Journal of Cleaner Production 88 (2015) 262–271

Contents lists available at ScienceDirect

Journal of Cleaner Production

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

Improving the sustainability of cement production by using numerical simulation of limestone thermal degradation and pulverized coal combustion in a cement calciner

Hrvoje Mikulčić*, Milan Vujanović, Neven Duić

Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Zagreb, Croatia

ARTICLE INFO

Article history:
Received 24 November 2013
Received in revised form
31 March 2014
Accepted 4 April 2014
Available online 15 May 2014

Keywords:
Sustainable cement production
Coal combustion
Calcination process
Cement calciner
Numerical modelling

ABSTRACT

The cement industry sector is one of the largest carbon emitting industrial sectors, and due to the effect of global warming sustainable cement production is increasingly gaining on importance. Controlling the combustion of coal and the thermal degradation of limestone, the two main thermo-chemical processes that occur inside a cement calciner, is of significant importance, since these processes have a direct influence on the cement quality, pollutant formation and overall energy efficiency of the cement manufacturing process. One of the possibilities for the improvement and control of these thermo-chemical processes are Computational Fluid Dynamics – CFD simulations. The results gained from these simulations are being increasingly used to enhance the efficiency of cement production, since they improve the understanding of the flow characteristics and transport phenomena taking place inside the cement calciner. The purpose of this paper is to present that a more energy efficient and sustainable cement production can be achieved by deploying CFD simulations in the process of cement production. The numerical models of limestone thermal degradation, also known as the calcination process, and pulverized coal combustion were developed and implemented within the commercial computational fluid dynamics code FIRE, which was then used for the analysis. The developed models are based on the solution of Navier–Stokes equations for the gas phase, and on the Lagrangian dynamics for the discrete particles. A three dimensional complex geometry of a real industrial cement calciner was used for the CFD simulation. The information obtained from this numerical simulation, such as the distribution of particles, distribution of temperatures and the concentrations can be used for better understanding of particle kinetics and pollutant emissions from the given cement calciner and also for its further investigation and optimization.

1. Introduction

Over the past five decades rapid increases in the concentrations of greenhouse gases – GHG in the atmosphere, mainly coming from the industrial sector, have resulted in global climate changes (IPCC, 2007). Due to this reason, cleaner and more sustainable production is becoming more and more important within all industrial sectors (Klemeš et al., 2012). The cement industry sector as an energy intensive industrial sector, where energy costs represent approximately 40% of the total production costs per ton of cement (Zhang et al., 2013), and one of the highest GHG emitting industrial sectors, accounts for around 5% of global anthropogenic GHG emissions (Wang et al., 2013). Therefore, improvement in energy efficiency is becoming increasingly important for fulfilling the CO_2 emissions limitations coming from this industrial sector (Dovì et al., 2009).

Currently, the most energy efficient technology for cement production is a dry rotary kiln process with a multi-stage preheater and a cement calciner (Benhelal et al., 2013). The latter, cement calciner, is a pyroprocessing unit found in front of the rotary kiln, and inside of which the raw material, mainly composed of limestone, undergoes the calcination process. The calcination process is a strong endothermic reaction that requires combustion heat released by the fuel, indicating that endothermic limestone calcination and exothermic fuel combustion proceed simultaneously (Mikulčić et al., 2013a). Controlling of these two thermo-chemical processes is of significant importance, since they have a direct influence on the cement quality, pollutant formation and overall

* 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), neven.duic@fsb.hr (N. Duić).

http://dx.doi.org/10.1016/j.jclepro.2014.04.011
0959-6526/

energy efficiency of the cement manufacturing process. There are various approaches for controlling and improving of the energy efficiency within industrial furnaces. However, using CFD simulations (Klemeš et al., 2010) to investigate and improve thermo-chemical processes is becoming increasingly important. Together with experiments and theory, CFD simulations have become an integral component of pyroprocessing unit's research. The results gained from CFD simulations can be used for the optimization of turbulent reacting fluid flow, the design of the pyroprocessing unit, and finally for the enhancement of the fuel efficiency, e.g. energy efficiency, making the cement production more sustainable.

Several studies have examined some of the numerical aspects of complex multiphase flow inside cement calciners. Oh et al. (2004) analyzed the turbulent gas-particle flow, coal combustion and heat transfer within a cement calciner. Their work showed that the numerically predicted results agreed well with the measured results. Hillers et al. (2005) numerically investigated processes that occur in cement calciners, e.g. they modelled the turbulence, radiation, calcination process, coal combustion, and NOx formation. Their study showed that CFD shows a great potential regarding emission control and fuel savings. Zheng et al. (2005) studied the effects of primary jet velocity and throat diameter on the two-phase gas–solid flow inside a cement calciner. Their study showed that for the simulated cement calciner, these two effects have a strong influence on flow structure and particle concentration. Dou et al. (2009) investigated the coal combustion and the decomposition of raw material inside a cement calciner. Their work showed that in order to increase the raw material decomposition and optimise the temperature inside the calculated cement calciner, the direction of the tertiary inlet needs to be tangentially adjusted, and that the raw material inlet needs to be opposite the coal inlet. Ha et al. (2010) studied the separation of coal particles and its corresponding influence on the decomposition of limestone inside a cement calciner. Their study showed that by combusting finer coal particles a negligible influence can be observed on the decomposition of limestone. Nance et al. (2011) using the mineral interactive computational fluid dynamics investigated the "Hot-Reburn" conditions inside a cement calciner. Their work showed that the proposed method greatly assists in the optimization of a cement calciner's operating conditions and design. Mikulčić et al. (2013a) numerically studied the impact of different inlet mass flows and fuel amounts, on the coal burnout rate, limestone decomposition rate, and pollutant emissions. Their study showed that CFD is a useful tool for a cement calciner's process optimization. All of these studies show that despite ongoing efforts in the development of both physical and chemical modelling, CFD simulation of the complex multiphase flow inside the cement calciners cannot as yet be considered fully predictive on a quantitative level and further research is required.

The processes occurring inside a cement calciner have a direct influence on cement quality, pollutant formation and the overall energy efficiency of the cement manufacturing process. Appropriate numerical models need to be used to numerically study the role and interaction of pulverized coal combustion and limestone calcination within a cement calciner. In this study a numerical model of pulverized coal combustion was developed and implemented within the commercial finite volume based CFD code FIRE. This code was used to simulate turbulent fluid flow, temperature field, species concentrations and the interaction of particles with the gas phase inside the complex three dimensional geometry of a real cement calciner, by solving the set of mathematical equations that govern these processes. The numerical model is based on the solution of Navier–Stokes equations for the gas phase, and on the Lagrangian dynamics for the discrete particles. Actual plant data were used to verify the accuracy of the modelling approach. The test of the numerical model's accuracy yielded satisfactory results and proper trends for the coal burnout rate as well as limestone degradation rate. The results gained by this real-plant example show that for better understanding of fluid flow, transport phenomena, and the thermo-chemical reactions taking place inside the cement calciner, the proposed model is a useful tool for investigation. Furthermore, the proposed model can assist in the improvement of the specific local conditions for the calcination process, the overall optimization of cement calciner's operating conditions, reduction of pollutant emissions, and the improvement of the cement calciner's design.

2. Numerical model

The continuous phase is described by solving conservation equations using the Eulerian formulation. These equations are based on the conservation laws for mass, momentum and energy. They are obtained by using the finite volume approach, where the fluid flow is divided into a number of control volumes and a mathematical description is developed for the finite control volume. The general form of conservation equation is fundamentally derived in integral form, taking into consideration the total amount of some property within the control volume:

$$\int_V \frac{\partial}{\partial t}(\rho\varphi)\mathrm{d}V + \int_S (\rho\varphi)u_j n_j \mathrm{d}S = \int_S \left(\Gamma_\varphi \frac{\partial \varphi}{\partial x_j}\right) n_j \mathrm{d}S + \int_V S_\varphi \mathrm{d}V, \tag{1}$$

where t is the time, ρ is the density, V is the volume, S is the surface, u_j velocity, n_j normal vector, x_j Cartesian coordinates, Γ_φ diffusion coefficient, and S_φ is the source term of the dependent variable φ. In Eq. (1) the first term is an unsteady term, the second term is convection, the third term is diffusion and the last term is source or sink. The first term on the left hand side represents the rate of change of the scalar property φ in the control volume. The second term on the left hand side is the net convective flux of this property across the control volume boundaries. The first term on the right hand side is the net diffusive flux across the control volume boundaries. The final term on the right hand side is the source or sink of the property φ. Two transport mechanisms can be distinguished across the control volume boundaries: convection – transport due to the motion of the fluid, and diffusion – transport due to the differences in concentration.

The motion and transport of the solid particles are tracked through the flow field using the Lagrangian formulation. Solid particles are discretized into finite numbers of particle groups, known as particle parcels, which are supposed to have same size and also the same physical properties. The trajectory of each particle parcel within the flow field is calculated using the Lagrangian scheme, which means that representative parcels are tracked by using a set of equations that describe their dynamic behaviour as they move through the calculated flow field. Furthermore, the equations of motion for each particle parcel based on the Lagrangian approach are coupled with the Eulerian representation of the continuous phase. This allows the decomposition of complicated and highly nonlinear systems of transport equations and describes the interactions between the control volumes and the system of equations that govern processes in individual control volumes, including the exchange between the solid particles and the gas phase. The systems of these equations are mainly integrated using a much shorter time step than the global time steps that are used for calculation of the gas phase. The coupling between the parcels and the gaseous phase is taken into account by introducing appropriate source terms for mass, momentum and enthalpy exchange.