CFD simulation and experimental validation of multiphase flow in industrial cyclone preheaters

Mirzaei, Mohamadali; Clausen, Sønnik; Wu, Hao; Zakrzewski, Sam; Nakhaei, Mohammadhadi; Zhou, Haosheng; Martin Jønck, Kasper; Arendt Jensen, Peter; Lin, Weigang

Published in:
Chemical Engineering Journal

Link to article, DOI:
10.1016/j.cej.2023.142757

Publication date:
2023

Document Version
Publisher's PDF, also known as Version of record

Link back to DTU Orbit

Citation (APA):

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain.
- You may freely distribute the URL identifying the publication in the public portal.

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.
CFD simulation and experimental validation of multiphase flow in industrial cyclone preheaters

Mohamadali Mirzaei a, Sønnik Clausen a, Hao Wu a, Sam Zakrzewski b, Mohammadhadi Nakhaei b, Haosheng Zhou c, Kasper Martin Jonck c, Peter Arendt Jensen a, Weigang Lin a,*,

a CHEC Research Centre, Department of Chemical and Biochemical Engineering, Technical University of Denmark, Kongens Lyngby 2800, Denmark
b FLSmidth A/S, Valby 2500, Denmark
c Rockwool A/S, Hedehusene 2640, Denmark

ARTICLE INFO

Keywords:
Dense dispersed gas-solid flow
Dense Discrete Phase Model (DDPM)
Gas-particle heat transfer
Industrial-scale cyclone preheater

ABSTRACT

A hybrid multiphase model (Dense Discrete Phase Model, DDPM) coupled with a k-ω sst turbulence model, which includes gas-solid heat exchange was developed for the simulation of industrial-scale cyclone preheaters operating under high solid loadings. The model results are validated by experimental measurements from an industrial cyclone preheater with a diameter of 1.6 m, with respect to the pressure drop, gas and particle exit temperature, local gas velocity profiles, measured using the LDA, and gas temperature profiles, measured using thermocouple and FTIR. Reasonable agreement between the data from experiments and simulations was obtained. In addition, the major trends, such as changes in the pressure drop, overall heat transfer rate, and flow pattern, caused by the changes in the operating parameters can be captured accurately by the model.

1. Introduction

Cyclones, primarily utilized for particle separation, are often used in high-temperature processes. In addition to their straightforward design and affordability, their ability to adapt to high-temperature conditions, especially over 900°C, make cyclones the most applied gas-particle separator in high-temperature processes [1]. Drying, decarburization, solidification, coal pyrolysis, gasification, combustion of solid fuels, and fluid catalytic cracking applications are some of the high-temperature processes where cyclone units play a critical role [2]. Cyclones are also commonly used as heat exchangers (cyclone preheaters) mostly in order to preheat solid material to improve the energy efficiency of high-temperature processes. Cyclone preheaters are key components in several processes such as cement production [2–5] and stone wool production [6,7] and are mostly operated at high solid loads (dense gas–solid flow).

CFD simulation of cyclone preheaters (or in general cyclones operating at elevated temperatures) have been reported in the literature including different levels of coupling of particle and gas phases and resolution: single-phase calculation [8,9], one-way coupling [10], two-way coupling [3,4], and four-way coupling [11–14]. Some of these studies are summarized in Table 1 which also includes CFD simulation of large-scale cyclone preheaters. As can be observed, the validation of these investigations is limited to overall experimental measurements (pressure drop, heat transfer rate, and separation efficiency), and validation in terms of flow pattern and temperature distribution are missing. Another point to mention is that Eddy Viscosity turbulence models such as (RNG and k-ε) are more common in the simulation of non-isothermal cyclones than isothermal ones.

Considering that in cyclones, particles are mostly concentrated close to the walls, in the case of highly loaded cyclones, particle–particle interactions might play a significant role in the hydrodynamics of multiphase flow. Therefore a four-way coupling method should be considered [11,20–29]. However, conventional four-way coupling models namely the Discrete Element Method (DEM) model and the Eulerian-Eulerian model might not be suitable options. DEM is too expensive to be implemented in industrial-scale simulations (Although coarse-graining strategies are available to reduce computational expenses of DEM [30–32], its current application in cyclones is still limited to lab-scale cases [22–27]), and the Eulerian-Eulerian model cannot easily handle the polydispersity of particles which is quite crucial in the performance of cyclones. On the other hand, a more novel hybrid multiphase model, the Dense Discrete Phase Model, that has been successfully validated for...
the simulation of hydrodynamics of multiphase flow in a highly loaded large-scale cyclone in our previous work [33,34] is a promising and feasible model.

In general, the existing experimental studies of cyclones operating at high temperatures, particularly in the case of highly loaded cyclone preheaters are limited to overall measurements of pressure drop, separation efficiency and temperature changes over the whole cyclone system in lab-scale devices [35]. Local measurements of velocity or temperature profiles in highly loaded industrial-scale cyclones are missing in the literature, to the best of the authors’ knowledge.

In the limited existing experimental studies, in the case of cyclones at high temperatures, a reduction in both the pressure drop and separation efficiency has been reported when gas temperature is elevated (at a fixed volumetric gas-flow rate) [36,37]. In the case of cyclone preheaters, the overall heat transfer rate is reported to increase with increasing solid and gas flow rates, and decreasing particle size [38,39]. The improvement with the increase in the solid flow rate is attributed to the reduction in the amount of gas bypassed to the vortex finder due to the reduced radial gas velocity observed when the solid load is increased. On the other hand, the improvement with an increase in the gas flow rate is explained by the increased driving factor for gas–solid heat exchange which is due to the greater availability of hot gas to transfer heat to the particles. Correlations for the prediction of the overall heat transfer coefficient in cyclone preheaters are reported in the limited experimental studies in lab-scale cyclone preheaters [38,39].

Regarding the limited availability of the experimental data, to obtain the lacking knowledge to bridge the gaps of understanding of what is happening inside industrial cyclone preheaters, a CFD model with detailed validation of flow and temperature distribution, to be utilized with confidence for simulation of highly loaded industrial cyclone preheaters is missing. To simulate highly loaded industrial cyclone preheaters, gas, solid, and gas–solid heat transfer modeling need to be added to the model developed for the simulation of hydrodynamics of multiphase flow in the previous work [33,34].

The main objective of the present work is to suggest a CFD model for the simulation of highly loaded industrial cyclone preheaters with detailed validation to improve understanding of gas–solid separation and heat transfer in this kind of device. For this purpose, gas-particle heat transfer modeling has been incorporated into a hybrid Eulerian-Lagrangian multiphase model (DDPM) coupled with a dispersed version of the k-ω SST turbulence model sensitized for flow rotation for simulation of the present cyclone preheater. The simulations are performed using Ansys Fluent 2021R1 and were validated by measurements of pressure drop, gas exit temperature, heat transfer rate, and also local flow and temperature patterns conducted under several operating conditions. The local measurements were provided by conducting water-

### Table 1

Details of CFD studies on gas–solid multiphase flow in cyclones investigating gas–solid heat exchange. The ones marked with * studied industrial-scale cyclones used for the purpose of preheating.

<table>
<thead>
<tr>
<th>Author</th>
<th>Cyclone diameter (m)</th>
<th>Temperature range (K)</th>
<th>Solid volume fraction</th>
<th>Turbulence model</th>
<th>Multiphase model</th>
<th>Validation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cristea et al. [15]</td>
<td>Industrial scale</td>
<td>300–1200</td>
<td>around 0.0007</td>
<td>RSM</td>
<td>Two Fluid Model</td>
<td>Pressure drop and separation efficiency measured by the authors were used for the validation.</td>
</tr>
<tr>
<td>Mikulic et al. [15]*</td>
<td>6</td>
<td>900–1200</td>
<td>0.0002</td>
<td>LES</td>
<td>one-way DPM</td>
<td>Pressure drop and heat transfer rate measured by the authors were used for validation.</td>
</tr>
<tr>
<td>Mothilal et al. [16,17]</td>
<td>0.1</td>
<td>300–500</td>
<td>up to around 0.0002</td>
<td>RNG</td>
<td>one-way DPM</td>
<td>The heat transfer rate measured by the authors.</td>
</tr>
<tr>
<td>Wasilewski et al. [4]*</td>
<td>3.5</td>
<td>300–900</td>
<td>around 0.0002</td>
<td>RSM</td>
<td>two-way DPM</td>
<td>The heat transfer rate measured by the authors.</td>
</tr>
<tr>
<td>Mariani et al. [11,13]*</td>
<td>6</td>
<td>600–850</td>
<td>around 0.0001</td>
<td>k-ω</td>
<td>Two Fluid Model</td>
<td>The heat transfer rate measured by the authors.</td>
</tr>
<tr>
<td>Cristea et al. [14,19]*</td>
<td>industrial scale</td>
<td>900–1100</td>
<td>around 0.0007</td>
<td>RSM</td>
<td>DDPM</td>
<td>Separation efficiency, pressure drop, and heat transfer rate measured by the authors.</td>
</tr>
</tbody>
</table>
2. Experimental study for model validation

2.1. Case description

Experiments were performed in an industrial cyclone preheater with a diameter of 1.6 m for validation of the CFD model, which is a part of the Integrated Melting Furnace (IMF) system [40], operated by Rockwool A/S. The IMF system melts mineral solid feed and then converts melted material into mineral wool and eventually isolation material. In the IMF system, there are two cyclone preheaters in cascade configuration and the top one is studied in the present work. A 3D view of the studied cyclone preheater is illustrated in Fig. 1. The experiments were done at various operating conditions (i.e., various particle feed loads, gas flow rates, and gas inlet temperatures). There are four main components in this system: the main cyclone body (4), the dust bin (6), the riser (3), and the particle feeding pipe (2). Mineral particles are injected into the riser, where they are carried up to the cyclone by the upward flow of the gas stream.

A PSD (Particles Size Distribution) of particles used during the experiments is obtained from sieving analysis and is presented in Fig. 2. The particles had a Sauter mean diameter of 375 \( \mu m \) with an approximate density of 2700 kg/m\(^3\), which is classified as a Geldart B type of particle, based on Geldart’s particle classification [41]. The particles are a mixture of natural rock materials [40]. Two slightly different mixtures of natural rock materials were used throughout the experiments (the composition was slightly changed due to the company production line demand). The two types of materials were tested in a DSC (NETZSCH model STA 449) and show slightly different heat capacities, which are presented in Fig. 3.

The experiments were conducted at 10 different operating conditions with varied gas and solid flow rates and slightly different gas inlet temperatures to provide validation data for the CFD study.

Table 2 summarizes the operating conditions (gas and solid flow rates, and gas and solid inlet temperatures), and types of measurements conducted in these 10 cases. The experimental data in the present study can be categorized into two groups: plant operation data and data from measuring campaigns:

2.2. Plant operation data

Plant operation data includes online measurements of pressure drop (difference measured by two plant’s pressure sensors shown in Fig. 1), gas inlet and exit temperature and particle exit temperature (using a thermocouple in a pile of particles accumulated in the dust bin). Furthermore, gas and solid flow rates are measured online.

2.3. Data from measuring campaigns

Flow pattern measurements and temperature profile measurements were conducted in measuring campaigns. Details of these measurements are presented as follows:

2.3.1. Flow pattern measurements

A fiber optic laser Doppler anemometer (LDA) system with a BSA processor (series F60) from Dantec Dynamics was used to perform 2D velocity measurements to provide axial and tangential velocity profiles in the cyclone main body (along the yellow line in Fig. 1). A commercial (Dantec Dynamics) 2D fiber optic LDA head mounted in a water-cooled probe with a protection window that can be inserted through the measurement port for mapping flow profiles was used in the present study [42]. The LDA head had a diameter of 27 mm diameter and a focal length of 160 mm (details of the LDA head geometry can be seen in Figure S-1 in the supplementary material). The LDA head was connected to a transmitter unit with a fiber-optic cable. The transmitter unit was a Dantec model 60X41 using a 300-mW air-cooled argon-ion laser (Stellar-Pro-L Select 300). Data was typically collected over 60–120 s for each measurement point using BSA processor series F60.

The LDA head is sensitive to temperature and damage to the optical lens. As Dantec recommends keeping the temperature of the LDA head below 45 °C, i.e., water-cooling was required for measurements inside the hot cyclone. Furthermore, in the studied cyclone preheater with fast-moving large particles, the fragile coated lens of the LDA head must be protected against erosion from particles moving towards the LDA optics during measurements. A tilted quartz window with a diameter of 25 mm and thickness of 3 mm was used as a protection window in front of the LDA head. Furthermore, a minor purge flow is used to avoid penetration of small gas-borne particles into the probe.

Measurement principles of LDA used in the present work can be found on the website of Dantec Dynamics [43]. The gas velocity can be
approximated by the LDA particle velocities measured. It is worth mentioning that a trace amount of micron-size particles which have good tracing properties are present in the system (coming with the inlet gas) reflecting the gas velocity in the LDA measurement. Also, the relatively coarse particles injected into the system (shown in Fig. 2) which have poor tracing properties are mostly concentrated in the near wall region, so they would not affect the LDA measurement in the cyclone main body when statistical averaging is done. The LDA measurement volume dimensions in the present setup are 0.074 mm in diameter and 1.57 mm long at 488 nm wavelength, with a fringe spacing of 5.2 µm and a total volume of approximately 0.007 mm³. The number of detected particles per mm³ is approximately 2 for an LDA data rate of 5 kHz and a velocity of 20 m/s, i.e., LDA signals in the cyclone are mainly generated by many fine particles that follow the gas flow. The data rate was fairly constant across the 1.6 m diameter of the cyclone which confirms that large particles contribute very little statistically to the obtained data. Thus, the gas velocity can be approximated well by the LDA particle velocities measured. High sampling rates were obtained in all positions across the cyclone, indicating that LDA signals mainly come from high-concentration fine particles.

As with any industrial-scale measurement, flow access and maintaining steady operation represent major challenges in this work rather than the operation of the LDA system under industrial conditions. Repeated LDA velocity measurements for 60–120 s give the same result under stable operating conditions of the preheater cyclone. However, it was difficult to control fully the operation conditions over time for a complex industrial production facility.

2.3.2. Temperature profile measurements
Two types of measurements were done to provide a gas temperature profile in the cyclone main body (along the yellow line in Fig. 1): Thermocouple measurement and FTIR measurement.

2.3.2.1. Thermocouple measurement. Thermocouple measurements were performed with a 3.0 mm diameter thermocouple of type N (Nickel-alloy thermocouple) connected to an Agilent data acquisition/data logger switch unit (Agilent 34970A). The thermocouple was mounted in a probe made of a stainless-steel tube and Swagelok fittings at the tip for a gas-tight seal. The inner part of the probe was filled with insulation material to avoid airflow movements inside the tube.

2.3.2.2. FTIR measurement. In FTIR measurement (infrared IR fiber-optic spectroscopic method [34]) the local gas temperature in front of the probe tip was extracted from the measured FTIR thermal emission spectrum; it can be found from FTIR thermal emission from CO₂ at around 2350 cm⁻¹ (4.3 µm) by scaling with Planckian curve. Details of the developed system can be found in the literature [44,45]. A portable blackbody was used for calibration before and after the cyclone measurements to capture any problems with unstable measurements due to dirt or particles on optics. Measurements at each point were in general made over several minutes. Data were analyzed and oscillations with a period of 5.4 s of the gas temperature were clearly seen in the FTIR measurements and weakly in thermocouple measurements due to the slow response time.

Preliminary measurements showed that an uncooled probe should be used instead of a water-cooled one as water-cooling offsets temperature measurements. It should be noted that the two methods provided similar average gas temperatures.

Fig. 2. (a) cumulative and (b) differential PSD of the natural rock materials used during the measurement campaigns by sieving analysis. This PSD is measured for Rock material 1; Rock material 2 has a similar PSD.

Fig. 3. Comparison of the specific heat capacity of the two mixtures used in campaigns as a function of temperature measured by an STA (NETZSCH model STA 449).
3. Numerical model

In the present work, a hybrid Eulerian-Lagrangian multiphase model, Dense Discrete Phase Model (DDPM) in Ansys Fluent (2021R1) solver coupled with a dispersed version of the k-ω st turbulence model sensitized to rotation and curvature is used. The development of the numerical model is presented in detail in our previous work [33,34]. This model was successfully validated for the simulation of hydrodynamics of a highly loaded large-scale cyclone in our previous work [33,34]. It was found that the revised Sarkar drag model [46] (which is a sub-grid drag model) provides slightly better predictions of important performance parameters of highly loaded large-scale cyclones. Therefore, this drag model is also used in the present work. Unlike the previous work, the particles in the present work are relatively large such that the agglomeration does not play a role and therefore is excluded in this study. How the numerical model is developed and the description of the model including the continuity and momentum conservation equations (Navier-Stokes equations) of the gas phase, equation of motion of particles, equations to model turbulence, gas-particle interactions, and particle–particle interactions, etc. can be found in our previous work [33,34]. Sub-models (models for particle–particle interaction force calculation, turbulence models, drag models and other gas-particle interactions) and model parameters (particle–particle restitution coefficient and particle–wall rebound coefficient) used in the present work are decided based on our previous work [33,34] and can be found there. It is worth mentioning that considering the reported impact of wall roughness modeling in CFD simulation of cyclones [47], a sand-grain roughness height of 5 μm was assumed in the present case.

A PDE (Partial Differential Equation) for energy conservation equation is added on top of six PDEs for multiphase and turbulence modeling in the Eulerian framework for gas phase calculation for modeling the gas–solid heat transfer. A term for the heat exchange between gas and particle appears in the energy conservation equation in the Eulerian phase (gas phase) calculation as follows:

\[ Q_{p-g} = h_{p-g} A_p (T_p - T_g) \]  

(1)

where \( h_{p-g} \) is the heat transfer coefficient between the gas and particle and is modeled in the present work by Ranz-Marshall correlation by assuming iso-thermal particles in the system.

\[ Nu_p = \frac{h_{p-g} d_p}{k_g} = 2 + 0.6 Re^{1/2} Pr^{1/3} \]  

(2)

While a simple heat balance is used in the Lagrangian (particle) calculation to update particle temperature as follows:

\[ m_p C_p \frac{dT_p}{dt} = h_{p-w} A_p (T_w - T_p) \]  

(3)

Table 3 summarizes the parameters used in the simulation. It should be mentioned that the raw material used during the experiments

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Case-1</th>
<th>Case-2</th>
<th>Case-3</th>
<th>Case-4</th>
<th>Case-5</th>
<th>Case-6</th>
<th>Case-7</th>
<th>Case-8</th>
<th>Case-9</th>
<th>Case-10</th>
</tr>
</thead>
<tbody>
<tr>
<td>G5.9-</td>
<td>G5.9-</td>
<td>G5.9-</td>
<td>G5.9-</td>
<td>G5.9-</td>
<td>G5.9-</td>
<td>G5.9-</td>
<td>G5.9-</td>
<td>G5.9-</td>
<td>G5.9-</td>
<td>G5.9-</td>
</tr>
<tr>
<td>S3.2-</td>
<td>S3.2-</td>
<td>S3.2-</td>
<td>S3.2-</td>
<td>S3.2-</td>
<td>S3.2-</td>
<td>S3.2-</td>
<td>S3.2-</td>
<td>S3.2-</td>
<td>S3.2-</td>
<td>S3.2-</td>
</tr>
<tr>
<td>T960</td>
<td>T960</td>
<td>T971</td>
<td>T974</td>
<td>T969</td>
<td>T991</td>
<td>T999</td>
<td>T975</td>
<td>T975</td>
<td>T1005</td>
<td>T1040</td>
</tr>
</tbody>
</table>

Table 2 Operating conditions of conducted tests. The cases are named in a way to present the main operating conditions of the case (e.g., in Case-1 G5.9-S3.2-T960, gas flow rate, solid flow rate, and gas inlet temperature are 5.9 kg/s, 3.2 kg/s, and 960 K respectively).
contains approximately 5.5 [wt.%] of moisture. In the preliminary simulations, the heating up and eventually evaporation of the moisture was included in the simulation using the wet combustion model available in Ansys Fluent 2021R1. As presented in Fig. 4 obtained from the preliminary simulation, the water content is immediately evaporated when the particles and hot gas get in contact, such that particles entering the cyclone are totally dry.

Therefore, to keep the model simple and applicable for all common cyclone preheaters operating in industries (which might not have evaporation in general), and to keep the model computationally cheap, water evaporation modeling was excluded from the calculation for the rest of the simulations. This is done by assuming that drying has already happened before particle injection. The energy required for the evaporation of 5.5 [wt.%] of moisture is around 25% of hot gas energy loss. As evaporation is not included in the final CFD model, the gas inlet temperature and solid flow rate are reconciled to compensate for the energy loss in the simulations.

This deduction appears in terms of reduction in the gas temperature at the gas inlet (the temperature reported in Table 2 as operating conditions are reduced temperatures. Also, this 5.5 [wt.%] of moisture is deducted from the solid flow and added to the gas flow). In addition, based on the overall energy balance of the system it can be observed that 7 [wt.%] of the gas flow rate reported in Table 2, is false air with ambient temperature entering the system through the gaps in the system. The effect of the cold false air, which is around 10% of gas energy loss, is also considered by modifying the gas inlet temperature reported in Table 2. It should be noted that these 5.5 [wt.%] of water content in particles and 7 [wt.%] of cold false air are uncertain and approximate assumptions. It is worth mentioning that based on both the operating data and also the CFD model in its final version used in the present study, the energy balance in the system is guaranteed.

4. Model validation, results and discussion

4.1. The necessity to have one solid phase for each particle size class

Although in Eulerian-Eulerian multiphase modeling approaches, having one solid phase for each particle size is common, in DDPM which is an Eulerian-Lagrangian approach, it was expected to provide a verified solution with only one solid phase for the whole PSD as it did in isothermal cases [29,33,34,48]. However, preliminary simulations showed that in the way the energy conservation equation and gas-particle heat exchange is implemented in the DDPM multiphase model in Ansys Fluent 2021R1, in cases with poly-dispersed particles, to have energy balance in the system (in non-isothermal cases), a specific procedure must be implemented. The procedure is to have one exclusive solid phase assigned for each particle size class present in the system. For example, if four particle sizes of 200, 250, 750, and 3000 µm are present in the system, four solid/dispersed phases are needed each to be assigned to one of the size classes to have an overall closed energy balance in the system. While if only one solid phase is used, a huge energy imbalance will be observed as shown in Fig. 5.

Such an imbalance seems to be an error in Ansys Fluent which has not been resolved up to at least version 2021R1. Due to limited documentation on heat transfer modeling in DDPM, it is unclear what is causing such an imbalance in the system. However, the suggested procedure can overcome the issue. It should be noted that without such a procedure the imbalance is large enough to make the predictions unreliable.

4.2. The effect of polydispersity of particles

To achieve an adequate energy balance, one exclusive solid phase is needed for each particle class included in the model in DDPM, as discussed. On the other hand, a greater number of solid phases results in heavier and more expensive numerical calculations. Therefore, it is important to find the minimum number of particle sizes required to adequately represent the actual PSD in the studied case. For this purpose, a study was designed where a various number of particle sizes were considered in different cases (from 1 to 7 particle sizes), all cases having the same Sauter mean diameter as the actual PSD (375 µm), to be compared and find the minimum number of particle sizes needed to provide similar results as the case with 7 particles sizes, which is the maximum number feasible in our computational resources. The 7 cases are summarized in Table 4.

Pressure drop and gas exit temperature predictions were taken as two

![Fig. 4. A snapshot of particles colored with moisture content obtained from preliminary simulation using wet combustion modeling.](image-url)
performance parameters of the cyclone preheater. The comparative results are presented in Fig. 6. In addition, the results of velocity and temperature profiles are also shown in Fig. 7. The comparison shows that even with one particle size (Polydispersity-Case-1), more or less, similar predictions can be achieved (in particular for pressure drop, gas exit temperature and velocity profiles). However, four particle sizes should be included (Polydispersity-Case-4) to minimize the difference in predictions from the most expensive case (Polydispersity-Case-7), in particular considering temperature distributions. This means that four particle sizes are sufficient in this study to cover the actual PSD. Therefore, for the rest of the simulations, 4 particle sizes were used (Polydispersity-Case-4).

As can be observed from Table 4, in the presented investigation, to keep the sauter mean diameter constant in all cases, even distribution of mass fraction is used (as the idea was to keep the sauter mean constant in all cases on the mimic the actual PSD). However, a few cases with uneven distributions were investigated as well showing that also with uneven distributions, as far as the sauter mean diameter is kept constant, the predictions do not deviate notably.

4.3. Mesh, parcel size, and time step size independence

A mesh independence study was carried out for Case-1, using five levels of mesh resolution (polyhedral mesh with prism layers). Some information about these five levels of mesh, including the average cell volume, the grid resolution, and the grid-to-parcel size ratio for each level are presented in Supplementary Material.

A comparison of predicted pressure drop and gas exit temperature with different levels of grid resolution is presented in Fig. 8. The results show that the predictions did not deviate substantially from the prediction of the finest mesh, even for mesh level 5. Furthermore, the comparison of flow pattern and gas temperature distributions in the five levels of mesh is presented in Supplementary Material. The grid with 0.8 million computation cells is used for the rest of simulations. If coarser grids (0.5 and 0.6 million cells) are used minor changes, in particular, in temperature distributions can be observed.

In addition, the Supplementary material contains convergence calculations for the three finest grid levels using the approach suggested by Roache [49,50] to analyze grid convergence and grid resolution error [51].

The coarse-graining strategy used in this study was to have the same parcel size (same parcel mass) for different particle sizes. According to preliminary simulations, the presence of too large parcels leads to variations in heat transfer calculations (e.g., gas exit temperature) and a loss of overall energy balance and therefore must be avoided. So, a study was carried out to find the largest parcel size (fastest calculation)

<table>
<thead>
<tr>
<th>Table 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Summary of seven simulation cases designed for the study of the effect of particle polydispersity.</td>
</tr>
<tr>
<td>Case name</td>
</tr>
<tr>
<td>---</td>
</tr>
<tr>
<td>Polydispersity-Case-1</td>
</tr>
<tr>
<td>Polydispersity-Case-2</td>
</tr>
<tr>
<td>Polydispersity-Case-3</td>
</tr>
<tr>
<td>Polydispersity-Case-4</td>
</tr>
<tr>
<td>Polydispersity-Case-5</td>
</tr>
<tr>
<td>Polydispersity-Case-6</td>
</tr>
<tr>
<td>Polydispersity-Case-7</td>
</tr>
</tbody>
</table>

Fig. 6. Comparison of (a) pressure drop, and (b) gas exit temperature in seven simulation cases designed for the study of the effect of particle polydispersity. The relative computation speed is also presented for each case. Presented deviations in % are based on a comparison with case 7 particle sizes (Polydispersity-Case-7).
providing overall energy balance (up to 1% energy imbalance is considered acceptable) and parcel-independent predictions which are presented in the Supplementary Material.

In addition, a time-step independence study was done; Six Courant numbers of 100, 200, 300, 400, 800, and 1600 equivalent to 1, 2, 3, 4, 8, and 16 ms respectively were examined. It was demonstrated that the results were independent of time-step size over the investigated range.

The findings of this study were obtained from the parcel size of 5 mg and the courant number of 300 resulting in around 3 million parcels in the domain and a time-step size of $3 \times 10^{-3}$ s upon achieving a steady-
state solution. An Intel(R) Xeon(R) Gold 6226R with 32 processors at 2.90 GHz and with 64 GB of RAM were used with a high-speed Infini-
Band network connection to perform simulations. Using the solution-
independent mesh, parcel size, and time step, it took <2 h to progress
1 s of simulation time. This corresponds to no more than 2.5 days to
reach a steady-state solution after around 20–30 s of simulation time.
Given available computing resources, the developed model can be easily
applied and is affordable for industrial-scale cyclone modeling.

The developed model provides stable solutions that produce
reasonable predictions of particle motion, flow pattern, pressure drop,
and heat transfer in fairly good agreement with experimental mea-
surements that will be discussed in detail in the following sections. It is
worth mentioning that the complexities of an actual industrial cyclone
preheater such as potential internal buildups (changing the geometry),
oscillating operating conditions, etc. make this validation more
challenging.

4.4. Overview of the multiphase flow

For one of the simulation cases, an overview of the simulation results
in the cyclone preheater is shown in Fig. 9, presenting average contours
of gas tangential and axial velocities, gas temperature, solid volume
fraction and pressure, in addition to velocity vectors. The predicted flow
characteristics are consistent with flow characteristics expected in cy-
clones, i.e., upward flow in the inner vortex (swirling up), downward
flow in the outer vortex (swirling down), and high concentration of
particles near the wall (Fig. 9 (d)) due to the centrifugal force acting on
the particles because of swirling motion. Fig. 9 (a, f) illustrates the
swirling motion expected in cyclones with a radial distribution of
tangential velocity, i.e., the swirling flow consists of an outer free-vortex
(also known as near loss-free vortex) and a forced vortex (also known as
near solid body rotation) at the core as expected from cyclones.

Fig. 9 (b, f) highlights descending flow near the walls carrying par-
ticles concentrated in the near-wall region towards the dust bin and also

![Fig. 9. Mean (averaged over 5 s after achieving steady solution) flow pattern contours in case-1 (G5.9-S3-T960): tangential velocity (a), axial velocity (b), gas
temperature (c), solid volume fraction (d), and static pressure (e), in addition to velocity vectors (f). The gas temperature distribution (c) is presented with two
different legends; one to show temperature changes in the cyclone main body and the other to show temperature changes in the riser.]
ascending flow far from the walls, around the center as expected in cyclones. A pressure drop is observed when approaching from the wall to the center (Fig. 9 (e)), which is a characteristic of swirling motion. Regarding the gas temperature profile in the cyclone, approaching from the wall towards the center, first, a slight increase is observed and then the temperature falls down, reaching its minimum at around the center (Fig. 9 (c)) that is in line with expectations in the cyclone preheaters [52]. Fig. 9 (c) shows that the gas temperature changes in the riser are way greater than in the cyclone main body meaning that the heat transfer is mostly occurring in the riser, and the main task of the cyclone is particle separation rather than heat transfer. As the gas-particle heat exchange occurs mostly in the riser, variation of gas temperature in the cyclone main body is minor such that gas and particles reach the cyclone at temperatures relatively close to the equilibrium temperature. On the other hand, a significant variation in gas temperature is observed in the riser where gas-particle heat exchange is happening.

In the following sections, the validity of the model in terms of prediction of pressure drop, heat transfer (as two important performance parameters of cyclone preheaters), flow pattern and temperature profile will be presented. It is worth mentioning that considering the relatively coarse size of injected particles, the separation efficiency as another important performance parameter of cyclone preheaters is reportedly 100% under all operating conditions, and the present CFD model predicts the same.

4.5. Pressure drop

A comparison between the model predictions and the experimental results, for all cases, is shown in Fig. 10. The predicted overall pressure drop is in acceptable agreement with experimental data, with the predicted values deviating from experimental measurements in the range of 16–29%. The deviation does not seem to be due to inappropriate modeling of turbulence and drag, as the same model provided better predictions in our previous investigation of a large-scale highly loaded cyclone [33,34]. However, this general underprediction of the pressure drop might be explained by the fact that the simulated geometry presented in Fig. 1, to some level, might be different from the actual geometry; this deviation might be caused by having an unaligned lining of refractory bricks, changes due to wall erosion, etc.

Considering all cases, the model is accurate enough to capture the trends observed in the measurements, i.e., the reduction/increase in the pressure drop when the operating conditions are changed. For example, with case-6 and case-8 having identical gas flow rates, measurements showed a slight reduction in pressure drop as the solid load increased. This reduction is due to the dampening of swirling motion when the solid load is increased [53], and simulations indicated the same (from case-6 to case-8, measurements showed a 3.5% reduction in the pressure drop and CFD simulation predicted a 3.6% reduction). Also, as the gas flow rate is increased comparing cases-5, case-6, and case-7 which have identical solid flow rates, measurements and simulations both exhibit a considerable increase in pressure drop. Gas flowing at a higher rate results in a stronger swirl and more dissipation of flow energy due to higher friction at the walls leading to a greater pressure drop [53]. More importantly, the pressure in the central region of the cyclone preheater decreases as the gas flow rate increases which results in lower pressure in the gas outlet pipe, and consequently greater pressure drop. Overall, experimental measurements indicated that the overall pressure drop responded more to gas flow rate rather than solid flow rate, and this was also observed in the CFD simulations.

4.6. Heat transfer

In evaluating the model performance for heat transfer in the cyclone preheater, the gas exit temperature and heat transfer rate (calculated based on particle temperature changes, \( Q = \dot{m}_p \int_{T_{p,\text{in}}}^{T_{p,\text{out}}} C_p \,dT \)) are used as the indicator and compared with the experimental data at the applied operating conditions. The results are presented in Fig. 11 and Fig. 12. From Fig. 11 and Fig. 12, it can be seen that the model is able to simulate heat transfer quite accurately since the predicted values match well with the experimental measurements with deviations not greater than 1.1%
and 9% in terms of the gas exit temperature and heat transfer rate, respectively. The observed systematic underprediction of heat transfer rate can be explained, to some extent, by uncertainty and inaccuracy in assumptions for the amount of water content of particles (5.5 [wt.%]) and the amount of false air (7 [wt.%]).

Similar to the pressure drop predictions, the simulation of heat transfer appears to be sufficiently accurate in reflecting the trends observed in the measurements. According to simulation results, in each pair of the two cases, the temperature at the gas exit and the heat transfer rate are predicted to change in a similar manner as observed in the measurements.

Comparing case-6 and case-8, which have identical gas flow rates, measurements showed that the heat transfer rate increased as the solid load increased, and simulations pointed in the same direction as well. Furthermore, both measurements and simulations show an increase in heat transfer rate when the rate of gas flow increases (comparing cases-5, case-6, and case-7 which have identical solid flow rates). In comparison with a general heat exchanger, both observations are intuitive (an increase in the flow rate of each medium leads to an increase in the heat transfer rate).

The heating up of particles in the cyclone preheater can be seen in snapshots of particles colored with their temperature presented for each
particle size in the system separately in Fig. 13 (a). As can be observed, gas-particle heat exchange is mostly occurring in the riser, in particular for fine particles, and gas and particles enter the cyclone with temperatures close to the equilibrium temperature. A higher heat transfer rate is observed in the riser where the gas-particle temperature difference is larger. As expected, large particles (3000 µm) are heated up more slowly because of their smaller specific surface area, and lower heat transfer coefficient (based on equation (2), the larger particle, the lower the heat transfer coefficient), such that they leave the cyclone with considerably lower temperature.

Snapshots of particles colored with the Nusselt number are shown in Fig. 13(b) for each of the particle sizes within the system, in separate images. The larger the particle, the greater the Nusselt number as expected from equation (2). Particles experience the highest heat transfer coefficient where they enter the cyclone from the riser (upper part of cyclone barrel) where they experience the highest Reynolds number as a swirling motion is made in the flow. Meanwhile, in the conical part of the cyclone, the lowest heat transfer coefficient can be observed.

In general, gas-particle heat exchange consists of two stages: (1) the transfer of the heat from the bulk of the gas to the surface of the particle through the gas film adjacent to the particle (an extrinsic process) and (2) the propagation of heat by conduction inside the particle to reach isothermal condition within the particle (an internal process). In cases with $Bi < 2$, the heat exchange is controlled by the external process (a gas-

![Fig. 13. Snapshots of a converged solution of case-1 showing particles colored with their (a) temperature, (b) Nusselt number, (c) Biot number.](image-url)
film-controlled process) [39]. In the present study, to model gas-particle heat exchange, a gas-film controlled process (temperature of the particles are assumed uniform and dependent on time alone) is assumed and snapshots of particles colored with their Biot number (Bi) presented in Fig. 14(c) validate the assumption as Bi < 1 for all particles in the system.

4.7. Flow pattern

The axial and tangential velocity profiles obtained from LDA measurements and CFD simulations are presented in Fig. 14(a, b) for three relatively similar operating conditions (Case-1 G5.9-S3.2-T960, Case-2 G6.1-S3.1-T971, Case-9 G5.8-S3.3-T1035). Despite there being some minor differences between the profiles obtained from LDA and CFD simulations, the agreement is satisfactory (with a maximum and average deviation of 23% and 10% respectively for tangential velocity and 38% and 22% respectively for axial velocity), considering the complexity of industrial-scale measurements and CFD simulations. Overall, as can be observed in the figure, the CFD model has a good ability to capture the experimentally determined profiles both in the core and in the near-wall regions. The swirl intensity, the location, and the extent of peaks in axial and tangential velocity profiles, the widths of the vortex cores (radial distance between the peaks of tangential velocity), and the location of the boundary between swirling up and swirling down vortices (with approximately 15% error based on the distance from the center) are captured by the CFD model.

To see whether the model is able to capture the effect of operating conditions on the flow pattern, LDA measurements were performed under two relatively different conditions (Case-3 G6.6-S2.9-T974). The velocity magnitude (\( \sqrt{V_\theta^2 + V_z^2} \)) obtained from measurements and CFD simulations at these conditions is compared with those of Case-1 G5.9-S3.2-T960 in Fig. 14(c). As can be observed, the model is able to capture the profile of velocity magnitude. In addition, the model is able to capture the shift in the flow pattern (increase in velocity magnitude) when the gas flow rate is increased considerably from case-1 to case-3.

4.8. Temperature pattern

To understand the temperature profile in a cyclone preheater, the measurements by using a thermocouple and FTIR were conducted at two operating conditions (Case-4 G7.0-S2.9-T975, Case-10 G5.3-S3.2-T1040). A comparison is shown in Fig. 15 of the temperature profiles.
though the location of the maxima is not precisely agreed in the simulation, incorporating gas-particle heat exchange. For the simulation of the cyclone, first swirls down (close to the wall) to the bottom of the cyclone and then from there swirls up in the central region up to the vortex wall towards the center) also observed in other studies [52] can be observed first, and then it declines, reaching a minimum in the center. A similar trend is observed from all measurements and CFD simulations, though the location of the maxima is not precisely agreed in the simulations and the measurements.

The observed trend (reduction in the temperature moving from the wall towards the center) also observed in other studies [52] can be explained by how the gas flows in a cyclone. The gas flow entering a cyclone first swirls down (close to the wall) to the bottom of the cyclone and then from there swirls up in the central region up to the vortex finder and gas exit. This means that at a specific height, the gas flow in the central region has been in the cyclone and in contact with particles for a longer time compared to the gas flow close to the wall. So, the gas flow in the central region has had a longer time to exchange heat with particles and therefore it has a relatively lower temperature. The decrease in gas temperature as approaching the wall in the region very close to the wall might be explained by the increases in particle concentration as approaching the wall (the more particle, the more heat exchange, and the lower gas temperature).

5. Conclusions

Industrial cyclone preheaters with high solid load have been simulated, by using a hybrid multiphase model, Dense Discrete Phase Model (DDPM), incorporating gas-particle heat exchange. For the simulation of the process, with a dispersed version of the k-ω SST turbulence model that is sensitive to flow rotation. The CFD model is validated by the data from an industrial cyclone preheater operating under various conditions. The validation of the model, in addition to the pressure drop and gas and particle exit temperature as overall measurements, successful local measurements of gas velocity profiles, using the LDA, and gas temperature profiles, using thermocouple and FTIR were conducted. The measurements and validation were conducted on a full-scale industrial cyclone preheater that is part of a factory producing isolation material.

The ability and accuracy of the developed CFD model in describing relevant phenomena present in such a device were demonstrated and tested with detailed measurements. Furthermore, the following conclusions can be stated from this study:

- An exclusive solid phase must be attributed to each particle size class present in the system in order to achieve energy balance in the system when using the DDPM model in Ansys Fluent 2021R1. Furthermore, it was concluded that the PSD in the present study can be adequately represented with four particle size classes (and consequently four solid phases) included in the model.
- The heat transfer calculations and energy balance in the system are sensitive to parcel size in a certain range. A sensitivity analysis must be performed to determine the largest parcel size and fastest calculation that will provide an overall energy balance and parcel-independent predictions. In the present case, this parcel size turned out to be a parcel with a mass of 5 mg.
- The presented model is sufficiently accurate to reflect the trends in performance parameters and flow patterns observed in the measurements when operating conditions are altered. For example, the reduction in pressure drop and the increase in total heat transfer rate when the solid load is increased in addition to the increase in the pressure drop and total heat transfer rate when the gas flow rate is increased are well predicted by the presented model.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

Data will be made available on request.

Acknowledgement

This work was supported by the ProBu project: Process Technology for Sustainable Building Materials Production (grant number: 8055-000148) funded by Innovation Fund Denmark, as well as FLSmidth A/S, Rockwool A/S and Technical University of Denmark.

Appendix A. Supplementary data

Supplementary data to this article can be found online at https://doi.org/10.1016/jcej.2023.14757.

References


