

PAPER • OPEN ACCESS

Numerical simulation of CaCO_3 fouling formation in double pipe heat exchanger

To cite this article: Zhou qinqiao *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **371** 042047

View the [article online](#) for updates and enhancements.

You may also like

- [Pressure Loss on Corrugated Annulus of Double Pipe Heat Exchanger](#)
Putu Wijaya Sunu, I Made Suarta, D S Anakottapary *et al.*
- [Experimental Investigation of Double Pipe Heat Exchanger Performance based on Alumina and Copper Oxide Working Nanofluids](#)
Fawziea M. Hussien, Johain J. Faraj and Ahmed J. Hamad
- [Experimental Study of Heat Transfer in Concentric Triple Pipe Heat Exchanger](#)
P Tanish, N R Ramanathan, C S K Balasekhar *et al.*



The Electrochemical Society
Advancing solid state & electrochemical science & technology

243rd ECS Meeting with SOFC-XVIII

Boston, MA • May 28 – June 2, 2023

**Abstract Submission Extended
Deadline: December 16**

[Learn more and submit!](#)

Numerical simulation of CaCO_3 fouling formation in double pipe heat exchanger

Zhou qinqiao¹, Zheng yangyan¹, Tang hancheng^{2*}

¹Special equipment safety supervision inspection institute of Jiangsu province, china

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

*Corresponding author's e-mail: qq519157345@163.com

Abstract: Fouling of heat exchangers in the industry is a common problem. This paper studies the fouling formation of CaCO_3 in double pipe heat exchanger, and a numerical simulation of CaCO_3 fouling formation in double pipe heat exchanger has been established from the perspective of heat and mass transfer. This study uses a fouling model with the CFD framework by a discrete phase model in commercial software (Ansys Fluent). According to the simulation, the change of the rate of fouling formation with Sediment particle diameter, particle mass flow, inlet velocity has been discussed here. The results indicate that smaller particles display a higher deposition rate, and the faster the inlet velocity, the smaller the deposition rate. Also, a higher mass flow rate results in a high deposition rate.

1. Introduction

Energy is the key problem of industrial development in today's world, with more and more applications of heat pipe technology in the energy field, which has a large proportion in today's industry. However, the fluid medium is almost uncleaned, while the impurity particles in the fluid medium could affect the performance of the heat exchanger and even lead to the failure of the heat exchanger. In order to avoid such an accident, more and more scholars begin to study the fouling formation of heat exchangers, and then much attention has been paid to fouling formation in heat pipe exchangers. Hence, it's imperative to explore the influencing factors of fouling formation.

Fouling formation is a complex physicochemical process, but CFD is an effective tool to solve problems including fouling formation and heat transfer. ^[1]Timothy J. Rennie, Vijaya G.S. Raghavan modeled a double-pipe helical heat exchanger for heat transfer characteristics under different fluid flow rates and tube sizes by using CFD. The simulation results fell within the range found in the literature, and they also found that the annulus Nusselt number was correlated with a modified Dean number. ^[2]Bipan Bansal, Hans Müller-Steinhage compared the fouling in plate heat exchanger with the double pipe heat exchanger, and they developed the conclusion that the fouling in plate exchanger was less than the double pipe heat exchanger. ^[3]M. Esawy and M.R. Malayeri developed a model of crystallisation fouling of finned surfaces during nucleate pool boiling. They found that finned tubes could easily retarded the deposition process, and this research gave us a new phenomenological model of CaSO_4 fouling formation in finned tube. Athanasios G. Konstandopoulos^[4] found the effect of rebound coefficient tangential and impact angle on distribution of particle deposition rate along the tube surface. Bouris et al.^[5] studied the effect of pipe arrangement and pipe shape on scaling rate, the asymmetric tube bundle and elliptic-shaped tube bundle had effective descaling performance. A simple method was



presented for the calculation of deposition to smooth surfaces, it also suggested that deposition rates were extremely related to the wall surface coefficient^[6]. Another study was carried out by Song-Zhen Tang et al.^[7] to investigate the fouling morphology and the effect of fouling on heat transfer performance. W.S.J. Uijtewaal, and R.V.A. Oliemans^[8] developed a study about the motion of dense particles in a turbulence gas flow, and the results had shown that the motion of small and large particles was related to their response time to the turbulence integral time scale.

At this stage, no general model for fouling formation has been raised. Considering that the study object of this paper is CaCO₃, a three-dimensional CFD simulation of CaCO₃ fouling formation is established, The main objective of this work is to establish a numerical simulation of CaCO₃ fouling formation in double pipe heat exchange. The principal purpose of this study is to provide reference for conducting fouling experiments. This study discusses the influence of these parameters on the deposition in the tube from three aspects: particle diameter, inlet velocity, and fluid mass flow rate.

2. physical model

2.1 geometric model

The physical model uses a double pipe heat exchanger which owns an internal heat pipe with 8mm diameter and an external heat pipe with 18mm diameter. Considering the impact of the entry section and according to the Nikolaez experiment, here, the length of the tubes is set to 1000mm. The shell side both inlet and outlet section is a drainage tube with a diameter of 10mm and a length of 40mm. the front view of the casing heat exchanger and regional distributions are shown in the figure below.

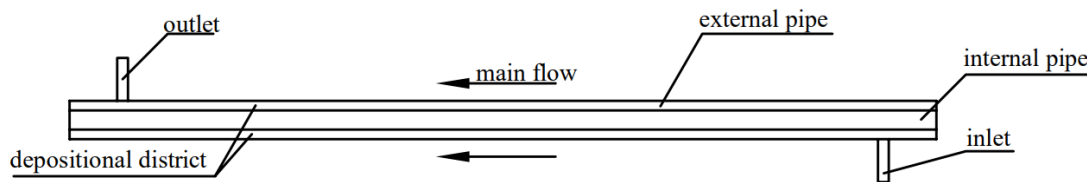


Fig. 1 schematic diagram of the physical model

2.2 Grid design and division

The influence of heat transfer is not considered here and the characteristic parameters of flow and fouling formation are assumed to be based on steady state. Take the direction of flow as the X-axis and the direction of gravity as the negative z-axis. The inlet interface is set to inlet, and the export interface is set to outlet. The whole watershed is divided into tetrahedral grids. Boundary layer grids are divided according to requirements for the wall area grid according to the wall function method. The boundary layer height of 1.1mm, the growth rate of 1.2 and the number of boundary layers are set as 8 layers.

The deposition process is a mass transfer process, so the mesh encryption is performed on the pipe wall. Since grid density has a great impact on the accuracy of numerical calculation, the number of grids should be analyzed for independence before numerical calculation. By adjusting the mesh size, we find that when the minimum mesh size reaches 1mm, the simulation data does not change with the change of mesh density, so the minimum mesh size is 1mm and a total of 2351183 grids are generated.

3. Mathematical model and boundary conditions

In order to obtain the velocity field, temperature field, mass concentration field and fouling formation deposition rate of the whole calculation region, the continuity equation, momentum squared equation, energy equation and mass transfer equation are used to solve this question comprehensively.^[9]The main equations applied are as follows:

The continuity equation:

$$\frac{\partial(\rho u_z)}{\partial z} + \frac{\partial(\rho u_r)}{\partial r} + \frac{\rho u_r}{r} = 0 \tag{1}$$

The momentum squared equation:

Radial direction:

$$\rho \left(\frac{\partial u_r}{\partial t} + u_r \frac{\partial u_r}{\partial r} + u_z \frac{\partial u_r}{\partial z} \right) = -\frac{\partial p}{\partial r} + \rho f_r + \mu \left[\frac{\partial^2 u_r}{\partial r^2} + \frac{\partial}{\partial r} \left(\frac{u_r}{r} \right) + \frac{\partial^2 u_r}{\partial z^2} \right] + S_r \tag{2}$$

Axial direction:

$$\rho \left(\frac{\partial u_z}{\partial t} + u_r \frac{\partial u_z}{\partial r} + u_z \frac{\partial u_z}{\partial z} \right) = -\frac{\partial p}{\partial z} + \rho f_z + \mu \left[\frac{\partial^2 u_z}{\partial r^2} + \frac{1}{r} \frac{\partial u_z}{\partial r} + \frac{\partial^2 u_z}{\partial z^2} \right] + S_z \tag{3}$$

(Supplementary instruction):

$$S_r = \left(\mu' + \frac{\mu}{3} \right) \left[\frac{\partial^2 u_r}{\partial r^2} + \frac{\partial}{\partial r} \left(\frac{u_r}{r} \right) + \frac{\partial^2 u_z}{\partial r \partial z} \right]$$

$$S_z = \left(\mu' + \frac{\mu}{3} \right) \left[\frac{\partial^2 u_z}{\partial z^2} + \frac{1}{r} \left(\frac{\partial u_r}{\partial z} \right) + \frac{\partial^2 u_r}{\partial z \partial r} \right] \tag{4}$$

The energy equation:

$$\frac{\partial T}{\partial t} + u_z \frac{\partial T}{\partial z} + u_r \frac{\partial T}{\partial r} = a \left[\frac{\partial^2 T}{\partial r^2} + \frac{1}{r} \frac{\partial T}{\partial r} + \frac{\partial^2 T}{\partial z^2} \right] \tag{5}$$

The mass transfer equation:

$$\frac{\partial c_f}{\partial t} + u_z \frac{\partial c_f}{\partial z} + u_r \frac{\partial c_f}{\partial r} = D \left[\frac{\partial^2 c_f}{\partial r^2} + \frac{1}{r} \frac{\partial c_f}{\partial r} + \frac{\partial^2 c_f}{\partial z^2} \right] - \frac{h_m}{\delta_T} (c_f - c_F) \tag{6}$$

Where u_r is the velocity component in the radial direction, u_z is the velocity component in the axial direction. ρ and μ are the density and dynamic viscosity of the fluid, P is the intensity of pressure, μ' is the second viscosity coefficient, f_r and f_z are radial and axial mass forces respectively. T is the thermodynamic temperature, t and a are the time and thermal diffusivity, respectively. This Mathematical model is closed and solvable under given initial and boundary conditions. Due to in our lives, most flows are turbulent. So in this study, the standard k-ε equation model is selected as the turbulence model.

Boundary conditions: When $t=0$, the fluid inlet velocity is 0.48m/s, the temperature of the fluid is 300K, it's assumed that the wall temperature of the internal heat pipe is 320K, and the CaCO_3 concentration in the fluid is 0.8kg/m^3 . Since the concentration of CaCO_3 in the fluid is not high, the fluid is considered as the physical property of water at 320K. Due to using the discrete phase in Fluent, So a three-dimensional steady-state heat transfer model is selected.

4. The DPM model of CaCO_3 fouling formation process

This process is influenced by particle diameter, wall roughness, deposition direction and other factors^[10].

In this study, the CaCO₃ fouling formation mainly occurred on the outer wall of the internal pipe and the inner wall of the external pipe^[11]. To simplify the model, the following assumptions are made:

- (a) Particle scale was precipitated at one time
- (b) Particles were spherical and distributed in diameter
- (c) The normal and tangential bounce coefficients on the wall did not change with the fluid flow

In this study, based on the above several assumptions, and the volume fraction of particle phase in solution is less than 10%, DPM model is adopted to simulate the deposition of impurity CaCO₃ in solution. When the exact boundary conditions are known, CaCO₃ is treated as discrete particles, so that the orbit of each particle can be tracked. Thus, we can obtain the deposition law of CaCO₃.

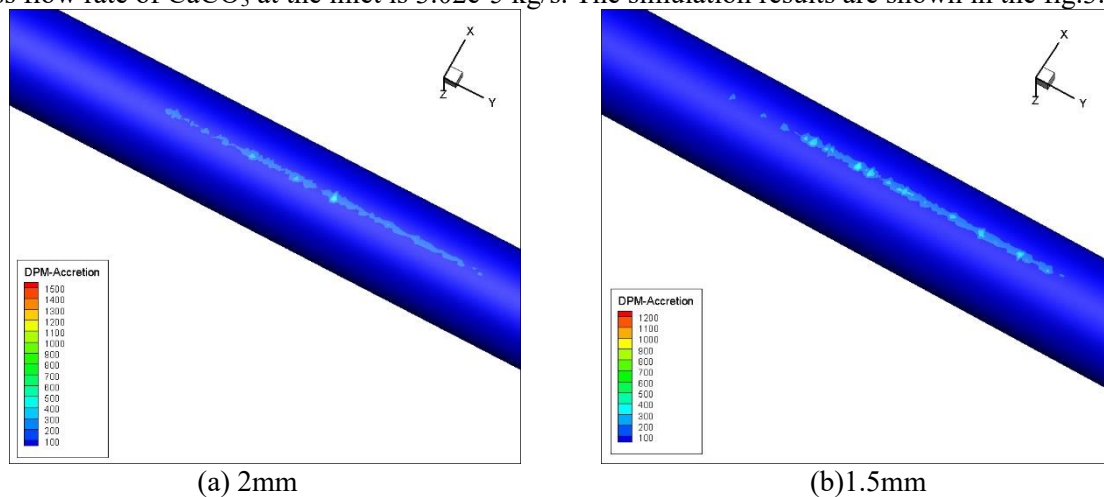
5. Results and discussions

5.1 Effect of particle diameter on wall deposition

Table 1 specific simulation program of particle diameter

Condition for change	Initial simulation condition	Comparative analysis of simulation condition
Minimum particle diameter/mm	1	0.5 0.75 1.25 1.5
Maximum particle diameter/mm	2	1.5 1.75 2.25 2.5
Mean particle diameter/mm	1.5	1.0 1.25 1.75 2.0

DPM model is adopted to study the influence of different particle diameters on wall deposition. Using the mass concentration of 0.8kg/m³ CaCO₃ solution as the working medium, the inner and outer tubes of the casing heat exchanger are both stainless steel tubes. The initial velocity at the inlet is 0.48m/s, the mass flow rate of CaCO₃ at the inlet is 3.02e-5 kg/s. The simulation results are shown in the fig.3.



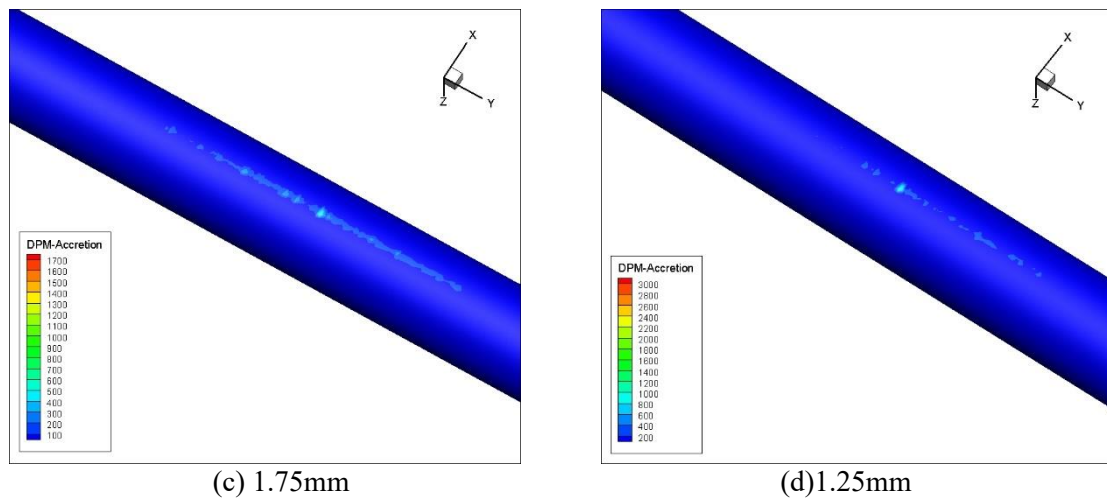


Fig.2 Tube wall fouling formation diagram with different particle diameters

In fig.2, it can be seen that when the inlet velocity is the same, the growth of fouling formation with different particle diameters is similar, the inlet does not contain CaCO_3 , But a portion of CaCO_3 generates some distance from the inlet. With the flow of the fluid, CaCO_3 finally forms scale near the pipe wall and settles down. And also the diagrams show that small particles of impurities are more likely to deposit on the tube walls. Most of the deposition forms on wall of the external pipe near the outlet. From the Fig.3, we can see that the inner deposition rate is much smaller than the outer deposition rate, by using the Ansys Fluent calculation, the average surface deposition rate is also shown in the Fig.3, it is inversely proportional to the particle diameter in solution.

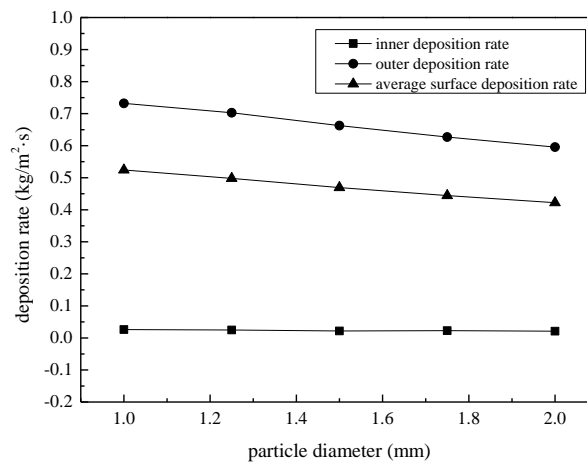


Fig.3 The relationship between particle diameter and deposition rate

5.2 Effect of the flow velocity on wall deposition

In order to study the influence of fluid inlet velocity on wall deposition, the inlet velocity is changed to 0.58, 0.68, 0.98, 1.28m/s, and the distribution of deposition rate on the wall of casing pipe is shown in the figure below.

According to the Fig. 4, with the increase of velocity, the wall deposition rate of CaCO_3 decreases gradually, and the higher the inlet velocity, the more obvious the effect of fluid on wall surface. Correspondingly, the deposition of CaCO_3 is less.

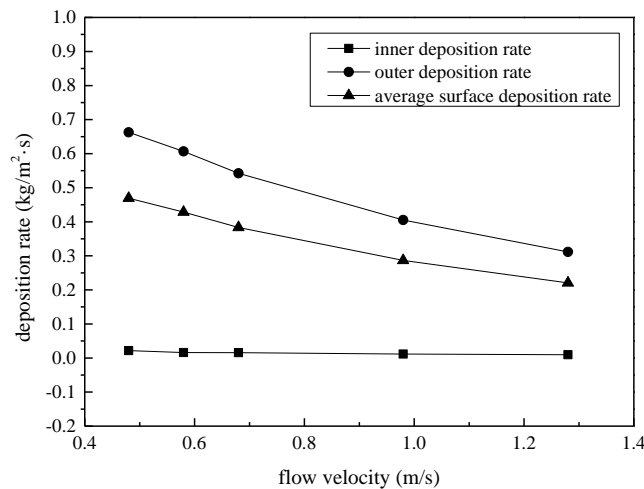


Fig.4 The relationship between flow velocity and deposition rate

5.3 Effect of the particle mass flow on wall deposition

The particle mass flow also has an effect on the scale deposition. Increase the concentration of particles in the tube by 1.5, 2, 2.5 and 3 times, and the variation rule of wall deposition rate is shown in the figure below.

In Fig.5, obviously, particle mass flow is positively correlated with deposition rate. With the increase of particle mass flow, the deposition rate on the wall also increases, correspondingly, the amount of CaCO₃ precipitation generated also increases.

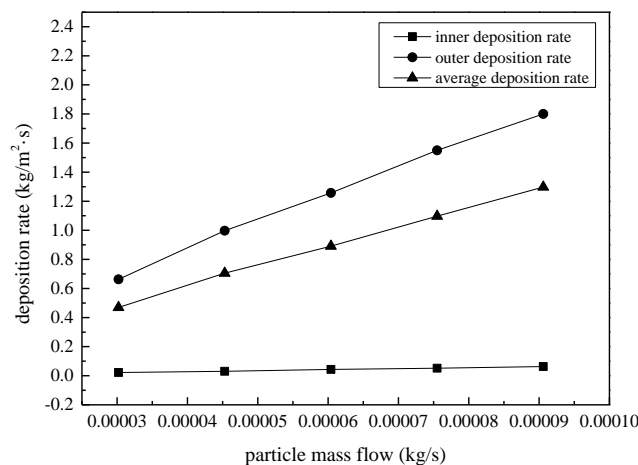


Fig.5 The relationship between particle mass flow and deposition rate

Through comparative analysis on appeal, the research can draw the following conclusions:

(a) Particle diameter has an effect on wall deposition, and small particles of impurities are more likely to deposit on the tube walls. Within a certain range, the wall deposition rate is negatively correlated with particle diameter.

(b) With the increase of velocity, the wall deposition rate of CaCO₃ decreases gradually. The higher the inlet velocity, the smaller the wall deposition rate is. This is because as the velocity increases, the fluid will wash the wall violently. As a result, the deposition of CaCO₃ will decrease correspondingly.

(c) The deposition rate on the wall increases with the increase of particle mass flow, and these two

parameters own a linear positive correlation.

6. Conclusion

This study is based on the principle of deposition, applying continuity equations as well as momentum, energy and mass transfer equations. The DPM mathematical model is used to establish numerical simulation of CaCO₃ fouling formation in casing heat exchanger.

This study finds that most of the deposition is not in the inlet section, but on the tube wall at a distance from the inlet section due to the flow of medium and the effect of gravity. Through the calculation of FLUENT, the formation and deposition process of CaCO₃ particles in the tube wall are simulated. This study quantitatively analyzes the effects of particle diameter, inlet velocity, and fouling particle concentration on the deposition, and finds that particle diameter, inlet velocity and deposition rate are negatively correlated linearly, on the contrary, Particle concentration is positively correlated with deposition rate. The influence law is consistent with that obtained by other scholars through simulation and experimental verification, which fully verifies the applicability of the model.

References

- [1] Rennie T J, Raghavan V G S. Experimental studies of a double-pipe helical heat exchanger[J]. *Experimental Thermal & Fluid Science*, 2005,29(8):919-924.
- [2] Bipan Bansal H M L X. Comparison of Crystallization Fouling in Plate and Double-Pipe Heat Exchangers[J]. *Heat Transfer Engineering*, 2001,22(5):13-25.
- [3] Esawy M, Malayeri M R. Modeling of CaSO₄ crystallization fouling of finned tubes during nucleate pool boiling[J]. *Chemical Engineering Research and Design*, 2017.
- [4] Konstandopoulos A G. Particle sticking/rebound criteria at oblique impact[J]. *Journal of Aerosol Science*, 2006,37(3):292-305.
- [5] Bouris D, Konstantinidis E, Balabani S, et al. Design of a novel, intensified heat exchanger for reduced fouling rates[J]. *International Journal of Heat & Mass Transfer*, 2005,48(18):3817-3832.
- [6] Wood N B. A simple method for the calculation of turbulent deposition to smooth and rough surfaces[J]. *Journal of Aerosol Science*, 1981,12(3):275-290.
- [7] Tang S, Wang F, Ren Q, et al. Fouling characteristics analysis and morphology prediction of heat exchangers with a particulate fouling model considering deposition and removal mechanisms[J]. *Fuel*, 2017,203.
- [8] Uijtewaal W S J, Oliemans R V A. Particle dispersion and deposition in direct numerical and large eddy simulations of vertical pipe flows[J]. *International Journal of Multiphase Flow*, 1996,8(10):2590-2604.
- [9] P Kk Nen T M, Ojaniemi U, P Ttikangas T, et al. CFD modelling of CaCO₃ crystallization fouling on heat transfer surfaces[J]. *International Journal of Heat & Mass Transfer*, 2016,97:618-630.
- [10] Jamialahmadi M, Müller-Steinhagen H. Heat Exchanger Fouling and Cleaning in the Dihydrate Process for the Production of Phosphoric Acid[J]. *Chemical Engineering Research & Design*, 2007,85(2):245-255.
- [11] Brahim F, Augustin W, Bohnet M. Numerical simulation of the fouling process[J]. *International Journal of Thermal Sciences*, 2003,42(3):323-334.