Fuel 189 (2017) 120–130

Contents lists available at ScienceDirect

Fuel

journal homepage: www.elsevier.com/locate/fuel

Full Length Article

The contribution of differently-sized ash particles to the fouling trends of a pilot-scale coal-fired combustor with an ash deposition CFD model

Manuel García Pérez *, Esa Vakkilainen, Timo Hyppänen

Lappeenranta University of Technology, Energy Technology, P.O. Box 20, FIN-53851 Lappeenranta, Finland

HIGHLIGHTS

- A CFD model for ash fouling with a newer drag law for fine particles is presented.
- A whole heat exchanger is modeled with ash particles ranging from 0.015 μm to 15 μm.
- The gas velocity seemed more important for the deposition than the tube deposits.
- The behavior of particles below 4 μm seemed to be identical for each studied case.
- This suggests a limiting particle diameter for the inertial impaction.

ARTICLE INFO

Article history:
Received 20 August 2016
Received in revised form 17 October 2016
Accepted 18 October 2016
Available online 25 October 2016

Keywords:
Fouling
Coal combustion
Ash deposition
Computational fluid dynamics
Boilers

ABSTRACT

This work is concerned with the research field of ash deposition phenomena. An unsteady CFD model of the convective section of a 100-kW PC pilot-scale combustor is implemented aiming to simulate and study the ash deposition trends. Emphasis is placed at analyzing how particles of different sizes may contribute differently to the deposit buildup. A discrete particle tracking and a rebound–stick submodel are used. Four simulation cases are implemented to investigate the effect of the inlet flow velocity and the fouling conditions (clean vs. fouled tubes).

The sticking efficiencies decreased with the particle diameter d_p after being constant and higher than 95% for $d_p < 1$ μm. It also seemed to decrease with the flue gas inlet velocity. The normalized particle arrival rates to the tubes were rather constant up to particle sizes of $d_p = 8$ μm. The studied cases of faster flow inlet velocity showed higher values for both the deposition and arrival trends as a consequence of a higher overall particle Stokes' number and the total ash mass input to the model. The cases with fouled tubes presented higher deposition rates (approximately, 7.91% more for the fast velocity case) than what is observed for clean tubes.

1. Introduction

The fouling and slagging of heat transfer surface typically represent the major challenge in boiler design and operation. The boiler performance becomes compromised entailing significant economic losses [1]. Consequently, ash-related issues are of vital importance [2–4] for these industrial units establishing a motivation for active research to tackle the complicated multidisciplinary problem of the deposits.

The ash accumulation on tube surfaces leads to the growth of coating deposits which represent a resistance to the heat transfer and consequently to the overall boiler performance. Different models and predictive tools of diverse kinds are of high interest for these phenomena. Particularly, numerous researchers utilize CFD simulations of specific areas of industrial furnaces in an attempt to understand and explain slagging and fouling implications from a rich variety of points of view and with multiple applications. Vuthaluru and Vuthaluru [5] attempted to optimize the locations of the secondary air injection ports regarding an efficient combustion while maintaining reasonable slag trends. Pyykönen et al. [6] modeled the ash fume deposition rates regarding potential corrosion issues in platen heat exchangers. Leppänen et al. [7] investigated the alkali vapor condensation and formation of fume ash particles from their origin by using an extensive computational domain comprising the furnace and the superheater region. These studies of Pyykönen et al. and Leppänen et al. were concerned with kraft recovery boiler applications. Han et al. [8] modeled different tube array arrangements and considered different tube shapes (e.g., elliptical tubes) regarding the minimization of the ash depos-

* Corresponding author.
E-mail address: manuel.garcia.perez@lut.fi (M. García Pérez).

http://dx.doi.org/10.1016/j.fuel.2016.10.090
0016-2361/

its. Balakrishnan et al. [9] implemented a mechanistic sticking model for fully or partially-molten ash particles to predict the growth of the slag deposits in the lower furnace of a pulverized-coal boiler. Beckmann et al. [10] addressed different sticking-rebound approaches for CFD contrasting their results with experimental measurements. Tong et al. [11] included in his CFD models an ash-removal approach and used both the lattice Boltzmann method and the finite volume method.

Nevertheless, it was stated [3] that the ash deposition modeling may be still in a somewhat early stage for predictions to be fully reliable due to their complicated and multidisciplinary nature. This is a consequence of the often needed assumptions and simplifications [12] which are required to tackle the complex and numerically demanding ash deposition issues. In addition, the uncertainties derived from measuring and obtaining field data (ash particle chemical composition and properties, diameters, velocities, temperatures, mass flows) makes it challenging to establish reliable model parameters. In the same line, the sticking behavior of the ash particles is still not well fixed understood and determined, resulting into a variety of approaches [10]. Other issues arise when try to model this complex reality. Due to these aspects, simplifications need to be assumed and analyzed carefully and critically.

Especially when modeling tube arrays, it is essential to perform an unsteady treatment of the flow since the flow experiences periodical oscillating patterns [13,14]. These unsteady flow patterns affect considerably the trajectories of the smaller ash particles [12]. In addition, a relatively small amount of works [15] have been concerned so far with the deposition trends on already ash-coated surfaces. Most of previous referenced works were concerned with the deposits on fresh, clean tube surfaces. Moreover, most of ash deposition CFD models have considered the deposits on a single tube, or on a row of a few tubes at most although it is known that the deposition trends observed on a probe differ from those observed in actual heat exchangers [16]. This limitation is usually a consequence of the very high computational costs these numerical models demand.

These mentioned limiting aspects noted in the current state-of-the-art of ash deposition modeling motivated the present study. This work presents a new unsteady CFD model of the deposition mechanisms and trends which could be observed in a whole tube array. The target of this model will be the water-cooled exchanger of a pilot-scale combustor burning a subbituminous Wyoming coal [17]. Different combinations of the conditions for the gas upstream velocities and deposit thicknesses are simulated in this work. These conditions have been selected in order to perform a reasonable case study set and do not match with all the conditions of the empirical measurements at once and, moreover, the deposition rates on the target heat exchanger were not provided in the study [17]. Consequently, there is no empirical data in [17] on the ash deposition trends of the heat exchanger to validate this CFD model. A previous ash deposition CFD approach [12] is adapted here to implement the firing and flow properties (temperature, geometry, materials) of the aforementioned combustor operation. The solver is unsteady, allowing thus to account for the particular time-dependent oscillating flow patterns past tube arrays. The two-dimensional solver is enhanced with a Lagrangian particle tracking scheme and a sticking-rebound mechanistic submodel [18–21] which is explained further in the model description. In addition, this tool accounts for accuracy guidelines on grid resolution for proper flow solving suggested by Weber et al. [22].

The model is used with the aim to explain fouling phenomena; focusing on the deposition rates as a function of the incoming flow velocity u_∞. Two different values of this upstream velocity are tested (1.1 m/s and 1.8 m/s), corresponding to the minimum and maximum gas velocities in the field study [17]. In addition, the effect of the fouling itself on the ash deposition trends is studied by executing different simulations with two different grids: one with clean tubes and a different one with layers of deposits coating those tubes. Emphasis is placed on the behavior of different particles as a function of their diameter.

This work is, naturally, not exempt of certain limitations. This model does not consider particle-particle interaction (collisions), or the deposit removal phenomena (modeled, as mentioned, by Tong et al. [11]). Due to the fine domain meshing required and the short time-step considered, the selected two-dimensional yielded results after about a month of calculations. Thus, increasing the model complexity, expanding the domain to a larger heat exchanger, or attempting a three-dimensional approach, would have been prohibitive. In addition, ash deposition models are hard to validate since, as it has been stated, relevant uncertainties arise while trying to obtain empirical values to contrast computed model results. However, due to the increasing computing resources and available approaches and tools, there exists an increasing reliability of these predictive tools.

2. Model description

The heat exchanger under study has five staggered plates, each one consisting of a pipe performing four passes. The diameter of all the pipes is $D = 1.27$ cm. In between plates, there is a transversal separation of 2.54 cm ($2 \cdot D$); with a longitudinal staggering offset of 4.44 cm ($3.5 \cdot D$).

Regarding the notation and nomenclature in this work, the *first* tube (or pass) is the tube located most upstream of a given plate, and the remaining tubes are numbered successively according to the flow path. The plates are named so that the two plates surrounding immediately the *central* plate are the *interior* plates. The two outermost plates are referred to as *edge plates*.

The heat exchanger tubes are submerged in a long horizontal cavity with a square section of 15.24 cm side. A two-dimensional longitudinal cross section of this cavity is modeled in this work. Therefore, the model domain consists of a rectangle with twenty staggered circular holes representing the tubes.

The model simulates the operating conditions of the specific coal tests [17]. The upstream flue gas temperature is 490 °C. The outlet boundary condition is set at 89.3 kPa. The tube wall surfaces are set as 29 °C (for the cases with fouled tubes, this corresponds to the inner temperature). The flue gas is modeled as an ideal gas with the typical properties of combustion gases at the aforementioned temperature and pressure [23]: viscosity $\mu = 3.114 \cdot 10^{-5}$ kg/(m s), thermal conductivity 0.0506 W/(m K), specific heat 1239 J/(kg K) and molecular weight 28.38 kg/kmol.

The selected physical properties of the deposits (for the cases with fouled tubes) are: density 795 kg/m^3, thermal conductivity 0.14 W/(m K), and specific heat 985 J/(kg K); based on the measured ash composition [17] and on the dependence of the thermal conductivity on the porosity [2,24].

2.1. Computational domain and meshes

Two separate grids have been generated: one grid for the clean tube cases and a second grid with fouled tubes. Fig. 1 shows the meshed computational domain for the clean case. Fig. 2 shows the meshing of the tube deposits for the fouled case. The top and bottom edges of the domains are adiabatic walls. The velocity inlet boundary is the left side of the rectangle, at $5 \cdot D$ upstream of the center of the first tubes of the central and edge plates. The pressure outlet boundary is located at $7 \cdot D$ cm downstream of the center of the last tubes of the interior plates. The total height of the rectangle is 15.24 cm ($12 \cdot D$).