文章编号: 1001－2060(2007)05－0500－04

电站给水泵汽轮机头部流场的数值计算与结构改进

冀春俊, 周子筠
(大连理工大学 能源与动力学院, 辽宁 大连 116023)

摘　要: 采用 SMPISO 算法和高 Reynolds $k-\varepsilon$ 湍流模型, 对某型号电站给水泵汽轮机的头部流场进行了数值计算与分析, 计算域包括主汽速关阀、调节阀及调节级在内的完整头部流场, 通过尽量保证几何无失真来确保计算结果的准确性。在流动分析的基础上, 利用数值试验的方法对进气室、调节阀和调节级叶型进行了改进, 有效地降低了各部分的流动总压损失, 提高了汽轮机的效率的同时, 保证了调节阀的工作稳定性。

关　键　词: 汽轮机; 主汽速关阀; 调节阀; 调节级; 数值模拟; 结构改进

中图分类号: TK263. 6　　文献标识码: A

引　言

提高电站给水泵汽轮机的效率对于降低能耗和发电成本有着重要的意义。由于主汽速关阀、调节阀及喷嘴进汽道在内的汽轮机头部流场对汽轮机的性能有着重要的影响, 内部流动损失占汽轮机总的流动损失的比重较大[1]。目前对于完整的头部流场进行数值模拟一直未见报道, 现有的研究工作一般局限于局部区域, 无论从模型的完整性还是边界条件的设定问题, 研究工作的深度和范围受到限制。由于缺乏可用的数据资料, 设计者有时只能根据经验设计其结构。另一方面, 在调节级中由于部分进气度的影响气流三维流动特性显著, 同时对于叶片的变工况性能有较高的要求, 因此对调节级的研究工作非常迫切。随着计算机技术和计算流体力学的发展, 通过数值模拟方法了解其中的复杂流动的基本情况已成为可能。本文利用计算流体力学软件 STAR-CD 对某型号电站给水泵汽轮机的头部流场系统地进行数值计算, 分析了内部流动和压力损失在各部分的分布, 提出了结构改进方案, 数值试验结果表明这些改进措施可以有效地降低总压损失, 从而有效地提高机器的效率, 具有重要的工程实际意义。

1　物理模型

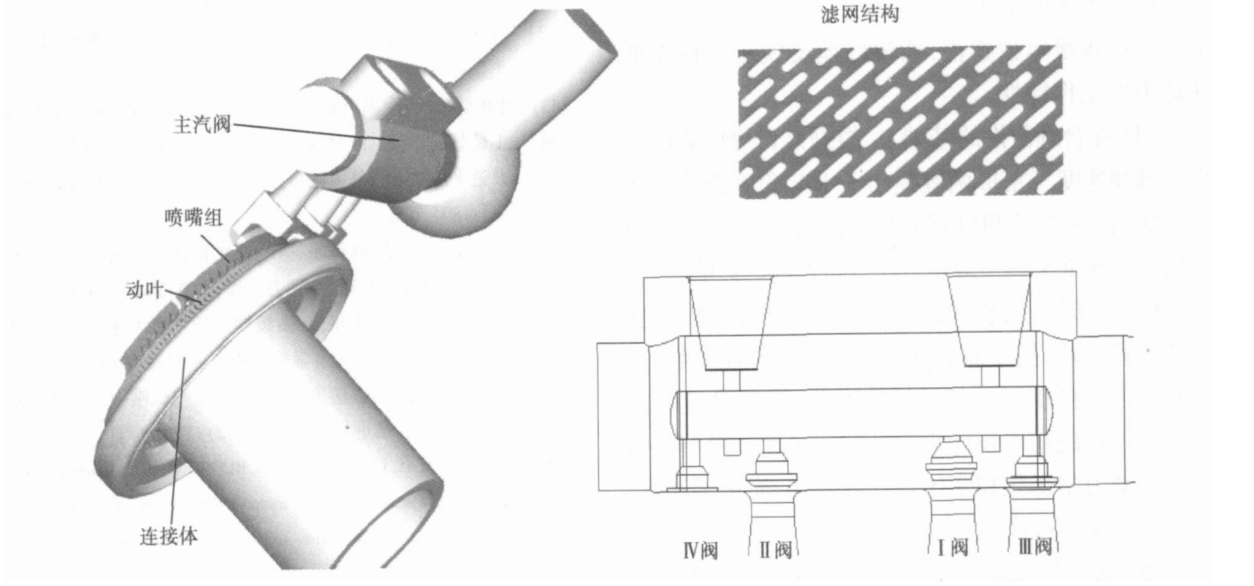


图 1　汽轮机头部流场几何模型及滤网, 调节阀的示意图

收稿日期: 2006－10－25;　修订日期: 2006－11－27

作者简介: 冀春俊(1962－), 男, 河南镇平人, 大连理工大学教授, 博士.

图 1 是汽轮机头部流场模型, 包括主汽速关阀, 调节阀, 调节级及调节级与反动级的连接体。该汽轮机正常工作时, 群阀提板上的Ⅰ、Ⅱ、Ⅲ调节阀开, Ⅳ阀关闭。为提高头部流场计算正确性, 将调节级作为头部流场计算的延伸, 每个调节阀对应一个喷嘴组, 每个喷嘴组由 7 个叶片组成。动叶包含 131 个叶片, 动叶部分和连接体内壁为转动部件(转速为 5 270 r/min), 其余部分均为静止部件。

2　计算方法, 网格及边界条件

2.1　控制方程

计算流体力学(CFD)法是对流场的控制方程用计算数学的方法将其离散到一系列网格节点或中心上求其数值解的一种方法。这些控制方程分别是连续性方程, 动量方程和能量方程。它们联立组成的方程组称为 $N-S$ 方程组, 是流体流动所遵守的普遍规律[3]。

2.2　计算网格的生成

图 2　计算网格

由于主汽调节阀的结构比较复杂, 流速较低, 并有滤网存在, 对其进行了四面体网格划分。滤网厚度为 1 mm, 生成网格时忽略了其厚度。调节级与反动级的连接体流速较高, 因此对其进行了六面体网格的划分, 整个计算模型所用的网格总数约 210 万, 生成网格的质量均达到了计算的要求, 可以保证计算的精度, 计算网格如图 2 所示。

2.3　湍流模型及介质属性

本算例采用 SMPISO 算法和高 Reynolds $k-\varepsilon$ 湍流模型。在近壁区域则用壁面函数来处理。工作介质为水蒸气, 由于流速较高, 作为可压缩流体处理。

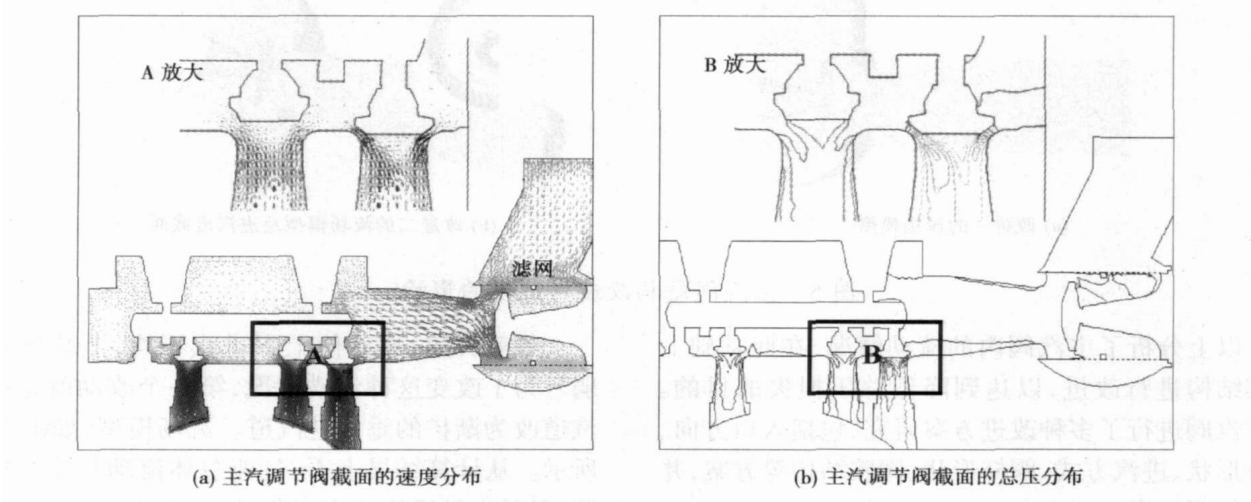
2.4　动静耦合及边界条件

动叶和静止部件间的耦合选用隐式的多重坐标系方法。基本思想是通过指定动叶和静止部件不同的旋转速度, 在离散方程中加入适当的旋转项以模拟指定区域的旋转运动, 实际上网格并不旋转。这种方法在处理旋转机械时比较灵活, 计算采用稳态进行。在计算时选择的坐标系为圆柱坐标系。根据已知条件在进口和出口处设置边界条件, 进口给定条件包括气流的总温 $T_{intot}=643.28$ K, 总压 $P_{intot}=1.102\ 25$ MPa。出口给定压力 $P_{out}=0.486\ 69$ MPa。滤网处为 BAFFLE 边界, 延伸部分的内壁为旋转速度为 5 270 r/min的壁面边界, 其余均为静止壁面边界。

3　计算结果分析及结构改进

在搭建的数值平台上, 对汽轮机头部流场进行了数值计算, 得到了汽流在汽轮机头部的流动情况。同时利用整体计算的结果分别对主汽阀、调节汽阀进行结构改进并用数值计算进行验证。

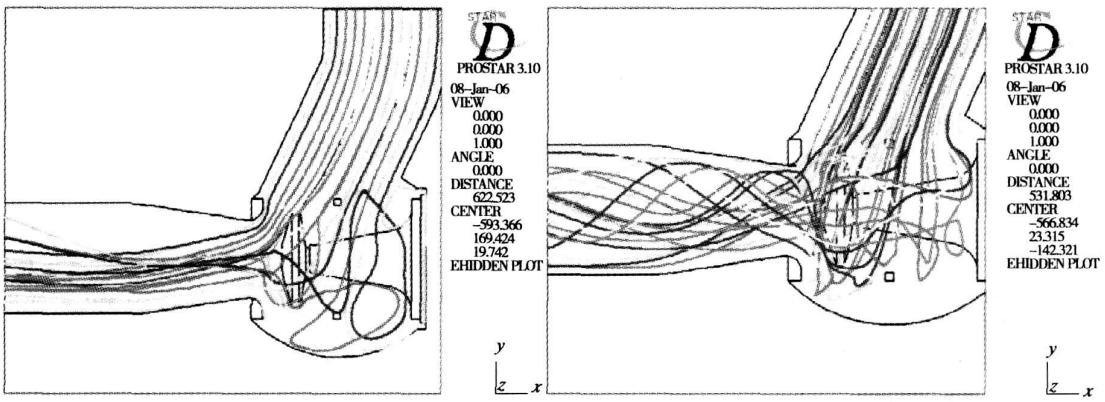
3.1　主汽速关阀计算结果分析及改进



(a) 主汽调节阀截面的速度分布　　(b) 主汽调节阀截面的总压分布

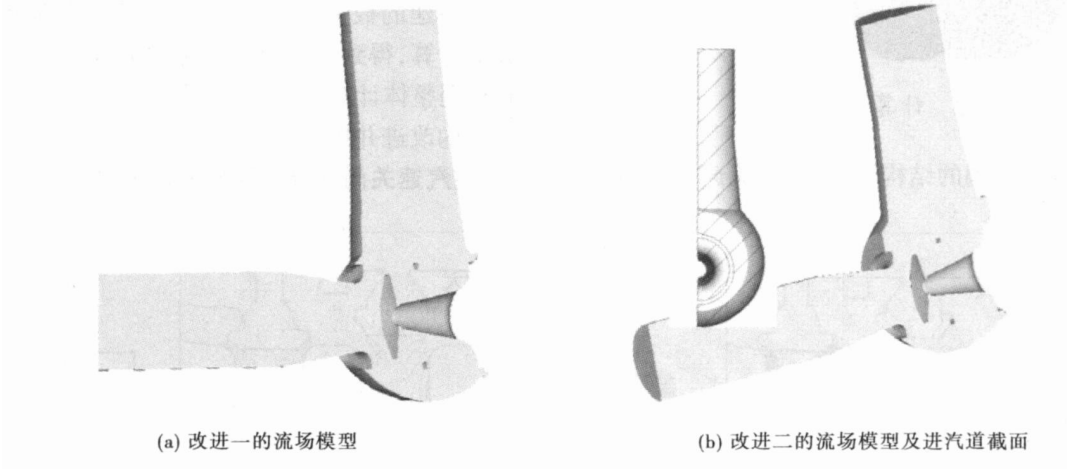
图 3　头部流场计算结果

图3 显示了主汽速关阀中截面的速度矢量和总压分布。从图 4 的流线图上看出，气流在阀腔内的流动非常复杂，尤其在气流流过滤网后绕流阀碟流入喉口时，多处流线螺旋缠绕。中科院工程热物理研究所曾对主汽阀滤网流动进行了实验研究[3]，指出滤网的出现改变了阀腔中的流场分布，在一定程度上抑制了“流动失稳”现象；主汽阀中的滤网的存在虽然有一定的压损，但对稳定阀门内部流场具有一定的作用。图 4(a)为无滤网时数值计算的流线图。此时气流多数直接从进汽腔以较大的角度流入喉口，在主汽阀喉口后的扩散段流体在较大的逆压梯度作用下产生了流动分离。加入滤网后，比较计算结果可看出，滤网大大改变了气流流动，稳定了速度场，这一点与实验结果吻合。阀腔内速度流场的分布详尽地反映了气流的流动情况，气流在进汽道内流动比较均匀，而后通过滤网流入阀腔。由于滤网上的孔均匀分布，只有少量气流从上端直接流过滤网，其余气流在阀壳与滤网构成的等截面通道中流动，流动过程中气流逐渐流入阀腔。从图 3(a)可以看出，从滤网下端进入阀腔的气流速度明显大于上端气流速度，气流流过阀碟后气流有上扬的趋势。在阀碟表面 A 区出现了流动分离现象。



(a) 无滤网时主汽阀流线图　　(b) 主汽阀原结构流线图

图 4　滤网对主汽内流动的影响



(a) 改进一的流场模型　　(b) 改进二的流场模型及进汽道截面

图 5　主汽阀结构改造对流动的影响

以上分析了主汽阀内的流动情况，在此基础上对其结构进行改进，以达到降低总压损失的目的。对主汽阀进行了多种改进方案研究，包括入口方向、入口形状、进汽方式、阀杆形状、阀腔结构等方案，并提出如下方案。

3.1.1　方案一——垂直进汽

如前所述，滤网使上下进汽不均，上端气流较弱。为了改变这种流动状况，第一个改动就是把进汽道改为渐扩的垂直进汽道。流场模型，如图 5(a)所示。从计算结果上看，气流总体流动与原结构相同，只是上端进汽速度有所增强，下端上扬的趋势有所减弱，总压损失有所降低。

3.1.2　方案二—垂直进汽, 阀壳与滤网通道截面为渐缩截面

为了进一步改善流动情况, 使滤网上下进汽均匀, 减小下端气流上扬, 采用垂直进汽, 阀壳与滤网通道为渐缩截面, 如图 5(b)所示。计算结果显示, 此结构一定程度上减弱了气流上扬趋势, 流动更加稳定。

3.1.3　方案三—垂直进汽, 阀壳与滤网通道截面为渐缩截面, 阀碟下端为 $R=20$ 的倒圆角

计算分析时已经发现气流绕流阀碟时出现的流动分离。为了解决这个问题, 对阀碟下端倒了一个 $R=20$ 的圆角, 从计算结果来看, 这种方法有效地消除了流动分离, 总压损失进一步降低。表 1 给出了几种改进方案后的总压损失情况。

表 1　几种改进方案的总压损失

	进口总压 /MPa	出口总压 /MPa	总压损失 /%
原型	1.012 3	0.990 4	2.163
方案一	1.012 3	0.991 32	2.073
方案二	1.012 3	0.991 72	2.033
方案三	1.012 3	0.992 15	1.991

3.2　调节阀的计算结果分析及改进

气流流经调节阀, 多数主流直接从阀碟流入, 少数绕流过阀碟从另一侧流入。气流在流道中心相撞, 造成阀碟中心附近的低压区, 阀碟正下方存在微小分离, 阀碟下方出现高损失区。从计算结果来看, 阀门开度越小, 调节阀喉部速度越不均匀, 总压损失越大。修改阀门型线和在阀碟下方加一个延长段是改变此处低压区的有效措施, 并进行改进。

3.2.1　改进一—阀碟型线为抛物线

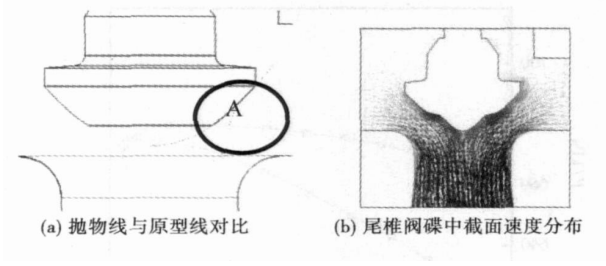


图 6　抛物线与原型线对比

对阀门型线的修改尝试了许多方案, 包括抛物线型、双曲线型等。经过计算比较, 其中有效的线型为 $y=0.01x^2$, 如图 6(a)所示。该种改进是在原结构阀碟最大直径和阀碟高度不变的前提下, 仅对气流流动影响最大的阀碟下端进行修改。计算结果表明气流流动较顺畅, 低压区面积也有所减小, 总压损失进一步降低。

3.2.2　改进二—阀碟型线为 $y=0.01x^2$ 的抛物线, 尾椎阀碟

为消除低压区, 尝试设计一种尾椎型阀碟, 其底面与阀碟底面均匀过渡。对这种结构进行了不同尺寸的数值实验, 得到一组总压损失最小的结构, 图 6(b)为该结构的流场。计算结果表明, 尾椎阀碟和阀座之间形成具有一定型线的流道, 改变了通道的面积关系, 具有约束气流的作用。此结构有效地破坏了阀碟底部的低压区, 气流流动紧贴阀碟表面, 没有发生两侧气流的对撞, 提高了阀门的气动性能的同时, 使阀头的工作稳定性得以加强。

表 2　改进调节阀 I 后总压损失情况

	进口总压 /MPa	出口总压 /MPa	总压损失 /%
原型线	0.989 07	0.952 05	3.743
改进一	0.989 07	0.959 49	2.991
改进二	0.989 07	0.959 11	3.029

4　结　论

对某型号电站给水泵汽轮机主汽调节阀及调节级流场在设计工况进行了内部三维粘性流动数值计算, 并对其中的流动进行了分析, 得出各部分的总压损失, 结果表明:

(1) 完整的头部流场计算可以保证边界条件的正确性从而提高计算精度, 同时有利于比较各部分总压损失情况, 为结构改进指明重点方向。

(2) 主汽阀中的滤网虽然造成了一定的压损, 但对稳定阀门内部流场具有一定的作用。

(3) 主气阀碟型线对于流动有着重要的影响, 一定的导角圆滑可以有效消除气流分离, 从而使流动损失较小。

(4) 合适的抛物线线型可以有效地降低调节阀的总压损失。采用尾椎阀碟可以基本上消除流动漩涡, 从而改善气流流动、减小低压区并有效地防止气流对撞以提高阀门的气动性能和工作稳定性。

参考文献:

[1] 王仲奇, 秦　仁. 透平机械原理[M]. 北京: 机械工业出版社, 1998.

[2] JOHN D, ANDERSON J R. Computational fluid dynamics[M]. 北京: 清华大学出版社, 2002.

[3] 徐克鹏, 蔡　虎. 600 MW 汽轮机高压联合进汽阀内部流场的数值模拟[J]. 工程热物理学报, 2001, 22(5): 555—558.

(编辑　渠　源)

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

By using SMPISO algorithm and high Reynolds $k-\varepsilon$ turbulent models, a numerical calculation and analysis was performed of the flow fields in the head portion of a steam turbine destined for power-plant feedwater pumps. The calculation domain includes the entire flow fields in the head portion, mainly involving the main steam quick-closing valve, regulating valves and stages. Maximum efforts have been made to ensure the absence of geometric distortions in order to guarantee the accuracy of calculation results. On the basis of flow analyses, an improvement in the steam admission chamber, regulating valves and profiles of the regulating stages has been conducted by employing a numerical test method, effectively lowering the total pressure loss of flows in various parts. With a simultaneous enhancement of steam turbine efficiency the operating stability of the regulating valves has been ensured. **Key words**: turbine, main steam quick-closing valve, regulating valve, regulating stage, numerical simulation, improvement in structure

某氦气压气机三维优化设计= **The Three-dimensional Optimized Design of a Helium Compressor**[刊, 汉] / CHEN Ying, ZOU Ji-guo (Compressor Design Department, Harbin No. 703 Research Institute, Harbin, China, Post Code: 150036), WANG Song-tao (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). —504 ~511

By analyzing the features of the three-dimensional numerical simulation results for the design scheme of a helium compressor, given were the design direction and guiding principle for its three-dimensional optimized modification design. By employing such methods as adjusting the distribution of blade thickness and the curvature of blade trailing edge profile as well as terminal-bend technology etc., a full three-dimensional optimization design was performed of the aerodynamic design version of the prototype helium compressor. A contrast analysis was conducted of the numerical simulation results before and after the three-dimensional optimized design. The efficiency of the compressor after the three-dimensional optimized design has been increased by 2 percent, effectively limiting the further development of secondary flows. **Key words**: helium compressor, blade modeling, three-dimensional optimization, controllable diffusion blade profile

煤高温气化—高温贫氧燃烧一体化系统的研究与开发= **Research and Development of an Integrated System Featuring Coal High-temperature Gasification-high-temperature Oxygen-deficient Combustion**[刊, 汉] / CAI Jiu-ju, TIAN Hong, WANG Lian-yong, et al (National Key Laboratory on Environmental Protection and Ecological Industry, Northeastern University, Shenyang, China, Post Code: 110004) // Journal of Engineering for Thermal Energy & Power. — 2007, 22(5). —512 ~516

To address the energy source and environment protection problems of the majority of medium and small-sized industrial furnaces or kilns in China due to the use of traditional direct coal-combustion technology, the authors have independently designed and developed an integrated system of coal high-temperature gasification and high-temperature oxygen-deficient combustion along with a description of the process chart and thermodynamic characteristics of the system. By a combination of experimental study and mathematical simulation, studied was the coal gasification and combustion characteristics of the integrated system. The research results show that by raising the temperature of the air gasification agent it is possible to reduce the air-coal ratio and enhance the coal-gas heat value, gasification efficiency and intensity. When the air gasification agent temperature is increased from the normal temperature to 1 050 ℃, the coal gas heat value will be increased by 33%, the air-coal ratio reduced by 43%, and the gasification intensity almost doubled. To raise the temperature of the combustion-supporting air can enlarge the flame volume of combustion gas in the heating furnace and the furnace temperature distribution tends to be uniform, leading to a significant increase in thermal efficiency and a dramatic reduction in NO_x generation concentration. The thermal efficiency of the above system will be more than doubled as compared with that of a normal-temperature coal-gasification furnace and a heat-exchange type of steel-rolling heating furnace system. Moreover, its unit product energy consumption will be reduced by 50%. **Key words**: coal gasification, high-temperature oxygen-deficient combustion, temperature of gasification agent, combustion-supporting air temperature, thermal efficiency, thermal connection