文章编号:1001-2060(2003)02-0163-03

非均匀布风流化床的 DEM 模拟

尹　斌,章明川,宋玉宝,吴　江
(上海交通大学能源系,　上海　200240)

摘　要:对二维非均匀布风流化床内的颗粒运动进行了数值模拟,用欧拉方法处理气相场的同时用拉格朗日方法处理离散颗粒场,直接跟踪颗粒场中的每一个颗粒。模拟结果表明,非均匀布风流化床内存在颗粒的内循环运动,因此颗粒的混合特性优于均匀布风流化床。

关　键　词:非均匀布风;流化床;DEM

中图分类号:TK221　　　文献标识码:A

1　引　言

DEM(Discrete element method)模型是目前研究稠密气固两相流(特别是流态化领域)的热点。以前的研究者曾使用拟流体模型进行稠密气固两相流的研究,但该模型中的局域平衡假设及固相连续性假设从本质上消弱了气固两相流中空间梯度变化很大的整体不均匀性和局部不均匀性。DEM 模型由于能够模拟每个颗粒的运动变化,所以能够较真实的反映颗粒运动的实际情况。

目前 DEM 模型主要有两种形式:软球方法(Tsuji 等),硬球方法(Humans 等)。软球方法由 Tsuji 等人在 1993 年提出。软球方法假定颗粒的弹性系数较低,时间步长取得很小,通常小于 10^{-5}s,所以计算所需时间长,这主要是考虑了数值计算的收敛性和稳定性,但软球方法可以处理一个颗粒与多个颗粒的碰撞。硬球方法中,颗粒的弹性系数接近实际值,只考虑两个颗粒的碰撞,且假定碰撞后形状不变。该方法中着重考虑颗粒碰撞后的弹性恢复系数及颗粒的摩擦系数对数值模拟的影响[1~2]。

本文采用软球方法对二维非均匀布风流化床内的颗粒运动进行 DEM 模拟。

2　数学模型

在模型中,气体的运动规律用两相耦合的 Navier-Stokes 方程进行描述,而颗粒的运动则通过随机轨道模型描述,同时考虑由于颗粒碰撞所产生颗粒动量的改变。

2.1　颗粒模型

颗粒没有发生碰撞时,主要受到流体曳力及颗粒自身重力的作用,颗粒的运动如下式所示:

$$m\frac{\mathrm{d}v}{\mathrm{d}t}=F_g+mg \tag{1}$$

其中流体对颗粒的作用力 $F_g=\frac{1}{8}\pi d_p^2 C'_d\ \rho_g \varepsilon^2\ |u-v|(u-v)$,式中 u 为流体速度,v 为颗粒速度,d_p 为颗粒直径,有效曳力系数与单颗粒曳力系数及空隙率有关[3],即:

$$C'_d=C_d\varepsilon^{-4.7}$$

单颗粒曳力系数如下式:

$$C_d=\begin{cases}\dfrac{24}{Re_p}(1+0.15(Re_p)^{0.687}) & Re_p<1\,000\\ 0.44 & Re_p\geq 1\,000\end{cases}$$

颗粒雷诺数 $Re_p=\dfrac{\varepsilon\rho_g\ |u-v|\ d_p}{\mu_g}$

通过式(1)可以计算出颗粒的加速度,然后可以计算出颗粒的速度和位置。

根据基本物理定律,两个球形颗粒发生弹性碰撞时,首先在接触点处发生弹性变形,颗粒在前进方向受到阻力,该阻力的大小与法向变形位移、颗粒硬度等有关,达到最大变形位移时,颗粒停止运动,沿原来运动的方向反弹。对非完全弹性碰撞,碰撞后颗粒的动能发生损失,损失的大小与阻尼系数及颗粒的法向相对速度有关。当两个颗粒发生偏心碰撞,由于受到切向力的作用,出现切向转矩,从而使颗粒产生旋转。上述物理过程可以通过如下数学模型描述[4]:

$$\vec{F}_{cnij}=(-k\delta_n-\eta\vec{V}_{ij}\vec{n}_{ij})\vec{n}_{ij} \tag{2}$$

$$\vec{F}_{ctij}=(-k\delta_t-\eta\vec{V}_{ij}\vec{t}_{ij})\vec{t}_{ij}\quad(|\vec{F}_{ctij}|<\mu_f\ |\vec{F}_{ctij}|) \tag{3.1}$$

收稿日期:2002-05-14;　修订日期:2002-09-20
作者简介:尹　斌(1975-),男,四川武胜人,上海交通大学博士研究生.

$$\vec{F}_{ctij} = -\mu_f |\vec{F}_{cn}| \vec{t}_{ij} \quad (|\vec{F}_{ct}| \geqslant \mu_f |\vec{F}_{cn}|) \tag{3.2}$$

$$\vec{V}_{tij} = \vec{V}_{ij} - (\vec{V}_{ij}\vec{n}_{ij})\vec{n}_{ij} + r(\vec{\tilde{\omega}}_i + \vec{\tilde{\omega}}_j)\vec{n}_{ij}$$

$$I\frac{d\tilde{\omega}}{dt} = \Sigma T, I = \frac{2}{5}mr^2$$

$$\vec{t}_{ij} = \frac{\vec{V}_{tij}}{\vec{V}_{|tij|}}$$

$$\vec{n}_{ij} = \frac{X_j - X_i}{\sqrt{(X_j - X_i)^2 + (Y_j - Y_i)^2}} i + \frac{Y_j - Y_i}{\sqrt{(X_j - X_i)^2 + (Y_j - Y_i)^2}}$$

式中：$\vec{F}_{cnij}$ —法向力，$\vec{F}_{ctij}$ 切向力，k —颗粒硬度，η —阻尼系数，δ_n —法向变形位移，δ_t —切向变形位移，$\vec{n}_{ij}$ —法向单位向量，$\vec{t}_{ij}$ —切向单位向量，μ_f —摩擦系数；$\vec{V}_{ij}$ —颗粒 i 与 j 的相对速度；$\vec{V}_{tij}$ —切向相对速度，$\tilde{\omega}$—颗粒的旋转角速度；T—转矩；I—转动惯量，(X_i, Y_i)和(X_j, Y_j)—颗粒 i、j 的位置坐标，r—颗粒半径。

通过式(2)和式(3)求出颗粒碰撞所产生的法向力和切向力后，根据牛顿第二定律可以计算出碰撞过程中所产生的加速度，从而确定颗粒碰撞后的速度，即：

$$V = V^0 + a \times \Delta t \tag{4}$$

其中：V^0 为初速度，a 为加速度，Δt 为时间步长。

2.2 气相模型

气体的流动规律用气固两相耦合的 $N-S$ 方程描述。

连续性方程为：

$$\frac{\partial(\varepsilon\rho_g)}{\partial t} + \Delta(\varepsilon\rho_g \vec{u}_g) = 0 \tag{5}$$

动量方程为：

$$\frac{\partial(\rho_g \varepsilon \vec{u}_g)}{\partial t} + \Delta(\rho_g \varepsilon \vec{u}_g \vec{u}_g) = -\varepsilon\Delta p - \beta(\vec{u}_g - \vec{V}) + \Delta(\varepsilon\tau) - \rho_g \varepsilon g \tag{6}$$

其中：ε 为空隙率，β 为相间动量交换系数，τ 为粘性应力。相间动量交换系数可通过相应的经验公式确定，即：

$$\beta = \begin{cases} 150\dfrac{(1-\varepsilon)^2}{\varepsilon}\dfrac{\mu_g}{d_p^2} + 1.75(1-\varepsilon)\dfrac{\rho_g}{d_p}|u_g - V|, & \varepsilon < 0.8 \\ \dfrac{3}{4}C_d\dfrac{\varepsilon(1-\varepsilon)}{d_p}\rho_g |u_g - V| \varepsilon^{-2.65}, & \varepsilon \geq 0.8 \end{cases}$$

2.3 计算方法

(1) 首先采用 SIMPLEC 方法在给定的时间步长内数值求解式(5)和式(6)，确定气相流场；(2) 根据式(1)，采用四阶变步长 $R-K$ 方法计算全部颗粒经过一个时间步长后的位置和速度；(3) 判断颗粒是否发生碰撞。若发生碰撞，则用式(2)～式(4)确定颗粒碰撞后的速度；(4) 统计每个流场网格内的空隙率，并计算网格内颗粒平均速度，然后计算颗粒对流场的反作用源项；(5) 修正两相耦合的 $N-S$ 方法，得出新的流场；(6) 重复 2～5 步骤。

3 非均匀布风流化床的 DEM 模拟

3.1 模拟对象

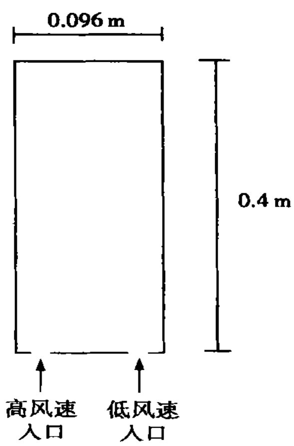

图 1 流化床的结构

数值模拟时采用的流化床结构如图 1 所示。床宽为 0.096 m，床高为 0.4 m。流化床底部布置两个进口，宽度均为 0.012 m。其中高风速进口距左界面为 0.018 m，低风速进口距右界面也为 0.018 m。

构造初场时，颗粒首先随机分布在床内，然后在不送风的状态下用 DEM 模拟颗粒的自由下沉过程。经过一段时间后，颗粒在床内自然堆积达到平衡，此时的颗粒位置即为颗粒初场，见图 2 中 0.0 s 时的情形。静止床层高度为 0.168 m。

3.2 模拟参数

表 1 进行模拟时使用的参数

固 相		气 相
颗粒直径 4mm	颗粒密度 2 500 kg/m	气体密度 1.205 kg/m^3
颗粒数目：1 000	颗粒硬度 8 000 N/m	动力粘度：1.82×10^{-5} Pa·s
阻尼系数：0.17	弹性恢复系数：0.9	流场网格数：10×22
摩擦系数：0.3	时间步长 10^{-5} s	网格大小：0.012 m×0.02 m

3.3 模拟结果

模拟时，高低风速进口的表观流化速度分别为 7 m/s、2.5 m/s，模拟结果见图 2。

从图 2 中可以看出，左边高风速区很明显产生了气泡，并且气泡沿床层上升过程中不断长大，当气泡运动到床层界面时破裂，将尾涡中的颗粒抛向自由空间。

由于高风速区向上运动的颗粒流率高，因此高

风速区上端的颗粒会向低风速区扩散, 而低风速区的颗粒向下运动过程中又会向高风速区扩散, 这样就形成了床料颗粒的内循环。图 3 为不同时刻的颗粒矢量, 可以看到颗粒存在大尺度的内循环运动。

由于非均匀布风流化床存在颗粒的内循环运动, 因此床料的横向混合特性较好, 而且通过调节高风速区和低风速区的流化速度, 可以控制内循环的强度, 所以适应性较好。

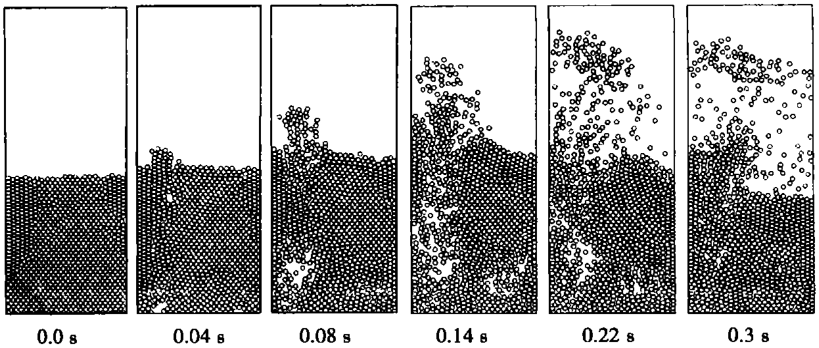

图 2　非均匀 布风流化床的 DEM 模拟

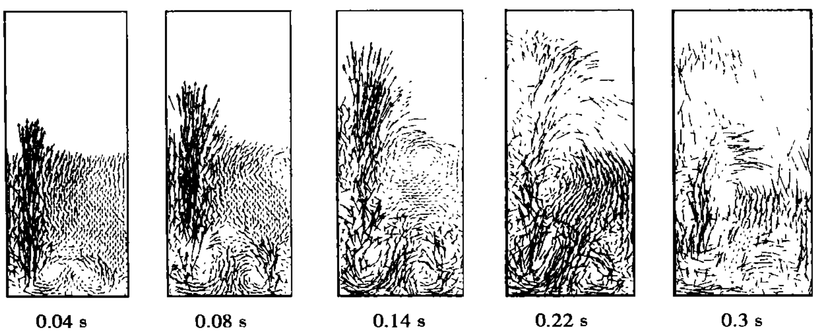

图 3　颗粒矢量

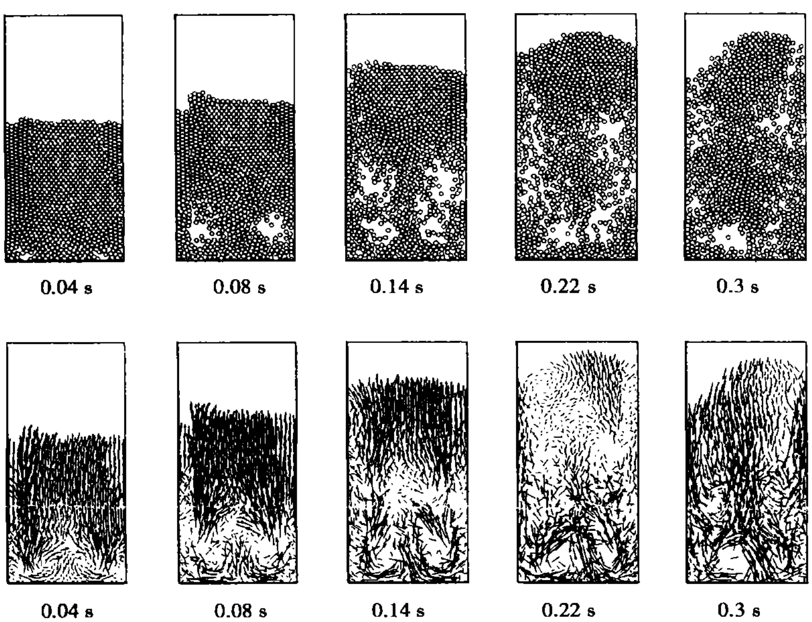

图 4　均匀布风流化床的 DEM 模拟

3.4　与均匀布风流化床的比较

在同样条件下对均匀布风流化床进行数值模拟的结果如图 4 所示。从图中可以看出, 均匀布风流化床只会出现局部的颗粒循环, 而不会象非均匀布风流化床那样存在宏观尺度的横向颗粒循环。因此对于非均匀布风流化床来说, 颗粒的循环流动将对床内的传质、传热、燃烧及化学反应过程产生重要影响。

4　结　论

非均匀布风流化床的 DEM 模拟表明, DEM 模型能够较好的模拟稠密气—固两相流的运动特性, 它的发展十分有潜力。

非均匀布风流化床存在颗粒的内循环运动。通过与均匀布风流化床的模拟结果相比较可以看出, 非均匀布风流化床中颗粒的横向扩散特性较好。

参考文献:

[1]　HOOMANS B P B, KUIPERS J A M. Discrete particle simulation of bubble and slug formation in a two-dimensional gas-fluidized bed : a hard sphere approach[J]. **Chemical Engineering Science**, 1995, **51**(1): 99—118.

[2]　TSUJI Y, KAWAGUCHI T, TANAKA T. Discrete particle simulation of two dimensional fluidized[J]. **Powder Technol**, 1993, **77**(1)79—87.

[3]　TSUJI Y, TANAKA T, YONEMURA S. Cluster patterns in circulating fluidized beds predicted by numerical simulation (discrete particle model versus two-fluid model)[J]. **Powder Technol**, 1998, **95**: 254—264.

[4]　XU B H, YU A B. Numerical simulation of the gas-solid flow in a fluidized bed by combining discrete particle method with computational fluid dynamics[J]. **Chemical Engineering Science**, 1997, **52**(16): 2785—2809.

(何静芳　编辑)

By using a IFA300 constant-temperature hot wire anemometer a six-sensor vortex probe was localized for the first time with the help of a three-dimensional movable coordinate stand. Meanwhile, a servomotor was used to drive its fixed base. The vorticity field at the tail section of the multi-platenlets in the foreign-made tangentially fired pulverized-coal furnace was measured by the use of a condition-sampling technique. With the multi-platenlets being assumed as thin-plate airfoils investigated were the separated vortices resulting from the combined action of spiral ascending gas flows and the flows past the thin-plate airfoils. The magnitude of the separated vortices was determined. The results of an experimental study indicate that there exist evident separated vortices at the tail section of a separated platen (front platen) and at the right half of the furnace. Furthermore, some of the separated vortices have been accompanied by other vortices assuming an opposite direction. The measurement results of velocity field and vortex field show that in the neighborhood of a right lateral wall there emerged not only a high flow speed but also a high degree of vorticity. All the above has provided a theoretical basis for the study of in-boiler pulverized-coal combustion and NOx emission control technology. **Key words**: tangentially fired furnace, multi-platenlets, spiral ascending flow, thin-plate airfoil, separated vortex, six-sensor vortex probe

六角切圆燃烧锅炉燃烧器区域煤粉粒径分布的试验研究= **Experimental Investigation of Pulverized-coal Particle Diameter Distribution at the Burner Zone of a Hexagonal-tangentially Fired Boiler** [刊, 汉] / LI Rui-yang, ZHU Qun-yi, ZHAO Yu-xiao, QIN Yu-kun (School of Energy Science and Engineering under the Harbin Institute of Technology, Harbin, China, Post Code: 150001) //Journal of Engineering for Thermal Energy & Power. — 2003, 18 (2). 155—158

A cold-state model test was conducted of a 670 t/h tangentially fired pulverized-coal boiler with burners being arranged at the six corners of a furnace. The distribution of pulverized-coal particle diameter and velocity at the outlet zone of burners was studied, using a phase Doppler anemometer to make relevant measurements. Moreover, the impact of the above distribution on the slag formation on heating surfaces was analyzed. By way of comparison one can also perceive a significant alleviation of slag formation following the introduction of a larger imaginary tangential-circle diameter of secondary air fed to the top-layer burners. On the basis of the above a series of effective measures were taken to solve the boiler slagging problem. **Key words**: tangential firing with burners located at six corners, phase Doppler anemometer, particle diameter distribution, slag formation

电厂直接空冷系统风效应风洞模拟实验研究= **Experimental Investigation of the Wind Tunnel Simulation of Wind Effects on a Directly Air-cooled System for a Power Plant** [刊, 汉] / GU Zhi-fu, ZHANG Wen-hong, LI Hui (Department of Mechanics and Engineering Science, Beijing University, Beijing, China, Post Code: 100871), PENG Ji-ye (Shanxi Provincial Electric Power Exploration and Design Institute, Taiyuan, Shanxi Province, China, Post Code: 030001) //Journal of Engineering for Thermal Energy & Power. — 2003, 18 (2). 159—162

The similarity criteria to be met during the wind tunnel simulation-based experimental study of the wind effects on a directly air-cooled system of a power plant are probed along with a description of the relevant experimental methods. The conception of recirculation ratio is proposed to describe the wind effect on the efficiency of an air-cooled condenser. Furthermore, by citing the wind tunnel simulation-test results of a specific case the authors explained the impact of outside wind speed and direction on the efficiency of a specifically configured air-cooled system of a power plant. In this connection it is of vital importance to conduct a pertinent wind tunnel simulation in the light of local meteorological data during the preliminary design of the directly air-cooled system of a power plant. **Key words**: power plant, air cooled system, condenser efficiency, wind tunnel simulation

非均匀布风流化床的 DEM 模拟= **Discrete Element Method-based Simulation of a Fluidized Bed with Non-uniformly Distributed Air** [刊, 汉] / YIN Bin, ZHANG Ming-chuan, SONG Yu-bao, et al (Department of Energy Engi-

neering, Shanghai Jiaotong University, Shanghai, China, Post Code: 200240)//Journal of Engineering for Thermal Energy &Power. — 2003, 18(2). 163—165

A numerical simulation was conducted of the particle movement in a fluidized bed with two-dimensional non-uniformly distributed air. In addition to dealing with a gas-phase field by the use of Euler method a Lagrangian method is employed to treat a discrete particle field, directly keeping track of each particle in the particle field. The results of simulation indicate that there exists in the fluidized bed with non-uniformly distributed air an inner circulation movement of the particles. Hence, the mixing characteristics of the particles in the fluidized bed under discussion are superior to those in a conventional fluidized bed with uniformly distributed air. **Key words**: non-uniformly distributed air, fluidized bed, discrete element method

地源热泵竖直埋管的有限长线热源模型＝**A Model of Finite-length Linear Heat-source for the Vertical Embedded Pipe of a Ground-source Heat Pump** [刊, 汉] /ZENG He-yi, DIAO Nai-ren, FANG Zhao-hong (Research Institute of Ground-source Heat Pumps under the Shangdong Institute of Architectural Engineering, Jinan, China, Post Code: 250014)//Journal of Engineering for Thermal Energy &Power. — 2003, 18(2). 166—169

Analyses and discussions were conducted of a non-steady heat transfer model for the vertical embedded pipe of a geothermal heat exchanger. With the use of a virtual heat source and Green function method obtained is an analytical solution expression for the non-steady temperature field generated by a finite-length linear heat source in semi-infinite large media. By way of comparison with a steady-state temperature field solution discussed is the time required for the temperature field to attain a nominal "steady state". Meanwhile, an analysis is performed of the temperature field when it has reached a steady state. In this connection a mistake that appeared in current textbooks was indicated. Two representative steady-state borehole wall temperatures, i.e., the temperature at the middle of the borehole and the integral mean temperature along the borehole, are defined. A comparison of the difference between these two temperatures has led to a simplified calculation formula, suitable for engineering applications. On the basis of the above analyses discussed further is the impact of the annual imbalance between heating and cooling loads of geothermal heat exchangers on their long-term performance. **Key words**: ground-source heat pump, geothermal heat exchanger, heat conduction, heat transfer model

大型锅炉长期动态特性研究中的烟气计算模型＝**Flue-gas Calculation Model Used in the Study of Long-term Dynamic Characteristics of a Large-sized Boiler** [刊, 汉] /LI Yun-ze, YANG Xian-yong (Department of Thermal Energy Engineering, Tsinghua University, Beijing, China, Post Code: 100084)//Journal of Engineering for Thermal Energy &Power. — 2003, 18(2). 170—172

The study of long-term dynamic characteristics of a large-sized turbogenerator set requires a simplified, accurate and rapid calculation of flue-gas heat release rate. To eliminate the main deficiency of currently used flue-gas heat release simulation models the authors have deduced from Dybosky-Broch formula of large-sized boiler thermodynamic calculation a new flue-gas heat release model for the simulation and analysis of dynamic characteristics. Moreover, through a concise analysis, simplification and deduction obtained was a method for calculating flue-gas outlet temperatures and heat release rates in a dynamic process for various boiler heat exchange surfaces. The recommended easy-to-use flue-gas calculation model offers a relatively accurate and simple expression of the variation mechanism of flue gas temperature and heat release rate of high-temperature gases in the furnace and various gas-pass heat exchange surfaces of a large-sized boiler during a dynamic process. The flue-gas calculation model has been used to simulate the dynamic characteristics of a 600MW supercritical power generation unit with rational and accurate simulation results being obtained. **Key words**: large-sized boiler, long-term dynamic characteristics, simulation, flue-gas calculation model

膜式水冷壁壁温影响因素的数值分析＝**Numerical Analysis of Factors Having an Impact on the Wall Tempera-**