute Housing**[刊,汉]/CHU Wu-li, YANG Yong, WU Yan-hui, et al (Power and Energy Source College under the Northwest China Polytechnical University, Xi' an, China, Post Code: 710072)// Journal of Engineering for Thermal Energy &Power. — 2006, 21(3). —255~258, 263

A Fine/Turbo module of commercial software Numeca was used to conduct the whole-machine calculation for an oblique flow blower incorporating an oblique flow impeller and a volute housing as an integrated whole. Moreover, on the basis of having achieved a relatively good agreement with the already available test data, a detailed numerical analysis is performed of its inner flow field, which confirms that inside the oblique flow impellers, there also exist classic jet-flow wake patterns specific to a centrifugal impeller. The research results show that due to a highly nonsymmetrical nature of the volute housing, the jet flow-wake inside various impellers also features totally different patterns. A further study indicates that the basic reason leading to the emergence of this phenomenon lies in the presence of a nonsymmetrical volute housing, which changes the blade-tip leakage flow at the top of the impellers. **Key words**: oblique flow impeller, volute housing, wake/jet flow, blade-tip leakage flow

总压畸变对小型风扇气动影响的数值模拟=**Numerical Simulation of the Impact of Total-pressure Distortion on the Aerodynamic Performance of Small-sized Fans**[刊,汉]/ SUN Peng, FENG Guo-tai (Energy Source College under the Harbin Institute of Technology, Harbin, China, Post Code: 150001), KUI Dong-wei (Harbin Vocational College of Technology, Harbin, China, Post Code: 150040)// Journal of Engineering for Thermal Energy & Power. — 2006, 21(3). —259 ~263

A full three-dimensional non-steady numerical simulation method was employed to solve the parameters of the flow field of a small-sized fan under the condition of a square-wave type total-pressure distortion at the inlet single-distortion zone. The effect of the total-pressure distortion on the parameters of the flow field of the fan has been analyzed in detail. Moreover, from the standpoint of amplitude and frequency analysis discussed are the phenomena of attenuation of total-pressure distortion via the fan and the interference between stator blades and moving ones. The calculation and analysis results indicate that the method under discussion can clearly depict the distribution of flow field parameters along the circumferential direction. The total-pressure distortion at the inlet along the circumferential direction may cause a redistribution of the inlet parameters, resulting in a marked circumferential lateral-flow at the boundary of high and low pressure areas and triggering a total-temperature distortion, which took place simultaneously with an attenuation via the fan. In addition, when the distortion degree at the inlet is identical, with an increasing distortion angle, the distortion disturbance and the interference between stator blades and moving ones will undergo a marked intensification. **Key words:** non-steady numerical simulation, fan, total-pressure distortion

花瓣稳燃器流场的数值模拟与特性分析＝**Numerical Simulation and Characteristic Analysis of the Flow Fields in a Petal-shaped Combustion Stabilizer**[刊,汉]/ZHAO Ling-ling, ZHOU Qiang-tai, ZHAO Chang-sui (Power Engineering Department, Southeast University, Nanjing, China, Post Code: 210096)// Journal of Engineering for Thermal Energy & Power. — 2006, 21(3). —264 ~266, 274

A calculation model of three-dimensional (360°) curvilinear coordinates and complex curved-surface geometry has been set up to conduct the numerical simulation of the flow-field of a petal-shaped combustion stabilizer along with a theoretical analysis of the flow field characteristics and mixing-dilution ones of the stabilizer. The operating principle of the latter's combustion stabilization has been studied. The specific design of the petal-shaped combustion stabilizers has made it possible to lengthen the thermal mixing boundary between "air"-"pulverized-coal"- "gas-flow" and flue gas return-flow. Apart from a central return-flow zone, a radial return-flow zone has been formed behind each petal to fuse with the central return-flow zone. By introducing a convection mixing of pulverized-coal gas flow with high-temperature flue gas flow, the heat-mass exchange strength of both flows was enhanced, and the circulation time of the pulverized-coal in the return zone prolonged, thus providing a stable heat source for the ignition and burning of pulverized-coal particles. This will be conducive to the ignition and burning-off of the pulverized-coal, especially for low-volatile coal and in case of operation at a low load. The research results of the authors can provide a theoretical basis for the optimized design of petal-shaped combustion stabilizers. **Key words:** petal-shaped combustion stabilizer, numerical simulation, stable combustion, flow field characteristics

可吸入颗粒粒径声学夹带法测量的实验研究＝**Experimental Study of the Measurement of Inhalable Particle Diameter by the Use of an Acoustic Entrainment Method**[刊,汉]/YAO Gang, ZHAO Bing, YANG Lin-jun, et al (Education Ministry Key Laboratory of Coal Clean Combustion under the Southeast University, Nanjing, Jiangsu, China, Post Code: 210096)// Journal of Engineering for Thermal Energy & Power. — 2006, 21(3). —267 ~269

To date, the measurement of a single particle with a diameter of about 1 $\mu$m is still a technically intractable problem. A new approach is proposed for the measurement of micron grade and sub-micron grade single particle size by the use of acoustic wave entrainment. This approach makes full use of the dynamic characteristics of the two-dimensional force field composed of the horizontal vibration and gravity-free sedimentation of a single particle at the pressure node location in a standing-wave acoustic field. Through an analysis the relationship between a fine particle size and the particle vibration in