文章编号: 1001－2060(2008)05－0527－04

两种内翅片管对流换热特性数值模拟

吴　峰, 邓志安, 陈军斌, 何光渝
(西安石油大学 石油工程学院, 陕西 西安 710065)

摘　要: 应用层流模型及湍流模型数值模拟方法, 结合两种边界条件处理方法分别对两种纵向内翅片管内的流动与传热性能进行了研究, 其中湍流计算采用可实现 $k\sim\varepsilon$ 两方程模型。将两种计算模型数值结果与实验结果进行了对比, 结果表明: 湍流模型数值模拟结果较层流更接近于实验值, 同时发现两种内翅片管内流动从层流发展到湍流的临界雷诺数远小于传统光管的临界雷诺数。针对湍流模型模拟结果分别拟合出了两种内翅片管 Nu—Re 及 f—Re 的关联式, 拓宽了实验数据的应用范围; 通过场协同原理定量对比分析了两种内翅片管强化换热机理, 研究表明纵向突起内翅片管的场协同程度好于纵向平翅片管, 起到强化传热的作用。

关　键　词: 内翅片管; 强制对流; 传热特性; 数值模拟

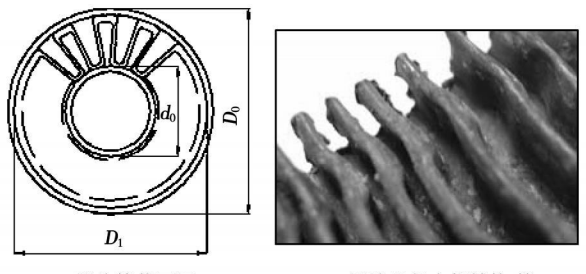
中图分类号: TK124　　文献标识码: A

引　言

强化传热技术由于其能使各种换热设备的效率提高、重量和体积减少而广泛应用于各工业生产及科学研究工作中[1~3]。在强化传热技术中, 翅片得到广泛的应用[4~8]。一般认为翅片在强化传热中的作用主要是增加了热阻大的这一侧流体的面积[9], 从而减少了以总面积计算的热阻, 其中纵向内翅片管起强化传热的作用由此而来。压缩机中冷器是提高多级压缩机整体效率的重要设备, 研发适合于这种换热设备的高效换热管具有十分重要的意义。文献[10]针对压缩机中冷器的上述要求, 对两种几何尺寸和形状基本相同的内翅片管进行了实验研究, 分别得到了努塞尔数、阻力系数随雷诺数变化的关系式。其中一根管的翅片为直片(文中称为管一), 截面如图 1(a)所示, 另外一根管在翅片上加有周期性凸起(文中称为管二), 内部结构如图 1(b)所示, 两种内翅片管的几何尺寸如表 1 所示。文献[11~12]分别对这两种内翅片管建立了数值模型并进行了一定的分析, 表明两种内翅片管较同等尺寸结构下的环形套管起到了强化传热的作用, 但并没有对两种内翅片管内流动提前向湍流过渡的临界雷诺数大小进行具体分析; 同时对于两种内翅片管的强化传热机理并没有深入进行定量分析和对比; 本研究针对文献[10]中两种内翅片管及以上文献中的一些研究不足之处, 通过数值模拟方法对两种内翅片管内流动与传热性能进行分析, 得出两种内翅片管内流动从层流向湍流过渡的临界雷诺数取值范围; 通过数值模拟方法拓宽了内翅片管实验关联式的参数范围, 并通过场协同原理定量对比分析了两种内翅片管强化换热机理。

表 1　翅片管几何尺寸　　(mm)

	管一	管二
有无凸起	无	有
波纹数 M	15	
管长 l	400	
翅片展开长度 l_f	242.9	
外管外径 D_o	28	
外管内径 D_{in}	26	
芯管外径 d_o	14	
翅片厚度 δ_f	0.2	
当量直径 D_h	2.67	



(a) 翅片管截面图　　(b) 翅片凸起内部结构(管二)

图 1　实验管结构图

收稿日期: 2007－10－11;　修订日期: 2008－01－10

作者简介: 吴峰(1978－), 男, 江西广丰人, 西安石油大学讲师.

1 物理问题及控制方程

针对管一、管二结构进行数值建模，由于内翅片管翅片分布及边界条件的对称性与周期性，另外考虑到管二内沿流动方向带有周期性的突起，选择一个波纹的外加两个半波纹组成的单元作为本文数值计算区域，以管二结构为例，如图 2 所示。对于管一而言，*ac*、*bd* 边界条件设定为速度及温度对称分布。

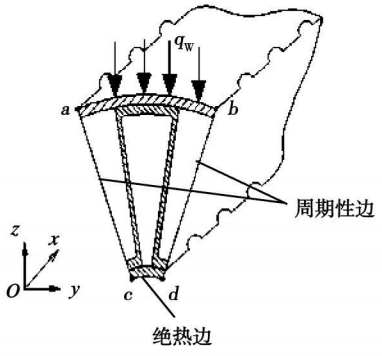


图 2 计算模型区域与边界条件(管二)

计算分别采用三维稳态层流和湍流变物性强制对流模型，流体工质为空气，其 ρ、λ、μ 等物性参数在 20～60 ℃之间线性插值。其连续性方程、动量方程和能量方程的通用形式为：

$$\nabla(\rho\vec{V}\phi) = \nabla(\Gamma_\phi \nabla\phi) + S_\phi \tag{1}$$

层流时不同变量的扩散系数及源项的具体形式如表 2 所示。

表 2 不同变量所对应的 ϕ、Γ_ϕ 及 S_ϕ(层流)

	ϕ	Γ_ϕ	S_ϕ
连续性方程	1	0	0
u 方程	*u*	μ	$-\partial p/\partial x$
v 方程	*v*	μ	$-\partial p/\partial y$
w 方程	*w*	μ	$-\partial p/\partial z$
能量方程	*T*	μ/*Pr*	0

采用 Fluent 商业软件进行数值求解，计算湍流时采用可实现 k～ε 两方程模型，其通用控制方程如式(1)，与层流不同的是广义变量中增加了 k，ε 两项，同时广义扩散系数也有变化，具体表达式可参见文献[13]。

采用有限容积法对计算区域进行离散，六面体网格划分节点数为 180(x)×38(y)×40(z)，固体区域采用均分网格，流体区则采用非均分网格进行划分，其中近壁面处网格较为密集。采用 SIMPLEC 算法处理速度和压力的耦合问题，对流项的离散格式为 QUICK，如图 2 所示，计算模型边界条件定义为：

进口截面：
$$\begin{cases} \dot{m} = \text{const},\ T_{in} = \text{const} \\ \text{固体区域：} \dfrac{\partial T}{\partial z} = 0,\ u = v = w = 0 \end{cases} \tag{2}$$

出口截面：

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = \frac{\partial w}{\partial x} = \frac{\partial T}{\partial x} = 0 \tag{3}$$

通道上壁面 *ab*：

$$\begin{cases} u = v = w = 0 \\ q_w = \text{常数} \end{cases} \tag{4}$$

通道下壁面 *cd*：

$$u = v = w = \frac{\partial T}{\partial y} = 0 \tag{5}$$

通道左右侧面 *ac*、*bd*：

$$\text{管一：} \frac{\partial v}{\partial x} = \frac{\partial w}{\partial x} = \frac{\partial T}{\partial x} = 0,\ u = 0 \tag{6}$$

$$\text{管二：} \begin{cases} T(x,y,z)\big|_{ac} = T(x,y,z)\big|_{bd} \\ v(x,y,z)\big|_{ac} = v(x,y,z)\big|_{bd} \end{cases} \tag{7}$$

2 计算结果及讨论

2.1 临界雷诺数分析

采用层流及湍流数值模型分别对两种内翅片管内传热特性进行数值分析，图 3 和图 4 所示为两种内翅片管内平均 *Nu* 数随 *Re* 数的变化情况。图 3 表明当 *Re*=1 200～1 500 时，采用层流模型的计算结果逐渐偏离实验数据，而湍流模型在整个实验数据范围内与实验数据符合较好，这说明管一内流动由层流向湍流过渡的临界雷诺数位于 *Re*=1 200～1 500范围内。同理，由图 4 可知管二内流动由层流向湍流过渡的临界雷诺数位于 *Re*=800～1 000 范围内，较管一提前进入湍流状态，且远小于光管内层流向湍流过渡的临界雷诺数 *Re*=2 300。这主要是

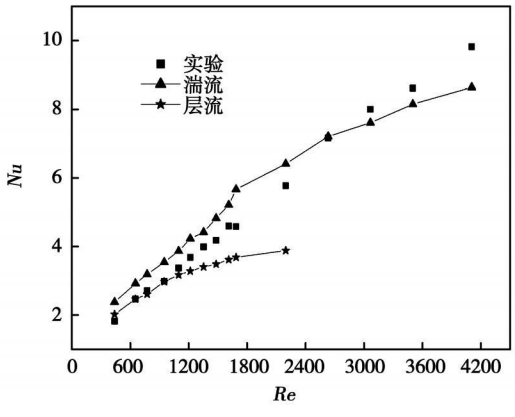


图 3 管一内平均 *Nu* 数随 *Re* 数的变化

由于管二内翅片纵向周期性凸能够不断破坏管内的流动边界层, 加剧了管内流体的湍动, 因此管二内流体流动更易达到湍流状态。这一结论类同于文献[14]中有关波纹管内流动与传热特性的分析研究。

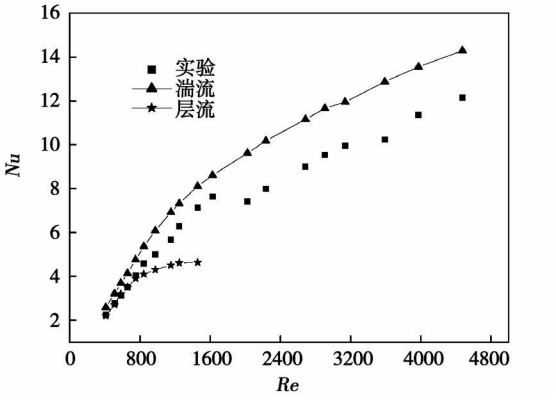


图 4　管二内平均 Nu 数随 Re 数的变化

2.2　数值模拟拓宽参数

本研究翅片管的当量直径用下式求得:

$$D_h = \frac{\pi(D_i^2 - d_o^2) - 4\delta l_f}{\pi(D_i + d_o) + 2l_f} \tag{8}$$

式中: D_i — 外管内径; D_o — 芯管外径; δ — 翅片厚度; l_f — 翅片展开长度。平均努塞尔数的定义为:

$$Nu = hD_h/\lambda \tag{9}$$

Reynolds 数的定义为:

$$Re = u_m D_h/v \tag{10}$$

式中: u_m — 来流的平均流速, m/s, 阻力系数采用 Darcy 定义:

$$f = \frac{-(\mathrm{d}p/\mathrm{d}x)D_h}{\rho u_m^2/2} \tag{11}$$

对管一、管二采用湍流模型在较大的 Re 数范围内计算了不同 Re 数下 Nu 数及阻力系数的变化情况, 如图 5 和图 6 所示, 同时分别拟合出两种内翅片管 Nu — Re、f — Re 关联式分别整理为

$$\text{管一}\begin{cases} Nu = 0.052\,8Re^{0.62} \\ f = 0.282Re^{-0.245} \end{cases} \tag{12}$$

$$\text{管二}\begin{cases} Nu = 0.056\,4Re^{0.664} \\ f = 0.639Re^{-0.258} \end{cases} \tag{13}$$

其 Re 数范围为 $10 \sim 2\times10^4$, 比实验参数的雷诺数范围 $4.3\times10^2 \sim 5\times10^3$ 要大, 从而拓宽了应用范围。

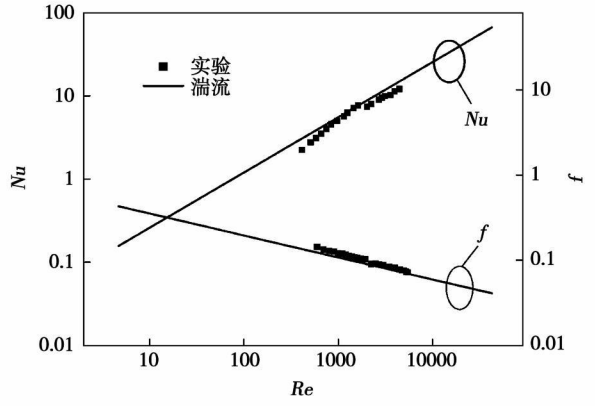


图 6　突翅片管对比

2.3　强化换热机理分析

对于强化传热的本质究竟是什么, 长期以来相关文献没有一个统一的解释[15]。文献[15~19]表明, 对流换热的强度, 不仅取决于流速、温差及流体物性, 还取决于速度场与温度梯度场的协同程度即场协同原理, 这一思想的提出对于对流换热的强化概念有了更加深刻的认识, 能够在定量分析上给予明确的分析。图 7 中为两种内翅片管及环形套管全场速度与温度梯度的平均夹角 θ 随 Re 数变化关系。由图可知在相同的 Re 数下, 管二的全场平均协同角最小, 其次为管一, 环形套管的平均协同角最大, 即协同性最差。

图 8 中 η_{int} 为对流项的积分值 $\rho c_p \int |V||\nabla T| dv$ (V 为速度矢量, ∇T 为温度梯度, dv 为体积元), 其值代表了换热量的大小。图 7、图 8 表明, 随着流速的增大, 换热量增大, 对流项的积分值 η_{int} 增大, 同时温度梯度与速度矢量的平均夹角变大, 即协同性变差, 在相同的 Re 数下, 管二的对流换热能力要好于管一及环形套管。文献[15, 20]通过数理分析结合工程实际深入探讨了强化单相对流传热技术的机理, 得出如下结论: 减薄热边界层及增加流体中的扰动其实质都是要减少速度与温度梯度间的夹角, 即现有文献中关于强化传热单相对流换热的 3 种机理与场协同原理是一致的。通过对纵向直内翅片管及

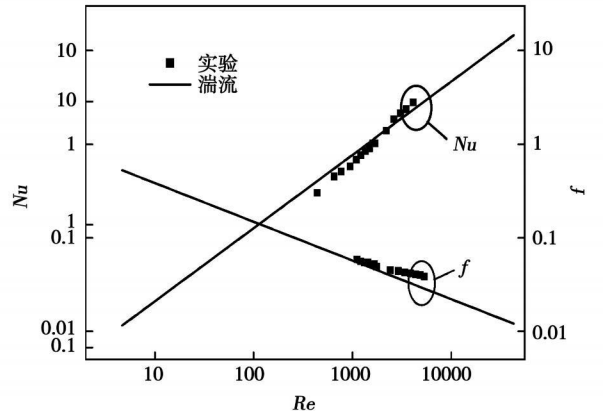


图 5　直翅片管对比

纵向周期性突起内翅片管的研究再次表明，场协同理论对于强化传热技术理论的统一性。

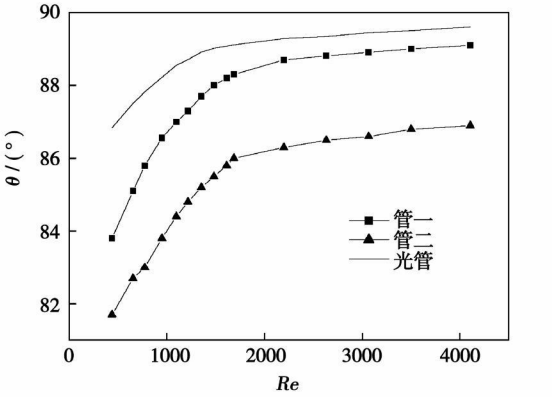


图 7 平均协同角随 *Re* 数的变化

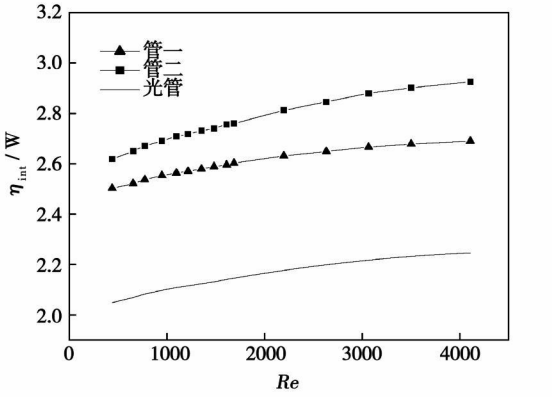


图 8 积分值随 *Re* 数的变化

3 结 论

(1) 采用数值计算方法对两种纵向内翅片管分别进行了层流和湍流两种模型的数值模拟，研究发现两种内翅片管内流动临界雷诺数均远小于光管，其中直内翅片管临界雷诺数位于 $Re=1\,200\sim1\,500$ 范围内，带突起内翅片管临界雷诺数位于 $Re=800\sim1\,000$范围内。

(2) 采用湍流计算模型在较宽 *Re* 数范围内模拟了两种内翅片管的流动与传热特性，得出两种内翅片管 $Nu-Re$、$f-Re$ 的关联式，拓宽了实验结果的应用范围。

(3) 通过场协同原理定量对比分析了两种内翅片管强化传热机理。研究表明纵向翅片的周期性凸起结构能够改善管内的温度场、速度场协同程度，提高了对流项的换热量，较纵向直内翅片管及环形套管而言更能起到强化换热作用。

参考文献：

[1] 崔海亭，彭培英. 强化传热新技术及其应用[M]. 北京：化学工业出版社，2006.

[2] 林宗虎，汪 军，李瑞阳，等. 强化传热技术[M]. 北京：化学工业出版社，2007.

[3] 钱颂文，朱冬生. 管式换热器强化传热技术[M]. 北京：化学工业出版社，2003.

[4] SAAD A E, SAYED A E, MOHAMED E A, et al. Experimental study of turbulent flow inside a circular tube with longitudinal interrupted fins in the streamwise direction[J]. Experimental Thermal Fluid Science, 1997, 15(1): 1—15.

[5] HUQ M, HUQ A, RAHMAN A M, et al. Experimental measurements of heat transfer in an internally finned tube[J]. Int Comm Heat Mass Transfer, 1998, 25(5): 619—630.

[6] BRAGA C V M, SABOYA F E M. Turbulent heat transfer, pressure drop and fin efficiency in annular regions with continuous longitudinal rectangular fins[J]. Experimental Thermal Fluid Science, 1999, 20(2): 55—56.

[7] NAIK S, PROBERT S D, BRYDEN I G. Heat transfer characteristics of shrouded longitudinal ribs in turbulent forced convection[J]. International Journal of Heat and Fluid Flow, 1999, 20(4): 374—384.

[8] DAGTEKIN I, OZTOP H F, SAHIN A Z. An analysis of entropy generation through a circular duct with different shaped longitudinal fins for laminar flow[J]. International Journal of Heat and Mass Transfer, 2005, 48(1): 171—181.

[9] INCROPERA F P, DEWITT D P. Fundamentals of heat and mass transfer: fifth ed[M]. New York: John Wiley &Sons Co, 2002.

[10] 田 林，罗来勤. 两种内翅片管对流换热与阻力特性的实验研究[J]. 工程热物理学报，2005, 26(3): 471—473.

[11] 吴 峰，林 梅，田 林，等. 纵向内翅片管对流换热特性的实验及数值研究[J]. 化工学报，2005, 56(11): 2065—2068.

[12] 吴 峰，林 梅，田 林，等. 纵向带突起内翅片管流动与传热特性的实验及数值研究[J]. 热能动力工程，2006, 21(3): 290—293.

[13] 陶文铨. 数值传热学(第二版)[M]. 西安：西安交通大学出版社，2000.

[14] 肖金花，钱才富，王凤林. 波纹管对高黏度介质的强化传热研究[J]. 北京化工大学学报，2007, 34(1): 53—57.

[15] 过增元，黄素逸. 场协同原理与强化传热新技术[M]. 北京：中国电力出版社，2004.

[16] GUO Z Y, LI D Y, WANG B X. A novel concept for convective heat transfer enhancement[J]. Int J Heat Mass Transfer, 1998, 41(14): 2221—2225.

[17] TAO W Q, GUO Z Y, WANG B X. Field synergy principle for enchancing convective heat transfer—its extension and numerical verificaion[J]. Int J Heat Mass Transfer, 2002, 45(18): 3849—3856.

[18] CHEN Y P, QU Z G, TAO W Q, et al. Numerical design of efficient slotted fin surface based on the field synergy principle[J]. Numerical Heat Transfer: Part A, 2004, 45: 1—22.

[19] 武俊梅，陶文铨. 纵向涡强化换热的数值研究及场协同原理分析[J]. 西安交通大学学报，2006, 40(7): 757—761.

[20] 杨世铭，陶文铨. 传热学[M]. 第 4 版. 北京：高等教育出版社，2006.

(编辑 何静芳)

Compared with ordinary pressure type nozzles, airflow type nozzles have such problems as easy to be worn out and a poor atomization effectiveness. However, ordinary airflow type nozzles are incapable of atomizing any gas—liquid mixture. To solve the above problem, a new type of airflow nozzle was designed by the Institute and an experimental study has been conducted of its atomization characteristics. In addition, the atomized particles and droplets were measured and analyzed. The measurement results show that under the test conditions, the particle diameters atomized by the nozzles in question are less than 50 μm and the atomization effectiveness is superior to that of ordinary airflow type nozzles. In the meantime, an analysis and study of the factors influencing atomization effectiveness have been conducted. Measurements were performed at different gas—liquid ratios. When the gas—liquid ratio increases, the atomized droplet particle diameter will decrease. At the same time, measurements were also performed at an air speed of 150 m/s and 250 m/s respectively. The greater the relative speed between the gas and liquid, the smaller the atomized droplet particle diameter. The results of the study show that the type of nozzles under discussion can atomize the gas—liquid mixture and can be applied to the alkaline solution separation in the separation towers of petrochemical industry and the water atomization for temperature reduction through water injection into a compressor. **Key words**: airflow type, nozzle, atomization, experimental study

气流搅拌流场中不同通气结构的 CFD 模拟= **CFD (Computational Fluid Dynamics) Simulation of Different Aeration Structures in a Gas—liquid Agitation Flow Field**[刊, 汉] / ZHANG Chao—ping, XU Tao, KE Chang—hua (Shandong Shanda Hua—te Environmental Protection Engineering Co. Ltd., Jinan, China, Post Code: 250001), LI Yan—fen (Beijing Strength and Environment Research Institute, Beijing, China, Post Code: 100827)// Journal of Engineering for Thermal Energy & Power. — 2008, 23(5). —519 ~ 522

By using a numerical simulation method, studied were the features of a gas—liquid two phase flow field in a same kind of agitation reactor with four types of different oxidation spray gun structures. With a Euler—Euler dual fluid model a flow field was analyzed by adopting RNG k—ε two—equation turbulent flow model. A mortar agitation zone was processed by using a multiple reference system method. As a result, the air content in the flow field of different oxidation spray gun structures was obtained. It has been found that 90 degree tube—bend finned structure can be used to attain a better agitation effectiveness, a more rational velocity field, a more ideal air bubble trajectory and a higher air content on a typical surface. **Key words**: two—phase flow, agitation, oxidation spray gun

钛管 $CaCO_3$ 污垢特性的实验研究= **Experimental Study of $CaCO_3$ Fouling Characteristics of Titanium Tubes** [刊, 汉] / XU Zhi—ming, QIU Zhen—bo (Postgraduate shool, Northeast Dianli University, Jilin, China, Post Code: 132012), ZHANG Zhong—bin (College of Energy Source and Power Engineering, North China Electric Power University, Baoding, China, Post Code: 071003)// Journal of Engineering for Thermal Energy & Power. — 2008, 23(5). —523 ~ 526

An experimental study of titanium tube fouling characteristics was conducted, which were compared with those of stainless steel tubes and copper bare tubes. The results of the study show that the fouling induction period of the titanium tubes will increase with an increase of concentration, and decrease with an increase of flow speed. However, the law governing the change of fouling heat resistance with the concentration and flow speed is rather complicated. Under the same operating conditions, the induction period of titanium tubes is shorter than that of stainless steel tubes but longer than that of copper tubes. The fouling heat resistance asymptotic value of the titanium tubes is smaller than that of copper tubes, but bigger than that of stainless steel tubes. This indicates that the fouling resistant characteristics of titanium tubes, though superior to those of copper tubes, are inferior to those of stainless steel tubes. **Key words**: titanium tube, $CaCO_3$ fouling, fouling characteristics, induction period

两种内翅片管对流换热特性数值模拟= **Numerical Simulation of Convection Heat Exchange Characteristics of Two Types of Internally—finned Tubes**[刊, 汉] / WU Feng, DENG Zhi—an, CHEN Jun—bin, HE Guang—yu (Petroleum and Gas Storage and Transportation Engineering Department, College of Petroleum Engineering, Xi'an Shiyou University, Xi'an, China, Post Code: 710065)// Journal of Engineering for Thermal Energy & Power. — 2008, 23(5). —527 ~ 530

With a laminar flow model and a turbulent flow model—based numerical simulation method being adopted and in combi-

nation with two kinds of boundary—condition treatment methods, a study has been conducted respectively of the flow and heat transfer performance of two types of longitudinally and internally finned tubes. During the study, a realizable $k-\varepsilon$ two equation model was used for the turbulent flow calculation. The numerical simulation results obtained from the two calculation models were compared with test results. It has been found that the simulation results obtained from the turbulent flow model are closer to the test values than those obtained from the laminar flow model. In the meanwhile, it has also been found that the critical Reynolds Number for the flow in both internally finned tubes when developing from a laminar flow to a turbulent one is far less than that for the traditional bare tube. In the light of the simulation results obtained from the turbulent flow model, correlation formulae were obtained through a fitting for the two types of internally finned tubes of $Nu-Re$ and $f-Re$ respectively, thus extending the applicable scope of the test data. Through a field synergy principle, a contrast analysis of intensified heat exchange mechanism for both types of internally—finned tubes was quantitatively conducted. The results of the study show that the field synergy degree of longitudinally—ridged and internally—finned tubes is better than that of longitudinally flat—finned tubes, thereby playing a role of intensified heat transfer. **Key words**: internally—finned tube, forced convection, heat transfer characteristics, numerical simulation

波纹倾角 β 对空气在波纹流道内的流阻与传热影响的数值分析=**Numerical Analysis of the Influence of Corrugation Inclination Angle β on the Air Flow Resistance and Heat Transfer in Corrugated Flow Passages**[刊, 汉] / WU Hua—xin, SUN Gang (College of Architectural Engineering, Harbin Engineering University, Harbin, China, Post Code: 150001), ZHOU Song (College of Power Engineering, Harbin Engineering University, Harbin, China, Post Code: 150001), TIAN Bao—ren (Jitai Heat Exchanges Manufacturing Factory, Siping, China, Post Code: 136001) // Journal of Engineering for Thermal Energy & Power. — 2008, 23(5). —531 ~534

An air—water corrugated plate type heater differs from a traditional ribbed tube type heater. The former is a primary surface type heat exchanger and the surface of which in contact with a heat transfer medium directly takes part in a heat transfer process, featuring a compact structure and high efficiency. It can be used in such cases as air—water heating and heat energy recovery systems etc. to serve as hot wind curtains and air conditioner heaters etc. The authors have simulated the air flow resistance and heat transfer characteristics in flow passages at various corrugation inclination angles of $\beta=30^\circ, 45^\circ, 60^\circ$ by utilizing CFD (Computational Fluid Dynamics) software Fluent, and calculated the surface friction coefficient f, Nusselt Number Nu and the ratio of heat transfer factor and surface friction coefficient j/f. The results of the study show that the corrugation inclination angle is an important factor influencing the flow resistance and heat transfer process of air in corrugated flow passages. Through relevant comparisons, the above research findings can provide definite guidelines for the further optimized design of air—water corrugation plate type heaters. **Key words**: air—water corrugated plate type heater, flow resistance, heat transfer, numerical characteristic analysis

高转速强漩流离心法聚氧聚氮装置研究=**Exploratory Study of a High Rotating Speed and Strong—whirlpool Centrifugal Method—based Oxygen and Nitrogen Collecting Device**[刊, 汉] / WANG Xi—kui, CHEN Zheng—ju, HONG Guang—huan, MENG Zhao—jun (Energy Collection Research Institute, Shengyang Institute of Engineering, Shenyang, China, Post Code: 110136), // Journal of Engineering for Thermal Energy & Power. — 2008, 23(5). —535 ~538

To solve the difficult problem of oxygen and nitrogen separation from air, an innovative structure of "rotating casing type axial compressor" is proposed to replace the traditional "rotating hub type axial compressor". The main difference of the two structures lies in the fact that the blades of a "rotating hub type compressor" are installed on the wheel hub and can only realize air compression, while those of a "rotating casing type compressor" are installed on the inside of an inner casing, rotating in an outer casing, thus effecting air separation. The authors have described the theories of oxygen and nitrogen centrifugal separation and the centrifugal pressure boosting of the rotating casing compressor. The merits, technical feasibility and application approach of air whirlpool separation were also expounded. A sketch of the centrifugal method—based test rig for oxygen and nitrogen collection was given, which can realize an integration of three devices, namely: an air compressor, an oxygen and nitrogen separator and a combustion whirlpool generator. The ultimate aim is to remarkably enhance energy source application efficiency, fully utilize resources as well as to save energy and protect the environment. **Key words**: oxygen and nitrogen collection, rotating casing type compressor, oxygen and nitrogen centrifugal separation, strong whirlpool separation, theoretical analysis, alternative version exploration