Numerical simulation of fouling deposition in boundary layer of corrugated tube

Y Y Zheng¹, Y J Ye¹, C Y Xie²,³, K Zheng¹ and Y P Yu¹

¹ Special equipment safety supervision inspection institute of Jiangsu province
² School of energy science and engineering, Nanjing Tech University, Nanjing, Jiangsu
³ E-mail: xcy0719@126.com

Abstract. A compared study on deposition condition in corrugated tube predicates that flow parameters (inlet velocity and mass flow rate of particles) influence fouling deposition with numerical simulation method. The particulate fouling of calcium carbonate/water is selected as medium. The results show that: in the boundary layer of the corrugated tube, as radial velocity changes rapidly, the particles effected by dominate drag force descend to accretion. The effect of inlet velocity and concentration of particles are decisive. As the inlet velocity increases, the accretion rate occur negative changes. In contrast, the effect of concentration is positive.

1. Introduction

Fouling deposition on the surface of tube in heat exchanger has been the main obstacle to improve heat transfer coefficient. The thermal conductivity of fouling is approximately 0.18-2.56 W/m·K, while thermal conductivity of metal is 108-180 W/m·K. In contrast, the thermal conductivity of metal is 70-1000 times than fouling, that is to say, fouling resistance is 70-1000 times than the metal [1]. The mechanism of fouling deposition is complicated. Due to the effect of viscous force, the fluid can form an extremely thin boundary layer on the surface of tube. The trajectory of particles flowing into boundary layer is complex with a large velocity gradient occur in the radial direction. The particles with low velocity will occur deposition phenomenon when contact the rough surface.

Owing to the advantage of reducing deposition and strengthening efficiency of heat exchange, the corrugated pipe is attracting attention and widely applied in industry. Based on the traditional tube and shell heat exchanger, the corrugated pipe is processed with the smooth pipe. Kallio [2] described Lagrange-based, stochastic model which provides a detailed and realistic simulation of particle deposition in turbulent duct flows. Zhang [3] simulate flow and particle motion first on a single sphere, then on different sets of linear arrays of 8 spheres that have a range of inter-sphere spacing.

Rouhiainen and Stachiewiz [4] proposed that deposition models based on Stokes stopping distance need high radial velocities.

The theoretical approaches of studying two phase flow in numerical simulation include Euler method and Lagrangian method. The Lagrangian method is applied in this study which predicts particle trajectories preferably when the discrete phase is coupled with the continuous phase [5]. This method treats solid particles as discrete phase and considers interaction between turbulent fluid and solid particle which asks volume concentration of discrete phase is less than 10% [6].
2. Numerical simulation analysis

Numerical simulation method has become an effective approach to study two phase flow. In this study, the Euler-Lagrangian method is used to simulate fluid and track the particle [7]. The liquid is treated as a continuous phase and described by the Navier–Stokes equations, while particles are treated as a discrete phase and mathematically described by Newton's second law.

2.1. Mathematical model

The Navier-Stokes equations are employed for incompressible fluid. The continuity equation is shown as in a three dimensional model:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0
\]  

(1)

and momentum equation is shown as:

\[
\frac{\partial}{\partial t} \left( \rho \vec{v} \right) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla P + \nabla \cdot (\tau) + \rho g + \vec{F} + \vec{S}_M
\]  

(2)

where \( \rho \) is the fluid density; \( \vec{v} \) is the instantaneous velocity vector; \( P \) is the pressure; \( g \) is the acceleration of gravity; \( \vec{S}_M \) is the source terms which can be ignored.

The stress tensor \( \tau \) is given as:

\[
\tau = \mu \left[ \nabla \vec{u} + \nabla \vec{u}^T \right] - \frac{2}{3} \nabla \cdot \vec{u} I
\]  

(3)

where \( \mu \) is fluid viscosity and \( I \) is the unit tensor.

The Euler-Lagrange approach is applied to study solid-fluid two phase. The dispersed phase is solved by tracking a large number of particles [8]. The discrete phase is coupled with continuous phase. This approach is made considerably simpler when particle-particle interactions can be neglected, and this requires that the dispersed second phase occupies a low volume fraction.

For turbulent flow of fluid, the RNG \( k-\varepsilon \) model is employed in this study [9]. Some researchers have proved that the RNG \( k-\varepsilon \) model is more accurate to simulate two phase flow in tube which has a good agreement with experimental data.

2.2. Particle motion model

The force acting on solid particles is complicated and diverse, such as thermophoretic force, Brownian force, Saffman’s lift force, gravity force and drag force, etc [10]. Among of those, drag and gravity force play the leading role. Due to diameter of particles is greater than 1 \( \mu m \), the Brownian force is ignored. The temperature of fluid is kept at 303.15K, the thermophoretic force is also ignored.

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

\[
\frac{d\vec{u}_p}{dt} = F_D(\vec{u} - \vec{u}_p) + \frac{g(\rho_p - \rho)}{\rho_p} + \vec{G}
\]  

(4)

where \( F_D(\vec{u} - \vec{u}_p) \) is the drag force per unit particle mass and

\[
F_D = \frac{18 \mu}{\rho_p d_p^2} \frac{C_D Re}{24}
\]  

(5)
Here, $u$ is the fluid phase velocity, $u_p$ is the particle velocity, $\mu$ is the molecular viscosity of the fluid, $\rho_p$ is the fluid density, $\rho$ is the density of the particle, and $d_p$ is the particle diameter. $C_D$ is a drag function based on the relative Reynolds number ($Re$). The study selects the Syamlal-O’Brien model [5], where the drag function has a form derived by Dalla Valle:

$$C_D = \left(0.63 + \frac{4.8}{\sqrt{Re/f \cdot v_s}}\right)$$

Re is the relative Reynolds number, which is defined as

$$Re = \frac{\rho d_p (u_p - u)}{\mu}$$

The exchange coefficient $K_{sl}$ is an important factor to embody fluid-solid two phase flow which can be written in the following general form:

$$K_{sl} = \frac{\alpha \rho_f}{\tau_s}$$

$\tau_s$, the particulate relaxation time, is defined as:

$$\tau_s = \frac{\rho d_p^2}{18 \mu}$$

The particle rebounds the off the boundary in question with a change (shown in Figure 1) in its momentum as defined by the coefficient of restitution.

```
\begin{figure}
\centering
\includegraphics[width=0.8\textwidth]{boundary.png}
\caption{Boundary condition of the discrete phase.}
\end{figure}
```

The normal coefficient of restitution defines the amount of momentum in the direction normal to the wall that is retained by the particle after the collision with the boundary.

$$e_n = \frac{v_{2,n}}{v_{1,n}}$$

Where $v_n$ is the particle velocity normal to the wall and the subscripts 1 and 2 refer to before and after collision, respectively. Similarly, the tangential coefficient of restitution, $e_t$, defines the amount of momentum in the direction tangential to the wall that is retained by the particle. A normal or tangential coefficient of restitution equal to 0 implies that the particle retains none of its normal or tangential momentum after the rebound, i.e., the particle deposited on the surface of tube [11]. Nonconstant coefficients of restitution can be specified for wall zones with the reflect type boundary condition.

2.3. Geometry model

The basic geometrical structure and partial parameters of corrugated tube is shown as Figure 2 and Table1.
2.4. Boundary condition.
The velocity inlet condition is selected which 2.0 m/s, 2.5 m/s, 3 m/s, 3.5 m/s, 4 m/s, 4.5 m/s and 5 m/s. The outflow and no slip boundary condition are used to define wall and outlet boundary condition. The particles flow into the tube with uniform distribution on the cross section. The inlet velocity of particles is consistent with fluid. The properties parameter of continuous and discrete phase is show follow as Table 2 and Figure 3.
Table 2. Material parameters.

|                  | Density       | Viscosity       |
|------------------|---------------|-----------------|
| Water (liquid)   | 971.79 kg/m³  | $3.551 \times 10^{-4}$ Pa·s |
| Calcium carbonate| 2800 kg/m³    | -               |

To simplify the simulation, some assumptions need to be proposed.  
(1) Due to homogeneous temperature of fluid, the particle movement in the mainstream area is not affected by temperature.  
(2) The particle is assumed to be a rigid sphere, no physical change occur in the movement.  
(3) The process merely includes mass transfer, i.e., chemical reaction isn’t considered.

3. Grid independence text
In this paper, CFD modeling is done with the commercial CFD software ANSYS FLUENT. In the investigation, unstructured of boundary layer is applied. In addition, the grid independence test is performed, and it is found that mesh with 500,500 elements is accurate enough. Grid independence test results are shown in Figure 4. The calculated simulation value change 2% with the previous value which can be considered simulation reach the requirement of grid precision, at the same time the computational efficiency is also high. Figure 5 shows the grid at the bellows section.
4. Results and discuss
The corrugated tube produces pressure drop with the periodically changes of cross section than smooth tube (Figure 6). The shear stress acting on the surface of corrugated tube is larger than smooth tube which results in that the corrugated tube has good anti-scaling properties. Figure 7 shows the distribution of deposition in the pipe.

Figure 5. Grid structure of the corrugated tube.

Figure 6. The pressure drop with velocity increases.

Figure 7. The contour of accretion.
Figure 8. The effect of mass rate on calcium carbonate of accretion rate.

Figure 9. The effect of velocity on accretion rate.

The results indicate the accretion rate increases from $1.33 \times 10^{-2}$ kg/m$^2$·s to $9.24 \times 10^{-2}$ kg/m$^2$·s with a linear increase as the mass rate of discrete phase increase (Figure 8). The accretion rate decreases from $2.7 \times 10^{-2}$ kg/m$^2$·s to $1.98 \times 10^{-2}$ kg/m$^2$·s as velocity increases from 2 m/s to 5 m/s (Figure 9). The relation between accretion rate and mass rate of discrete rate and velocity is fitted by linear regression method:

$$y = 0.05238 + 0.032x_1$$  \hspace{1cm} (11)

$$y = 3.191 - 0.227x_2$$  \hspace{1cm} (12)

where $y$ is accretion rate (kg/m$^2$·s), $x_1$ is mass rate of discrete phase (mg/L) and $x_2$ is velocity (m/s). The maximum fitting error shall not exceed 8%. Based this model, accretion rate has a linear relationship with velocity and mass flow respectively.

5. Conclusions

The simulation based on finite volume method discretizes governing equations. The Euler-Lagrange method is the main mean of this investigation. For anti-fouling performance, the different structures of corrugated tube play different effects.

Fluid flow in the corrugated tube creates bigger pressure drop than smooth tube which needs more driving work. The fluid in high Reynolds number flow region (from $4.8 \times 10^4$ to $1.2 \times 10^5$) damages velocity boundary layer and creates eddy on the surface of crest. The accretion rate increases almost linearly as the mass rate of discrete phase increases and inlet velocity decreases. The ratio of increase is related with physical model size of the corrugated tube and the properties of continuous and discrete phase, etc.

References

[1] Dehbi A, Martin S 1989 CFD simulation of particle deposition on an array of spheres using an Euler/Lagrange approach Nuclear Engineering & Design 241 3121-3129

[2] Kallio G A, Reeks M W 1989 A numerical simulation of particle deposition in turbulent boundary layers International Journal of Multiphase Flow 15 433-446.

[3] Zhang J, Li A 2008 CFD simulation of particle deposition in a horizontal turbulent duct flow Chemical Engineering Research & Design 86 95-106

[4] Rouhiainen P O, Stachiewicz J W 1970 On the Deposition of Small Particles from Turbulent Streams Heat Transfer 92 169
[5] M Syamlal and T J O’Brien 1989 Computer Simulation of Bubbles in a Fluidized Bed AIChE Symp Series 85 22–31

[6] Hojat N, Goodarz A, Mclaughlin J B 2009 A DNS study of effects of particle-particle collisions and two-way coupling on particle deposition and phasic fluctuations Journal of Fluid Mechanics 640 507-536

[7] Lin J H, Chang K C 2016 Particle Dispersion Simulation in Turbulent Flow Due to Particle-Particle and Particle-Wall Collisions Journal of Mechanics 32 237-244

[8] Xu Z, Zhao Y, Han Z, et al. 2018 Numerical simulation of calcium sulfate (CaSO₄) fouling in the plate heat exchanger Heat & Mass Transfer 1-11

[9] Li C W, Ma F X 2001 3D Numerical Simulation of Deposition Patterns due to Sand Disposal in Flowing Water Journal of Hydraulic Engineering 127 209-218

[10] Vicente P G, Garcia A, Viedma A 2004 Experimental investigation on heat transfer and frictional characteristics of spirally corrugated tubes in turbulent flow at different Prandtl numbers International Journal of Heat & Mass Transfer 47 671-681

[11] Stasiek J A 1998 Experimental studies of heat transfer and fluid flow across corrugated-undulated heat exchanger surfaces International Journal of Heat & Mass Transfer 41 899-914