文章编号:1001－2060(2004)02－0131－03

流化床密相区流动特性的数值模拟

徐 祥,向文国,秦成虎
(东南大学 洁净煤发电及燃烧技术教育部重点实验室,江苏 南京 210018)

摘 要:流化床内气固两相流动一直是实验研究和数值模拟的热点。基于Eulerian双流体模型,本文建立了流化床内的气固两相流动模型,采用FLUENT软件对流化床密相区两相流动特性、床内气泡的产生运动和爆裂等特性进行了数值模拟。模型中,将颗粒相看作是连续介质,建立与气相相同形式的数学模型;采用了离散介质动力理论,引入颗粒温度来描述固相粘性应力,并用气固曳力进行气固两相耦合。模拟得到了气泡产生、运动和爆裂的变化过程,与实验结果相一致。采用不同的曳力模型对流化床稠密两相流动进行了模拟,与Kuipers实验对比,结果表明采用Gidaspow曳力模型描述流化床稠密两相流动特性更准确。

关 键 词:Eulerian模型;气固两相流;气泡;流化床
中图分类号:O359 文献标识码:A

1 引 言

流化床在动力化工等领域得到愈来愈多的应用,而到目前为止,对其中的稠密气固两相流动、化学反应、传热等物理化学现象和作用规律并没有彻底的了解。随着两相流动理论的发展和计算机计算能力的提高,已有一些学者采用数值模拟方法研究流化床内气泡行为特性。如Wachem等人采用CFX计算软件[1],应用Eulerian模型,对鼓泡流化床里气泡尺寸、速度进行模拟,所得结果和Darton(1977)总结的气泡大小经验关系式符合良好。Goldschmidt等人运用Eulerian模型研究了颗粒碰撞还原系数对流化床密相区流动特性的影响[2]。祁海鹰等人[3]运用欧拉双流体模型模拟流化床密相区的气泡生长、发展和爆裂,并在考虑颗粒团聚效应的基础上对气固曳力模型做修正。

本文使用商业软件Fluent进行流化床密相区流动特性数值模拟,在计算模型的基础上,加入模型修正,如影响稠密两相流动的重要因数——气固曳力系数,采用不同的气固曳力模型研究其对流动的影响。Fluent可以方便克服计算收敛的困难,提高计算精度,得到较为满意的结果。

2 计算模型

Eulerian方法的应用是在一定浓度条件下,把离散的固体颗粒相看作一种假想的连续介质,即所谓“拟流体”假设,这样颗粒相就具备与气相相似的动力学特性,也就能用形式相同的流体力学守恒方程加以描述。颗粒相在空间中量的多少用体积百分比浓度来表示。由于流动空间为气固两相所分割,它们的当地流体力学参数就必须通过颗粒体积分数加以修正。气固相互作用则是通过气固曳力予以耦合,其它影响因素如湍流效应均可通过模型封闭方法予以考虑。

下面具体给出模型的控制方程和封闭方程:

$$\sum_{i=1}^{n}\alpha_i = 1 \quad (1)$$

$$\frac{\partial}{\partial t}(\alpha_i\rho_i)+\nabla\cdot(\alpha_i\rho_i\vec{v}_i)=0 \quad (2)$$

用空隙率 α_i 来表示每一相在流体空间中所占的体积份额,i 标识不同的相。文中下标g表示气相,s表示固相。式(2)为适用于气固两相的连续性方程。ρ_i 和 $\vec{v}_i$ 为第 i 相的密度和速度向量。

气相动量守恒方程:

$$\frac{\partial}{\partial t}(\alpha_g\rho_g\vec{v}_i)+\nabla\cdot(\alpha_g\rho_g\vec{v}_g\vec{v}_g)=-\alpha_g\nabla p+\nabla\cdot\bar{\bar{\tau}}_g+\alpha_g\rho_g\vec{g}+K_{sg}(\vec{v}_s-\vec{v}_g) \quad (3)$$

颗粒相动量守恒方程:

$$\frac{\partial}{\partial t}(\alpha_s\rho_s\vec{v}_s)+\nabla\cdot(\alpha_s\rho_s\vec{v}_s\vec{v}_s)=-\alpha_s\nabla p-$$

收稿日期:2003－06－09; 修订日期:2003－11－03
基金项目:国家自然科学基金资助项目(虚拟流化床仿真研究50176010)
作者简介:徐 祥(1979－),男,江苏句容人,东南大学硕士研究生.

$$\nabla p_s + \nabla \cdot \bar{\bar{\tau}}_s + \alpha_s \rho_s \vec{g} + K_{gs}(\vec{v}_g - \vec{v}_s) \quad (4)$$

颗粒相的体积粘度(bulk viscosity)λ_s:

$$\lambda_s = \frac{4}{3}\alpha_s \rho_s d_s g_{0,ss}(1 + e_{ss})\left(\frac{\Theta_s}{\pi}\right)^{0.5} \quad (5)$$

式中:$g_{0,ss}$ 径向分布函数,e_{ss} 为颗粒碰撞还原系数,本文中设定 $e_{ss} = 0.95$。$\alpha_{s,max}$ 为颗粒最大空隙率,值通常在 0.6 ~ 0.7 之间。颗粒温度 Θ_s 是与颗粒运动的波动动能成比例的,下文详细介绍。$K_{gs}(\vec{v}_g - \vec{v}_s)$ 为气固曳力项,相间动量交换系数(interphase momentum exchange coefficient)$K_{gs} = K_{sg}$,曳力模型:

(1)Gidaspow(1992)曳力模型:

$$K_{gs} = \frac{18}{\alpha_g Re_s}[1 + 0.15(\alpha_g Re_s)^{0.687}] \times \frac{\alpha_s \alpha_g \rho_g |\vec{v}_s - \vec{v}_g|}{d_s}\alpha_g^{-2.65} \qquad \alpha > 0.8 \quad (6)$$

$$K_{gs} = 150\frac{\alpha_s(1 - \alpha_g)\mu_g}{\alpha_g d_s^2} + 1.75 \times \frac{\alpha_s \rho_g |\vec{v}_s - \vec{v}_g|}{d_s} \qquad \alpha_g \leq 0.8 \quad (7)$$

(2)Syamlal et al(1993)曳力模型:

$$K_{gs} = \frac{3\alpha_s \alpha_g \rho_g}{4v_{r,s}^2 d_s}C_D\left(\frac{Re_s}{v_{r,s}}\right)|\vec{v}_s - \vec{v}_g| \quad (8)$$

$$C_D = \left(0.63 + \frac{4.8}{\sqrt{Re_s / V_{r,s}}}\right)^2 \quad (9)$$

引入颗粒温度(Granular Temperature)概念,定义如式(10),v' 为颗粒速度波动。从定义可以看出颗粒温度是与颗粒随机运动的动能成比例的。从动能理论得到的输运方程(11):

$$\Theta_s = \frac{1}{3}v'^2 \quad (10)$$

$$\frac{3}{2}\left[\frac{\partial}{\partial t}(\alpha_s \rho_s \Theta_s) + \nabla \cdot (\alpha_s \rho_s \vec{v}_s \Theta_s)\right] = (-p_s \bar{\bar{I}} + \bar{\bar{\tau}}_s) : \nabla \vec{v}_s + \nabla \cdot (k_{\Theta_s} \nabla \Theta_s) - \gamma_{\Theta_s} + \phi_{gs} \quad (11)$$

3 计算方法

本文用 Fluent 对 Eulerian 模型进行计算。采用 *GAMBIT* 对所计算的几何区域划分网格系统,在局部如气体进口和近壁面处进行网格加密,以适应流场的变化。计算区域为二维矩形,左右两边设为壁面。区域的上部设为压力出口边界,该边界条件意味着在出口处无回流,为充分发展流场。底部中心为一个进气口,即一次风入口,此处颗粒的向下速度设为零。其它进口以及边界、初始条件在表 1 中给出。

表 1 模型参数列表

物理量	值	说明
床高 H/m	1	固定值
床宽 W/m	0.4	固定值
颗粒直径 d_s/m	5×10^{-4}	统一粒径,无宽筛分
颗粒密度 ρ_s/kg·m^{-3}	2 660	
入口气体速度 U_0/m·s^{-1}	10	可在在一定范围内变化
颗粒碰撞还原系数 e_{ss}	0.95	由 Boemer et al.(1995)
固体最大空隙率 α_{max}	0.58	Syamlal et al.(1993)
静止床高 h/m	0.5	
时间步长 Δt/s	5×10^{-5}	达到收敛要求

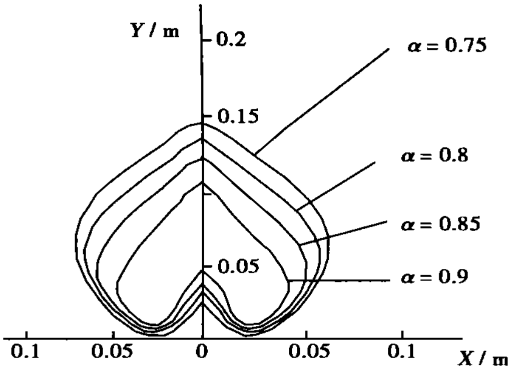

图 1 不同空隙率下气泡的形状和面积

4 计算结果及分析

气泡在床内所占空间的大小通过气体空隙率大于某一固定值来表示。图 1 显示了当 α_g 在 0.75 ~ 0.9 之间变化时气泡面积的差异。不同的研究者对 α_g 采用了不同的取值,C.Guenther 在研究中 α_g 取 0.7[4],J.J.Nienwland 等人 α_g 取 0.85[5]。在本文中 α_g 取 0.8,下面的结果都是在这个条件下得到。图 2 为 0 ~ 0.75 s 时间内气泡从产生、上升及爆裂的发展过程。

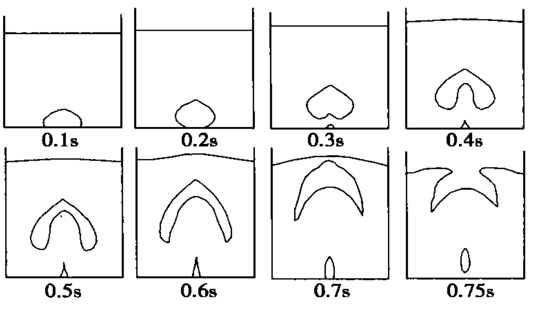

图 2 单 气泡产生变 化过程

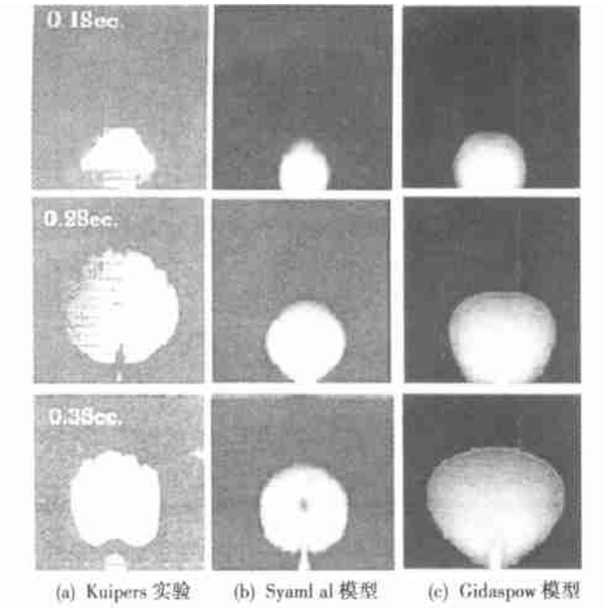

图 3　实验和模拟结果对比

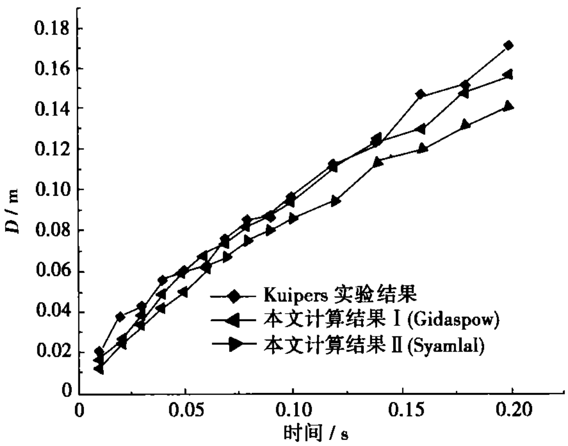

图 4　不同模型模拟结果和实验所得气泡直径随时间变化关系

图 3 为在 0.1 s,0.2 s 和 0.3 s 时模拟结果和实验结果的对比。其中(a)为 Kuipers (1990)[6] 实验所得到的气泡图象;(b)为曳力系数采用 Syamlal et al (1993)模型计算所得气泡图象;(c)表示曳力系数采用 Gidaspow(1992)模型计算所得气泡图象。图 4 为模型预测气泡发展与 Kuipers 实验数据的比较图。计算采用的曳力模型分别是 Gidaspow 和 Syamlal 模型。从图中明显可以看出,在模拟流化床两相流动时,Gidaspow 曳力模型比 Syamlal 曳力模型更符合实验结果。

5　结　语

本文总结了一套应用于流化床密相区流动计算和预测气泡特性的模型,并用 Fluent 软件进行数值模拟,所得结果与实验吻合良好。通过本文的工作,发现气固曳力是影响流动情况的重要因数,采用不同的曳力模型将很大程度上决定了数值模拟的合理性、正确性。结果表明 Gidaspow 曳力模型适用于流化床的两相流动模拟。Eulerian 模型为稠密气固两相流动的数值模拟提供了一个有利的工具,今后稠密气固两相流动理论的发展应着重于两相之间的相互作用,以使得模型更加精确与完善,另外一个方面就是向床内传热和化学反应等方向发展。

参考文献:

[1] WACHEM B G M VAN, SCHOUTEN J C, KRISHNA R, et al. Eulerian Simulation of Bubbling Behaviour in Gas-Solid Fluidised Beds[J]. **Compuers chem Engng**, 1998, **22**: 299—306.

[2] GOLDSCHMIDT M J V, KUIPERS J A M, SWAAIJ W P M VAN. Hydrodynamic modelling of dense gas-fluidised beds using the kinetic theory of granular flow: effect of coefficient of restitution on bed dynamics[J]. **Chemical Engineering Science**, 2001, **56**: 571—578.

[3] 祁海鹰,由长福,徐旭常. 稠密气固两相流动欧拉数值模拟的理论与实践 Fluent 第一届中国用户大会 http://www.hikeytech.com/content/43.pdf.

[4] GUENTHER C. The Effect of Numerical Diffusion on Isolated Bubbles in a Gas-Solid Fluidized Bed. www.mfix.org/pub/num-bub2.pdf.

[5] NIENWLAND J J, VEENENDAAL M L, KUIPERS J A M. Bubble formation at a single orifice in gas fluidized beds[J]. **Chemical Engineering Science**, 1996, **51**: 4087—41026.

[6] KUIPERS J A. Two fluid micro blance model of fluidized bed. PhD thesis, University of Twente, The Netherlands, 1990. J. Ding and D. Gidaspow. A Bubbling Fluidization Model Using Kinetic Theory of Granular Flow[J]. **AIChE J**, 1990, **36**(4): 523—538.

(渠　源　编辑)

of Bubble Physics &Natural Circulation under the Chinese Research and Design Institute of Nuclear Power Engineering, Chengdu, China, Post Code: 610041), SUN Qi, YANG Rui-chang (Thermal Power Engineering Department, Tsinghua University, Beijing, China, Post Code: 100084) //Journal of Engineering for Thermal Energy &Power. — 2004, 19 (2). —124～126, 174

Void fraction is one of the basic parameters in steam-liquid two-phase flows. To date, the study of methods for calculating subcooled-boiling void fraction mainly focused on high-mass flow rates. Moreover, a huge amount of technical literature now available on void fraction models is hardly suitable for low flow-rate subcooled- boiling operating conditions. Based on a theoretical model of NVG (net vapor generation) point in low flow-rate subcooled boiling the authors have developed a fitting model for calculating the distribution of subcooled-boiling void fraction. Within a relatively wide range of pressures, mass flow rates, heat flux density and flow path dimensions the results of model calculations are compared with the at present available void-fraction experimental data. Under low flow-rate operating conditions the data obtained from the model are in fairly good agreement with those of experiments. This shows that the model is well suited for low flow-rate subcooled-boiling operating conditions. **Key words**: two-phase flow, void fraction, subcooled boiling, net vapor generation

不同煤种挥发氮析出过程的数值模拟与试验研究＝**Numerical Simulation and Experimental Study of the Separation-release Process of Volatile Nitrogen from Various Sorts of Coal** [刊,汉]/ ZHOU Hao, WENG An-xin, CEN Ke-fa, FAN Jian-ren (Education Ministry Key Laboratory of Clean Utilization of Coal and Environmental Engineering under the Zhejiang University, Hangzhou, China, Post Code: 310027) //Journal of Engineering for Thermal Energy &Power. — 2004, 19(2). —127～130

With the help of a numerical model a study was carried out regarding the following items. They include the separation-release of volatile nitrogen, the generation of intermediate nitrogen-containing product HCN and the process of the latter's transformation into NO. All the above took place in the course of pyrolysis and combustion of various types of coal, such as bituminous coal, lean coal and anthracite, etc. A finite volume method was utilized to perform a discrete solution for the equations of mass, chemical components, momentum and heat energy conservation. Through calculations a whole variety of data were obtained, such as the coal particle ignition time of various ranks of coal, pyrolysis process, porosity factor, HCN and NO formation rate, etc. In addition, the above data were also compared with the test results of a subsidence furnace followed by a pertinent analysis. **Key words**: pulverized coal, nitrogen fuel, numerical simulation

流化床密相区流动特性的数值模拟＝**Numerical Simulation of Flow Characteristics in the Dense-phase Zone of a Fluidized Bed** [刊,汉]/ XU Xiang, XIANG Wen-guo, QIN Chen-hu (Education Ministry Key Laboratory of Clean Coal Power Generation and Combustion Technology under the Southeastern University, Nanjing, China, Post Code: 210096) //Journal of Engineering for Thermal Energy &Power. — 2004, 19(2). —131～133

Gas-solid two-phase flows in a fluidized bed have always been a focused objective of experimental research and numerical simulation. Based on an Eulerian two-fluid model, the authors have set up a model for the gas-solid two-phase flow in a fluidized bed. By making use of FLUENT software a numerical simulation was conducted of the two-phase flow characteristics in the dense-phase zone of the fluidized bed, and the characteristic features of the generation, movement and explosion/cracking of in-bed gas bubbles. In the model a particle-phase is treated as continuous media. A mathematical model identical in form to the gas phase was established. By utilizing the dynamic theory of discrete media the concept of granular temperature was introduced to describe the viscous stress of a solid phase. Moreover, a gas-solid two-phase coupling was accomplished by the use of a gas-solid drag force. By way of simulations a transformation process was obtained of the generation, movement and explosion/cracking of gas bubbles, which has been found to be in full agreement with experimental results. By using different drag force models a simulation was performed of the fluidized bed dense two-phase flows. A comparison with Kuipers experimental results shows that the use of a Gidaspow drag force model can result in a more accurate description of the dense two-phase flows in the fluidized bed. **Key words**: Eulerian model, gas-solid two-phase flow, gas bubble, fluidized bed

环栅式动力除尘器的两相流数值模拟=**Numerical Simulation of Two-phase Flows of a Cascade-ring Type of Aerodynamic Dust Collector** [刊,汉] / LIN Feng, LIN Long (Harbin No. 703 Research Institute, Harbin, China, Post Code: 150036), HU Qi-di (Shenzhen Meishi Electric Power Industrial Co. Ltd., Shenzhen, China, Post Code: 518000), ZHANG Shi-lei (Thermal Power Plant of Jilin Petroleum Group Co. Ltd., Jilin, China, Post Code: 138000) //Journal of Engineering for Thermal Energy &Power. — 2004, 19(2). —134~136, 190

A numerical simulation of two-dimensional two-phase flows was conducted of a cascade-ring aerodynamic dust collector in an effort to enhance its performance through an improved design. During the simulation a particle trajectory model has been employed with a k-ε model being used to simulate gas-phase turbulent flows and a Stochas model used to describe the turbulent diffusion of a particle phase. The flow conditions of ash particles and gas, and the characteristics of the dust collector were studied and analyzed. **Key words**: two-phase flow, numerical simulation, dust removal, air purification

二阶全展开 ETG 有限元方法在方腔自然对流模拟中的应用=**The Application of a Second-order and Full-extension ETG Finite Element Method for the Simulation of Natural Convection in a Square Cavity** [刊,汉] / WEI Ying-jie (Astronautics Engineering and mechanics Department &Engineering under the Harbin Institute of Technology, Harbin, China, Post Code: 150090), HE Zhong-yi (School of Municipal &Environmental Engineering under the Harbin Institute of Technology, Harbin, China, Post Code: 150090) //Journal of Engineering for Thermal Energy &Power. — 2004, 19(2). —137~139

A second-order and full-extension ETG finite element method was employed to carry out a discrete solution for a N-S equation and an energy equation. With a square-cavity natural convection problem of zero initial value serving as an example a numerical simulation was conducted. The flow and temperature fields of natural convection in a square cavity at different Rayleigh numbers were calculated. The steady-state results being finally attained are in very good agreement with those of a standard numerical solution. Moreover, the time evolution process depicting the flow and temperature fields has been reflected quite well. Especially worth mentioning here is the capture of the change of vortex structure in the flow field before and after bifurcation. All the above shows that the second-order and full-extension ETG finite element method features a relatively fair stability and precision, and has its definite merits when used to evaluate the time evolution process of temperature and flow fields. **Key words**: second-order and full-extension ETG finite element, square cavity, natural convection

表面活性剂减阻流体湍流空间结构试验研究=**Experimental Research on the Turbulent Spatial Structure of a Drag Reducing Fluid with a Surfactant being Added** [刊,汉] / WANG De-zhong, HU You-qing, WANG Song-ping, ZHOU Rong-sheng (Institute of Mechanical and Power Engineering under the Shanghai Jiaotong University, Shanghai, China, Post Code: 200030) //Journal of Engineering for Thermal Energy &Power. — 2004, 19(2). —140~143

An experimental study was conducted of the turbulent flow field of a CTAC drag reducing fluid in a two-dimensional flow