

# 离心叶轮内三维湍流流场的数值分析

谭大治, 袁 新

(清华大学 热能工程系, 北京 100084)

**摘 要:** 应用 LU-SGS-GE 隐式格式和改良型高阶 MUSCL TVD 格式, 通过求解全三维可压缩 Reynolds 平均的 Navier-Stokes 方程和低 Reynolds 数  $q-\omega$  双方程湍流模型, 对后弯角  $30^\circ$  的 Krain 离心叶轮在设计工况下通道内复杂的三维流动进行了计算。计算结果与实验数据符合良好, 说明该方法适用于大逆压梯度流动的数值模拟, 也可以用来深入研究离心压气机内各种复杂的三维流动现象。

**关 键 词:** 数值模拟;  $q-\omega$  湍流模型; 高分辨率格式; MUSCL TVD 格式; LU-SGS-GE 隐式格式

中图分类号: TK474.8 文献标识码: A

## 1 前 言

离心压气机由于结构简单、单级增压比高, 在各工业行业中有着广泛的应用。在过去的几十年里, 由于实验仪器、实验手段的进步和数值方法的发展, 我们对离心压气机内复杂的三维流动现象和机理得到进一步的了解。其中文献 Eckardt<sup>[1]</sup> 和 Krain<sup>[2~3]</sup> 的工作最具代表性。为了进一步提高压气机的效率, 需要更加深入地了解压气机内部流场的细节。而实验测量由于受实验经费、实验方案和实验周期的限制, 不可能大规模的进行, 这就需要依赖于大量快速而准确的数值计算来进行定量分析。本文采用高分辨率、高精度和高收敛率的数值计算方法求解可压缩 Reynolds 平均的 Navier-Stokes 方程和低 Reynolds 数  $q-\omega$  双方程湍流模型, 对 Krain 离心叶轮在设计工况下的内部流场进行了详细的数值模拟。

## 2 数值计算方法

### 2.1 控制方程

对于任意曲线坐标系  $\xi_i$ , 可压缩 Reynolds 平均的 Navier-Stokes 方程和双方程湍流模型建立在旋转角速度为  $\Omega$  的相对转动坐标系上, 可写成:

$$\frac{\partial Q}{\partial t} + \frac{\partial F_i}{\partial \xi_i} + \frac{1}{Re} D + S = 0 \quad (1)$$

式中:  $Q$ —未知变量的一般形式;  $F_i$ —对流通量矢量;  $D$ —粘性项;  $S$ —源项。离心力和 Coriolis 力已经附加到源项当中。

$$Q = J \begin{bmatrix} \rho \\ \rho w_1 \\ \rho w_2 \\ \rho w_3 \\ \rho E \\ \rho \vartheta_1 \\ \rho \vartheta_2 \end{bmatrix}, \quad FF_i = J \begin{bmatrix} \rho W_i \\ \rho w_1 W_i + \xi_{i,1} p \\ \rho w_2 W_i + \xi_{i,2} p \\ \rho w_3 W_i + \xi_{i,3} p \\ \rho H^r W_i \\ \rho \vartheta_1 W_i \\ \rho \vartheta_2 W_i \end{bmatrix},$$
$$D = - \frac{\partial}{\partial \xi_i} J \xi_{i,j} \begin{bmatrix} 0 \\ \tau_{j1} \\ \tau_{j2} \\ \tau_{j3} \\ \tau_{j1} w_1 + \omega_j(k) - q_j \\ \omega_j(\vartheta_1) \\ \omega_j(\vartheta_2) \end{bmatrix},$$
$$S = - J \begin{bmatrix} 0 \\ 0 \\ \rho \Omega (\Omega x_2 + 2w_3) \\ \rho \Omega (\Omega x_3 + 2w_2) \\ 0 \\ S_{\theta_1} \\ S_{\theta_2} \end{bmatrix} \quad (2)$$

收稿日期: 2003-03-03

基金项目: 国家重点基础研究专项经费资助项目(G1999022306); 国家自然科学基金资助项目(50076019)

作者简介: 谭大治(1978-)男, 湖南湘潭人, 清华大学硕士研究生

其中:  $\rho$ —密度;  $w_1, w_2, w_3$ —相对速度矢量的 3 个分量;  $E'$ —相对滞止内能;  $H'$ —转焓;  $J = \partial(x_1, x_2, x_3) / \partial(\xi_1, \xi_2, \xi_3)$  为坐标变换 Jacobian;  $\xi_{i,j} = \partial \xi_i / \partial x_j$ ;  $W_i = \xi_{i,j} w_j$  为逆变速度。

## 2.2 湍流模型

基于涡旋粘度假设的双方程模型能够有效地模拟分离流、剪切流等复杂的流动现象, 而逐渐取代 B—L 模型成为现在较为常用的湍流模型。应用最为广泛的应属 Jones—Launder、Chien 等人的  $\kappa-\epsilon$  模型, 而 Wilcox 的  $\kappa-\epsilon$  模型, Coakley 的  $q-\omega$  模型<sup>[4]</sup> 等在  $\kappa-\epsilon$  模型基础上发展而来的新型双方程模型也得到广泛的关注。文献[5] 详细地比较了这三种湍流模型。 $\kappa-\epsilon$  模型的适用范围广, 但有对初场敏感的弱点。而  $q-\omega$  模型计算量小, 收敛快, 湍流边界条件处理简单, 适用于可压缩湍流计算, 并对初场较为不敏感。本文采用  $q-\omega$  双方程湍流模型, 可表示为:

$$\begin{aligned} \mu_t &= Re C_{\mu} f_{\mu} \frac{\rho q^2}{\omega}, \theta_1 = q = \sqrt{k}, \theta_2 = \omega = \frac{\epsilon}{k}, \\ S_q &= \frac{1}{2} \left[ C_{\mu} f_{\mu} \frac{\varphi}{\omega^2} - \frac{2}{3} \frac{\Theta}{\omega} - 1 \right] \rho \omega q \\ S_{\omega} &= \left[ C_1 f_1 \left( C_{\mu} \frac{\varphi}{\omega^2} - \frac{2}{3} \frac{\Theta}{\omega} \right) - C_2 \right] \rho \omega^2 \end{aligned} \quad (3)$$

$$f_{\mu} = 1 - e^{-0.02 Re_t}, f_1 = 1 + 9 f_{\mu}, Re_t = Re \frac{\rho q \gamma}{\mu}$$

$$C_{\mu} = 0.09, C_1 = 0.055, C_2 = 0.83,$$

$$Pr_q = 0.8, Pr_{\omega} = 2.0$$

其中:  $\Theta$  和  $\varphi$  是湍流速度场的散度和应变率不变量:

$$\Theta = \frac{\partial u_i}{\partial x_i}, \varphi = \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \Theta \right] \frac{\partial u_i}{\partial x_j} \quad (4)$$

## 2.3 数值计算格式

为了求解任意曲线坐标系下的 Navier—Stokes 方程 采用了 LU—SGS—GE 隐式格式<sup>[6]</sup> 和改良型高分辨率 MUSCL TVD 格式<sup>[7]</sup> 以加快收敛速度及提高解的精度和分辨率。LU—SGS—GE 格式是在 Jameson 和 Yoon 的 LU—SGS 隐式格式基础上发展起来的, 由于采用了相似变换而不是近似处理来构造通量矢量的迎风 Jacobian 矩阵, 有效地避免了 LU—SGS 算法中构造近似的 Jacobian 矩阵所引入的数值耗散, 大大提高了计算的收敛速度。改良型高阶 MUSCL TVD 格式是一个四(五)阶高分辨率的 MUSCL TVD 类格式, 尤其适于捕捉粘性流场中的弱间断面和滑移面, 不仅可以精确地捕捉到

流场中出现的激波, 而且能够准确地模拟激波与边界层、激波与尾迹相互干扰等复杂的流动现象。

## 2.4 计算网格和边界条件

现在普遍采用的网格生成方法是偏微分方程网格生成技术, 但是这种方法在网格的疏密控制上存在难度, 特别是对疏密度无法事先控制, 对疏密的调节程度也十分有限。除此之外, 该方法的收敛速度也比较慢。本文所采用的网格生成技术的主体是几何法。这种方法的优点是容易控制网格的疏密, 而且调整方便, 可以根据需要确定网格的最小间距。这种方法还可以对网格的任意局部进行正交和光滑, 生成高质量的网格, 而且几何法的收敛速度很快。但几何法对初始网格比较敏感, 所以本文把几何法与求解偏微分方程方法结合在一起使用。首先用求解偏微分方程的方法对初始网格进行几十步迭代, 然后用几何法进行网格的最后生成。采用这种方法生成的计算网格正交性和光滑性都很好, 网格的生成速度很快。计算所采用的网格是 221(流向)×31(周向)×41(径向)的 C 型网格, 忽略叶顶间隙。

边界条件为: 进口边界给定进气总温、总压和绝对气流角, 出口边界给定出口静压, 叶轮上下游延伸段周向边界采用周期性边界条件, 固壁给定无滑移边界条件。计算中, 通过调整出口静压, 使进口质量流量与实验值相符。

## 3 计算结果及分析



图 1 Krain 离心叶轮

计算采用的 Krain 离心叶轮(见图 1)是德国宇航局(DFVLR)设计的高亚音速离心压气机半开式叶轮, Krain.H 对设计工况下该叶轮内部流场做了详细的激光测量。该叶轮后弯角达  $30^\circ$ , 共有 24 个叶片, 设计压比是 4.7, 设计转速为 22 363 r/min, 叶尖顶部最高速度为 470 m/s, 设计流量为 4 kg/s, 对应的叶轮出口压力为

265.978 kPa, 其具体的几何数据和测量结果可参考文献 [2~3]。图 2 是测量截面示意图, 其中各测量截面 I~VI 分别位于子午面轮廓曲线的 0, 0.2, 0.4, 0.6, 0.8, 1.0 处。

### 3.1 无因次子午速度与相对气流角

图 3 给出的是无因次子午速度 ( $C_m/U_2$ ,  $C_m$  为子午速度,  $U_2$  为叶尖处旋转速度) 在 III、IV 截面处的等值线图, “PS” 表示压力面, “SS” 表示吸力面, “u” 表示叶轮转动方向。可以看出计算结果与实验结果虽然在定量上有一些差距, 但总体吻合良好。

计算结果与实验结果在定量上的差距, 主要是由于计算中忽略了叶顶间

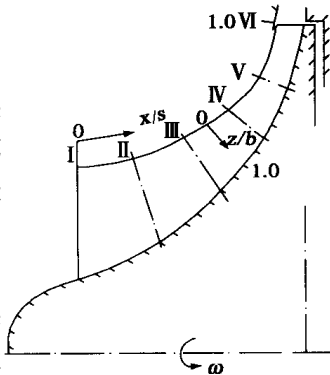


图 2 测量截面示意图

隙的缘故。在叶轮通道前半部分, 叶顶间隙流的影响并不显著, 故 II、III 截面的  $C_m/U_2$  等值线的分布很相似, 并且与实验结果吻合得很好。III 截面以后, 叶顶间隙流的影响逐渐增大, 在轮盖和吸力面附近形成一股低速流, 并且向叶轮通道中间扩展, 低速流的中心远离吸力面。在 III、V 截面, 计算结果也模拟出了轮盖附近的低速区, 但是由于未考虑间隙流的影响, 低速区的位置与实验结果不太相符。

图 4 给出的相对气流角 ( $\beta = \arctg(W_m/W_T)$ ,  $W_m$  为子午速度,  $W_T$  为相对周向速度) 在 III、IV 截面处的等值线图, 总的来说, 等值线的分布与实验值是一致的。在叶顶附近的计算结果则要小于实验结果, 这也是由于数值计算中忽略了叶顶间隙的缘故。

### 3.2 极限流线与二次流

图 5 为压力面的极限流线图, 由图中清晰可见压力面中部有一条分离线, 这可能是因为流道中部转折较大引起的, 但分离并不严重。压力面靠近根部处还有一条附着线。

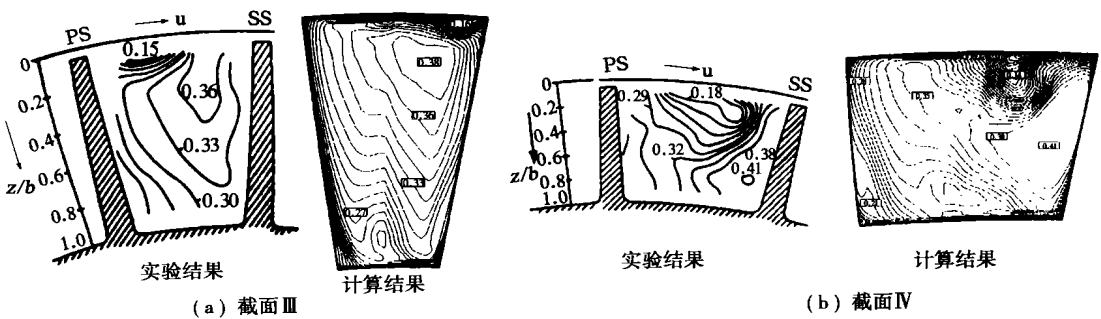


图 3  $C_m/U_2$  分布图

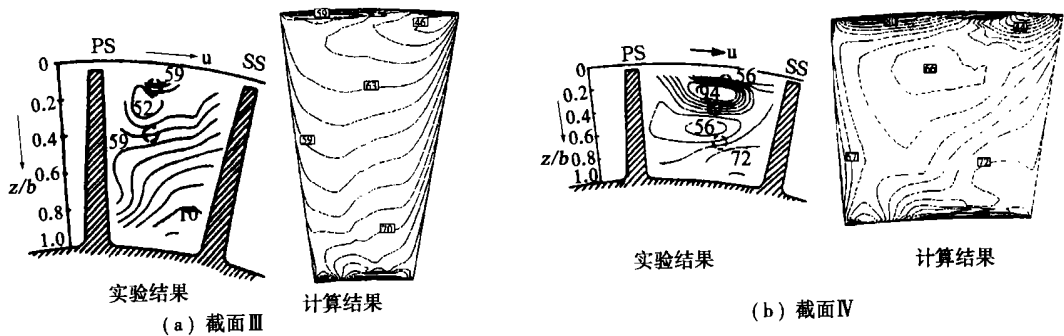


图 4 相对气流角分布图

为了提高各种类型叶轮机械的效率, 展开了内流流动和损失机理的研究, 而二次流的研究就是其中很重要的一部分。叶轮中高度三维性的二次流是很难计算、控制和优化的。在轴流式叶栅研究中有一些相关报道, 而在离心式方面文献很少。本文中所定义的二次流速度矢量为三维 NS 计算所得的粘性速度矢量与其在流动方向坐标网格投影的向量差。这个二次流速度矢量在  $S_3$  坐标面上投影, 并作微团的面示踪, 就可显示出二次流旋涡。

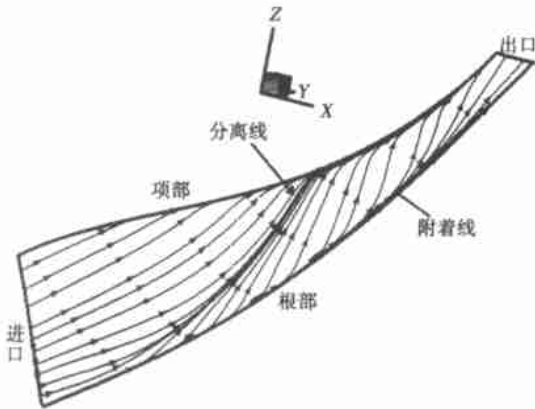


图5 压力面上的分离线与附着线

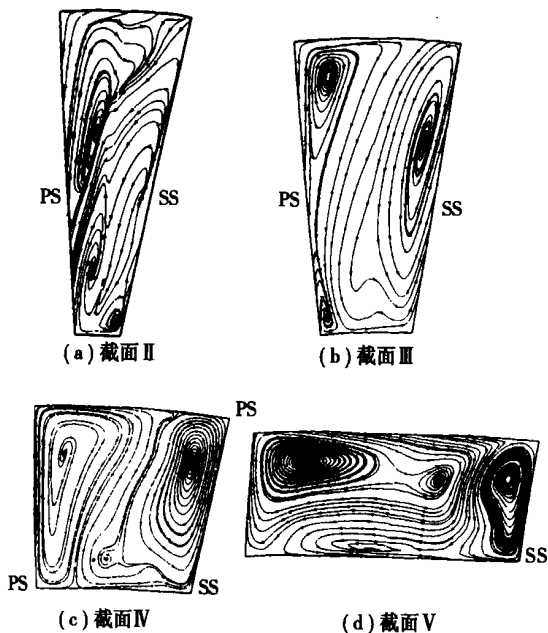


图6 II~V截面上的二次流旋涡微团示踪图

图6给出了II~V截面的二次流微团示踪图, PS为压力面, SS为吸力面。总的看来, 流道内存在一对明显的二次流旋涡, 靠近压力面的为顺时针旋涡, 靠近吸力面的为逆时针旋涡。

由图中可以清楚看出二次流旋涡的发展过程, 从叶轮进口到出口, 靠近压力面的顺时针旋涡影响区域不断扩大, 而吸力面附近的逆时针旋涡则被压缩, 这点在截面V上最为明显, 顺时针旋涡几乎影响了整个流道的3/4。

## 4 结 论

应用LU-SGS-GE隐式格式和高精度、高分辨率的MUSCL TVD格式数值模拟了Krain离心叶轮设计工况下的湍流流场。数值计算的结果与实验结果符合良好, 说明本文的方法适用于大逆压梯度流动的计算, 为以后深入研究离心机械内复杂的三维湍流现象打下基础。

## 参考文献:

- [1] ECKARDT D. Detailed flow investigations within a high-speed centrifugal compressor impeller[J]. *ASME Journal of Fluids Engineering* 1976 98(3): 390-402.
- [2] KRAIN H, HOFFMAN W. Verification of an impeller design by laser measurements and 3D-viscous flow calculations[R]. ASME Paper, 89-GT-159 1989.
- [3] HAH C, KRAIN H. Secondary flows and vortex motion in a high-efficiency backswep impeller at design and off-design conditions[J]. *ASME J of Turbomachinery*, 1990, 112: 7-13.
- [4] COAKLEY T J, HUANG P G. Turbulence modeling for high speed flows [R]. AIAA Paper 92-0436 1992.
- [5] 袁 新. 可压缩粘性流动中双方程湍流模型的选择[J]. *工程热物理学报*, 1998, 19(4): 427-432.
- [6] YUAN XIN, DAIGUJI HISAAKI. A specially combined lower-upper factored implicit scheme for three-dimensional compressible navier-stokes equations[J]. *Computers and Fluids*. 2001, 30(3): 339-363.
- [7] DAIGUJI H, YUAN XIN, YAMAMOTO S. Stabilization of higher-order high resolution scheme for the compressible navier-stokes equations[J]. *Int J of Numerical Methods for Heat & Fluid Flow*, 1997, 7(2/3): 250-274.

( 辉 编辑)

emerge with regard to the definition of the exergy efficiency. In such a case a similar method for coping with the defect can be put into use. **Key words:** exergy efficiency, definition method, analysis, mathematical expression, heat pump  
壳管式海水换热器污垢状况的可用评价方法研究 = **An Investigation of the Method for Evaluating the Exergy Loss Relating to a Shell-and-tube Seawater Heat Exchanger Fouling Condition** [刊, 汉] / JIANG Zhu-xing, LIU Xiao-hong (Guangzhou High Technical School of Navigation, Guangzhou, China, Post Code: 510725) // Journal of Engineering for Thermal Energy & Power. — 2003, 18(6). — 558 ~ 560

An analysis is given of the impact, which the fouling of a shell-and-tube seawater heat exchanger at the tube side and the resulting changes in heat transfer intensity and flow pressure drop will have on the exergy loss of the heat exchanger. A method is proposed to evaluate heat exchanger fouling condition by making use of the exergy loss factor. This method has been found to be more comprehensive and straightforward for evaluating heat exchanger fouling than the one based on heat resistance detection and measurement. **Key words:** heat exchanger, exergy loss, fouling

城市污泥和煤混燃特性的热重分析法研究 = **A Study of the Characteristics of Mixed Burning of Municipal Sewage Sludge and Coal by a Thermogravimetric Method** [刊, 汉] / GU Li-feng, CHEN Xiao-ping, ZHAO Chang-sui, et al (Education Ministry Key Laboratory on Clean Coal Power Generation and Combustion Technology under the Southeastern University, Nanjing, China, Post Code: 210096) // Journal of Engineering for Thermal Energy & Power. — 2003, 18(6). — 561 ~ 563

A thermogravimetric method is employed to study such parameters as ignition temperature, activation energy and comprehensive combustion characteristics in connection with the burning of municipal sewage sludge and coal as well as their blends. The result of the study indicate that as compared with the case of burning only coal the combustion of blends has resulted in an enhanced activation energy with a lowering of the ignition temperature and a reduction of comprehensive combustion performance. During the blend burning process the municipal sewage sludge and coal have basically maintained their respective devolatilization characteristics with the coal burning exhibiting a more conspicuous behavior in this respect. **Key words:** municipal sewage sludge, coal, mixed combustion of sewage sludge and coal, thermogravimetric method

余热多级动力回收系统及其优化 = **Waste Heat Multi-stage Recovery System and It Optimization** [刊, 汉] / LIU Ye-kui, WANG Li, YAN Wen-jun, et al (College of Environmental and Chemical Engineering under the Xi'an Jiaotong University, Xi'an, China, Post Code: 710049) // Journal of Engineering for Thermal Energy & Power. — 2003, 18(6). — 564 ~ 567, 576

Taking into account the commonly seen phenomenon of incomplete recovery of waste heat from reactors a multi-stage system of reaction heat recovery is proposed based on a single-stage system of waste heat recovery. With the net power output serving as an objective function an optimization of the multi-stage recovery system was conducted. It can be demonstrated that the multi-stage system of waste heat recovery is markedly superior as compared to the single-stage one in terms of heat recovery efficiency. **Key words:** heat recovery, waste heat, optimization, multiple stage, model

离心叶轮内三维湍流流场的数值分析 = **Numerical Analysis of a Three-dimensional Turbulent Flow Field in a Centrifugal Impeller** [刊, 汉] / TAN Da-zhi, YUAN Xin (Department of Thermal Engineering, Tsinghua University, Beijing, China, Post Code: 100084) // Journal of Engineering for Thermal Energy & Power. — 2003, 18(6). — 568 ~ 571

Through the use of a LU-SGS-GE implicit scheme and an improved version of high-order MUSCL TVD scheme and by solving for a full three-dimensional compressible Reynolds time-averaged Navier-Stokes equation and low Reynolds number  $\tau\omega$  dual equation turbulent flow model calculated is the complicated three-dimensional flow in the impeller passage of a centrifugal compressor. The results of the calculation are in good agreement with those obtained by tests. This shows that

the method under discussion is applicable for the numerical simulation of flows featuring great inverse-pressure gradients. Moreover, it can also be employed to conduct the in-depth research of various complicated three-dimensional flow phenomena in a centrifugal compressor. **Key words:** numerical simulation,  $q-\omega$  turbulent flow model, high-resolution scheme, MUSCL TVD scheme, LU-SGS-GE implicit scheme

流道几何参数对低压导向器气动性能的影响 = **The Impact of Flow-passage Geometric Parameters on the Aerodynamic Performance of a Low-pressure Guide Vane Assembly** [刊, 汉] / AN Bai-tao (Institute of Engineering Thermophysics under the Chinese Academy of Sciences, Beijing, China, Post Code: 100080), WANG Shong-tao, HAN Wan-jin, WANG Zhong-qi (Harbin Institute of Technology, Harbin, China, Post Code: 150001) // Journal of Engineering for Thermal Energy & Power. — 2003, 18(6). — 572 ~ 576

By using a numerical calculation method a numerical simulation was conducted of a prototype and modified low-pressure guide vane assembly. The meridian flow passage of the latter has a relatively long casing transition section with the casing assuming a rather great outer-wall divergence angle. Gas flow is liable to separate at the outer wall, resulting in a deterioration of the cascade inlet flow of the guide vane assembly. The results of the numerical study indicate that the optimized design of the casing outer wall profile and blade contour has improved the overall aerodynamic performance of the guide vane assembly. In the case of identical calculation-domain initial conditions there is a marked reduction of outlet energy loss of the guide vane assembly. **Key words:** low-pressure guide vane assembly, numerical simulation, twisted blade, end wall profile, optimized design

CFB 锅炉煤成灰特性的 6 参数模型研究 = **Six-parameter Model Study of Ash Formation Characteristics of Coals in a Circulating Fluidized Bed Boiler** [刊, 汉] / YANG Hai-rui, LU Jun-fu, XIAO Xian-bin (National Key Laboratory of Clean Coal Combustion Technology under the Tsinghua University, Beijing, China, Post Code: 100084), M. WIRSUM (Siegen University, Siegen, Germany) // Journal of Engineering for Thermal Energy & Power. — 2003, 18(6). — 577 ~ 581

An experimental method involving static combustion and cold state vibration screening is employed to study the ash formation characteristics of several Chinese coal ranks with the ash formation data-base being given for these coals. Through an analysis of the ash formation data-base of different coals it is found that the distribution of ash formation of an arbitrary narrow sieve fraction assumes a bimodal nature, i. e. being composed of relatively fine and soft ash components and relatively coarse and hard ash ones. These two kinds of ash components can simultaneously satisfy a Rosin-Rommler distribution. A six-parameter model has been set up, which can predict the ash mass portion and relevant particle size distribution of each kind of ash components. By the use of a weighting method the ash formation distribution of coal with a wide sieve fraction can be obtained. **Key words:** ash formation characteristics, soft ash, hard ash, Rosin-Rommler distribution, six-parameter model

近红外光谱分析技术预测煤质挥发分含量模型的研究 = **A Model Study Concerning the Use of Near Infrared Spectral-analysis Technology for Predicting Volatile Content of Coals** [刊, 汉] / LI Feng-rui, TANG Yu-guo (Changchun Research Institute of Optical Precision Instruments and Physics under the Chinese Academy of Sciences, Changchun, China, Post Code: 130021), XIAO Bao-lan (Department of Thermal Engineering, Jilin University, Changchun, China, Post Code: 130025) // Journal of Engineering for Thermal Energy & Power. — 2003, 18(6). — 582 ~ 583

A near infrared spectral method is used to conduct the on-line analysis of various kinds of coal with an emphasis on the determination of coal volatile content. On the basis of reading numerous spectrograms and with the use of a multiple regression method for data analysis and processing the authors have set up a multi-linear model. The correlation factor between the model-derived forecast volatile content of coal and the artificially tested standard value is found to be 0.96. The