IOP Conf. Series: Earth and Environmental Science **189** (2018) 052025 doi:10.1088/1755-1315/189/5/052025

Numerical simulation of solid - liquid two - phase flow deposition on the surface of tube bundle

Zheng yangyan[1], ye youjun[1], xie chuan yang[2,*], zheng kai[1] and yu yanping[1]

[1] Special equipment safety supervision inspection institute of Jiangsu province

[2] School of energy science and engineering, Nanjing Tech University, Nanjing, Jiangsu

[*] Corresponding author: victory0719@163.com

Abstract. Based on finite volume method, the Reynolds number of fluid affecting solid-liquid two phase deposition on the outer surface of tubes has been predicted. A 3D geometric model of fluid flow cross tube bundles is established. The particle trajectory was tracked by Euler - Lagrange model. The results show that: different Reynolds number of fluid impact dynamic pressure which is consistent with the distribution of fouling deposition. That is to say, dynamic pressure can be treated as indicator when monitor the fouling deposition on the surface of tube bundle. The deposition rate of particles is positively correlated with dynamic pressure. In addition, based on this geometry model, with the increment of Reynolds number, average accretion rate tend to stabilize to， rather than increase indefinitely. The location of the most serious accretion rate is always on the joint between the pipe and the shell, especially, on the surface of windward.

1. Introduction

It is reported that 90% of heat exchangers occur fouling deposition with different degree [1]. The fouling of boiler or heat exchanger is a substance that is attached to the heating surface in solid form. According to the main chemical composition, scaling is divided into calcium and magnesium scale, silicate scale, iron oxide scale, copper scale and phosphate scale. In this study, calcium carbonate is selected to be main scaling. The calcium carbonate scale deposited on the equipment in the form of dense crystal. The fouling accumulates on the heat transfer surface. Comparing to metal, owing to low thermal conductivity, fouling reduces the heat transfer coefficient severely. The selection of fouling characteristics often depends on the experience of field operation which is inaccurate to reflect the real situation of scaling in the heat exchanger. Tubular heat exchanger is a kind of general heat exchanger which consists of shell and tube. When the fluid is swept across the tube bundles, the solid particles will have a serious deposition effect on the surface of the upstream tube bundle.

Kern [2] developed the earliest deposition model, mass balance model, where the change of fouling thermal resistance over time is the scaling model of the difference between deposition rate and decomposition rate. Wood [3] reviewed the theoretical and experimental research on particles deposition in turbulent flow. A simple analytical method is presented for calculation of deposition to both smooth and rough surfaces. Mahmood Izadi et al. [4] adopted RSM to simulate particle deposition of calcium carbonate inter water in a horizontal tube domain. They developed monitoring system for fouling phenomenon which can be used to monitor fouling in a wide range of tubular heat exchangers. Xu[5] et al. constructed a mathematical model in view of the $CaCO_3$ fouling process in the inner tube. Guo [6] developed a two-phase flow mathematical model of particle dynamics coupled with turbulent kinetic energy.

In this work, based on numerical simulation method, compare fouling accretion rate with Reynolds number of fluid. Then compare different boundary condition effect fouling accretion rate on the outer surface of tube bundle.

2. Numerical simulation

2.1. Governing equations

In this study, CFD modelling is done with commercial software ANSYS FLUNET 15.0.The governing equations for conservation of mass and momentum are solved with control volume based discretization method along with a pressure-based solver algorithm. The motion state of impressible fluid is solved by navier-stokes equations:

$$\frac{\partial \rho}{\partial t} + \nabla\left(\rho \vec{u}\right) = 0 \quad (1)$$

In the continuity equation, ρ is the density of fluid which is considered as a constant, t and u represent time and velocity respectively. In the equation of motion, p is pressure and g is gravitational acceleration, F is external body force.

$$\frac{\partial}{\partial t}\left(\rho \vec{u}\right) + \nabla \cdot \left(\rho \vec{u}\vec{u}\right) = -\nabla P + \nabla \cdot \left(\overline{\overline{\tau}}\right) + \rho \vec{g} + \vec{F} \quad (2)$$

In this work, standard k-ε turbulence model is used, which allows determination of both the turbulent length scale and the time scale by solving two separate transport equations, one for the dissipation kinetic energy, and one for the dissipation rate[7].Turbulent viscosity is calculate by combining k-ε according to：

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + Gk + Gb - \rho\varepsilon - Y_M + S_k \quad (3)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k + C_{3\varepsilon}G_b\right) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (4)$$

2.2. Geometric model and boundary condition

The schematic of computational domain is shown below figure1. The specific size is shown in table 2.

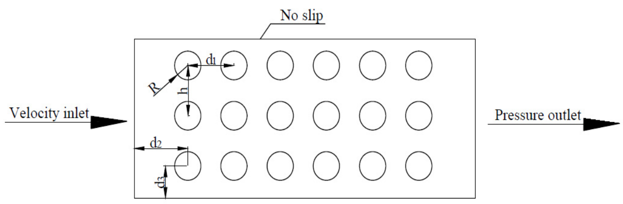

Fig. 1. Front view of geometry model

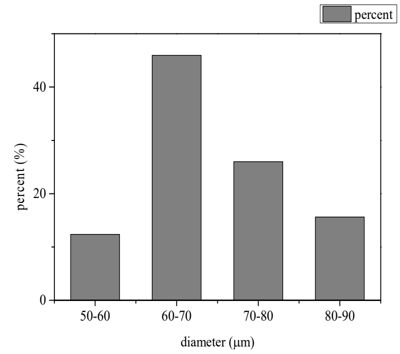

Fig. 2. Diameter distribution of particle.

Table 1.the size of geometry model

Num.	Parameter/symbol	unit	value
1	Diameter of tube/R	mm	20
2	transverse space/d_1	mm	30
4	vertical spacing/h	mm	30
5	Distance/d_2	mm	35
6	Distance/d_3	mm	25

Table 2.Boundary condition:
The inlet velocity of the particle is consistent with the fluid and distributed uniformly.

	Type	value
Inlet	Velocity inlet	2.5~5m/s
Outlet	Pressure outlet	
Shell wall	No slip	
tube	No slip	

The materials and properties of continuity and discrete phase show in table3. The diameter distribution of particles obey rosin-rammler model shown as figure2. The maximum diameter is $200\mu m$, the minimum diameter is $60\mu m$ and the mean diameter is $131\mu m$.

Table3. Materials and properties of medium

Parameters	Continuity phase	Discrete phase
Material	water (liquid)	Calcium carbonate
Density	998.2kg/m^3	2800kg/m^3
Viscosity	0.001003Pa·s	-
Mass rate	-	200mg/l

In this simulation, Euler-Lagrange method is adopted to consider fluid as continuous phase and particles as discrete phase. This method is suitable for dilute discrete phase occupying less volume (≤10%). Owing to the complexity of two phase flow, some assumptions need to be raised:

i. The particles are sphere;

ii. Ignoring the effect of pipe diameter changes;

iii. Ignoring the interaction between particles.

Model predicts the trajectory of a discrete phase particle (or droplet or bubble) by integrating the force balance on the particle, which is written in a Lagrange reference frame [8]. This force balance equates the particle inertia with the forces acting on the particle

The dispersion of particles due to turbulence in the fluid phase can be predicted using the stochastic tracking model. The stochastic tracking (random walk) model includes the instantaneous turbulent velocity fluctuations on the particle trajectories through the use of stochastic methods [9].

When the flow is turbulent, model will predict the trajectories of particles using the mean fluid phase velocity, $\bar{u}$, in the trajectory equations.

2.3 Equation of motion for particles

Euler- Lagrange model predicts the trajectory of a discrete phase particle by integrating the force balance on the particle, which is written in a Lagrange reference frame. This force balance equates the particle inertia with the forces acting on the particle, and can be written as:

$$\frac{d\vec{u}_P}{dt} = F_D(\vec{u} - \vec{u}_{\rm p}) + \frac{\vec{g}(\rho_P - \rho)}{\rho_P} \tag{5}$$

where $F_D(\vec{u} - \vec{u}_{\rm p})$ is the drag force per unit particle mass and

$$F_D = \frac{18\mu}{\rho_P d_p^2}\frac{C_D \mathrm{Re}}{24} \tag{6}$$

Here, $\vec{u}$ is the fluid phase velocity, $\vec{u}_P$ is the particle velocity, μ is the molecular viscosity of the fluid, ρ is the fluid density, ρ_P is the density of the particle, and d_P is the particle diameter.

2.4Turbulent dispersion of particles

The dispersion of particles due to turbulence in the fluid phase can be predicted using the stochastic tracking model [10]. The stochastic tracking (random walk) model includes the effect of instantaneous turbulent velocity fluctuations on the particle trajectories through the use of stochastic methods.

2.5Laws for drag coefficients

Spherical drag law is used to depict interaction between fluid and particles

$$C_D = a_1 + \frac{a_2}{\mathrm{Re}} + \frac{a_3}{\mathrm{Re}^2} \tag{7}$$

where a_1, a_2 and a_3 are constants that apply over several ranges of Re, which are equal to 0.5191, -1662.5 and 5.1467×10^6 respectively in this simulation.

In this simulation, the particle deposition rate adopts the Eq5. the expression of particle relaxation time is shown as Eq6.

$$u_d^+ = \frac{N_d / t_d}{N_0 / y_0} \tag{5}$$

$$\tau_p^+ = \frac{(\rho \mathrm{p} - \rho)\cdot d_p^2 \cdot u^2}{18\cdot \rho \cdot v^2} \tag{6}$$

3.Results and discuss

This simulation on kinetic theory of granular simplifies solid-liquid two phase flow reasonably. The particle deposition mathematical model can predict the surface deposition rate accurately.

3.1Reliability of the numerical simulation

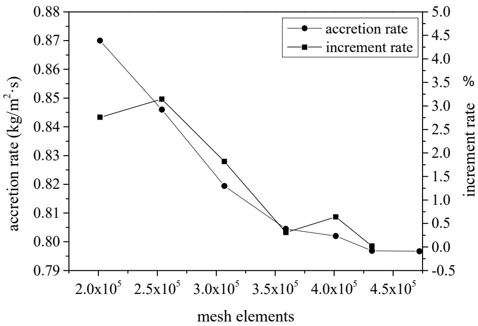

Fig. 3. Grid independence test result

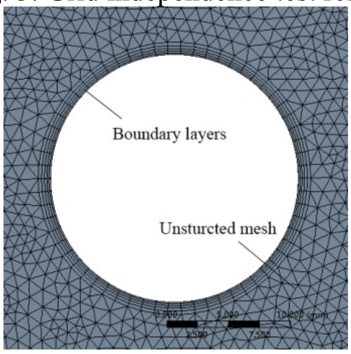

Fig. 4. Grid structure

In the present investigation, unstructured mesh of 5 boundary layers is applied to calculate accurately. From the picture, the conclusion can be drawn that an unstructured grid with 432,045 is sufficient and accurate. In addition, the result is that deviation between simulation and kern's mass balance model is within 15%. For immature and complicated two phase flow model, this deviation is acceptable.

3.2. Accretion rate and dynamic pressure

The study predicts the distribution of facet average accretion rate and dynamic pressure on the outer surface of tubes. The results show that, dynamic pressure of the surface of tubes is related closely to particle deposition. That is to say, dynamic pressure is a parameter to reflect location of fouling deposition.

The linear regression equation is obtained by regression analysis of deposition rate and dynamic pressure：

$$Y=\left(D-5789.78\right)\div 8288.04 \qquad (7)$$

Where, Y is accretion rate, $kg/m^2 \cdot s$;

D is dynamic pressure, Pa.

In working condition, dynamic pressure is easier and low cost to be supervised than fouling deposition.

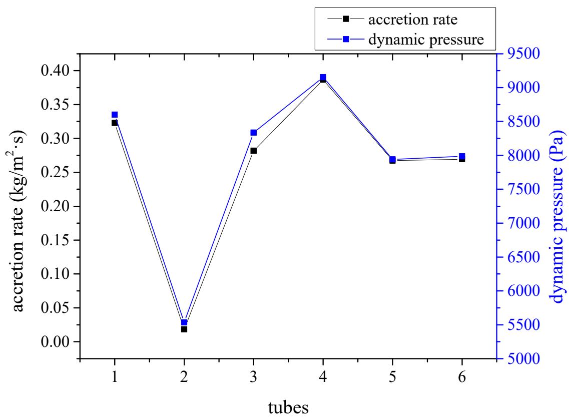

Fig. 5. Facet average accretion rate and dynamic pressure on every row of tube bundle

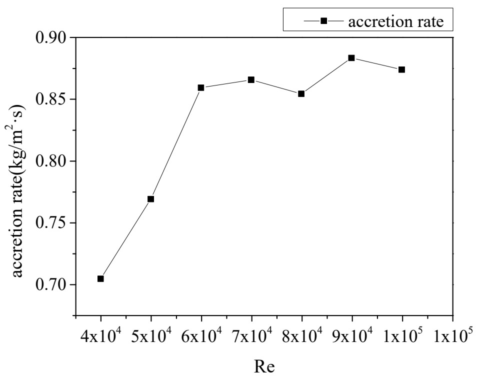

Fig. 6.Facet average accretion rate with different Re

The accretion rate doesn't increase with the increment of fluid Reynolds number. In this geometry model, Re within the range of 39928-59892, the accretion rate increase at linearly. The facet average accretion rate tends to be stable to around, however, when the Re exceeds 598892. The aligned arrangement of tube bundles may be a reason for steady facet average accretion on the back row, which is in line with reality that the accretion rate of back tube bundles stabilizes.

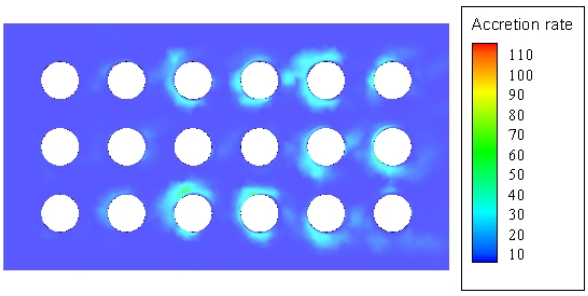

Fig. 7. Top view of geometry model

The figure7 shows that the prediction of sedimentary condition on the joint between with tube bundles and shell. In this place, the flow forms a vortex that promotes particle deposition, where the maxium accretion rate access 110kg/m^2·s. In addition, the first two rows and the middle tubes occurs much less deposiment than the later and tubes near the wall.

IOP Conf. Series: Earth and Environmental Science **189** (2018) 052025 doi:10.1088/1755-1315/189/5/052025

Reference:

[1] Hongxia Li, Guanqiu Li, Wei Li. *Analysis of in tubes particulate fouling characteriistic.* J ZJ university **46**,7(2012)
[2] DQ Kern，RE Seaton. *A theoretical analysis of thermal surface fouling.*(1959)
[3] Wood N B. *A simple method for the calculation of turbulent deposition to smooth and rough surfaces.* J Aerosol Science**12**,16(1981)
[4] Izadi M. *Experimental and numerical investigation of fouling in heat exchangers* .Dissertations & Theses - Gradworks(2011)
[5] Xu Z, Wen X, Sun Y, et al. *Forecasting fouling characteristics of plain tube based on least squares-support vector machine for regression*. J Chemical Industry & Engineering Society of China (2009)
[6] Guo Yin-cheng. *Modeling of gas/particles flows using a comprehensive model of kinetic theory and turbulent kinetic energy.* Computers and applied chemistry(2003)
[7] Lin T, Ahmadi G. *Particle deposition in turbulent duct flows—comparisons of different model predictions.* Journal of Aerosol Science**38**,21(2007)
[8] Kulick J D, Fessler J R, Eaton J K. *Particle response and turbulence modification in fully developed channel flow*. Journal of Fluid Mechanics,**277**,26(1994)
[9] Kuan B, Yang W, Schwarz M P. *Dilute gas–solid two-phase flows in a curved 90°duct bend: CFD simulation with experimental validation.* Chemical Engineering Science**62,**21(2007)
[10] Elsharkawy A. *Predicting the Properties of Sour Gases and Condensates: Equations of State and Empirical Correlations*(2002)