第22卷第4期
2007年7月

热　能　动　力　工　程
JOURNAL OF ENGINEERING FOR THERMAL ENERGY AND POWER

Vol.22, No.4
Jul., 2007

文章编号: 1001—2060(2007)04—0367—04

基于双流体模型的湿蒸气凝结流动三维数值模拟

吴晓明, 李　亮, 李国君, 丰镇平
(西安交通大学 能源与动力工程学院, 陕西 西安 710049)

摘　要: 建立湿蒸气凝结流动的双流体模型, 考虑了湿蒸气汽液两相流动中相间速度滑移、耦合以及湍流扩散作用的影响。针对蒸汽透平叶栅中流动的湍流特性, 在单相湍流计算中数值模拟精度相对良好的两方程 SST $k-\omega$ 湍流模型基础上, 参照颗粒湍能输运方程理论, 推导建立了湿蒸气两相流动 SST $k-\omega-k_p$ 湍流模型, 模型中引入了液相粘性、导热及扩散系数等拟流体概念。对一直列叶栅中存在自发凝结的湿蒸气流动进行了三维数值模拟, 结果表明: 与中心截面相比, 端壁附近汽流首先出现大量凝结核, 并较早恢复到平衡状态, 由于涡系结构的存在使得沿叶高汽液两相叶栅出口马赫数之间存在一定差异。本文建立的模型提高了湿蒸气凝结流动三维数值模拟的精度, 更好地揭示叶栅中凝结流动的相间作用。

关　键　词: 透平叶栅; 湿蒸气; 凝结流动; 双流体模型; 两相湍流模型; 数值模拟

中图分类号: O354　　文献标识码: A

符号说明

v_i—汽相速度分量	v_{pi}—液相速度分量
ρ—湿蒸气密度	ρ_g—蒸汽密度
ρ_p—液相表观密度	N—水滴数
Y—湿度	$\dot{m}$—质量凝结速率
J—成核率	v_p—液相湍流粘性系数
k_p—液相湍能	τ_{ip}—液相拟粘性应力

下　标

g—汽相参数	p—液相参数

引　言

汽轮机中的湿蒸气流动是伴随有相变和传热过程的两相非平衡凝结流动, 再加上其固有的三元、非定常及湍流特性, 使得流动十分复杂。目前湿蒸气流动的测量很困难, 而且系统的试验研究成本十分昂贵, 在这种情况下, 有必要建立合理、准确、可靠的数值模型, 从数值模拟角度进行研究。

湿蒸气凝结流动数值模型可分为单流体模型、颗粒轨道模型以及双流体模型3种。其中: 单流体模型也称作无滑移模型, 它忽略了由于凝结造成的动力学不平衡, 仅求解汽相 $N-S$ 方程和液相组分方程, 其特点是简单, 但误差较大; 颗粒轨道模型在 Lagrange 坐标系中处理液相问题, 考虑液滴与蒸汽间速度的滑移及温度的差异, 而且认为这些滑移与扩散漂移无关, 最初的确定轨道模型不考虑扩散现象[1], 后来发展的随机轨道模型必须用 Monte-Carlo 法计算[2], 由于轨道数目必须足够多, 因此计算量非常大。

考虑到湿蒸气凝结流中的颗粒相液滴具有尺寸小、数量巨大的特点, 本文采用双流体模型来模拟湿蒸气两相凝结流动, 其液相同样在 Euler 坐标系中处理, 按照该模型的概念直接封闭湍流两相时均守恒方程组, 因此是比较完整和严格的两相湍流模拟方法。

对于颗粒相湍流的处理, 传统的颗粒湍流代数模型以颗粒追随当地流体湍流为基础[3], 即颗粒只取决于当地流体的湍流特性, 颗粒的扩散与脉动只能小于流体, 颗粒越大其脉动越小、扩散越慢, 这与不少学者在气固两相流实验中观察到的现象相反。鉴于此, 周力行提出了颗粒湍能输运方程理论[4~5], 得出 $k-\varepsilon-k_p$ 两相湍流模型, 应用于有旋突扩气固两相流动模拟中取得了良好的效果。本文仿照该理论, 基于透平叶栅内的单相湍流计算中表现良好的 SST $k-\omega$ 模型, 推导建立了 SST $k-\omega-k_p$ 两相湍流模型, 并将其应用于透平叶栅中存在自发凝结的湿蒸气两相流动模拟中。

1　两相凝结流动的湍流时均方程组

文献[6]基于无滑移假设建立了湿蒸气非平衡凝结流动的单流体数值模型, 在 Euler 坐标系中求解汽相 $N-S$ 方程和液相的连续方程和组分方程。

湿蒸气凝结流属于稀疏悬浮流, 相对于流动时间, 液滴间的碰撞可忽略不计。本文从两相湍流时

收稿日期: 2006—12—15
基金项目: 国家自然科学基金资助项目(50336050)
作者简介: 吴晓明(1982—), 男, 湖北江陵人, 西安交通大学博士研究生.

均守恒方程组出发[4]，按双流体模型概念进行封闭，完整地考虑了湿蒸气两相流动中相间滑移、耦合及湍流扩散的影响。

忽略非定常关联项汽相密度的脉动、传质的脉动及阻力项的脉动，并对汽相动量、能量及组分方程中关联项取与单相流相同的模拟方法，同时对液相方程中关联项取梯度模拟，即：

$$\rho'_g=0,\ \dot m'=0,\ \overline{Y'(v'_i-v_{pi}')}=0,$$

$$-\overline{N'v_{pi}'}=\frac{\nu_p}{\sigma_p}\frac{\partial N}{\partial x_i},$$

$$-\overline{Y'v_{pi}'}=\frac{\nu_p}{\sigma_p}\frac{\partial Y}{\partial x_i},\ -\overline{Y'v_{pj}'}=\frac{\nu_p}{\sigma_p}\frac{\partial Y}{\partial x_j},$$

$$-\overline{(Y'v_{pj}'v_{pi}'}+Y\overline{v_{pj}'v_{pi}'})=\nu_p Y\left(\frac{\partial v_{pi}}{\partial x_j}+\frac{\partial v_{pj}}{\partial x_i}\right)$$

可得湿蒸气两相凝结流动的湍流时均方程组：

汽相连续方程：

$$\frac{\partial\rho_g}{\partial t}+\frac{\partial(\rho_g v_j)}{\partial x_j}=-\rho\dot m \tag{1}$$

汽相动量方程：

$$\frac{\partial(\rho_g v_j)}{\partial t}+\frac{\partial(\rho_g v_i v_j)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\times\left[\mu_e\left(\frac{\partial v_i}{\partial x_j}+\frac{\partial v_j}{\partial x_i}\right)\right]+\frac{\rho_p}{\tau_{rp}}(v_{pi}-v_i)-\rho\dot m v_i \tag{2}$$

汽相能量方程：

$$\frac{\partial E}{\partial t}+\frac{\partial[(E+p)v_j]}{\partial x_j}=\frac{\partial q_j}{\partial x_j}+\rho\dot m(h_{gp}-h_t) \tag{3}$$

液相连续方程：

$$\frac{\partial(\rho N)}{\partial t}+\frac{\partial(\rho N v_{pj})}{\partial x_j}=\rho_g J+\frac{\partial}{\partial x_j}\left(\frac{\rho\nu_p}{\sigma_p}\frac{\partial N}{\partial x_j}\right) \tag{4}$$

液相动量方程：

$$\frac{\partial(\rho Y v_{pi})}{\partial t}+\frac{\partial(\rho Y v_{pi}v_{pj})}{\partial x_j}=\frac{\rho_p}{\tau_{rp}}(v_i-v_{pi})+\rho\dot m(v_i-v_{pi})+\frac{\partial}{\partial x_j}\left[\rho Y\nu_p\left(\frac{\partial v_{pi}}{\partial x_j}+\frac{\partial v_{pj}}{\partial x_i}\right)\right]+\frac{\partial}{\partial x_j}\left[\frac{\rho\nu_p}{\sigma_p}\left(v_{pi}\frac{\partial Y}{\partial x_j}+v_{pj}\frac{\partial Y}{\partial x_i}\right)\right] \tag{5}$$

液相组分方程：

$$\frac{\partial(\rho Y)}{\partial t}+\frac{\partial(\rho Y v_j)}{\partial x_j}=\rho\dot m+\frac{\partial}{\partial x_j}\left(\frac{\mu_e}{\sigma_Y}\frac{\partial Y}{\partial x_j}\right) \tag{6}$$

式中：$\pm\frac{\rho_p}{\tau_{rp}}(v_{pi}-v_i)$—相间滑移的源项；$-\rho\dot m v_i$ 和 $\rho\dot m(v_i-v_{pi})$—相间传质的源项；$\rho\dot m(h_{gp}-h_t)$—凝结过程中汽相总能的变化；液相连续方程右端后一项为液滴湍流扩散所引起的质量输运；液相动量方程右端最后两项分别表示液滴湍流粘性与液滴质量扩散所引起的动量输运；液相组分方程右端后一项为流体湍流扩散所引起的液相组分输运。

2　SST $k-\omega-k_p$ 两相湍流模型

文献[4]根据以 $k-\varepsilon$ 方程为基础的湍流模型和颗粒湍能输运方程 k_p，建立了 $k-\varepsilon-k_p$ 两相湍流模型。SST $k-\omega$ 模型在透平叶栅内的单相湍流计算中表现出良好性能[7]，结合湿蒸气两相流动的特点，仿照颗粒湍能输运方程理论，推导建立了如下的 SST $k-\omega-k_p$ 湿蒸气两相湍流模型：

$$\frac{\partial(\rho_g k)}{\partial t}+\frac{\partial(\rho_g k v_j)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\frac{\mu_e}{\sigma_k}\frac{\partial k}{\partial x_j}\right)+G+G_p-C_\mu\rho\omega k \tag{7}$$

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho\omega v_j)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\frac{\mu_e}{\sigma_p}\frac{\partial\omega}{\partial x_j}\right)-\frac{\rho\gamma}{\mu_t}(G+G_p)-\beta\rho\omega^2+2(1-F_1)\rho\sigma_{\omega2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j} \tag{8}$$

$$\frac{\partial(\rho Y k_p)}{\partial t}+\frac{\partial(\rho Y k_p v_j)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\frac{\rho Y\nu_p}{\sigma_p}\frac{\partial k_p}{\partial x_j}\right)+P+P_g-\rho Y\varepsilon_p \tag{9}$$

其中：G_p—液相对汽相湍流应力造成的源项；P_g—汽相对液相湍流应力造成的源项；P—k_p 方程中液滴湍能生成项；$-\rho Y\varepsilon_p$—相应的耗散项，各项表达式为：

$$G_p=\frac{2\rho Y}{\tau_{rp}}(c_p^k\sqrt{kk_p}-k)$$

$$P_p=\rho Y\nu_p\left(\frac{\partial v_{pi}}{\partial x_j}+\frac{\partial v_{pj}}{\partial x_i}\right)\frac{\partial v_{pi}}{\partial x_j}$$

$$P_g=\frac{2\rho Y}{\tau_{rp}}(c_p^k\sqrt{kk_p}-k),\ \nu_p=c_{\mu p}\frac{k_p^2}{|\varepsilon_p|}$$

$$\varepsilon_p=-\frac{1}{\tau_{rp}}\left[2(c_p^k\sqrt{kk_p}-k_p)-\frac{1}{Y}\left(\frac{\nu_p}{\sigma_p}\frac{\partial Y}{\partial x_j}\right)(v_i-v_{pi})\right]$$

与文献[5]中模拟有旋突扩气固两相流动的守恒方程组相比，部分液相参数输运方程中的标量采用了与湿度 Y 乘积的形式，这是针对湿蒸气凝结流中的颗粒相液滴具有尺寸小、数量巨大的特点而作出的改进，采用宏观参数能有效地减少计算中的舍入误差。

3　计算方法及求解过程

运用商业软件 Fluent 中 UDF(User-Defined Function)模块提供的编程接口，对建立的流动控制方程和湍流模型方程进行了编程。其中液相参数通过该模块中 UDS(User-Defined Scalar)求解输运方程得到。各方程输运项离散均采用二阶迎风格式。

4　数值模型验证

对文献[8～9]中的透平叶栅顶部截面叶型进行

二维数值模拟，与应用文献[6]中的单流体模型结合 SST $k-\omega$ 单相湍流模型计算结果对比。图 1 给出了叶片表面沿轴向弦长的静压分布，可以看出，本模型的计算结果与实验值吻合良好，特别在吸力面的后半段，即同时出现自发凝结与边界层分离的区域，较单流体模型有很大改进。

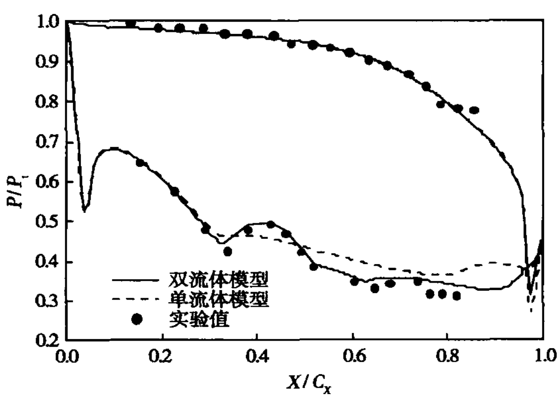

图 1　叶片表面压力分布

5　算例及分析

对上述叶型，在叶高方向取一倍轴向弦长得到三维直叶片。计算采用 H 型网格，网格数为 271×81×41，网格如图 2 所示。

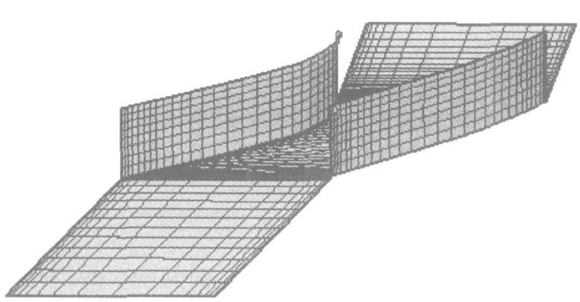

图 2　计算网格

图 3～图 5 分别给出了 5％叶高和 50％叶高截面的汽相马赫数、过冷度及湿度分布。从图 3 可以看到，吸力面尾缘发生了分离，在靠近端壁的 5％叶高处更强烈，使得尾迹分布与中心截面处有较大差异；由于端壁的影响，叶栅出口汽流速度下降明显。图 4 中，在叶栅喉部附近凝结核大量形成并快速生长使得过冷度迅速降低，湿蒸气流恢复到平衡状态；在尾缘激波后，由于汽流速度降低，温度上升使得汽流出现过热状态，可能会导致液滴蒸发。可以看出，端壁附近的吸力面发生分离处，汽流较早达到过热状态，而端壁附近的叶栅出口汽流速度下降较快也使得这一区域的过热度稍高于中心截面。图 5 中，5％叶高处更早出现凝结，但受端壁的影响，叶栅出口处的过冷度稍低，使得该区域的叶栅出口汽流湿度略低于中心截面。

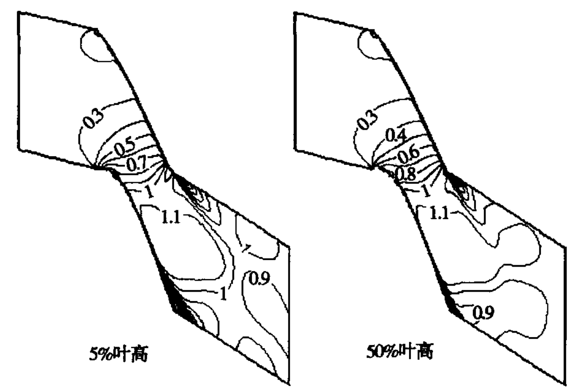

图 3　马赫数分布

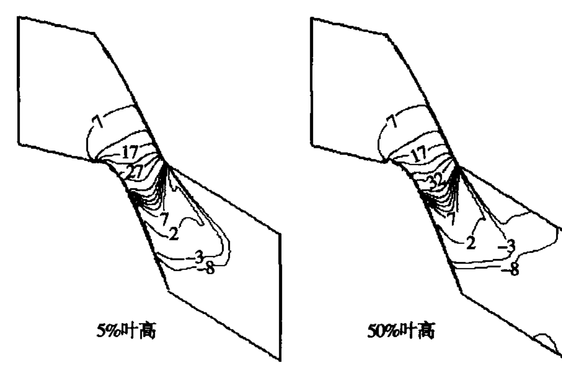

图 4　过冷度分布

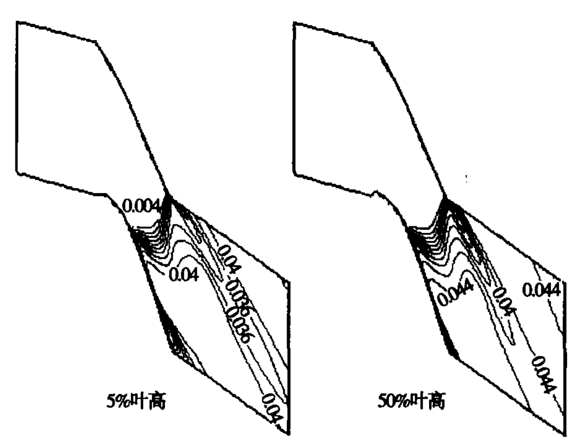

图 5　湿度分布

图 6 给出了叶栅出口截面的马赫数、过冷度及湿度分布。由图 6(a)可以看出，在端壁附近吸力面侧和压力面侧都有涡系结构存在，其中吸力面附近的分离涡强度要大很多；由图 6(b)可以看出，吸力

面侧发生分离的区域内，湿蒸气流的过热度已经很高，在实际流动中此区域的液滴可能会再次蒸发为水蒸气；由图 6（c）可以看出，吸力面附近的气流膨胀速率大，引起湿度梯度大。

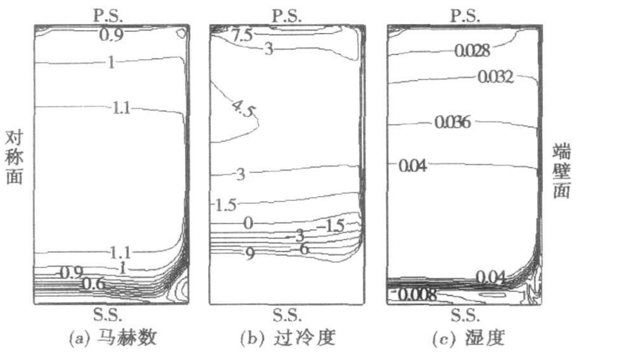

图 6 叶栅出口截面参数分布

图7 给出了 5％叶高和 50％叶高处的汽液两相出口马赫数。由于本文的数值模型不考虑二次水滴，汽液两相速度的大小与方向差别不大，相间速度差最大处也不超过 10 m/s。图 7 中，5％叶高处的吸力面和压力面附近各有一部分区域内的液相马赫数略大于汽相，这应该是受到该区域内涡系结构的影响，同时作为连续介质处理的液滴由于脉动惯性的作用，其脉动的扩散和耗散要比汽相慢；50％叶高处，在靠近吸力面的部分区域液相马赫数略大，这体现了吸力面侧的分离对两相流动产生的影响。图 8 给出了两相沿叶高与轴向的出口气流角。可以看到，在端壁附近由于壁面的影响，液相出口气流角略大；在叶栅中部，液相由于惯性气流角变化不大，而汽相由于在斜切部分继续膨胀发生偏转，使得汽相出口气流角略小。

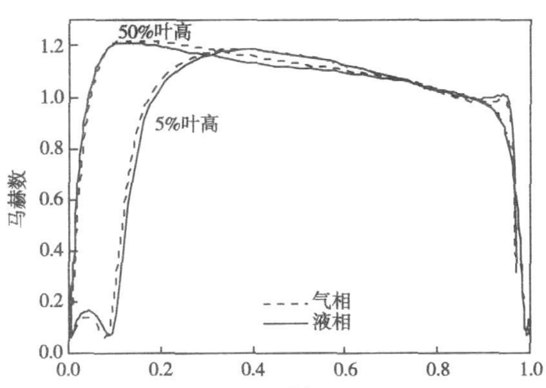

图 7 汽液两相出口马赫数

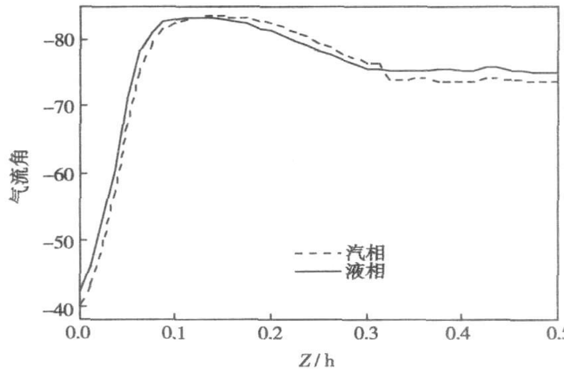

图 8 50％叶高处的汽液两相出口气流角

6 结 论

建立了湿蒸气两相凝结流动的双流体方程组和湿蒸气两相 SST $k-\omega-k_p$ 湍流模型。对一直列叶栅中存在自发凝结的湿蒸气流动进行了三维数值模拟，研究表明：与中心截面相比，端壁附近汽流首先出现大量凝结核，并较早恢复到平衡状态，由于涡系结构的存在使得不同叶高、汽液两相的叶栅出口马赫数之间存在一定差异；本文建立的模型应用于模拟三维流场，能够较好地得出汽液两相的流动特性及参数分布，从而更好地揭示叶栅中凝结流动的相间作用。

参考文献：

[1] CROWE C T, SHARMA M P, STOCK D E. The particle-source-in-cell (PSIC) method for gas-droplet flows[J]. Fluid Eng, 1977, 99(2): 325－332.

[2] GOSMAN A D, IOANNIDES E. Aspects of computer simulation of liquid-fuelled combustion // AIAA 19th Aerospace Science Meeting[C]. St Louis: McGraw-Hill, 1981. 1285－1293.

[3] SOO S L. Fluid dynamics of multiphase systems[M]. Mass: Blaisdell Pub Co, 1967.

[4] 周力行. 湍流两相流动与燃烧的数值模拟[M]. 北京：清华大学出版社, 1991.

[5] 周力行. 多相湍流反应流体力学[M]. 北京：国防工业出版社, 2002.

[6] 李 亮. 存在自发凝结的湿蒸气两相非平衡凝结流动数值研究[D]. 西安：西安交通大学, 2002.

[7] MENTER F R. Two equation eddy viscosity turbulence models for engineering applications[J]. AIAA J, 1994, 32(8): 1598－1605.

[8] BAKHTAR F, EBRAHIMI M, WEBB R A. On the performance of a cascade of turbine rotor tip section blading in nucleating steam-part 1[J]. Proc Instn Mech Engrs, 1995, 209(2): 115－124.

[9] BAKHTAR F, MAHPEYKAR M R. On the performance of a cascade of turbine rotor tip section blading in nucleating steam-part 3[J]. Proc Instn Mech Engrs, 1997, 211(3): 195－210.

（编辑 何静芳）

基于双流体模型的湿蒸气凝结流动三维数值模拟＝**A Three-dimensional Numerical Simulation of Wet Steam Condensation Flows Based on a Dual Fluid Model**[刊，汉] / WU Xiao-ming, LI Liang, LI Guo-jun, et al (College of Energy Source and Power Engineering under Xi'an Jiaotong University, Xi'an, China, Post Code: 710049) // Journal of Engineering for Thermal Energy & Power. －2007, 22(4). －367～370

A dual fluid model for wet-steam condensation flows was established with due consideration of such effects as inter-phase speed slip, coupling and diffusion of turbulent flows in wet-steam steam-liquid two-phase flows. In the light of the turbulent characteristics of flows in steam turbine cascades, derived and established was the SST $k-\omega-k_p$ turbulent flow model of wet steam two-phase flow by consulting the conveyance equation theory of particle turbulent energy and on the basis of the turbulent flow model involving two equations SST $k-\omega$, which have a relatively good numerical simulation accuracy in single-phase turbulent flow calculations. In the above model, introduced were several quasi-fluid conceptions, such as liquid-phase viscosity, heat conductivity and diffusion coefficient etc. On this basis, a three-dimensional numerical simulation was conducted of a wet steam flow of spontaneous condensation existing in a rectilinear cascade. It can be shown that compared with a central section, a large quantity of condensation nuclei first emerged from steam flows in the vicinity of the end walls and regained their balance state relatively early. There exists a certain difference among the mach numbers at the outlet of the steam-liquid two-phase flow cascade along the blade height due to the existence of a structure of vortex system. The model established by the authors can enhance the three-dimensional numerical simulation accuracy of wet steam condensation flows and better reveal the interphase function of condensation flows in a cascade. **Key words**: turbine cascade, wet steam, condensation flow, dual fluid model, two-phase turbulent flow model, numerical simulation

凝汽器水侧流动的三维数值模拟＝**A Three-dimensional Numerical Simulation of Water Flows at the Water Side of a Condenser**[刊，汉] / JIANG Jian-fei (Architectural and Thermal Energy Engineering Department of Pingdingshan Institute of Technology, Pingdingshan, China, Post Code: 467044), HUANG Shu-hong, WANG Kun, et al (College of Energy Source and Power Engineering under the Central China University of Science and Technology, Wuhan, China, Post Code: 430074) // Journal of Engineering for Thermal Energy & Power. －2007, 22(4). －371～374

A model for the calculation of flows in tube bundles of a condenser has been established and by employing a CFD (computational fluid dynamics) method, a three-dimensional numerical simulation conducted of the flow field at the water side of the condenser. By using a sub-regional symmetric calculation method, the number of mesh can be greatly reduced, making it possible to conduct a detailed prediction of the flow characteristics in the inlet and outlet water chambers and their connecting pipes at the condenser water side as well as in the cooling water tube bundles. The calculation results clearly indicate that there exist numerous vortexes in the inlet and outlet water chambers of the condenser, resulting in an increase of flow resistance and a deterioration of the flow conditions. The velocity distribution inside the water chambers is not uniform. As a result, the flow velocity in the tubesheet central area of the inlet water chamber is relatively high and that in the marginal area is relatively low, indicating the presence of a certain structural problem. The structure of the outlet water chamber, however, is relatively rational. This fully corresponds with the conclusion of the simulation conducted by using a porous medium model, giving further proof that it is correct and feasible to use the porous-medium model to calculate the condenser. The calculation results show simultaneously that the distribution of flow rates and velocity in the cooling water tube bundles is not uniform, i.e. the cooling water tubes in the central area have a relatively big flow rate while those in the peripheral area a relatively small flow rate. The biggest difference in flow rates can amount to 38%. The margin of decrease in the flow rate of neighboring tubes has something to do with the layout of the cooling water tubes. The difference in cooling water flow rate and velocity in the cooling water tubes will influence the heat exchange performance of a heat exchanger. The calculation results can provide a foundation for the analysis of heat exchange efficiency problems caused by a non-uniform flow in tube banks and also a basis for the design and structural optimization of condensers. **Key words**: marine condenser, water side, flow characteristics, numerical simulation

竖直通道内相邻气泡对上升的直接数值模拟＝**A Direct Numerical Simulation of Neighboring Air-bubble Rising Process in a Vertical Channel**[刊，汉] / LI Yan-peng, ZHANG Qian-long (College of Environmental Science and Engineering under Chang'an University, Xi'an, China, Post Code: 710064), BAI Bo-feng (National Key Laboratory on Multi-phase Flow, Power Engineering Department of the Xi'an Jiaotong University, Xi'an, China, Post Code: 710049) // Journal of Engineering for Thermal Energy & Power. －2007, 22(4). －375～379