 Simulation of aerosolized oil droplets capture in a range hood exhaust using coupled CFD-population balance method

Shuyuan Liu, Yong Zhang, Yu Feng, Changbin Shi, Yong Cao and Wei Yuan

1 Harbin Institute of Technology, Shenzhen Graduate School, Guangdong 518055, China;
2 Shenzhen Xinbaoying Science & Technology Co., Ltd, China.
3 fengyu85@hit.edu.cn

Abstract. A coupled population balance sectional method (PBSM) coupled with computational fluid dynamics (CFD) is presented to simulate the capture of aerosolized oil droplets (AODs) in a range hood exhaust. The homogeneous nucleation and coagulation processes are modeled and simulated with this CFD-PBSM method. With the design angle, $\alpha$ of the range hood exhaust varying from 60° to 30°, the AODs capture increases meanwhile the pressure drop between the inlet and the outlet of the range hood also increases from 8.38Pa to 175.75Pa. The increasing inlet flow velocities also result in less AODs capture although the total suction increases due to higher flow rates to the range hood. Therefore, the CFD-PBSM method provides an insight into the formation and capture of AODs as well as their impact on the operation and design of the range hood exhaust.

1. Introduction

Aerosolized oil droplet (AOD) is a main particulate air pollutant. The AOD may consist of carbon monoxide, carbon dioxide and hydronitrogen, which greatly harms the health of people [1]. The range hood is the important tool which is employed to clear the AOD in cooking, so enhancing suction efficiency is the key point in the design of range hood. Increase in the flow rate is the main way to enhance suction efficiency for the design of present range hood, which results in such problem as: high energy consumption, high noise, and so on. Therefore, the effective suction for the AODs has great influence on the performance of range hood.

At present, in order to enhance the suction efficiency of range hoods, many researchers focus on the optimization of local structure in range hood. The suction controlled by an artificial tornado in range hood was studied by Zhang [2]. The feasibility of enhancing suction efficiency for the artificial tornado was verified in Zhang’s study. For the high noise in range hood, Huang [3] and Son [4] optimized the centrifugal fan. In the above studies, the improvement of aerodynamic is to increase the performance of range hood by optimizing the local structures.

Rotary jet equipment is mainly used to collect the AOD in range hood. The small oil droplets with different dimensions are contained in AOD besides a large number of harmful gases. The oil droplets collide with each other and congregate on the wall of range hood. The gathered oil will be recycled. The movement of small oil droplets inflow is very complicated, so some oil droplets do not congregate on the wall of range hood and exhaust to the outside, which causes the pollution for the environment. Therefore, for the design of rotary jet equipment, on the one hand, it is effective entrainment for the
Therefore, in this paper, the CFD-PBSM was used to simulate the aerodynamic flow field in rotary jet equipment, and the collision and congregating among the small oil droplets were focused on in this study. In addition, to optimize the rotary jet equipment, the influence of design angle of the rotary jet equipment on the congregating of oil droplets was studied as well. In the present study, a population balance sectional method (PBSM) method proposed in our previous research [5] is used together with CFD method to simulate the homogeneous nucleation and coagulation processes in the oil droplet capture within a range hood exhaust. The effects inlet flowrates as well as the geometric structures of the range hood are investigated and analyzed.

2. Methodology

2.1. Governing equations

The governing equations of the coupled fluid-particle dynamics in incompressible flows include the continuity, momentum and energy equations i.e. Equations (1) and (2) as well as species transport equation i.e. Equation (3):

\[ \nabla \cdot \mathbf{u}(x, t) = 0 \quad (1) \]
\[ \frac{\partial u(x, t)}{\partial t} + (u(x, t) \nabla) u(x, t) = -\nabla P(x, t) + \mathbf{f}(x, t) \quad (2) \]
\[ \frac{\partial Y_a}{\partial t} + \nabla (u(x, t) Y_a) = D_a \nabla^2 Y_a + \dot{\omega}(Y_1, Y_2, \ldots, Y_m) \quad (3) \]

where \( u \) is the velocity of the carrier fluid phase, \( v \) is the kinematic viscosity which is assumed constant, \( x \) is the coordinates of particles, \( t \) is the time, \( P \) is the pressure, \( \rho \) is the fluid density, \( Y_a \) is the mass fraction of species (i.e., \( a = 1, 2, \ldots, m \)), \( D_a \) is the diffusion coefficient in composition space and \( \dot{\omega} \) is the source term determined by the aerosol dynamic processes (i.e., coagulation, nucleation and growth) in the present study. The following equations present the \( k-\varepsilon \) turbulence model in Cartesian tensor notation form and repeated indices mean summation [6]:

\[ \frac{\partial}{\partial t} [\bar{\rho} k] + \frac{\partial}{\partial x_j} [\bar{\rho} u_j k] = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G - \bar{\rho}e \quad (4) \]
\[ \frac{\partial}{\partial t} [\bar{\rho}e] + \frac{\partial}{\partial x_j} [\bar{\rho}u_j e] = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_e} \right) \frac{\partial e}{\partial x_j} \right] + \frac{\varepsilon}{k} \left[ C_{1\varepsilon} G - C_{2\varepsilon} \bar{\rho}e \right] \quad (5) \]

where Reynolds averaged quantities are shown with a bar on top while Favre averaged quantities are shown with a tilde on top. \( \rho \) is the mixture density, \( \mu \) is the molecular viscosity of the mixture, \( \mu_t \) is the turbulent viscosity, \( \sigma \) is the turbulent Prandtl number and \( G \) is the generation rate of turbulent kinetic energy. The constants of \( C_1, C_2, \sigma_k \) and \( \sigma_e \) are 1.44, 1.92, 1.0, and 1.3, respectively.

The PBE in terms of particle number density \( n(v, x, t) \), a function of particle volume as well as of space coordinates and time, can be written as Equation (4), in which \( n(v, x, t) \) is written as \( n(v) \) for simplicity,

\[ \frac{\partial n(v)}{\partial t} + \nabla (u n(v)) = D_a \nabla^2 n(v) + B(Y_1, Y_2, \ldots, Y_m, v) - \dot{\omega}(v) n(v) \]
\[ + \frac{1}{2} \int_{v_0}^{v_{\infty}} K(u, v-u) n(u, t) n(v, t) du - n(v, t) \int_{0}^{v_{\infty}} K(u, v) n(u, t) du \quad (6) \]

The terms in PBE i.e. Equation (6) from the left-hand side to the right-hand side are the accumulation term of the particle number density, the convection term, the particle diffusion term, the nucleation term, and coagulation term, respectively. The PBE shown in Equation (6) is solved by the population balance sectional (PBSM) method modified in [5].
2.2. Numerical setup
The simulation parameters are shown in Table 1. The simultaneous nucleation and coagulation processes are considered. An inlet air flow with uniformly distributed oil droplets (diameter 0.1-6.4μm) is introduced to the inlet of the simplified range hood with three different geometric structures (shown in Figure 1). Three inlet flow velocities i.e. 2m/s, 4m/s, and 8m/s are used to test the influence of inlet flowrate. Two-dimensional grid mesh is generated in domain consisting of a half of the cross section of the range hood as shown in Figure 1.

Table 1. Simulation parameters.

| Particulate phase | Arachidonic acid (C_{20}H_{32}O_2) |
|-------------------|-------------------------------------|
| Oil droplet density (kg/m$^3$) | 900 |
| Initial particle size range (μm) | 0.1–6.4 |
| Inlet velocity (m/s) | 2–8 |
| Coagulation model | Turbulent kernel (Saffman and Turner, 1956) |
| Nucleation rate (#/m$^3$⋅s) | 7.5×10$^6$ |
| Turbulence model | k-ε model with enhanced wall treatment |

Figure 1. The simplified geometric structure of a range hood exhaust (α=30°, 45° and 60° are tested in the present study).

3. Results and discussion
The effects of the design angle, α (as shown in Figure 1) and the inlet flowrates on the formation and distributions of AOD are studied and discussed in this section. The PBSM model used here has been modified and validated by a probability density function (PDF) based Lagrangian Monte Carlo (LMC-PDF) method in the authors’ previous study [5].

3.1. The effect of the geometric structures
With the design angle, α varying from 30° to 60°, three different geometric structures are obtained. The total number concentration of the captured AODs varies significantly for different geometric structures. This can be explained by the varying the entrainment effect of the eddy as shown in the small local view of the flow field as well as the varying cross section of the outlet of Figure 2. However, with the increase of AODs capture, the pressure drop also increases as shown in Table 2. In other words, the varying geometric structure results in not only increasing AODs capture but also increasing pressure drop, which implies a balance between the energy cost and the AODs capture should be achieved.
Figure 2. The number density (#/m³) distributions of AOD for different geometric structures.

Table 2. Pressure drop of different design angles

| α   | 30°  | 45°  | 60°  |
|-----|------|------|------|
| ∆P(Pa)| 175.75 | 24.13 | 8.38 |

3.2. The effect of inlet flow velocities
Figure 3 shows that the AODs capture decreases as the inlet flow velocities increases from 2.0m/s to 8.0m/s. This is because the increasing inlet flow velocity yields more intensive eddy near the obtuse angle corner, which inhibits the coagulation processes by transporting the droplets to the main flow stream, thus less AODs are captured. Therefore, the operating conditions should be optimized in order to obtain high suction as well as high AODs capture.

Figure 3. The number density (#/m³) distribution of oil droplets for different inlet flow velocities (the inlet flow velocities are 2.0m/s, 4.0m/s and 8m/s respectively from the left the right, the angle is 45 degree).

4. Conclusions
The CFD-PBSM method is applied to the simulation of aerosolized oil droplet formation in the range hood exhaust with consideration of simultaneous coagulation and nucleation processes. The conclusions are drawn as follows,

Smaller design angle leads to higher capture of AODs, but also results in higher pressure drop between the inlet and outlet of the range hood exhaust;

Although higher inlet flow velocities increases the suction of the range hood, it reduces the AODs capture by affecting the coagulation processes.

The coupled CFD-PBSM method can provide a better insight into the design and operation of range hood exhaust with consideration of aerosolized oil droplets formation and distribution in it.
Acknowledgement
This research work is supported by Shenzhen Technology Projects (CXZZ20150331160450786, GJHZ20160226200952216) and China Postdoctoral Science Foundation (No. 2017M621267).

References
[1] Mukherjee A, Agrawal M 2018 A global perspective of fine particulate matter pollution and its health effects Reviews of Environmental Contamination & Toxicology 244 5-51
[2] Zhang J S, Zhou S N 1996 Journal of China University of Mining & Technology 25 1-5
[3] Huang J D, Yin M, Deng Q, et al 2011 Journal of Shanghai Jiaotong University 35 1196-1199
[4] Son P N, Kim J W, Byun S M, et al 2012 Journal of Mechanical Science and Technology 26 1531-1538
[5] Liu S, Chan T L 2017 Aerosol Science and Technology 51 269-281
[6] Akridis P, Rigopoulos S 2015 In: Procedia Engineering 102 1274-1283
[7] Saffman P G, Turner J S 1956 Journal of Fluid Mechanics 1 16-30