热力涡轮机械

文章编号: 1001－2060(2009)05－0560－06

气膜孔形状对涡轮叶片气膜冷却效果的影响

戴　萍[1], 林　枫[2]

(1. 哈尔滨工程大学 动力与能源学院; 2. 中国船舶重工集团公司第七〇三研究所, 黑龙江 哈尔滨 150036)

摘　要: 基于控制容积法对三维定常不可压缩 $N-S$ 方程进行离散, 采用非结构化网格及两层 $k-\varepsilon$ 湍流模型, 在吹风比 M 为 0.6 和 1.2 的情况下, 数值模拟了气膜孔几何形状对涡轮叶片气膜冷却效果的影响, 得到了气膜孔附近的流场分布。所选孔形为圆柱孔、前向扩张孔、开槽前向扩张孔及新型缩放槽缝孔。结果表明: 圆柱孔的冷却效率随着吹风比的增加而显著地降低; 开槽前向扩张孔的冷却效率优于圆柱孔和前向扩张孔; 缩放槽缝孔在不同吹风比下的冷却效率均高于其它 3 种孔形, 缩放槽缝孔和开槽前向扩张孔不同程度地抑制了反向涡旋对的产生, 提高了射流对壁面的贴附性, 增强了壁面的冷却效果。

关　键　词: 涡轮叶片; 缩放槽缝孔; 开槽前向扩张孔; 气膜冷却效率; 湍流模型; 数值模拟

中图分类号: TK474; TK124　　　文献标识码: A

引　言

提高透平进口温度是改善燃气轮机性能和提高其经济性的重要途径, 但却受到叶片材料耐热性能的限制, 因此必须采取有效的冷却措施对涡轮叶片加以保护, 使其免受高温腐蚀或损伤。在众多的冷却技术中, 气膜冷却已被广泛地应用于压气机、燃烧室尤其是涡轮上, 成为发动机热端部件的主要冷却方式之一, 因此, 准确预估气膜冷却效果对燃气轮机叶片等设计起着至关重要的作用。

影响气膜冷却效率的因素很多, 其中气膜孔的形状对冷却效率的影响不容忽视。近年来, 国内外学者对此进行了大量的研究, 朱惠人等人实验比较了圆柱形孔、圆锥形孔及簸箕形孔的冷却效率[1], 发现当吹风比较大时, 带有扩张形出口的气膜冷却效率优于圆柱形孔。Ekkad 和 Schmidt 等人研究了不同几何形状射流喷孔的气膜冷却效率及冷却效率随动量通量比的变化规律[2~3]。Gritsch 和 Kim 等人给出了不同吹风比情况下[4~6], 复合斜孔射流下游传热系数比的分布情况, 发现圆孔的传热系数比值最高, 由于具有扩张形出口的孔, 在出口处横截面积的增加, 射流动量降低, 导致传热系数比值降低。近几年, 有学者尝试采用新型缩放槽缝孔来对气膜孔的结构进行优化[7~8], 以期得到较好的气膜冷却效果和孔口气动性能。Lu 实验研究了月牙形孔、槽缝形孔、漏斗形孔孔口下游的传热特性[9], 发现月牙形孔和槽缝形孔在相邻孔间的气膜覆盖性较好, 漏斗形孔在孔口下游的冷却效率较高, 沿叶高方向的效率低于其它两种孔形。

新型孔优化了气膜孔的结构, 不同程度地提高了气膜冷却效率, 有望在今后发动机高温部件的气膜冷却设计中发挥重要的作用[10], 迄今这方面的工作仍在继续, 但大多以实验研究为主[11~13], 而用数值模拟方法对新型孔与其它孔形进行系统比较与分析的文献相对较少。为此本文应用两层 $k-\varepsilon$ 湍流模型, 模拟计算了圆柱孔、前向扩张孔、开槽前向扩张孔及新型缩放槽缝孔在射流下游处及叶高方向上的气膜冷却效率及流场分布, 并将 4 种孔形的计算结果进行了详细的对比分析, 以期更好地了解不同气膜孔孔口的流场流动特性和传热机理, 为实际气膜孔的优化设计提供参考。

1　数值模拟方法

1.1　湍流模型

采用的两层 $k-\varepsilon$ 湍流模型是介于壁面函数和低雷诺数模型之间的一种中间模型[14~15], 它在受粘性力影响的近壁面应用一方程模型[16], 外部核心流应用标准 $k-\varepsilon$ 模型, 两层 $k-\varepsilon$ 湍流模型的优势表现在: 第一, 与单纯低雷诺数方法相比, 在粘性边界层处可以用较少的网格点来比较精确地描述复杂的流动, 例如它在计算旋转的同性剪切流, 包括自由

收稿日期: 2008－06－02;　修订日期: 2008－12－08

作者简介: 戴　萍(1971－), 女, 黑龙江大庆人, 哈尔滨工程大学博士研究生.

流、回流、分离流以及旋涡流中有较强的优势；第二，对于存在逆压力梯度的边界层及气膜孔出口附近，两层 $k-\varepsilon$ 湍流模型较传统的 $k-\varepsilon$ 模型有更精确的预测[17~18]。

1.2　几何模型及网格生成

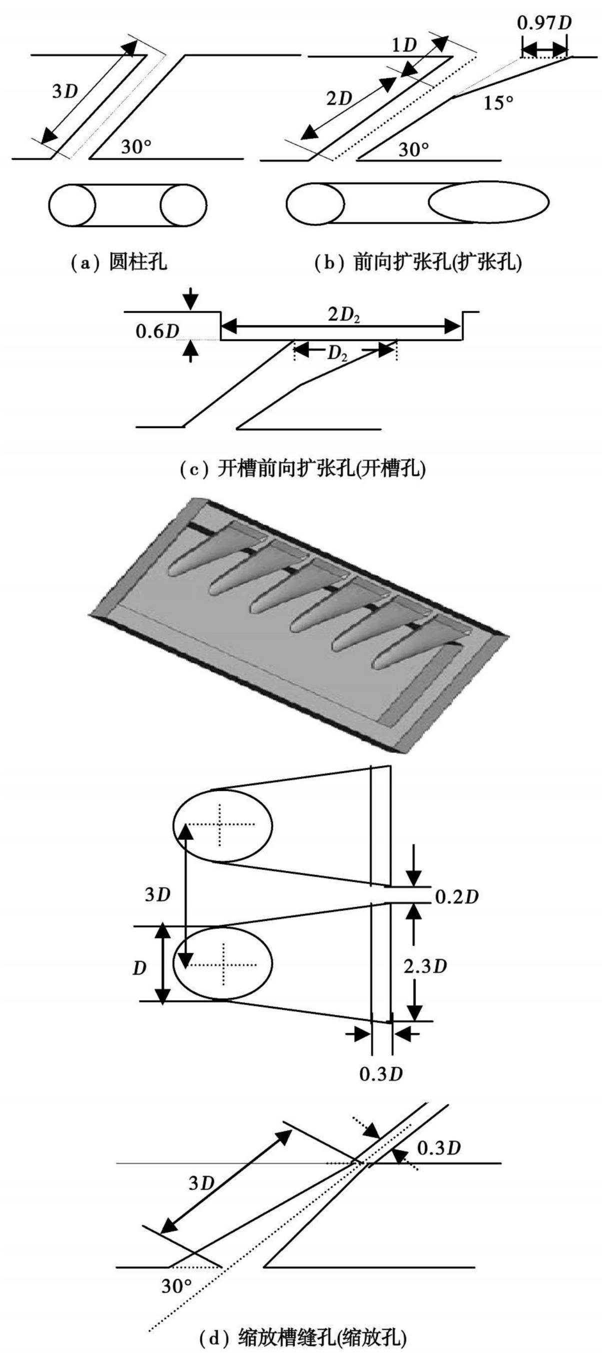


图 1　4 种气膜孔几何结构图

图 1 为本研究的 4 种形状不同的气膜孔结构图。孔排由 6 个孔组成，孔与孔之间的距离为 3D，孔长与孔径比 $L/D=3$，孔径 $D=9$ mm。孔轴线与流动方向的夹角均为 30°，前向扩张孔（扩张孔）的孔口张角为 15°，开槽前向扩张孔（开槽孔）的开槽宽度为前向扩张孔冷气出口宽度的两倍。缩放槽缝孔（缩放孔）在冷气入口处的横截面为圆形，从入口到出口孔壁逐渐扩张，在孔出口处收缩为槽缝，导致孔出口横截面积以及相邻两气膜孔出口处距离的减小。

将计算域划分为进气通道、冷气出口和供气腔三部分，如图 2 所示，并用非结构化网格划分整个计算域。在近壁面及气膜孔入口、出口附近使用加密网格，网格总数为 50 万左右。近壁面第一个网格单元中心的近壁距离满足壁面函数律的条件。

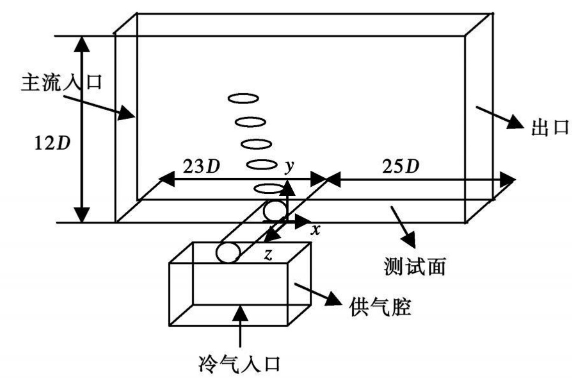


图 2　计算域及边界条件

1.3　边界条件及计算工况

由图 2 看出，供气腔提供的冷气经过气膜孔进入到主流区域，坐标原点位于沿着气膜孔轴线出口与主气流通道壁面的交点处。在坐标原点左侧 23D 处为主流入口速度边界，在测试面向上 12D 处为无滑移无渗透边界，在坐标原点右侧 25D 处为压力出口边界，气膜孔中心线为对称边界，气膜孔到气膜孔之间的流通面为周期边界，气膜孔的进出口都为交接面，孔壁为无滑移无渗透的物面边界。供气腔的顶壁与气膜孔进口交接，底面为流量进口边界，供气腔的两侧壁为周期边界，顶壁为无滑移物面。供气腔高度为 50 mm，宽度为 99 mm，主流入口和冷气入口温度分别为 337 和 189 K。为了使结果具有可比性，用于 3 种气膜孔计算的所有参数均保持不变。

吹风比定义为 $M=\rho_c v_c/\rho_\infty v_\infty$，式中 ρ_c 和 ρ_∞ 分别为冷却气体和主流气体的密度，v_c 和 v_∞ 分别为冷却气体喷射速度和主流气体的来流速度。这里 M 分别取 0.6 和 1.2，冷气射流与主流密度比 $\rho_c/\rho_\infty=2$，进口紊流度设定为 1%。主流速度 $U_\infty=10$ m/s，定义气膜冷却效率 $\eta=(T_{aw}-T_\infty)/(T_c-T_\infty)$，其中 aw 代表壁面，c 代表冷气，∞代表主流。

1.4　数值计算方法

采用 Fluent6.1 分离隐式求解器对 $N-S$ 雷诺时均紊流方程进行求解，湍流模型采用两层湍流模型，压力—速度耦合采用 Simplec 算法。采用非结构化网格技术，利用瞬态插值法避免压力场振荡，应用强隐式法求解代数差分方程。各参数的离散采用二阶精度的迎风格式，亚松弛求解，松弛因子在计算过程中逐步调整，解的收敛标准是相对残差小于 1×10^{-5}。

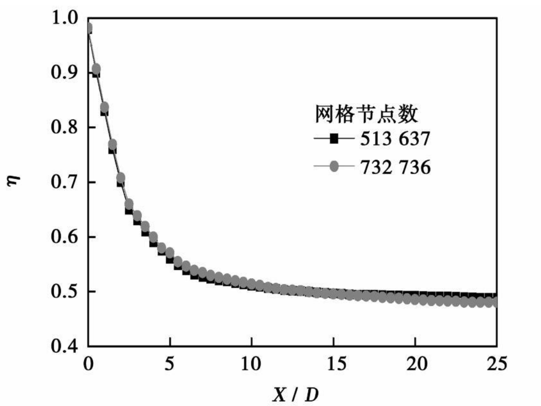


图 3　网格无关性检验

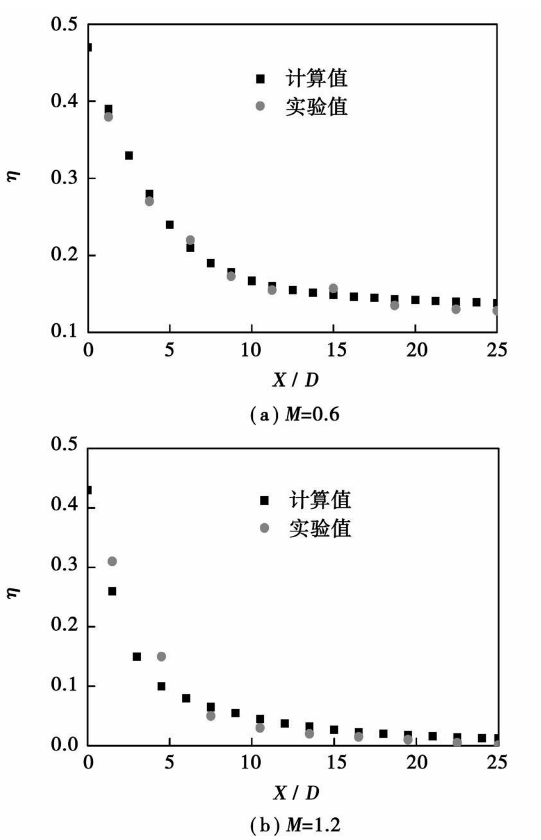


图 4　圆柱孔气膜冷却效率计算与实验结果的比较

图 3 给出了吹风比 $M=0.6$ 时，在对网格进行进一步加密后计算结果的比较，所提取的数据为气膜孔下游处沿射流方向的气膜冷却效率，从图中可以看出使用两种网格节点数所计算出的冷却效率非常接近，最大误差小于 1%，说明所采用的网格无关性很好。

为验证本研究的计算结果，将倾斜角为 35°的圆柱孔在 $M=0.6$ 和 1.2 时的计算数据与文献[19] 的实验进行了比较，如图 4 所示，最大相对误差为 20%，变化趋势基本一致。

2　结果与分析

2.1　不同吹风比时冷却效率的比较

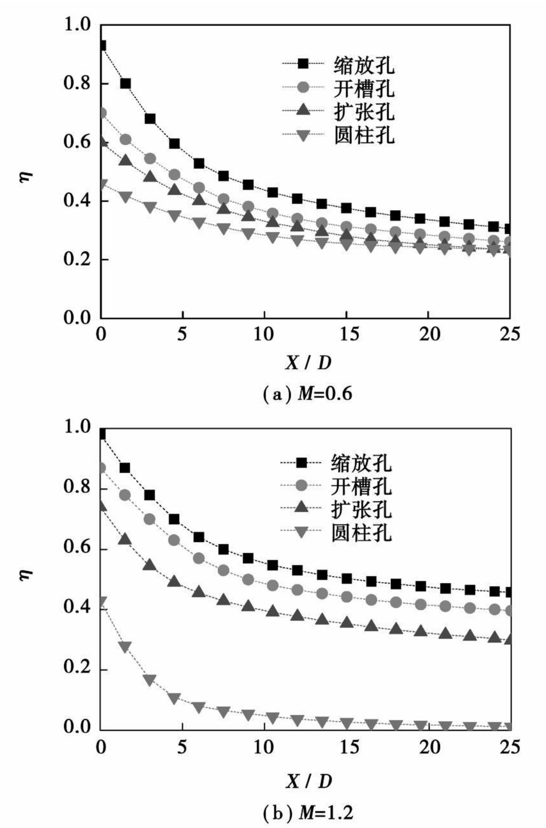


图 5　不同孔形下游平均气膜冷却效率

图 5 给出了 4 种孔形在吹风比 $M=0.6$ 和 1.2 的条件下，沿孔排下游平均冷却效率(冷却效率沿横向平均值)的对比情况。可以看出，当吹风比 $M=0.6$ 时，在孔口附近扩张孔和开槽孔的冷却效率高于圆柱孔，并随着 X/D 的增大三者均呈缓慢下降趋势，在孔排下游较远区域扩张孔与圆柱孔的冷却效率相差不大。当 $M=1.2$ 时，圆柱孔的冷却效率与低吹风比相比下降极为迅速，这是由于吹风比越低，圆柱孔的射流中心线越贴近壁面，对壁面的冷却效果越好。随着吹风比的增大，射流的初始动量逐渐增加，射流垂直方向的动量分量使得冷却射流穿

透主气流边界层，对主流区域的影响增加，能量损失增加，导致了冷却效率的降低。扩张孔和开槽孔的冷却效率优于圆柱孔的趋势随着吹风比的增加而增大，这是由于两者均不同程度地增大了出口截面面积，导致了截面上射流动量比的减小，射流穿透主流的能力明显减弱，从而使射流冷气能够较好地贴附在壁面上起到保护作用。在不同吹风比下，由于开槽孔的出口截面面积与扩张孔相比进一步增大，因此其冷却效果要好于扩张孔。缩放孔在两个吹风比下的冷却效率均高于其它 3 种孔形，特别是在孔口附近其高出的幅度更大，这与它的射流结构有关。缩放孔的出口形状使得高速气流紧贴出口底平面出流后直接冲刷表面，因此气膜冷却保护效果较佳。缩放孔有效抑制了射流下游的旋涡强度，从而提高了对壁面的冷却效果[7]，其复杂的流动机理还需要进一步的研究来揭示。

图6 为 2 种吹风比情况下不同孔形沿孔排下游冷却效率的分布，可以看出缩放孔的冷却效率最好，开槽孔次之，圆柱孔最差。

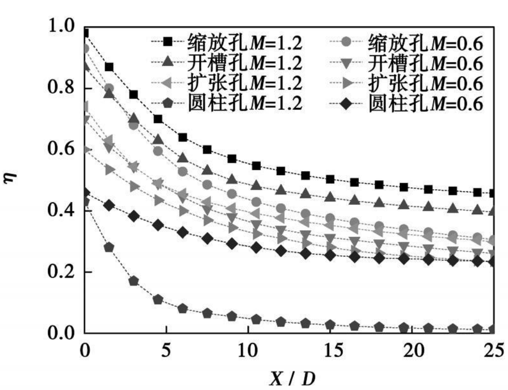
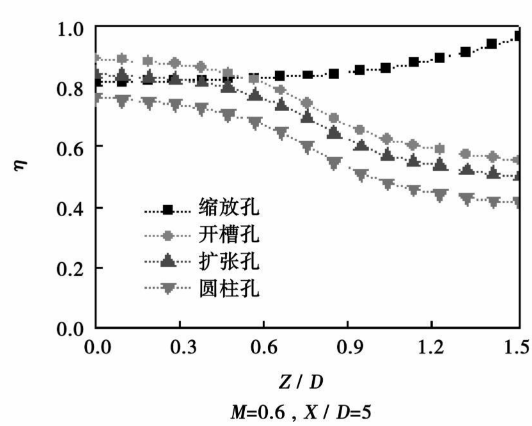


图 6　不同孔形在不同吹风比下平均气膜冷却效率

2.2　不同孔形沿叶高方向冷却效率的比较

图7 给出了所研究的 4 种孔形当吹风比 $M=0.6$ 时，在 $X/D=5$ 和 13 两个不同位置处，冷却效率沿叶高方向的分布。可以看出，缩放孔的冷却效率随着 Z/D 的增加而增加，特别是在 $Z/D>1.0$ 区域，其增加的趋势更加明显。当 $X/D=5$ 时，扩张孔和开槽孔的冷却效率明显高于圆柱形孔，并且在 $Z/D<0.4$ 的区域，两者的冷却效率高于缩放孔。当 $X/D=13$ 时，开槽孔的冷却效率下降缓慢，说明开槽孔沿叶高方向的冷却效果较好。圆柱孔沿叶高方向的冷却效率在两种情况下都呈快速下降趋势。在 $X/D=13$ 的位置处，4 种孔形沿测试面的冷却效率相差不大，均为 0.4 左右，但随着 Z/D 的增大，缩放孔的冷却效率优于其它 3 种孔形的趋势也越来越大。分析其原因，从结构上讲，圆柱孔的气流喷出比较集中，出流后直接穿过边界层进入主流区，使得冷气流在叶高方向上气膜覆盖性较差[20]，因此冷却效率较低。而具有扩张形出口的气膜孔出流后有较多的冷气贴附在叶片壁面附近，且其射流在叶高方向上的扩展较广，在侧向产生了较高的冷却效率。缩放孔的射流结构具有典型的三维特征，且出口横截面积减小，导致了气流从孔入口到出口速度的加大，这些具有较高速度的冷气射流沿叶高方向的动量分量形成了对壁面的冲刷，使得在射流孔附近 $X/D=5$ 处的冷却效果好于 $X/D=13$ 处的冷却效果。另一方面，缩放孔出口向外扩张，相邻两喷孔出口处距离较近，因此相邻两个孔喷出的气流，随着向两侧的扩展，气流发生混合，增强了冷却效果。而且孔轴线与流动方向倾斜，导致了较强的侧向动量和射流的更广扩展，使得沿叶高方向上的喷射出流连续，冷气覆盖面广，且可抑制孔后反向涡旋对的产生（在 2.3 节中将对此做分析），冷气不会被抬离叶片表面，因此在孔下游远处两个邻近孔之间的区域，沿叶高方向气膜冷却效率较高，这与文献[18] 所得出的结论是一致的。



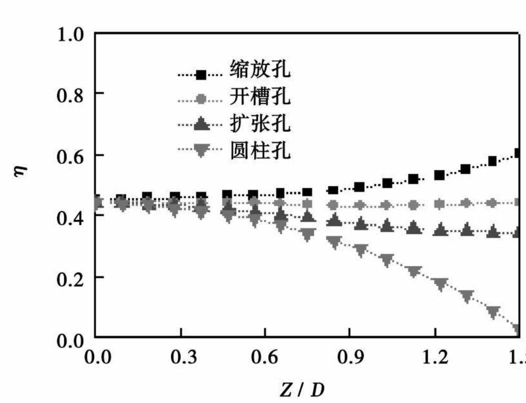


图 7　不同孔形沿叶高方向的冷却效率

图 8 给出了当吹风比 $M=0.6$ 时 4 种孔形在孔下游 $X/D=15$ 处横向冷却效率的比较，可以看出缩放孔的冷却效率明显高于其它 3 种孔形，最高冷却效率达到了 0.53，开槽孔为 0.4，扩张孔为 0.28，圆柱孔只有 0.16。横向的覆盖宽度也大于其它 3 种孔形，整体冷却效果最好。

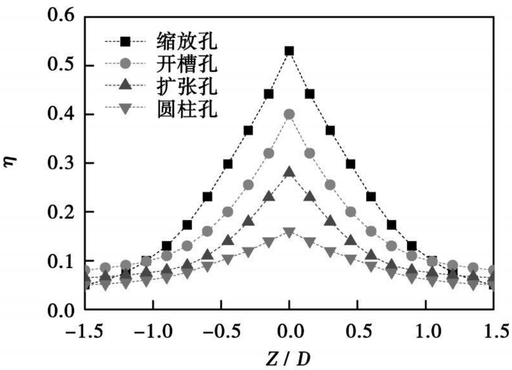


图 8　$X/D=15$ 处不同孔形横向冷却效率（$M=0.6$）

2.3　不同孔形冷气射流的速度场分析

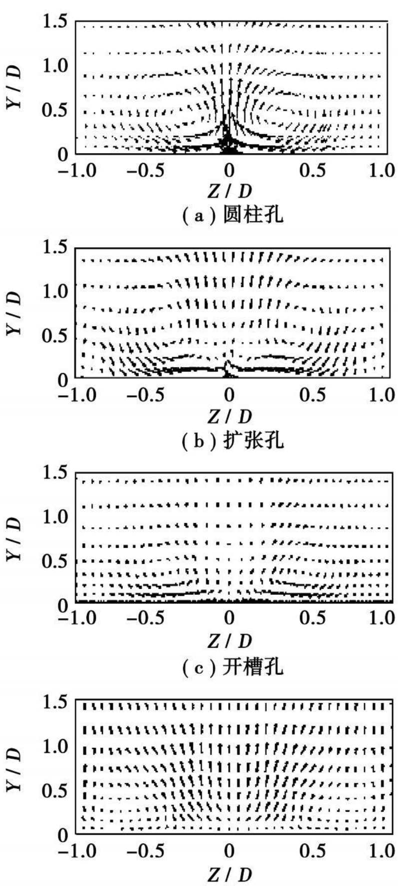


图 9　$X/D=1$ 处不同孔形的速度场（$M=1.2$）

图 9 给出了当吹风比 $M=1.2$ 时在 $X/D=1$ 处，4 种喷孔附近冷气射流的速度场分布。从 4 张图的对比来看，在圆柱孔附近生成的反向涡旋对，其强度和尺寸均大于其它 3 种孔附近产生的旋涡，反向涡旋对的产生是由于主流与射流的速度差产生的切向应力而引起的[21]。形成的反向涡旋对将冷气卷吸成两股冷气流，使冷气射流在向下游发展的开始阶段逐渐抬离壁面，反向涡旋对的强度和尺度越大，其卷吸冷气流并使其向四周掺混的程度也越大，从而导致冷却效率下降的越快。开槽孔和扩张孔使反向涡旋对的强度得到了一定程度的抑制，气膜出流向主流的垂直穿透能力减弱，因此其气膜覆盖性较好。缩放孔由于其结构上的特殊性，以及邻近气膜孔喷射气流流场的相互影响[22]，完全抑制了反向涡旋对的产生，因此其冷气射流能较好地贴附于壁面，气膜冷却效率较高。

3　结　论

（1）圆柱孔的气膜冷却效率随着吹风比的增加而显著地降低，且其冷却效率沿叶高方向呈快速下降趋势。

（2）前向扩张孔和开槽前向扩张孔的冷却效率随着吹风比的增大而增大，开槽前向扩张孔的射流在叶高方向上的扩展较广，在侧向孔间区域产生了较高的气膜冷却效率。

（3）缩放槽缝孔在不同吹风比下的冷却效率均高于其它 3 种孔形，并在孔口下游较远区域，两个孔之间沿叶高方向的气膜覆盖性较好，整体冷却效果最好。

（4）圆柱孔附近生成了强度和尺寸较大的反向涡旋对，前向扩张孔和开槽前向扩张孔附近生成的涡旋与圆柱孔相比明显减小，缩放槽缝孔抑制了反向涡旋对的产生，提高了射流对壁面的贴附性，增强了壁面的冷却效果。

参考文献：

[1] 朱惠人，许都纯，刘松龄．气膜孔形状对排孔下游冷却效率的影响[J]．航空学报，2002，23(1)：75－78.

[2] EKKAD S V，ZAPATA D，HAN J C. Film effectiveness over a flat surface with Air and CO_2 film injection through compound angle holes using a transient liquid crystal image method[J]. ASME Journal of Turbomachinery，1997，119：587－593.

[3] SCHMIDT D L，SEN B，BOGARD D G. Film cooling with compound angle holes：adiabatic effectiveness[J]. ASME Journal of Turbomachinery，1996，118：807－813.

[4] GRITSCH M，SCHULZ A，WITTIG S. Heat transfer coefficient mea-

surements of film cooling holes with expanded slots[R] . ASME Paper 98—GT—28 1998.

[5] GRITSCH M, COLBAN W, SCHAR H, et al. Effect of hole geometry on the thermal performance of fan—shaped film cooling holes[J] . ASME Journal of Turbomachinery, 2005, 127(4): 718—725.

[6] KIM Y J, KIM S M. Influence of shaped injection holes on turbine blade leading edge film cooling[J] . International Journal of Heat and Mass Transfer, 2004, 47: 245—256.

[7] SARGISON J E, GUO S M, OLDFIELD M L, et al. A converging slot—hole film—cooling geometry, part Ⅰ: low—speed flat—plate heat transfer and loss[J] . ASME Journal of Turbomachinery, 2002, 124: 453—460.

[8] SARGISON J E, GUO S M, OLDFIELD M L, et al. A converging slot—hole film—cooling geometry, part Ⅱ: transonic nozzle guide wane heat transfer and loss[J] . ASME Journal of Turbomachinery, 2002, 124: 461—471.

[9] LU Y. Effect of hole configurations on film cooling from cylindrical inclined holes for the application to gas turbine blades[D] . Louisiana State University, 2007.

[10] LU Y, EKKAD S. Predictions of film cooling from cylindrical holes embedded in trenches//9th AIAA Joint Thermophysics and Heat Transfer Conference[C] . San Francisco, 2006: 5—8.

[11] YU Y, YEN C H, SHIH TI, et al. Film cooling and heat transfer coefficient distributions around diffusion shaped holes[J] . Journal Heat Transfer, 2002, 124: 821—827.

[12] MCGOVERN K T, LEYLEK J H. A detailed analysis of film cooling physics: Part Ⅱ: Compound angle injection with cylindrical holes [J] . ASME Journal of Turbomachinery, 2000, 122(1): 113—121.

[13] LIGRANI P M, MITCHELL S W. Interaction between embedded vortices and injection from film—cooling holes with compound angle orientations in a turbulent boundary—layer[J] . ASME Journal of Turbomachinery, 1994, 116(1): 80—91.

[14] AZZI A, LAKEHAL D. Perspectives in modeling film cooling of turbine blades by transcending conventional two—equation turbulence models[J] . ASME Journal of Turbomachinery, 2002, 124: 472—484.

[15] KIM J, MOIN P, MOSER R. Turbulence statistics in fully developed channel flow at low Reynolds number[J] . Journal Fluid Mechincal, 1987, 177: 133—166.

[16] RODI W, MANSOUR N N, MICHELASSI V. One equation near wall turbulence modeling with the aid of direct simulation data[J] . ASME Journal of Engineering for Fluids, 1993, 115: 196—205.

[17] LIU XIAOCHEN, JIANG PEIXUE, SUGISHITA H, et al. Conjugate heat transfer analysis of film cooling flows[J] . Journal of Thermal Science, 2006, 15(1): 85—91.

[18] AZZI A, JUBRAN B A. Numerical modeling of film cooling from converging slot—hole[J] . Heat Mass Transfer, 2007, 43: 381—388.

[19] SCHMID D L, SEN B, BOGARE D G. Film cooling with compound angle holes: adiabatic effectiveness[R] . ASME Paper: 91—GT—127, 1991.

[20] 郭婷婷, 刘建红, 宋东辉, 等. 不同形状气膜孔对气膜冷却效果的影响[J] . 动力工程, 2006, 26(3): 333—336.

[21] 郭婷婷, 李少华, 徐　忠. 横向紊动射流流场结构的数值分析[J] . 中国电机工程学报, 2003, 23(7): 191—195.

[22] WALTERS D K, LEYLEK J H. A detailed analysis of film—cooling physics: Part Ⅰ: streamwise injection with cylindrical holes[J] . ASME Journal of turbomachinery. 2000, 122: 102—112.

（编辑　刘　伟）

新技术、新产品

45 型驱逐舰的第一艘舰通过航行试验

据《Gas Turbine World》2009 年 1—2 月号报道，HMS Daring 号导弹驱逐舰是英国海军新一级先进的 45 型驱逐舰的首舰在完成了它的初次海上航行试验后，已被它的主承包者 BVT Surface Fleet Ltd 移交给英国国防部。

该 7 500 t 战舰在进入服役以前，将经受几个月的补充试验和训练。

该现代化战舰将由一台在 ISO 条件下额定连续功率为 25 252 kW 的 WR—21 中间冷却回热式燃气轮机驱动，航速最高可达到 29 节。

该驱逐舰系统约 80％是新的设计，使得装备有直升机的 HMS Darring 号弹驱逐舰是英国海军技术上最先进的海上战斗平台。

（吉桂明　摘译）

燃蒸联合循环单轴发电机布置的启发＝**Edification from a Single-shaft Layout of Gas-steam Combined Cycle Power Generators**[刊，汉]/WANG Zhi-tao, LI Shu-ying, SUN Yu-feng (College of Power and Energy Source Engineering, Harbin Engineering University, Harbin, China, Post Code: 150001) // Journal of Engineering for Thermal Energy & Power. — 2009, 24(5). —549～553

Proceeding from the empirical data of various single-shaft layouts of latest gas-steam combined cycle power generator units supplied by GE, Mitsubishi, Siemens and Alstom company etc. for the world power markets, a preliminary argumentation was conducted of a version in which an IFEP (integrated full electric propulsion) marine combined power plant was used as a primer mover module. Moreover, the feasibility of the module in question based on a combined power plant was investigated and the improvement for full electric propulsion versions published for two warships of USA and one CVN (carrier vessel nuclear) warship of Great Britain was compared with some meaningful viewpoints being obtained. Summing up various circumstances, the authors have concluded that the use of a single generator can reduce the total weight by over 10% compared with the use of two generator sets having the same power output. The version thus obtained can reduce the number of generators by one half and decrease the land area occupied by over 10%, resulting in weight reduction and significant simplification of relevant equipment items. In the meantime, the reliability and maintainability of the system can be improved. **Key words**: gas-steam combined cycle power generator unit, integrated full electric propulsion (IFEP), combined power plant, prime mover module

压汽闪蒸海水淡化方法的研究与进展＝**Recent Advances in the Study of a Pressurized-steam Flash Evaporation Method for Seawater Desalination**[刊，汉]/CHOU Qiao-li, JIN Cong-zhuo (Hefei Swan Refrigeration Science and Technology Co. Ltd., Hefei, China, Post Code: 230088), JIN Cong-zhuo, SHU Peng-cheng (Xi' an Jiaotong University, Xi' an, China, Post Code: 710049)//Journal of Engineering for Thermal Energy & Power. — 2009, 24(5). —554～559

Analyzed, compared and summed up were the following problems in an open heat enthalpy process among the currently available seawater desalination methods, including the multistage flash evaporation method and multi-effective distillation one. The problems are: condensate latent heat loss of the secondary steam, fouling and corrosion of the distillation method, seawater pretreatment and unstable water quality of its product when a reverse osmosis method is adopted. Summing up all the merits of the methods under discussion, the authors have presented for the first time the flash evaporation method with the highest thermal efficiency to attain the best water quality of its product. Such a wholly new, optimum and integral pressurized-steam and seawater desalination process features low investment cost, and an independent flash evaporation operation coupled with a modularized combination production and other technical advantages. Due to the proven and comprehensive integration technology and the plant safe and reliable operation, the method in question will certainly replace the various currently available methods step by step with its excellent technology and cost-effectiveness, unifying the seawater desalination markets and initiating the further development of seawater desalination technologies. **Key words**: pressurized steam, flash evaporation, seawater desalination, plant

气膜孔形状对涡轮叶片气膜冷却效果的影响＝**Influence of Air-film Hole Shapes on Turbine Blade Air-film Cooling Effectiveness**[刊，汉]/ DAI Ping (College of Power and Energy Source Engineering, Harbin Engineering University, Harbin, China, Post Code: 150001), LIN Feng (Gas Turbine Research Department, CSIC (China Shipbuilding Industrial Corporation) Harbin No. 703 Research Institute, Harbin, China, Post Code: 150036)// Journal of Engineering for Thermal Energy & Power. — 2009, 24(5). —560～565

On the basis of a bulk flow control method, discretized was a three-dimensional steady incompressible *N-S* equation. By using a non-structured grid and two-layer k-ε turbulent flow model and under the condition of the air blowing ratio M being 0.6 and 1.2, numerically simulated was the influence of air-film hole shapes on turbine blade air-film cooling effec-

tiveness, and obtained was the flow field distribution around the air-film holes. The hole shapes chosen were cylindrical, forward flared, slot-shaped forward flared and new type converging-diverging slotted holes. It has been found that the cooling efficiency of cylindrical holes is reduced significantly with an increase of the air blowing ratio, the cooling efficiency of slot-shaped forward flared holes is superior to that of cylindrical and forward flared holes and the cooling efficiency of converging-diverging slotted holes at different air blowing ratios is invariably higher than that of the other three kinds of hole shapes. The converging-diverging slotted holes and slot-shaped forward flared holes can restrain the production of reverse vortex pairs to a certain extent and strengthen the wall-adhesion property of jet flows, thus enhancing the cooling effectiveness to wall surfaces. **Key words**: turbine blade, converging-diverging slot-shaped hole, slot-shaped forward flared hole, air-film cooling efficiency, turbulent flow model, numerical simulation

迷宫密封转子动力学特性的数值模拟＝**Numerical Simulation of the Kinetic Characteristics of a Labyrinth Gland Rotor**[刊, 汉] / YAN Xin, LI Jun, FENG Zhen-ping (Turbomachinery Research Institute, Xi' an Jiaotong University, Xi' an, China, Post Code: 710049), JIANG Yu-e (Henan Tailong Science and Technology Development and Application Co. Ltd., Zhengzhou, China, Post Code: 450007)// Journal of Engineering for Thermal Energy & Power. — 2009, 24 (5). —566 ~ 570

By way of seeking numerical solutions to a three-dimensional RANS (Reynolds-Averaged-Navier-Stokes) equation, studied were the kinetic characteristics of a 16-tooth labyrinth gland rotor and analyzed was the influence of the inlet pre-swirling on the kinetic characteristic coefficient of the rotor in question at two rotating speeds. In addition, in the absence of the inlet pre-swirling and at two pressure ratios, the variation relationship of the cross rigidity and direct damping coefficient of the labyrinth gland system with the rotating speed was calculated and the calculation results were compared with the test ones and the values calculated by using two volume controlled BF (bulk flow) methods. The research results show that the numerical method adopted can predict relatively well the kinetic characteristics of the labyrinth gland rotor and the calculated results are better than those obtained by using dual volume controlled BF methods. As regards labyrinth glands, the cross rigidity is approximately in direct proportion with the inlet pre-swirling and increases with an increase of the rotating speed. The direct damping is not sensitive to both the rotating speed and inlet pre-swirling, but increases remarkably with an increase of the pressure ratio. An excessively large inlet pre-swirling and rotating speed can invariably reduce the stability of the rotor. The labyrinth gland system operating at a bigger rotating speed can enhance its rotor stability by applying a rational inlet pre-swirling. **Key words**: labyrinth gland, rotor kinetic characteristics, inlet pre-swirling, BF (bulk flow) method, numerical simulation

畸变进气条件下风扇三维非定常流动数值模拟＝**Numerical Simulation Study of a Three-dimensional Unsteady Flow in a Fan Under Distorted Air Admission Conditions**[刊, 汉] / XU Kai-fu, QIAO Wei-yang, LUO Hua-ling (College of Power and Energy Source, Northwest Polytechnic University, Xi' an, China, Post Code: 710072)// Journal of Engineering for Thermal Energy & Power. — 2009, 24(5). —571 ~ 576

Analyzed was a three-dimensional compressible model for calculating three-dimensional flow fields and performance of an axial-flow fan/compressor, in which a streamline curvature method was adopted to obtain a source term of a turbine doing work to air flows by blades. Subsequently, by seeking solutions to a three-dimensional unsteady Euler equation involving the source term, a simulation was performed of an inner three-dimensional flow field and performance of a fan/compressor. By using the model, the three-dimensional flow field and performance of a transonic fan rotor were numerically simulated and analyzed, especially, the three-dimensional inner flow fields and aerodynamic performance in the absence and presence of an abnormal inlet distortion were analyzed and compared. The research results show that the three-dimensional-theory-based prediction model can effectively analyze the influence of an inlet air distortion on the performance and stability of an axial-flow fan. **Key words**: streamline curvature method, source term, semi-actuator disk, inlet distortion