热力循环

文章编号: 1001－2060(2007)04－0375－05

竖直通道内相邻气泡对上升的直接数值模拟

李彦鹏[1], 张乾隆[1], 白博峰[2]

(1. 长安大学 环境科学与工程学院, 陕西 西安 710064; 2. 西安交通大学 动力工程多相流国家重点实验室, 陕西 西安 710049)

摘　要: 采用 Level Set 方法和耦合表面张力模型的 Navier-Stokes 方程, 结合 ALE 数值算法, 直接模拟了竖直通道内两个相邻气泡的上升。重点研究不同空间布置的 8 mm 气泡对后面的尾迹流及其相互作用。数值模拟准确再现气泡对的变形、吸引及排斥行为, 气泡上升速度计算结果与经验式吻合。模拟结果表明, 两个气泡后面的尾迹流及其相互作用决定了上升气泡对的行为。并排上升的气泡对, 由于尾流区被一个射流流动分隔, 气泡对没有聚并; 然而当垂直上升气泡对中的尾随气泡有超过 50% 的投影面积进入到前头气泡的尾流区, 聚并现象发生。

关　键　词: 气泡对; 尾迹流; Level Set 方法; 直接数值模拟

中图分类号: O351.1; TK16　　文献标识码: A

引　言

气泡运动广泛存在于能源、动力和环境等工程设备与工艺中, 因此对气泡运动规律的研究具有十分重要的实际意义。气泡运动属于典型的带有相界面的气液两相流动, 以前对它的研究大都采用高速摄像技术以可视化气泡行为。近年来, 随着计算机技术和数值方法的发展, 出现了许多数值模拟方法来描述气泡的界面行为, 例如, 峰面跟踪法、VOF 方法和 Lattice Boltzmann 方法等[1~3]。

在现有的气泡运动数值模拟中, 大都是针对单个气泡在不同黏度和表面张力的液体中的上升过程进行计算, 对气泡对上升过程的模拟则很少。究其原因, 主要是气泡对的运动涉及到气液相界面的变形、破碎与聚并等复杂变化, 对于这类相界面拓扑结构变化很大的问题, 尤其在三维情况下, 上述方法实施起来非常复杂。Level Set 方法作为一种新的界面追踪方法, 隐式处理界面, 简单精确, 易于在三维中使用, 近年来得到了越来越多的关注与研究[4~5]。本文采用 Level Set 方法, 结合考虑表面张力的气液运动基本方程, 数值模拟不同配置的气泡对在竖直通道内的上升过程, 研究气泡对相互作用规律。

1　数学模型与数值方法

1.1　气液两相主流场控制方程

对于气泡在黏性液体中运动这样的气液两相流动, 可假定气泡内外的气液两相均为不可压缩且不相溶混的牛顿型流体, 则其流动可用下述 Navier-Stokes 方程描述:

$$\nabla\cdot u = 0 \tag{1}$$

$$\partial(\rho u)/\partial t + \nabla\cdot(\rho uu) = -\nabla p + \nabla\cdot(2\mu D) + \rho g + \sigma\kappa\delta(d)n \tag{2}$$

式中: ρ、μ, u 和 p—流体的密度、黏度、速度矢量与压强; D—应变率张量, 其表达式为 $D_{ij} = (\partial u_j + \partial u_i)/2$。式(2)右边最后一项代表界面上的表面张力, 由连续表面力模型给出[6], 其中的变量 σ、κ、δ、d 和 n 分别表示表面张力系数、界面曲率、Dirac Delta 函数、计算域中各点到界面的距离和界面外法向单位向量。需要注意的是, 上面的控制方程将相界面分隔的气液两相看作物性随空间变化的单一流体体系, 即流体的密度与黏度为:

$$\begin{aligned}\rho(x,t) &= \rho_f H(x,t) + \rho_g[1-H(x,t)]\\ \mu(x,t) &= \mu_f H(x,t) + \mu_g[1-H(x,t)]\end{aligned} \tag{3}$$

式中: x—计算域中各点的位置; 下标 f 和 g—液相和气相; $H(x,t)$—一个指示函数, 定义为液相中其值为 1, 气相中为 0。

1.2　界面运动方程

采用 Level Set 方法描述气液相界面。Level Set 方法把随时间运动的气液相界面 $\Gamma(t)$ 看作某个标量函数 $\varphi(x,t)$ 的零等值面, 即 $\Gamma(t) = \{x \mid \varphi(x,t) = 0\}$, 同时 $\varphi(x,t)$ 的初值应满足 $\Gamma(t)$ 附近法向

收稿日期: 2006－12－01;　修订日期: 2007－01－15
基金项目: 国家自然科学基金资助项目(50476049)
作者简介: 李彦鹏(1973－), 男, 陕西泾阳人, 长安大学副教授.

单调。一般可取 $\varphi(x,0)$为x 点到界面 $\Gamma(0)$的符号距离,通常取气泡内 φ 为负,气泡外 φ 为正。

为了保证任意时刻函数 φ 的零等值面就是运动界面,在气—液两相流中,φ 遵循下面的输运方程:

$$\partial\varphi/\partial t+u\cdot\nabla\varphi=0 \tag{4}$$

在任意时刻,只要求出 φ 的值,就可以确定气液界面的位置。为了求解方便,保持 $\varphi(x,t)$的符号距离性质是重要的,需要在每个计算步采用重新初始化(Reinitialization)技术[7],这可以通过求解下面初值问题的稳定解来实现:

$$\partial\varphi/\partial\tau=\mathrm{sign}(\varphi_0)(1-|\nabla\varphi|) \tag{5}$$

式中:τ—伪时间变量;φ_0—重新初始化前的 Level Set 函数;sign—符号函数,其定义为:

$$\mathrm{sign}(\varphi_0)=\frac{\varphi_0}{(\varphi_0^2+\Delta^2)^{1/2}} \tag{6}$$

式中:Δ—网格尺寸。

1.3　数值方法

虽然上述主流场控制方程将整个计算区域一起求解简化了计算,但是由于相界面的存在使流体性质不连续,尤其对于气水这样物性相差很大的两相流动,往往造成数值计算的困难。为此,采用光滑化处理技术,即人为假定界面具有有限厚度 ε,物性参数在界面层中连续变化。此时,式(3)中的指示函数 H 变为:

$$H_\varepsilon(\varphi)=\begin{cases}0 & \varphi<-\varepsilon\\ \dfrac{\varphi+\varepsilon}{2\varepsilon}+\dfrac{\sin(\pi\varphi/\varepsilon)}{2\pi} & |\varphi|\leqslant\varepsilon\\ 1 & \varphi>\varepsilon\end{cases} \tag{7}$$

光滑化处理的具体方法可参见文献[7]。数值试验表明,界面光滑化处理虽然部分模糊了界面,但是它极大地改善了数值稳定性,而且这种模糊化可以控制在有限几个网格内。

本文采用有限体积法对主流场和气液界面的控制方程进行耦合求解。求解步骤为:(1)给定初始场,特别要给出初始距离函数场;(2)使用 ALE (Arbitrary Lagrangian Eulerian)数值算法求解控制式(1)与式(2),其中有关参数的计算按式(3)与式(7)进行;(3)求解界面运动式(4);(4)重新初始化,解式(5)和式(6);(5)重复步骤(2)~(4),进入下一时间步的计算。

2　结果与分析

2.1　模型的验证

首先对单个球形气泡在静止水中的三维上升过程模拟,以验证模型的正确性。模拟区域为4 cm× 4 cm× 8 cm 的方截面竖直通道,直径 $d_b=8$ mm 的静止气泡位于距下边界 0.5 cm 的通道正中间。模拟中液体的物性参数为 $\rho_f=998$ kg/m^3,$\mu_f=0.001$ Pa·s,$\sigma=0.0728$ N/m;气体的物性参数为 $\rho_g=1.1$ kg/m^{3}和 $\mu_g=1.8\times10^{-5}$ Pa·s。计算区域的上下边界分别采用压力与速度入口边界条件,其它壁面采用无滑移条件。

经过网格无关性数值试验,发现为了正确模拟气泡行为,气泡在每一个方向上至少需要 16 个计算网格覆盖。因此,本文的模拟均采用 0.05 cm 的网格尺寸以满足这个条件。另外,对于下面气泡对的模拟,使用相同的气液两相物性参数与边界条件。

计算得到的单个气泡上升过程如图 1 所示。图中每两个气泡之间的时间间隔为 0.05 s。可以看出,气泡从初始的球形连续变化为扁椭球状。此时,Eotvos 数和 Morton 数分别为 8.6 和 2.5×10^{-11}。与 Fan 给出的气泡形状图谱比较[8],模拟的气泡形状完全吻合。根据气泡位置的变化可计算出气泡平均上升速度大约是 17.6 cm/s。Collins 给出了单个气泡在有限空间内的上升速度的经验公式[9]:

$$V_b=0.71\alpha\sqrt{gd_b} \tag{8}$$

式中:有限空间修正因子 α 由下式计算:$\alpha=1.12\exp(-d_b/D_T)$,$D_T$ 为通道的定性尺寸。这样计算得到的气泡上升速度 $V_b=17.9$ cm/s。显然,模拟结果与经验公式结果比较相近,表明本文的数值模型及数值算法是正确的。

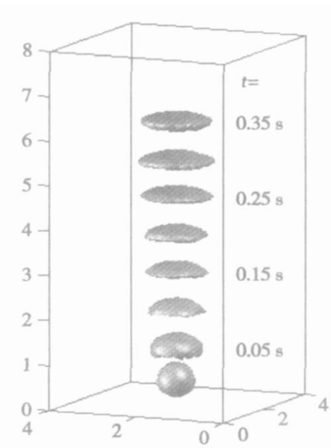


图 1　单个上升气泡的形状变化

2.2　水平并排气泡对的上升

考虑一个5 cm× 5 cm× 6 cm 大小的竖直通道,距底部0.8 cm 处并排放置两个 8 mm 直径的球形气

泡。两气泡中心的初始水平距离为 12 mm。图 2 示出了在这种配置下气泡对上升过程的模拟结果。由于没有更好的实验数据, 本文采用 Tokuhiro 等人使用高速 CCD 拍摄的两个当量直径为 9 mm 的椭球形气泡在水中的运动图片作为比较的对象[10], 如图 2 右图所示。显然, 模拟与实验都表明, 气泡变形为扁椭球形, 而且呈摇摆状上升。由于气泡对的摇摆, 它们在上升过程中出现了"靠近-分离-再靠近-再分离"的运动现象, 计算过程中没有出现两个气泡的聚并行为。

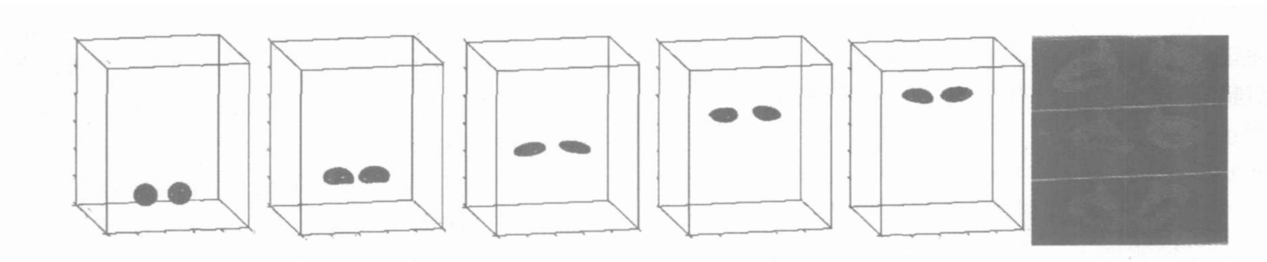


图 2　水平并排气泡对上升过程的模拟结果

图 3 示出了图 2 后 3 个时刻的中心纵截面的速度矢量图。为了观察气泡的尾流, 该速度矢量图是在减去气泡平均上升速度后得到的, 如图 3 所示。可以观察到, 每一个气泡的后面都出现有清晰的涡旋, 伴随涡旋的脱落形成各自的尾流区。两个尾流区没有直接相互作用, 而是被一个射流状液体流动区分隔。随着气泡的摇摆上升, 气泡对可以反向相对, 如图 3 所示, 可以正向相对, 如图 3 所示, 或者居于二者之间如图 3 中图所示, 尾流区域也逐渐变长, 形状也随之摇摆变化, 但是整个过程中两个尾流区没有重叠。

2.3　垂直同轴气泡对的上升

将水平并排的气泡对垂直排列, 两气泡中心同

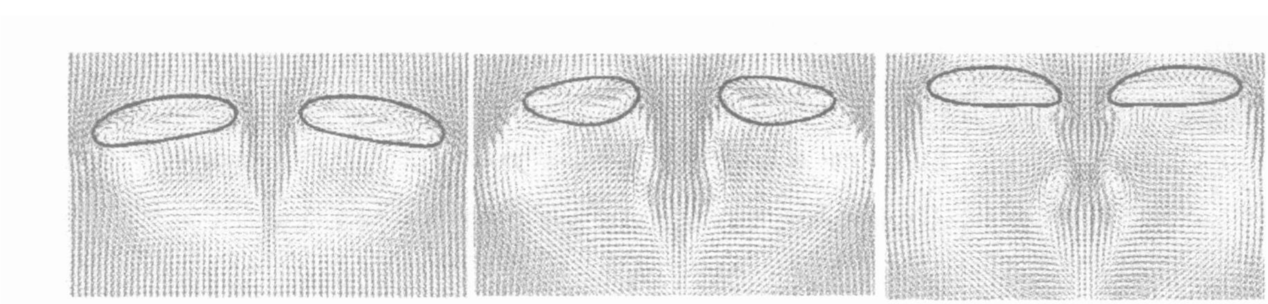
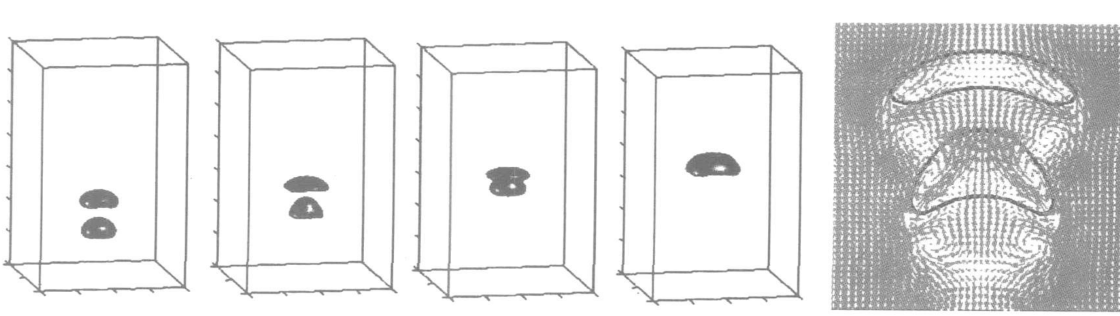


图 3　水平并排气泡对上升过程中气泡区域的速度矢量图



(a) 45 ms　(b) 75 ms　(c) 97.5 ms　(d) 120 ms　(e) 75 ms 时局部速度矢量图

图 4　垂直同轴气泡对上升过程的模拟结果

轴于通道中轴线，其中心距 10 mm。为了节约计算资源，将计算区域变为 4 cm×4 cm×6 cm。模拟得到的气泡对上升过程中其形状的三维变化，如图 4 所示。在这种配置下，气泡对在上升过程中发生聚并。其聚并过程分为 4 个阶段：首先两个气泡同轴上升，伴随有变形，如图 4(a)所示；当两个气泡接近时，尾随气泡被拉长并加速上升，如图 4(b)所示；然后尾随气泡追上前头的气泡，气泡对碰撞，如图 4(c)所示；最后聚并为一个大的球帽状气泡，如图 4(d)所示。整个过程与 Fan 等人的实验观测基本吻合[8]。为了分析气泡对的聚并行为，给出了 75 ms 时刻，中心纵截面上气泡区域的速度矢量图，如图 4(e)所示。从图中可以看到，两个气泡的下部均有两个对称分布的反向旋转的涡旋(尾流区)。在图示时刻，尾随气泡完全进入到前头气泡的尾流区，于是受到尾迹流诱导的局部压差作用，尾随气泡加速上升并与前头气泡碰撞聚并。

2.4　垂直交错气泡对的上升

为了进一步分析气泡对的相互作用，将垂直同轴气泡对交错布置。此时，两个气泡中心的竖直与水平间距分别为 10 mm 和 4 mm。这时模拟得到的气泡对上升过程以及气泡区域的二维速度矢量图，如图 5 所示。可以看到，尽管初始时刻气泡对之间呈交错垂直排列，但是由于初始时刻尾随气泡与前头气泡的投影面积有 50% 的重合，气泡对上升过程，尾随气泡还是会进入前头气泡的尾流影响区，并在此尾流区作用下加速上升，最终聚并为一个大气泡。与垂直同轴气泡对相比，尾随气泡在开始加速上升阶段(t=90 ms 左右)，并没有全部被前头气泡的尾流所捕获，但有大于 50% 的投影面积进入到前头气泡的尾流区。

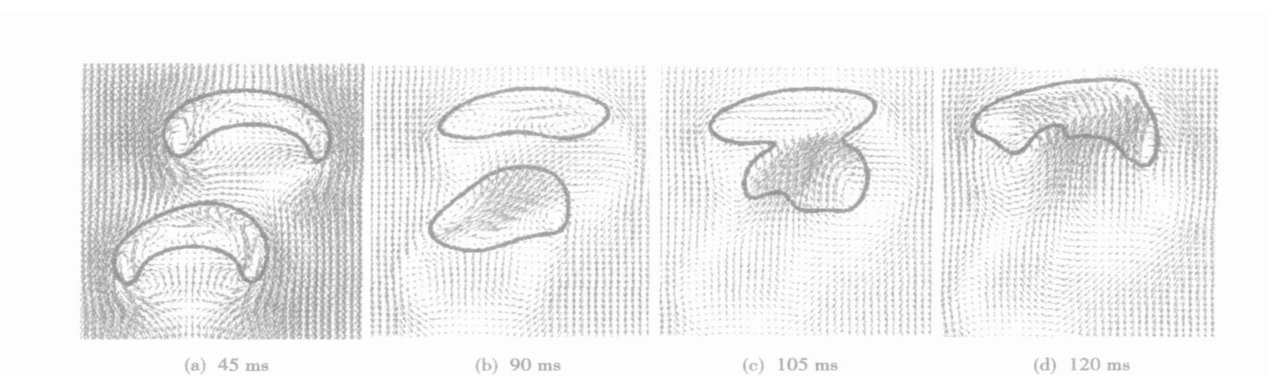


图 5　垂直交错气泡对上升过程及其速度矢量分布的模拟结果

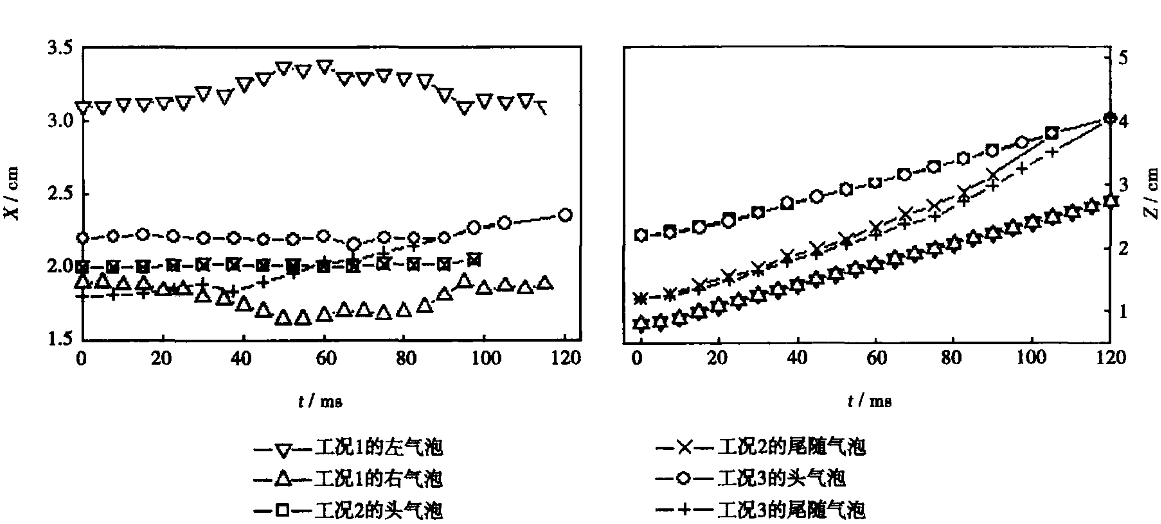


图 6　不同气泡对配置下气泡位置变化的比较

2.5　气泡相互作用对上升速度的影响

为了分析气泡对相互作用对气泡上升速度的影响, 比较了上述 3 种配置(工况 1、工况 2 与工况 3)气泡的垂直与水平位置变化, 如图 6 所示。可以看到, 水平并排上升的两个气泡的垂直位置(Z)变化曲线呈线性重合, 表明两个气泡的垂直上升速度相等, 由直线的斜率可得出此速度为 17.2 cm/s; 而两个气泡的水平位置(X)变化曲线则以容器的中轴线为对称, 这说明两个气泡水平运动反向, 这也进一步解释了气泡对在上升过程中靠近—分离—再靠近的运动特点。对于垂直同轴气泡对和交错气泡对的上升, 可以发现前一个气泡垂直位置的变化曲线也基本呈线性重合, 气泡上升速度大约为 17.2 cm/s, 表明前头气泡的上升没有受到尾随气泡的影响, 但尾随气泡的上升速度受前头气泡的影响显著。对于垂直同轴气泡对的上升, 尾随气泡以大约 20.8 cm/s 的速度上升到 75 ms 以后, 明显加速到 37.6 cm/s。这说明尾随气泡在 75 ms 之后受前头气泡的影响远远大于之前的影响, 显然这是尾随气泡在 75 ms 完全被前头气泡的尾流区捕获的缘故。垂直交错气泡对的上升有相似的影响, 不过相比与垂直同轴上升的气泡对, 由于两气泡初始距离大, 所以尾随气泡经过了更长的时间才赶上前一个气泡。

3　结　论

(1) 采用 Level Set 方法结合考虑表面张力影响的 Navier—Stokes 方程, 成功地数值模拟了两个直径 8 mm 的气泡在水中的上升、变形、吸引和排斥等行为。模拟结果与文献[8~10]中可用的实验观测和经验公式吻合较好。

(2) 气泡对周围的速度场模拟结果揭示了气泡后面的尾迹流特征。两个气泡后面的尾迹流及其相互作用决定了气泡对上升过程的形状与速度变化。模拟试验表明, 对于垂直上升的气泡对, 前头气泡的尾迹流对气泡间的相互作用有重要影响。当尾随气泡有大于 50% 的投影面积进入到前头气泡的尾流区, 则由于受到前头气泡尾迹流诱导的局部压差作用, 尾随气泡会产生一明显的加速上升, 最终导致两个气泡聚并为一个大气泡。

(3) 模拟试验表明, 对于水平并排上升的气泡对, 其尾流区之间存在一个类似射流的流动分布, 正是由于射流分布使气泡对在上升过程中不能直接相互作用, 因此没有发现气泡对的聚并现象。气泡对的摇摆作用使它们在上升过程中呈现出"靠近—分离—再靠近"的运动特征。

参考文献:

[1] UNVERDI S Q, TRYGGVASON G. A front-tracking method for viscous incompressible multi-fluid flows[J]. J Comput Phys, 1992, 29: 25—37.

[2] CHEN L, LI Y. A numerical method for two-phase flows with an interface[J]. Environmental Model & Software, 1998, 13: 247—255.

[3] SANKARANARAYANAN K, SHAN X, KEVREKIDIS I G, et al. Bubble flow simulations with the lattice Boltzmann method[J]. Chemical Engineering Science, 1999, 54: 4817—4823.

[4] OSHER S, FEDKIW R P. Level set methods: an overview and some recent results[J]. J Comput Phys, 2001, 169: 463—502.

[5] 李彦鹏, 白博峰. 气泡从浸没孔中生成与上升的数值模拟[J]. 水动力学研究与进展 A 辑, 2006, 21(5): 660—666

[6] BRACKBILL J U, KOTHE D B, ZEMACH C. A continuum method for modeling surface tension[J]. Journal of Computational Physics, 1992, 100(2): 335—354

[7] SUSSMAN M, FATEMI E, SMEREKA P, et al. An improved level set method for incompressible two-phase flows[J]. Computers & Fluids, 1998, 27(5/6): 663—680

[8] FAN L S, TSUCHIYA K. Bubble wake dynamics in liquids and liquid-solid suspensions[M]. Boston: Butterworth-Heinemann, 1990.

[9] COLLINS R. The effect of a containing cylindrical boundary on the velocity of a large gas bubble in a liquid[J]. Journal of Fluid Mechanics, 1967, 28: 97—112.

[10] TOKUHIRO A, FUJIWARA A, HISHIDA K, et al. Measurement in the wake region of two bubbles in close proximity by combined shadow image and PIV techniques[J]. Journal of Fluids Engineering, 1999, 121: 191—197.

(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

By adopting Level Set method and a Navier-Stokes equation coupled with a surface tension model and in combination with ALE (Arbitrary Lagrangian Eulerian) numerical algorithm, a direct numerical simulation was conducted of the rising process of two neighboring air bubbles inside a vertical channel. The emphasis was put on the study of the effect of 8 mm diameter air bubbles arranged at different spatial positions on the rear wake flows and their interactions. The numerical simulation can accurately reproduce the deformation, attraction and repellent action of the air bubble pairs. The calculation results of air bubble rising speed are in good agreement with those obtained by using an empirical formula. The simulation results show that the wake flows after the two air bubbles and their interaction determine the behavior of the rising air bubble pair. An air bubble pair rising side by side does not coalesce due to their wake zones being separated by a jet flow. When the air bubbles following the vertically rising air bubble pair have over 50% of their projection area entering the wake zones of their ahead air bubbles, however, a coalescence phenomenon will occur. **Key words**: air bubble pair, wake flow, Level Set method, direct numerical simulation

垂直自由下降液膜厚度的瞬时无接触测量研究= **A Study of the Transient Non-contact Measurement of Vertical Free-falling Liquid-film Thickness**[刊, 汉] / YAN Wei-ping, YE Xue-min (Education Ministry Key Laboratory on Condition Monitoring and Control of Power Plant Equipment under North China Electric Power University, Baoding, China, Post Code: 071003), LI Hong-tao (Power Engineering Department of Shenyang Engineering College, Shenyang, China, Post Code: 110136), GU Gen-dai (Department of Mathematics, North China Electric Power University, Baoding, China, Post Code: 071003) // Journal of Engineering for Thermal Energy & Power. —2007, 22(4). —380 ~384

On a vertical free-falling liquid-film test rig, by employing the image rapid-acquisition function of a CCD (Charge Coupled Device), the transient flow-pattern images of a free-falling liquid-film flow at different Reynolds numbers were obtained and then digitally processed on a computer by using an image processing method. Studied were the evolution characteristics of the transient liquid-film thickness along the flow direction in a certain zone and the time-dependent evolution characteristics of the above thickness at a certain location. Also given was an experimental formula showing the correlation of the average liquid film thickness with Reynolds numbers under the condition of different Reynolds numbers. The test results show that the measurement accuracy is comparatively high when the Reynolds number is less than 4000 and the measurement error is relatively big when the Reynolds number is over 4000 due to the influence of sampling facilities. The authors have undertaken a non-contact measurement of flow characteristics of liquid films, initiating a useful attempt for applying digital image processing technology to the study of thin-film flow characteristics. **Key words**: non-contact measurement, liquid film, thickness, evolution characteristics, correlation formula

激冷室内气体穿越液池过程气液固三相的数值模拟= **A Gas-liquid-solid Three-phase Numerical Simulation of the Syngas Passing Through a Cistern in a Quench Chamber**[刊, 汉] / WU Xuan, XIE Han-yan, LI Tie, et al (Education Ministry Key Laboratory on Clean Coal Power Generation and Combustion Technology, College of Energy Source and Environment under the Southeast University, Nanjing, China, Post Code: 210096) // Journal of Engineering for Thermal Energy & Power. —2007, 22(4). —385 ~390

A syngas passing through a cistern in the quench chamber of a coal slurry gasifier pertains to a sophisticated gas-liquid-solid three-phase flow process, which functions to further cool the syngas and capture and collect the slag contained therein. By combining the Euler method with Lagrange one, the authors have calculated particle collisions by using a direct simulation Monte Carlo's (DSMC) method and employing VOF model to track the gas-liquid interface. A numerical simulation was conducted of the gas-liquid-solid three-phase flow process of slag-laden syngas passing through a cistern. An exploratory study has been performed of the influence of the following factors on the gas-liquid flow field and separation of solid particles: particle diameter, gas flow velocity and submerged depth of the downcomer at the outlet. The research results show that the syngas undergoes an abrupt change in flow direction after it leaves the downcomer and the formation of gas and liquid exhibits a periodic wave-shaped flow pattern. The dust-laden gas, when it passes through the cistern, features a relatively high particle-capture efficiency. An increase in particle diameters can also enhance the particle capture efficiency. With an increase of gas flow velocity and submerged depth of the downcomer at the outlet, the perturbation of the liquid intensifies, producing more liquid drops and contributing to an enhancement of particle-capture efficiency. The influence of gas flow velocity and submerged depth of the downcomer on the particle-capture efficiency, however, will be gradually weakened. **Key words**: gasifier, quench chamber, gas-liquid-solid three-phase, capture efficiency, direct simula-