Energy xxx (2014) 1–8

Contents lists available at ScienceDirect

Energy

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

Large eddy simulation of a two-phase reacting swirl flow inside a cement cyclone

Hrvoje Mikulčić [a,*], Milan Vujanović [a], Moh'd Sami Ashhab [b], Neven Duić [a]

[a] Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Zagreb, Croatia
[b] Department of Mechanical Engineering, Hashemite University, Zarqa, Jordan

ARTICLE INFO

Article history:
Received 2 January 2014
Received in revised form
6 April 2014
Accepted 19 April 2014
Available online xxx

Keywords:
CFD (computational fluid dynamics)
LES (large eddy simulation)
Cement cyclone
Calcination process
Swirling flow
Numerical modelling

ABSTRACT

This work presents a numerical study of the highly swirled gas–solid flow inside a cement cyclone. The computational fluid dynamics – CFD simulation for continuum fluid flow and heat exchange was used for the investigation. The Eulearian–Lagrangian approach was used to describe the two-phase flow, and the large eddy simulation – LES method was used for correctly obtaining the turbulent fluctuations of the gas phase. A model describing the reaction of the solid phase, e.g. the calcination process, has been developed and implemented within the commercial finite volume CFD code FIRE. Due to the fact that the calcination process has a direct influence on the overall energy efficiency of the cement production, it is of great importance to have a certain degree of limestone degradation at the cyclone's outlet. The heat exchange between the gas and solid phase is of particular importance when studying cement cyclones, as it has a direct effect on the calcination process. In order to study the heat exchange phenomena and the flow characteristics, a three dimensional geometry of a real industrial scroll type cyclone was used for the CFD simulation. The gained numerical results, characteristic for cyclones, such as the pressure drop, and concentration of particles can thus be used for better understanding of the complex swirled two-phase flow inside the cement cyclone and also for improving the heat exchange phenomena.

1. Introduction

Global anthropogenic GHG (greenhouse gases) emissions have grown since pre-industrial times, having the largest increase of 70% between 1970 and 2004. Over that period, the emissions of CO_2, as the most important greenhouse gas, have increased by approximately 80%, from 21 to 38 Gt, and represented 77% of total anthropogenic GHG emissions in 2004 [1]. Global climate changes are happening precisely due to the build-up of GHG, and if researchers, policymakers, and the public do not change something over the upcoming years, the climate changes will have major environmental and social effects all over the world [2]. In response to this situation many industrial sectors are undertaking research initiatives for increasing the efficiencies of their production processes [3]. The cement industry sector is one of the major GHG emitters within the industrial sector, accounting for approximately 5% of global anthropogenic CO_2 emissions [4]. The high amount of CO_2 emitted from the cement manufacturing process is due to the fact that cement manufacturing is an energy intensive process, and per each tonne of cement produced, around a tonne of CO_2 is emitted [5]. The calcination process and the combustion of fossil fuels are the main processes contributing to high CO_2 emissions, where the first one contributes to around 50%, and the latter one contributes to almost 40% of CO_2 emitted from the cement manufacturing process. The remaining 10% comes from the transport of raw material and other manufacturing activities [6].

The most energy efficient cement manufacturing technology at this moment is a dry kiln process together with a multi-stage cyclone preheating systems and a cement calciner. The cyclone preheating systems have been developed to enhance the heat exchange process between the raw material and the flue gases [7]. The preheating system takes place prior to the calciner and the rotary kiln and can have several stages, depending on how many cyclones are used. At each stage of the preheating system, e.g. in each cyclone, the principle of the heat exchange is the same. Raw material is heated by moving counter to the flow of the hot flue gases coming from the rotary kiln [8]. This counter-flow movement effect is due to the particle separation phenomena occurring within the gas cyclones. The separation of the solid particles from the gas is done by the highly tangential flow entering the cyclone. The centrifugal force acting on the particles directs them to the wall,

* Corresponding author. Tel.: +385 1 6168 494; fax: +385 1 6156 940.
E-mail addresses: hrvoje.mikulcic@fsb.hr (H. Mikulčić), milan.vujanovic@fsb.hr (M. Vujanović), sami@hu.edu.jo (M.S. Ashhab), neven.duic@fsb.hr (N. Duić).

http://dx.doi.org/10.1016/j.energy.2014.04.064
0360-5442/

separating them from the flow, and due to the gravitational force the particles slide to the lower part of the cyclone. In contrast to the solid particles the gas flow has a different behaviour. Firstly the gas swirls downwards in the outer cyclone part, where the separation is done, and then in the lower part of the cyclone where the axial velocity reverses, the gas starts to swirl upwards in the inner cyclone region [9].

Over recent years, as the computer technology has evolved, CFD (computational fluid dynamics) simulations have gained significant importance for obtaining information about particle separation and flow characteristics inside the cyclones. There have been several studies that have investigated some of the complicated swirled two-phase flow characteristics inside cyclones. Bernardo et al. [10] analysed the impacts of different inlet angles on the flow characteristics, separation efficiency, and the pressure drop within calculated cyclones. Their work showed that the inlet angle can increase the cyclone's separation efficiency. Wang et al. [11] by using the Lagrangian model for the solid phase tracking, and the Reynolds stress model for calculating the gas flow turbulence, investigated the gas–solid flow in a Lapple cyclone. Their study showed good correlation between the numerical predictions and the experimental measurements. Karagoz and Kaya [12] studied the gas flow and the characteristics of the heat transfer in a cyclone with a tangential inlet. Their study showed that the inlet velocity has a direct influence on the swirling characteristics of the gas flow. Wan et al. [13] numerically investigated the concentrations of different particle sizes inside a scroll cyclone. Their work showed that the concentration of solid particles has an influence on the gas flow. Gronald and Derksen [14] by using different turbulence modelling approaches analysed the swirling gas flow inside a gas cyclone. Their study showed that for industrial applications the RANS (Reynolds-Averaged-Navier-Stokes) approach for modelling turbulent flow inside the cyclones provides reasonable results, however when it comes to studying swirl flow characteristics the LES (large eddy simulation) approach ought to be used. Chu et al. [15] using the CFD-DEM approach studied the two phase gas–solid flow inside a gas cyclone, and the impact of the solid loading ratio conditions on the flow. Their work showed that the proposed approach greatly assists in the investigation of key flow characteristics inside a gas cyclone. Costa et al. [16] by using the Euler–Euler approach analysed the influence of the number of solid phases in the Euler model, on the results of the collection efficiency and the pressure drop of the simulated cyclone. Their study showed that the Euler–Euler model is a promising modelling approach for cyclone multiphase research. Sgrott et al. [17] numerically investigated the optimisation of cyclone's geometry. Their study showed that with small geometrical changes in the standard cyclones, it is possible to have higher collection efficiency and lower pressure drop. Here it should be stated that only a few of these studies investigated real industrial cyclones, and most of them investigated small scale cyclones that could also be easily investigated experimentally. When it comes to actual plant cyclones, despite the ongoing developments, multiphase flows inside industrial cyclones are considered to be insufficiently understood, and due to this reason their further research is required.

To date, to the knowledge of the authors, there have been no studies that have investigated the reacting gas–solid flow inside a cement cyclone. In order to correctly study the gas–solid flow inside the cyclones and the interaction between the two phases, appropriate numerical models needed to be developed. In this study, a developed model for the calcination process that was extensively studied in our recent study [18], was used for investigating the reactive side of the gas–solid flow inside the cement cyclone. The commercial finite volume based CFD code FIRE was used for simulating the cement cyclone. The Eulearian–Lagrangian approach was used for the numerical computation of the two-phase flow and the LES method was used for correctly obtaining the turbulent fluctuations of the gas phase. The actual plant data were used to for the numerical investigation, in order to obtain comprehensive understanding of the complex swirling two-phase flow. The results obtained by the simulation show that for better understanding of the swirling flow, pressure drop phenomena, gas to particle heat exchange, and the thermo-chemical reaction taking place inside the cement cyclone, the presented numerical model would be a valuable tool for investigation. Even more, it was shown that the presented model can assist in the optimisation of a cement cyclone's design and operating conditions.

2. Mathematical modelling

The Eulearian–Lagrangian modelling approach for the numerical computation of the gas–solid flow used in the presented work was well documented in our previous study [19]. Therefore for brevity, this paper only provides a brief description of the modelling approach.

2.1. Gaseous phase

The gaseous phase is described by solving conservation equations using the Eulerian modelling approach. These equations are based on the conservation laws for mass, momentum and energy. The general form of the time averaged conservation equation for any dependent variable φ, of the gaseous phase in the differential form is:

$$\frac{\partial}{\partial t}(\rho\varphi) + \frac{\partial}{\partial x_j}(\rho\varphi u_j) = \frac{\partial}{\partial x_j}\left(\Gamma_\varphi \frac{\partial \varphi}{\partial x_j}\right) + S_\varphi, \tag{1}$$

where ρ is the density, u_j Cartesian velocity, Γ_φ diffusion coefficient, and S_φ is the source term of the dependent variable φ. The source term S_φ is used for the coupling of the gas and solid phase, e.g. the coupling of the Eulerian and the Lagrangian formulations. In Eq. (1) the first term from the left hand side to the right hand side is the unsteady term, the second term is the convection, the third term is the diffusion and the last term is the source or sink.

2.2. Turbulence modelling

With increasing computational power, LES methods, have become very popular over recent years. Even though currently most of the studies investigate controlled experimental flows, LES methods are increasingly gaining in importance regarding industrial flow investigation. In the presented work the LES method was used for correctly obtaining the turbulent fluctuations of the gas phase. This method is certainly superior to the RANS (Reynolds-Averaged-Navier-Stokes) methods in regard to strongly separated flows, as it only simulates directly the large turbulent structures and models only the influence of the sub-grid scales on the resolved ones. The Smagorinsky model with the default value for the Smagorinsky constant of The Smagorinsky model with the default value for the Smagorinsky constant of $C_s = 0.1$ was used for sub-grid scale modelling. Detailed information on the used LES method can be found in the FIRE's Manual [20]. The governing LES equations are given as:

$$\frac{\partial\left(\overline{u}_i\right)}{\partial t} + \frac{\partial(\overline{u}_i\overline{u}_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu\frac{\partial \overline{u}_i}{\partial x_j} - \left(\overline{u_i u_j} - \overline{u}_i\overline{u}_j\right)\right], \tag{2}$$