文章编号: 1001-2060(2004)05-0506-04

新型喷嘴结构下蒸汽喷射式热泵性能的数值研究

张少维[1], 徐海涛[2], 桑芝富[1]
(1. 南京工业大学 机械与动力工程学院, 江苏 南京 210009; 2. 江苏苏源环保工程股份有限公司, 江苏 南京 210024)

摘 要: 运用 CFD 数值模拟方法对比研究了普通喷嘴和新型喷嘴结构对蒸汽喷射式热泵操作性能的影响, 并分析了整流管长度和喷射系数之间的变化关系。结果表明, 在相同的操作条件下, 整流喷嘴结构能有效地提高喷射器的喷射系数。同时存在一最佳整流管长度, 对应于最大喷射系数。这种喷嘴结构能有效改善由于工作流体压力降低而造成的设备操作性能恶化, 提高蒸汽喷射泵运行的稳定性。

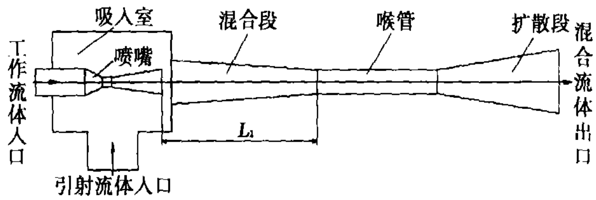
关 键 词: 喷射泵; 喷嘴; 喷射系数; 数值模拟

中图分类号: TB115 文献标识码: A

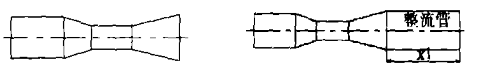
1 引 言

蒸汽喷射器利用吸入室内的喷嘴, 将高压流体介质(一般称为工作流体, Primary Motive Fluid)的压力能转化为动能, 在喷嘴出口处形成高速射流而造成局部真空, 以至使其压力低于被抽流体(称为引射流体, Second Entrained Fluid)的压力, 在此压力差的作用下, 引射流体由设备中被引射进入吸入室, 进而与工作流体混合并被高速射流带走, 经扩压器作用再把混合流体的动能转换为压力能而被排出泵外。由于其具有结构简单、没有运动部件、运转费用低廉和容易操作维修等优点, 并且对被抽介质无严格要求, 对于有毒性、易燃易爆、腐蚀性强乃至可凝性气体等几乎都适用, 加之其抽气量较大, 工作压力范围宽, 因而在制冷、能源动力、石油化工和冶金等领域得到了广泛应用。

喷射器的基本结构如图 1(a)所示, 主要由喷嘴、吸入室和扩散管组成, 扩散管又分为混合段、喉管和扩散段。图 1(b)和(c)分别为普通喷嘴和改进后的新型喷嘴, 即整流喷嘴结构示意图。和普通喷嘴相比, 整流喷嘴在普通喷嘴的出口处安装一整流管。整流管深入到扩散管的混合段内部, 通过稳定混合段流体流动, 抑制工作流体边界层的形成和发展, 改善混合段和喉管内激波系的分布, 从而在不增加额外消耗的情况下提高引射流体流量, 进而提高喷射泵的操作。



(a) 蒸汽喷射泵基本结构



(b) 普通喷嘴结构 (c) 整流喷嘴结构

图 1 蒸汽喷射泵基本结构及改进前后的喷嘴结构示意图

蒸汽喷射器优化设计工作主要集中在两个方面: 一是操作工况优化; 二是结构参数优化。操作工况的改变(如工作压力 P_p、引射压力 P_H 及背压 P_C 等)对喷射器的操作性能有很大影响。徐海涛和桑芝富等人的研究表明[1], 在不同的 P_p、P_H 和 P_C 下, 总存在一最优的操作条件对应于最大的喷射系数(表示为 u, 其值为引射流体和工作流体质量流量的比值)。结构优化涉及的内容比较广泛, 包括喷嘴结构的改进、喷嘴最佳位置的确定、扩散管长度和喉管面积的改变等。Hedge 和 Hill 通过对两种不同锥度喷嘴下喷射器内速度及壁面压力分布的测定表明[2], 喷嘴锥度的改变对喷射器性能的影响不大。Mastsuo 和 Nahdi 等人则研究了混合室截面积与喷嘴截面积之比对喷射泵操作性能的影响[3~5], 其研究结果表明对应于一定的面积比, 存在着一最大喷射系数和

收稿日期: 2003-10-08; 修订日期: 2003-11-27
作者简介: 张少维(1980-), 男, 陕西杨凌人, 南京工业大学在读硕士研究生.

最大压力比。Chang 和 Sujith 等人则对不同喷嘴形状(如花瓣型喷嘴和椭圆喷嘴等）进行了研究[6~7]。

本文利用计算流体动力学（Computer Fluid Dynamics，CFD）方法对比研究了自主开发的新型整流喷嘴和普通喷嘴结构对蒸汽喷射式热泵操作性能的影响，CFD 方法和传统的分析方法相比更能实现实际操作条件和理想工况的模拟，减小因经验公式中过多的简化而造成的计算误差。并且其可视化研究方法能够直接模拟喷射泵内部流体速度和压力等操作参数的分布规律，为优化设计提供直观信息。文献[8] 中利用 CFD 数值模拟的方法对喷射泵进行了优化设计，其计算结果已在中国扬子石化和金陵石化等企业广泛应用，并且实际使用效果和数值计算结果的吻合非常好，系统能够保证一定的真空度，同时蒸汽消耗量也有较大程度的下降。因此，在现有理论基础上利用 CFD 数值模拟方法进行蒸汽喷射泵研究完全可以满足工程实际需要，并能比较准确地反映喷射泵内部流动特性，对蒸汽喷射泵工程应用的优化设计和科学研究提供了参考依据。

2　数值计算模型

2.1　控制方程组

稳态可压缩流体流动满足如下积分形式的 N-S 方程：

$$\frac{\partial}{\partial t}\int_V W \mathrm{d}V + \oint [F - G] \cdot \mathrm{d}A = \int_V H \mathrm{d}V$$

其中：W、F 以及 G 的向量表示形式如下：

$$W = \begin{Bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{Bmatrix}, F = \begin{Bmatrix} \rho v \\ \rho v u + p\vec{i} \\ \rho v v + p\vec{j} \\ \rho v w + p\vec{k} \\ \rho v E + p v \end{Bmatrix},$$

$$G = \begin{Bmatrix} 0 \\ \tau_{xi} \\ \tau_{yi} \\ \tau_{zi} \\ \tau_{ij} v_j + q \end{Bmatrix}$$

式中：H 项包含了体力以及能量源项的影响；ρ、v、E 及 p 分别为密度、速度、单位流量的总能量和流体压力；τ 为粘性应力张量；q 为热源。

2.2　计算方法及计算网格

采用控制容积法（Control-Volume Method）对计算域作离散化处理，商用软件建模分网(网格划分如图 2 所示)，并使用网格自适应方法来获得与网格密度无关的解。考虑边界层，使用标准 κ-ϵ 湍流模型以实现湍流对流动的影响，近壁区域采取壁面函数修正法处理。计算采用可压缩流动中广泛使用的时间相关法，耦合求解连续性方程、动量方程和能量方程以期同时获得速度、压力和温度等变量，并对系数矩阵进行预处理(Preconditioning Method)，改变系数矩阵的特征值，以消去在低马赫数时的刚度。对于结构比较简单并且网格质量良好的计算，采用具有二阶精度的离散格式可以确保对激波的成功捕捉。求解过程隐式迭代格式以保证收敛的稳定性。

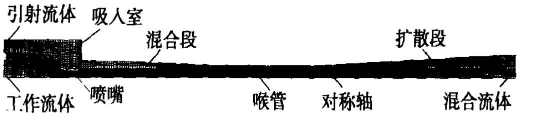


图 2　蒸汽喷射式热泵计算网格模型

2.3　流体物性及边界条件

流体介质为过热水蒸气，密度以理想气体法则处理，粘度由温度的指数函数形式确定。给定工作流体和引射流体入口压力及相应的温度和湍流条件；混合流体出口采用压力出口边界，给定静压和适当的回流条件；固体壁面采用不可渗透和无滑移绝热边界。

3　计算结果分析

3.1　两种喷嘴结构对喷射系数的影响

在其它热力参数保持不变的情况下，工作蒸汽压力由 1.2 MPa 逐渐提高到 2.268 MPa 时，喷射系数的变化规律如图 3 所示。由图可见，整流喷嘴对应的喷射系数较之于普通喷嘴提高近 30%。在给定的结构参数下，整流喷嘴最佳工作压力点相对普通喷嘴最佳工作压力点左移，这有利于整流喷嘴在较低的工作压力下仍然能够保持优良的工作性能。随着工作压力的进一步降低，工作蒸汽产生的动能不足以带动引射流体排出泵外，导致混合流体在出口出现回流，喷射泵的操作性能恶化。当 P_p 越过最佳压力点并继续增大时，由于动能的不可逆损失增大而导致工作流体携带能力降低，从而造成喷射系数下降。

但整流喷嘴对应的喷射系数始终高于普通喷嘴，这有利于削弱压力波动对喷射泵操作性能的影响。图 3 给出了两种喷嘴结构下喷射泵内部马赫数分布，可见，对于普通喷嘴，工作流体在喷嘴出口处的速度呈扩散状（见图 4(a)），存在较厚的边界层，这样可能造成引射流体有效流通截面积减小，产生流道堵塞现象。同时受喉管作用而产生的膨胀波和压缩波的相互作用，造成较大的动量损失，使得扩散段和喉管内流动马赫数衰减较快，特别是在扩散段，随着激波的推移和减弱，流体迅速由超音速衰减为亚音速流动，并在喉管内保持亚音速流动（见图 5）。而增加工作流体的压力，虽然可以在一定程度上弥补由于流速降低而造成的引射动力下降，但提高蒸汽压力所带来的额外蒸汽量同样会造成引射流体有效流通截面积减小，并且使波系向扩散段推移而产生新的膨胀波，导致不可逆动量损失，使得喷射系数降低。

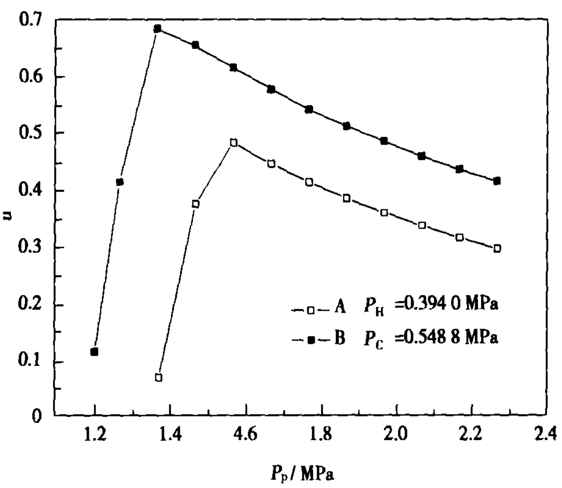


图 3 不同喷嘴结构下工作蒸汽压力对喷射系数的影响（A— 普通喷嘴；B— 整流喷嘴）

而对于整流喷嘴，其波系在整流管内几乎没有衰减，并且在整流管出口充分发展并继续向喉管推进（见图 4(b)，其中 A 和 B 两种结构形式分别对应相同的工作压力 P_P、引射压力 P_H 及背压 P_C），这说明整流管的存在能有效减弱喉管对混合段流动的不利影响，减少能量损失。具体表现在：(1) 安装整流管后，混合段内激波的作用范围减小，流动边界层减薄，有利于增加引射流体的有效流通截面积，使引射流体流动通畅；(2) 激波作用范围的减小并没有导致激波的迅速衰减，在整流管以及整流管出口和喉管之间的扩散段内，流体始终保持超音速流动（见图 5），这样可以增加工作流体的有效负载能力，在不消耗多余能量的前提下，提高设备的工作效率。在扩散段，混合流体的马赫数大于普通喷嘴结构下的马赫数，同时仍然保持亚音速流动，不会增加额外动能损失。在某一操作压力下，工作蒸汽产生的激波系刚好传递到喉管出口而又不进入扩散段，对应的喷射系数达到最大[8]。

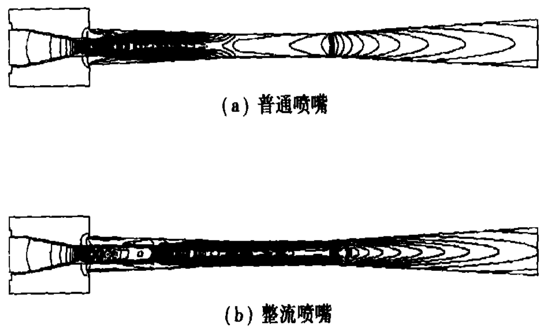


图 4 不同喷嘴结构下马赫数分布图

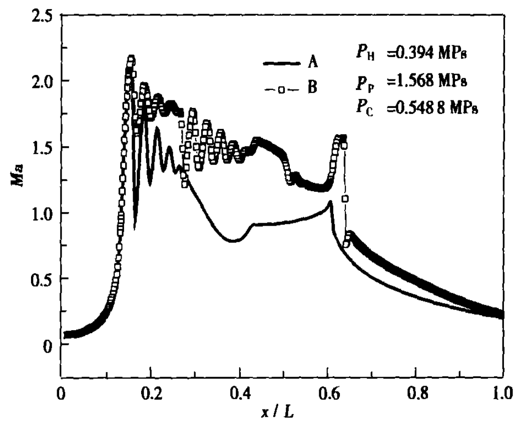


图 5 不同喷嘴结构下轴线马赫数分布（A— 普通喷嘴；B— 整流喷嘴）

3.2 整流管长度对喷射系数的影响

保持工作流体、引射流体以及混合流体的压力不变，逐渐增加整流管长度。以整流管长度的无因次数 λ 表示横坐标，喷射系数表示纵坐标，可得喷射器操作性能随整流管长度的变化规律，如图 6 所示。$\lambda = x_1/L_1$，x_1 表示整流管实际长度，L_1 表示普通喷嘴出口截面与喉管入口截面之间的距离（如图 1(c) 和图 1(a) 所示）。可见，对应于一定的操作参数，随着整流管长度的变化，喷射系数的变化呈近似的抛物状分布，并存在一最佳长度，对应于最大喷射系数。

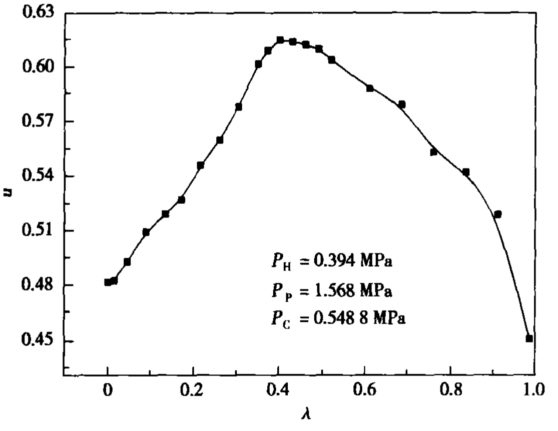


图 6　整流管长度对喷射系数的影响

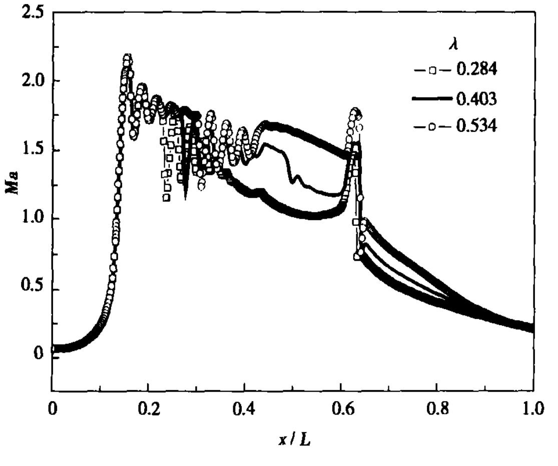


图 7　不同整流管长度下轴线马赫数分布

图 7 为整流管长度变化时, 喷射器轴线马赫数的变化。可见, 整流管长度对整流管以外混合段、喉管和扩散段的流动状态有很大影响。λ 较小时, 轴线上的波系逐渐向喉管推移, 喉管内流速增大, 流体携带能力增强, 喷射系数随 λ 的增大而增加。当 λ 达到某一定值时, 工作蒸汽产生的激波刚好通过喉管, 此时喷射系数达到最大(对应于图 6 中 u 值最大点)。随着 λ 的继续增大, 接近喉管入口时, 混合段流通截面减小对引射流体流动的影响加大, 同时工作蒸汽流动受喉管的影响而产生的波系向扩散段推移。可以设想, 当整流管出口截面与喉管入口截面重合时, 工作流体直接进入喉管而产生膨胀, 喷射泵内的真空度要小于工作流体在混合段膨胀而产生的真空度, 从而降低喷射泵的抽气能力, 并且随着波系向扩散管出口推移, 额外能量损失增大, 喷射系数降低, 喷射泵的操作性能恶化。

4　结　论

(1) 整流喷嘴能有效提高蒸汽喷射泵的工作效率。相同的操作条件下, 其平均喷射系数较普通喷嘴提高近 30%, 并且在工作蒸汽压力较低时仍能保持较高的喷射系数, 对于实现节能以及克服实际操作中由于工作蒸汽量不足而带来的操作失稳有重要意义。并且其结构简单, 容易实现生产装置使用。

(2) 随着整流管长度的增加, 喷射系数先增后减。对于给定的操作条件, 存在着一最佳长度, 对应于最大喷射系数。

参考文献:

[1]　徐海涛, 桑芝富. 蒸汽喷射式热泵变工况性能分析[J]. 热能动力工程, 2003, **18**(4): 395—398.

[2]　HEDGES K R, HILL P G. Compressible flow ejectors part Ⅰ-development of a finite difference flow model[J]. **ASME Journal of Fluids Engineering**, 1974, **96**: 272—281.

[3]　MATSUO K, SASAGUCHI K. Investigation of supersonic air ejectors, part 1: performance in the case of zero-secondary flow[J]. **Bulletin of JSME**, 1981, **24**(198): 2090—2097.

[4]　MATSUO K, SASAGUCHI K. Investigation of supersonic air ejectors, part 2: effects of throat-eara-ratio on ejector performance[J]. **Bulletin of JSME**, 1982, **25**(210): 1898—1905.

[5]　NAHDI E, CHAMPOUSSIN J C, HOSTACHE G, *et al*. Optimum geometric parameters of a cooling ejector-compressor[J]. **International Journal of Refrigeration**, 1993, **16**(1): 67—72.

[6]　CHANG Y J, CHEN Y M. Enhancement of a steam-jet refrigerator using a novel application of the petal nozzle[J]. **Experimental Thermal and Fluid Science**, 2000, **22**: 203—211.

[7]　SUJITH R L, RAMESH R, PRADEEP S, *et al*. Mixing of high speed coaxial jets[J]. **Experiments in Fluids**, 2001, **30**: 339—345.

[8]　徐海涛. 蒸汽喷射器的理论及数值研究[D]. 南京: 南京工业大学, 2003.

ergy &Power. — 2004, 19(5). —495～497

A new method for realizing an adjustable recirculation zone is proposed along with an investigation of its specific features. On the basis of fluid mechanics theory the possibility of adjusting a recirculation zone is realized in the absence of moving elements and high-temperature components (bluff body) to meet the requirement of various kinds of oil with regard to the recirculation zone during their normal combustion. The above-mentioned features are more pronounced, especially in the case of studying the combustion characteristics of "oil-in-water" type of emulsified oil with different water dilution rates. The present study also provides further support for industrial applications. The test and calculation results indicate that this recirculation zone can adapt to the requirements of various ranks of oil. **Key words**: adjustable recirculation zone, "oil in water" type emulsified oil, burner

电站锅炉鳍片管省煤器鳍片尺寸优化模型＝**Optimization Model for Economizer Fin Size in a Utility Boiler** [刊，汉] / YAN Wei-ping, MENG Xue-mei, LU Yu-kun (Power Engineering Department, North China University of Electric Power, Baoding, China, Post Code: 071003) //Journal of Engineering for Thermal Energy &Power. — 2004, 19(5). —498～501

On the basis of heat exchange mechanism and heat transfer control equations of finned tubes a mathematical model is derived for the selection of fin sizes with the maximum heat transfer under the condition of a given fin-metal mass. The objective is to optimize the height and thickness of rectangular straight fins under the typical operating conditions of a utility boiler economizer. Moreover, with the economizer of a utility boiler serving as an object of study the optimization method and procedures are outlined along with a calculation and analysis of the influence of flue-gas flow speed and pollution factor on the optimized sizes. **Key words**: utility boiler, economizer, finned tube, fin size optimization

内可逆四热源吸收式热泵生态学最优性能＝**Ecological Optimal Performance of Endoreversible Four-heat-source Absorption Type of Heat Pumps** [刊，汉] / QIN Xiao-yong, CHEN Lin-gen, SUN Feng-rui (Department of Nuclear Science and Engineering, Naval Engineering University, Wuhan, China, Post Code: 430033) //Journal of Engineering for Thermal Energy &Power. — 2004, 19(5). —502～505

On the basis of an energy-analysis viewpoint established were the ecological criteria reflecting the optimal compromise between four-heat-source absorption heat-pump heating load and entropy production rate. An analysis is given of the ecological optimal performance of endoreversible four-heat-source absorption heat-pumps under a linear (Newtonian) heat transfer law. The following items were derived: the optimization relationship between the ecological objective and pump heating factor, the maximum ecological objective value and its corresponding pump heating factor, pump heating load and entropy production rate. Ecological optimal selection scope of the cycle main parameters was determined. By way of numerical sample calculations analyzed was the relationship between pump heating rate objective and ecological objective. Calculation results indicate that the ecological criteria are a candidate optimization objective having a long-term effect for the optimal design of absorption heat-pumps. **Key words**: four-heat-source absorption heat-pump, ecological criteria, pump heating load, pump heat production factor, entropy production rate

新型喷嘴结构下蒸汽喷射式热泵性能的数值研究＝**Numerical Study of the Performance of Steam-jet Heat Pumps with an Innovative Nozzle Structure** [刊，汉] / ZHANG Shao-wei, SANG Zhi-fu (Mechanical and Power Engineering Institute under the Nanjing Polytechnical University, Nanjing, China, Post Code: 210009), XU Hai-tao (Jiangsu Suyuan Environmental Protection Engineering Co. Ltd., Nanjing, China, Post Code: 210024) //Journal of Engineering for Thermal Energy &Power. — 2004, 19(5). —506～509

By using a CFD (computational fluid dynamics) numerical simulation method a comparison study was conducted of the influence of a conventional nozzle structure and an innovative one on the operating performance of a steam-jet heat pump.

An analysis was performed of the variation relationship between the flow-straightening tube length and the steam-jet factor. The results of the analysis indicate that under identical operating conditions the nozzle structure of a straightened flow can effectively enhance the steam jet factor of the jet ejector. Meanwhile, there exists an optimum length of the flow-straightening tube corresponding to the maximum jet factor. In addition, the above-mentioned nozzle structure is conducive to effectively preventing the deterioration of equipment performance caused by a reduction of operating fluid pressure, thus increasing the stability of the steam jet pump operation. **Key words**: jet pump, nozzle, steam jet factor, numerical simulation

关于换热系统优化目标函数的探讨＝**An Exploratory Study Concerning the Target Function of Heat Exchange System Optimization** [刊, 汉] / XU Wen-zhong (Department of Architectural Environment and Equipment under the Civil Engineering College of Shandong University of Science & Technology, Taian, Shandong Province, China, Post Code: 271019), ZHANG Kai (Energy & Power Engineering Institute under the Shandong University, Jinan, China, Post Code: 250061), YANG Dong (Air Conditioning Department, Shandong Architectural Engineering Institute, Jinan, China, Post Code: 250061) // Journal of Engineering for Thermal Energy & Power. — 2004, 19(5). —510～512

Through an analysis of the various factors liable to affect expenses caused by the loss of available energy of an heat exchange system a comprehensive consideration has been given to the material consumption expenses of a heat exchanger. A target function based on the second law of thermodynamics was proposed for the optimization of the heat exchange system. Moreover, by making use of the theory of engineering thermodynamics, fluid mechanics and thermo-economics a derivation of the target function was carried out, resulting in the acquisition of a calculating formula for the target function. **Key words**: the second law of thermodynamics, heat exchange system optimization, target function, expenses due to the loss of available energy

液幕状气液两相流流动特性的实验研究＝**Experimental Investigation of the Flow Characteristics of a Gas-liquid Two-phase Flow Assuming the Form of a Gas-liquid Screen** [刊, 汉] / ZHOU Qu-lan, SONG Hong-peng, HUI She-en, et al (Thermal Energy Engineering Department, Xi'an Jiaotong University, Xi'an, China, Post Code: 710049) // Journal of Engineering for Thermal Energy & Power. — 2004, 19(5). —513～516

A new flow pattern, the so-called "gas-liquid screen", is proposed and an experimental investigation of the gas-liquid two-phase flow characteristics based on this new flow pattern conducted. The investigation of the flow characteristics has been focused on the relationship between the above-mentioned bed layer height of the gas-liquid two-phase flow and resistance characteristics on the one hand and gas phase and liquid phase flow speed on the other. As a result, correlation equations for calculating equivalent bed layer height, actual bed layer height and the resistance factor of the gas-liquid screen bed layer were obtained. These correlation equations provide basic scientific test data for the research of the gas-liquid two-phase flow assuming the form of a gas-liquid screen. **Key words**: gas-liquid two-phase flow, fluid dynamics, experimental research

节流件阻力特性的 CFD 研究＝**CFD (computational fluid dynamics) Research on the Resistance Characteristics of a Throttling Element** [刊, 汉] / FU Jian-qiang, CHEN Jun, YANG Yan-hua (Department of Nuclear Science and System Engineering, Shanghai Jiaotong University, Shanghai, China, Post Code: 200030) // Journal of Engineering for Thermal Energy & Power. — 2004, 19(5). —517～520

One of the major measures to suppress flow instability in a boiling tube consists in increasing the flow resistance at the boiling tube inlet section. With the help of a CFD (computational fluid dynamics) software "CFX" computations and analyses have been performed of a boiling tube inlet section fitted with a tiny throttling element and a formula for calculating the friction resistance of that inlet section was obtained. The calculation results agree well with the experimental data.