Fuel xxx (2015) xxx–xxx

Contents lists available at ScienceDirect

Fuel

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

2D dynamic mesh model for deposit shape prediction in boiler banks of recovery boilers with different tube spacing arrangements

Manuel García Pérez *, Esa Vakkilainen, Timo Hyppänen

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

HIGHLIGHTS

- A new CFD model for deposit shape prediction is proposed.
- This model was tested for different geometries of a boiler bank in a Kraft Recovery Boiler.
- A dynamic mesh deforms the mesh, and the deposit grows according to the deposition.
- Results highlight deposition rates, shapes, and heat transfer penalties.

ARTICLE INFO

Article history:
Received 10 November 2014
Received in revised form 18 March 2015
Accepted 28 April 2015
Available online xxxx

Keywords:
Kraft Recovery Boiler
Fume
Deposition
Dynamic mesh
Black liquor

ABSTRACT

CFD tools are of great value in the design and operation of boilers. One particular aspect that can be modeled by CFD is the ash deposition and plugging of heat transfer surfaces of boilers. Fouling and slagging are the most typical causes of unscheduled boiler shutdowns. Consequently, appropriate predictions of deposit geometries and rates are of considerable interest. CFD multiphase flow simulations are capable of modeling particle-laden streams and constitute a suitable tool for study of material deposition.

Fouling phenomena have a complicated and multidisciplinary nature involving thermo-fluid mechanics, sticking/rebounding of particles, sintering, etc. If the deposit growth rate has been accurately calculated by a multiphase model appropriate for particle-laden flows, CFD dynamic mesh techniques are able to move the interphase fluid-deposit according to the growth rate. This work develops and presents a CFD model for prediction of deposition shapes in a classical boiler bank of a Kraft Recovery Boiler by combining a multiphase and a dynamic mesh model. The effect of tube transversal spacing is also analyzed. The paper highlights the features and effects of the dynamic mesh model.

1. Introduction

Since their invention in 1934, recovery boilers have operated in the pulp and paper mill industry. As a part of the Kraft pulping process, they allow the recovery of inorganic cooking compounds which are residuals from the fiber extraction process, for further cyclical reutilization. As a secondary function, they also generate steam for the mill. Since the black liquor is not a typical fuel and there exist two main desired outputs (energy and reduction) from the boiler, their operation, modeling and design are usually more difficult than for other boiler applications [1]. In addition, the fact that black liquor is a very ashy fuel makes the operation of Kraft Recovery Boilers (KRBs) very challenging.

Modeling of boilers is a topic of considerable interest within both academia and industry. CFD provides a powerful tool to numerically solve the complex Navier–Stokes partial differential equations. By using CFD it is possible to resolve the complex phenomena in boilers such as fluid flow, turbulence, heat transfer, chemical reactions and combustion, transport of mass/particles, agglomeration, deposition, fouling, erosion, and pollutant formation [2–5].

A tool capable of modeling accurately and simultaneously all the involved phenomena is not yet available as this is a multidisciplinary problem which entails really complex formulations, and also because the domain itself (i.e., the target region to be modeled) is of high complexity. Even with a narrow domain it may happen that a particular problem is still hard to handle, e.g., an analysis of chemistry should take into account hundreds of reactions among different phases of reactants, including phase changes. As a consequence, CFD tools often try to model a specific

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

http://dx.doi.org/10.1016/j.fuel.2015.04.074

part of the boiler and solve only a limited set of the phenomena. Weber et al. [2] analyzed the state-of-the-art of CFD usage for ash behavior prediction. In their work, it was noted that CFD tools are not widely used for boiler operation analysis and that many results are still considered merely indicative. It was furthermore observed that little work has been done on models that include accurately the effect of flow patterns over tube arrays. For instance, some models fail to fulfill the resolution requirements necessary for accurate prediction of particle motion. It was also highlighted that not much work has been done on models that consider the flow over tube arrays, rather than a single tube. In addition, transient studies are often omitted, which means that the effects of Von Kármán vortex shedding and the Coanda effect are lost. These effects lead to a swinging fashion motion of the flow over tube arrays [4].

The model presented here simulates and calculates fume deposition on tubes of a boiler bank of a KRB by means of dynamic meshes. Special consideration is given to resolving the flow pattern around a tube array consisting of periodical repetitions in the transversal spacing of a row of 4 tubes. In order to illustrate the effect of different flow patterns, different transversal tube spacings are simulated. A discrete phase model is used to simulate the ash particles in the flow. The deposition rate in the tube surfaces is computed, and the deposit subdomain grows accordingly.

1.1. Study cases

Fouling is a major concern in KRB operation, especially in the boiler banks (Fig. 1) [1,5]. This work presents CFD calculations of fume deposit growth simulating the conditions in these heat exchangers. A parametric analysis with 8 different values for the transversal spacing between tube rows, s_t, is carried out. Each case presents a computational domain consisting of a row of 4 tubes, with a typical boiler bank tube diameter of $D = 50$ mm. The longitudinal pitch remains constant at $3 \cdot D = 150$ mm between the centers of the tubes. Translational periodic boundary conditions on the sides of the domain ensure an array configuration. Symmetry boundary conditions should not be applied since the flow pattern is not symmetrical although the geometry is. The transversal pitch s_t corresponds to the width of the domain. The parametric analysis of the transversal spacing consists of 8 different cases, varying the spacing from 1.25 to 3.00 times the tube diameter, in steps of $0.25 \cdot D$.

A discrete phase model is used to simulate the ash particles in the flow. The deposition rate in the tube surfaces is computed, and the dynamic mesh model deforms the mesh to make the deposits grow according to the deposition rates. Transient considerations are highlighted. Ansys FLUENT enhanced with User-Defined Functions is used in this work.

2. Model description

2.1. Discrete phase model

A Lagrangian approach is used to track the particles. This kind of model (also called the Discrete Phase Model) first solves the continuous phase (gas) alone and then models the discrete phase (particulate). The discrete parcels of particles are uniformly distributed and injected through the inlet boundary condition at each time-step. The number and mass of parcels is automatically set to match the fume concentration (6 g/m^3, as discussed in a later section). Each parcel is tracked separately. It is possible to neglect the particulate phase influence to the continuous phase, or consider coupling among phase calculations. Since the mass and volume load of fume in gas are very low, the coupling is not necessary here. The model has been shown to work well for flows where the volume fraction of particles is below 10% [6]. Higher volume fractions would lead to a too strong coupling between the dilute and continuous phase motions.

Tomeczek et al. [7] created a somewhat similar model for shape prediction in Fluent. Even though they performed a steady state model, they showed that the mass can be computed in FLUENT with the user-defined function DEFINE_DPM_EROSION. In this model, that routine was coded to add up the mass of each particle which sticks to a face.

2.2. Fume stream motion

A carryover-free stream is simulated. Due to inertial impaction, only the windward edges of the leading tubes would be prone to carryover deposits. This model focuses on fume deposition and its mechanisms.

Since fume is to be analyzed, special considerations must be taken. The motion of very small particles is significantly affected by flow drag. Therefore, inertial impaction (i.e., impaction due to the flow being unable to drag the particle away as it avoids the windward side of a tube) is not expected to be very significant. On the other hand, thermophoresis and Brownian motion constitute the major mechanisms of deposition for sub-micron particles [1,5]. These two phenomena are considered in this model by enabling their respective options in the software package.

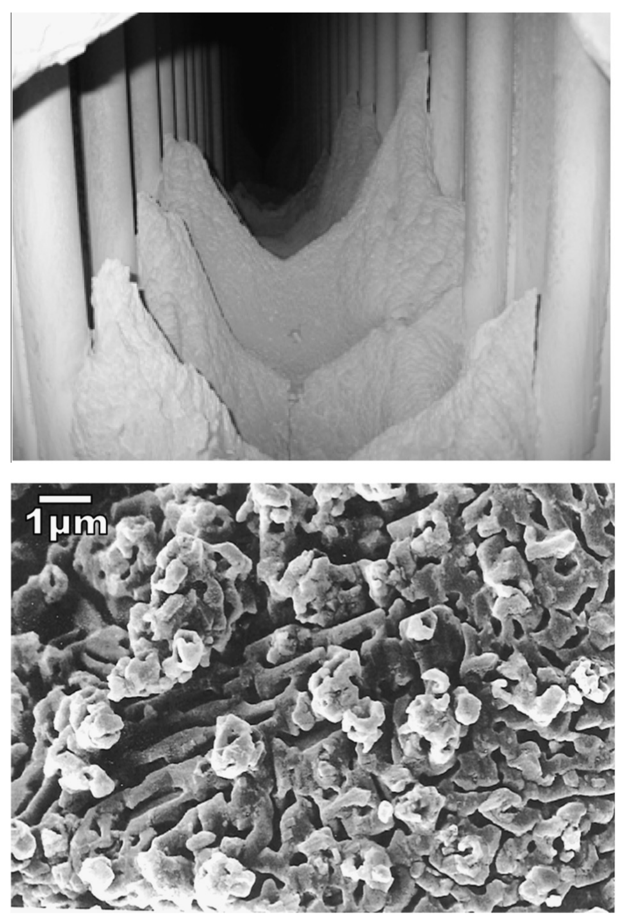

Fig. 1. Top: Deposit plug in a KRB shutdown between the boiler bank and the boiler bank screen. Deposits on the leading edge are close to plugging the whole space between them. Bottom: Micrograph of fume retrieved at an electrostatic precipitator of a KRB in eastern Finland.