CHEMICAL ENGINEERING TRANSACTIONS

VOL. 61, 2017

A publication of AIDIC
The Italian Association
of Chemical Engineering
Online at www.aidic.it/cet

Guest Editors: Petar S Varbanov, Rongxin Su, Hon Loong Lam, Xia Liu, Jiří J Klemeš

ISBN 978-88-95608-51-8; **ISSN** 2283-9216

DOI: 10.3303/CET1761037

Using an Advanced Numerical Technique for Improving Pulverized Coal Combustion inside an Industrial Furnace

Hrvoje Mikulčić[a,*], Eberhard von Berg[b], Xuebin Wang[c], Milan Vujanović[a], Houzhang Tan[c], Neven Duić[a]

[a]Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Ivana Lučića 5, 10000 Zagreb, Croatia
[b]AVL List GmbH, Hans List Platz 1, Graz, Austria
[b]MOE Key Laboratory of Thermo-Fluid Science and Engineering, Xi’an Jiaotong University, Xi’an 710049, China
hrvoje.mikulcic@fsb.hr

When it comes to industrial activities, one of the most greenhouse gas emitting industrial sectors is the cement industry. Cement industry as an energy intensive industry is alone responsible for a large amount of CO_2 emissions. Compared to the total global CO_2 emissions in 2014, cement production contributed to global anthropogenic CO_2 emissions with 8 %. Meaning that cement industry needs to adopt more energy efficient technologies and reduce its demand for fossil fuels. The aim of this paper is to present an advanced numerical technique for improving pulverized coal combustion inside an industrial furnace. With the development of increasingly affordable and powerful computers, use of numerical techniques, have become a very useful and cost-effective tool that is increasingly used in industry for detailed understanding of the combustion process and for the appropriate design and optimisation of different combustion units. In order to study the role of pulverized coal combustion inside a cement furnace, appropriate numerical models need to be used. In this study a numerical model of the pulverized coal combustion was used for the numerical analysis. A detailed three dimensional geometry of a cement furnace was simulated. The results gained by this simulation can be used for better understanding of complex thermo-chemical phenomena occurring inside the calculated furnace.

1. Introduction

Albeit its CO_2 emission is higher than that of other fossil fuels, coal as an abundant resource, is the most used solid fossil fuel in industry and power generating sector worldwide. In China, world’s largest greenhouse gas (GHG) emitter, coal accounts for approximately 70 % of the primary energy consumption, with coal-fired power plants producing 97 % of China’s thermal power capacity (Niu et al., 2015). Due to rapid increases in the GHG concentrations in the atmosphere, cleaner and more sustainable production is being increasingly emphasized within all industrial sectors (Yong et al., 2016).

Bearing in mind that cement is the most widely used material for housing and modern infrastructure, and that coal is the most used solid fossil fuel in the cement industry, the aim of this paper is to numerically analyse the coal combustion process inside the rotary kiln. With the development of increasingly affordable and powerful computers, use of Computational Fluid Dynamics (CFD) simulations, has become a very useful and cost-effective tool that is increasingly used in industry for detailed understanding of the combustion process and for the appropriate design and optimisation of different combustion units (Juřena et al., 2016).

Over the past five years there have been several studies that numerically investigated different cement production units inside which multi-phase phenomena occur. The cyclone preheating system, used for the heat exchange process between the raw material and the flue gases, was studied by Wasilewski (2017). The study showed that by optimizing the structure of the cyclone, an increase in separator efficiency can be achieved. In the subsequent study by Wasilewski and Brar (2017) the possibilities for improving the process of clinker burning was presented. The study showed that better solid particle collection results with reduction in the power consumption. In a study by Mikulčić et al. (2014) the reacting two-phase flow inside the cyclone was

Please cite this article as: Mikulčić H., von Berg E., Wang X., Vujanović M., Tan H., Duić N., 2017, Using an advanced numerical technique for improving pulverized coal combustion inside an industrial furnace, Chemical Engineering Transactions, 61, 235-240
DOI:10.3303/CET1761037

investigated. The study showed that CFD simulations are an effective tool for analysing of the heat exchange phenomena, and increasing of the separation efficiency.

The numerical investigation of fuel substitution inside a cement calciner has been the research topic in the study by Mikulčić et al. (2015a). The study showed that different operating condition need to be used is a standardly used fossil fuel is replaced with an alternative fuel. In the following study by Mikulčić et al. (2015b) it was shown that the CFD simulation method can serve plant operators and practical engineers in the optimization of cement calciner's operating conditions.

When it comes to the numerical investigation of the cement rotary kiln, over the past five years there have been only few studies that investigated the combustion process inside of the rotary kiln. As inside of the rotary kiln the highest energy demanding process during cement production occurs, the clinkering process, the numerical investigations of this furnace are entirely needed (Mikulčić et at., 2016a). Elattar et al. (2014) numerically investigated the confined non-premixed jet flames inside rotary kilns for gaseous fuels, to understand the flame behaviour and heat transfer. The study resulted with useful design guidelines and dimensionless correlations that characterize the flame length. Granados et al. (2014) numerically evaluated the flue gas recirculation effect during coal oxy-fuel combustion inside a cement rotary kiln. The study showed that the flame length in the oxy-fuel combustion cases were 30 % to 65 % shorter than that in air combustion with a higher intensity. Consequently the study reports that this could allow shorter kiln designs and might improve high-temperature clinkering reaction process. However the study states that more research is needed in this area. Ariyaratne et al. (2015) investigated the effects of meat and bone meal combustion inside a cement rotary kiln. The study showed the importance of fine fuel grinding, and that compared to coal particles, meat and bone meal particles needed more time to fully combust due to high moisture content and slower devolatilisation.

In order to study the role of pulverized coal combustion inside a rotary kiln, appropriate numerical model needs to be used. In this study a numerical model of the pulverized coal combustion was developed and implemented into a commercial CFD code AVL FIRE®, which was then used for the numerical analysis. Time-averaged conservation equations were solved for the gaseous and particulate phase, the former being treated by the Eulerian formulation, and the latter by the Lagrangian formulation. A detailed three dimensional geometry of a cement rotary kiln was simulated. The results gained by this simulation can be used for the improving the understanding of the complex thermo-chemical phenomena occurring inside the calculated rotary kiln. Furthermore, it needs to be stressed that this study analyses only the coal flame. For the raw material that is undergoing the clinkering process, different physical models need to be developed. The moving bed physics need to be carefully modelled in order to properly simulate the chemical reactions taking place inside of this bed. This is part of future work.

2. Numerical models

In this study the Eulerian-Lagrangian method for solving the multi-phase flow phenomena was used. The motion and transport of the particles, through the rotary kiln, are tracked through the flow field using a Lagrangian formulation, while the gas phase is described by solving conservation equations using an Eulerian formulation. The coupling between the solid and the gaseous phases is taken into account by introducing appropriate source terms for interfacial mass, chemical species, momentum and energy exchange. The mathematical model used for the coal combustion calculation is treated in the Lagrangian spray module, where heterogeneous chemical reactions occur inside a particle as well as homogeneous reactions between components released from the particles and gas phase species.

2.1 Pulverized coal combustion

When pulverized coal particles travel through gas and interact with the gas in the rotary kiln, they rapidly heat up and undergo a series of conversion processes. The pulverized coal combustion model used in this study includes four steps: drying, devolatilisation, char burning, and combustion of volatiles. The four-step process for the combustion of the biomass and the coal particles has been reported in a recent review study (Williams et al., 2012).

The coal particle first undergoes the drying process (see Eq. 1), after which the devolatilisation starts. After the drying process, the particle further heats up, and with an increase in the temperature, the devolatilisation process starts (see Eq. 2). During the devolatilisation, an important loss of weight occurs due to the release of volatile matter. The quantity and composition of the volatiles depend on the coal composition and the particle size and temperature. The composition of coal and the volatile matter released during the devolatilisation are estimated from the coal ultimate and proximate analysis. After the devolatilisation, only char and ash are left in the solid particle. Parallel to the devolatilisation and depending on the particle size and temperature, char oxidizes to CO or CO_2 (see Eq. 3), and subsequently, only ash remains. Then, the ash particle is considered inert, as only residual ash heating occurs.

The Eulerian-Lagrangian modelling approach for the numerical computation of the gas-solid flow used in the presented work was well documented in our previous study (Mikulčić et al., 2016b). In the mentioned study the numerical models were described in detail to accurately explain the thermo-chemical processes that govern the combustion process. The plausibility checks and quantitative checks and balances of the presented combustion model in the form of tests of a single particle in a single mesh cube are also given. Therefore for brevity, this paper only provides a brief description of the coal combustion modelling approach. The treated chemical reactions are:

$$wet\ \mathrm{Coal(s)} \xrightarrow{H_2O\ evaporation} dry\ \mathrm{Coal(s)}, \tag{1}$$

$$dry\ \mathrm{Coal(s)} \xrightarrow{pyrolysis} Volatile(g) + Char(\mathrm{s}), \tag{2}$$

$$Char(\mathrm{s}) + O_2(g) \longrightarrow CO_2(g), \tag{3}$$

$$Volatile(g) + O_2(g) \longrightarrow H_2O(g) + CO_2(g). \tag{4}$$

The heterogeneous reactions (Eq. 1, 2 and 3) cause mass transfer sources and sinks to the gas phase and particles. The homogeneous reactions (see Eq. 4) of volatile oxidation are treated within the gaseous phase reactions module of the CFD code used. Considered volatile species in this model are CO, CH_4, H_2, C_6H_6. For the oxidation of volatile species, a detailed chemistry approach is used for each of the homogeneous reaction.

3. Computational details

In order to numerically study the role of pulverized coal combustion inside a rotary kiln, a detail three dimensional geometry of an industrial cement rotary kiln was used. The rotary kiln is 45 m long and has a diameter of 3.4 m. The burner is 1 m inside of the rotary kiln and is centred along its axis. The high mass flow of pre-heated secondary air from clinker cooler enters on one side of the rotary kiln and together with the primary air and pulverized particles from the burner exits the rotary kiln on the other side of the rotary kiln.

The mesh quality testing, mesh dependency tests, testing of the time-step size, and validation of modelling approach for pulverised coal combustion was analysed in our previous study (Mikulčić et al., 2016b). Based on these findings, in the simulation of a rotary kiln 251,393 cells were employed to discretize the computational domain. A transient mode of simulation was applied. Due to fast chemical reactions, a time step of $2.5\text{x}10^{-4}$ was used. The differencing scheme used for momentum, continuity and enthalpy balances was MINMOD Relaxed and for turbulence and scalar transport equations an Upwind scheme was applied. Turbulence was modelled by the standard k-epsilon model. The P-1 radiation model was used to model the radiative heat transfer and the effects of particle radiation.

Figure 1 shows the boundary conditions that were used for simulation of the studied rotary kiln. As can be observed on the upper part of Figure 1, at the back of the rotary kiln, the outlet is positioned. Wall boundary condition is used for the complete cylindrical area of the rotary kiln. From this figure, the specific burner position can also be observed. The burner is positioned one meter inside of the rotary kiln, and after the secondary air inlet. The secondary air inlet depicted with red arrows, is the inlet of the hot gas stream coming from the clinker cooler. It was assumed that the secondary air enters the rotary kiln at a 45° angle. More detail description of the burner inlets can be seen on the lower part of Figure 4. The particular burner has a possibility to use three inlets, however in the studied case, only coal as a fuel was analysed, and therefore the secondary fuel inlet was considered as a wall boundary condition. It can be seen that the coal particles enter together with the carrying air through the inner inlet ring, whereas the swirl air, used for flame stabilization, enters through the outer ring.

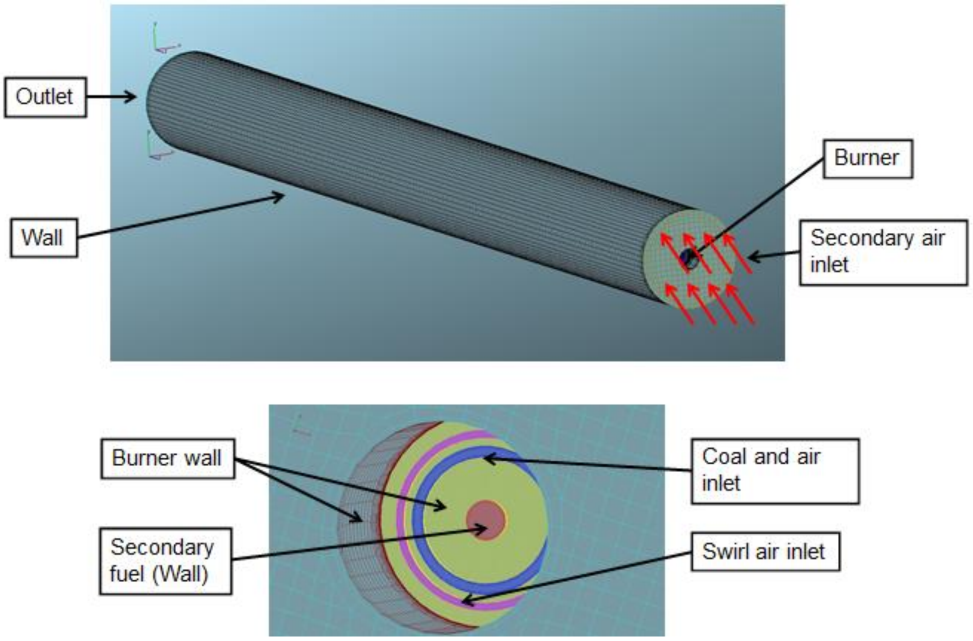

Figure 1: Rotary kiln boundary conditions

4. Results and discussion

In Figure 2, the temperature field inside the calculated rotary kiln is shown. It can be seen that due to the devolatilisation process, the highest temperatures are located close to the burner. This is due to the production of combustible volatiles and constantly available oxygen that is introduced with the secondary air. Furthermore, the flame shape can be observed. It can be seen that the flame is leaning towards the bottom of the rotary kiln. This is due to the high mass flow of secondary air, that is entering the rotary kiln at a 45° angle. The secondary air first goes to the top of the rotary kiln and then partly comes down, pushing the flame to lean towards the rotary kiln bottom. The higher temperatures that are observed away from the burner are a result of char oxidation. This statement is confirmed when looking at position of the particles in Figure 4.

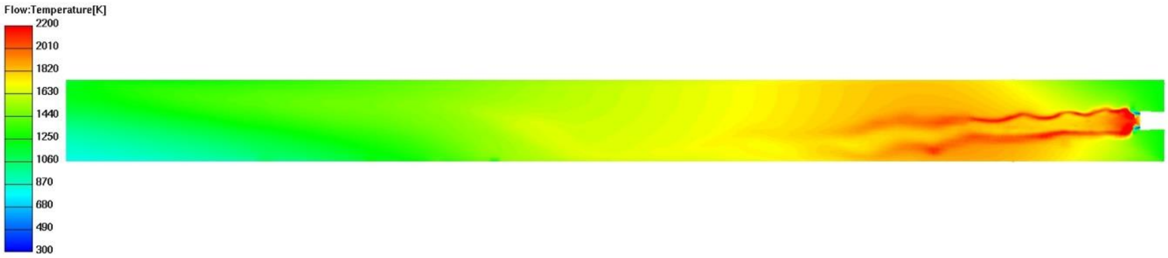

Figure 2: Temperature field inside the calculated rotary kiln

Figure 3 shows the velocity field inside the calculated rotary kiln. As can be seen the velocity is highest close to the burner where the primary air and swirl air inlet are positioned. On the right side of the figure, where the secondary air inlet is positioned, it can be observed that this air stream enters the rotary kiln at a 45° angle. The air stream from the burner first is pushed down and later at approximately middle of the rotary kiln, it goes to the rotary kiln top.

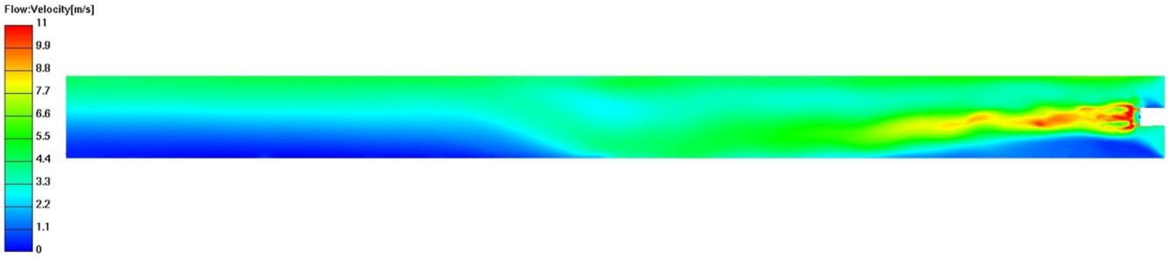

Figure 3: Velocity field inside the calculated rotary kiln

Figures 4 shows the ash mass fraction in particles. The distribution of ash particles inside the calculated calciner is also shown. In this figure the increase of ash mass fraction towards the outlet can be observed. Also, it can be seen that the ash particles are located in the whole rotary kiln. As can be seen, the ash particles are blown away by the high velocity air stream from the burner to the kiln outlet. As ash particles move through the kiln, due to the velocity profile, they agglomerate towards the kiln outlet. Here should be noted that ash deposition and particle interaction were not considered in the current investigation.

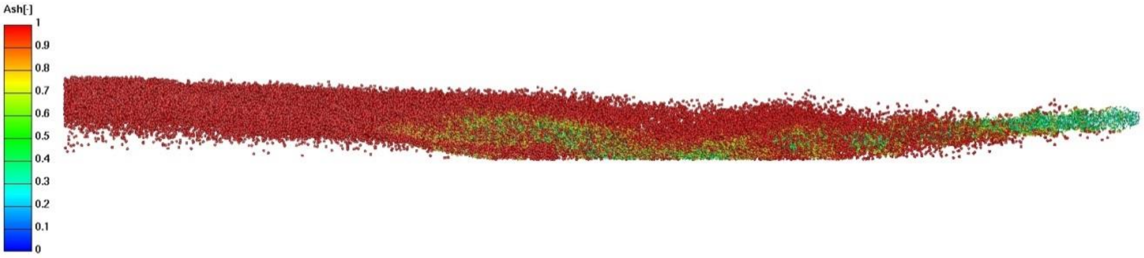

Figure 4: Ash mass fraction in particles

Though there are no available field measurements for this rotary kiln, that would allow some comparison with numerical predictions, validation of the pulverised coal combustion model, used for this calculation, was performed previously. The results gained by this simulation can be used for better understanding of complex thermo-chemical phenomena occurring inside the calculated furnace. Furthermore, the simulation results showed that, due to the high mass flow of secondary air, that is entering the rotary kiln at a 45° angle, the flame is leaning towards the bottom of the rotary kiln. Therefore, inlet condition parameter variation can give different operating scenarios that can be explored by numerical simulations, and the corresponding results can be used for decision making.

5. Conclusion

To satisfy both the need for more cement and the increased governmental and public environmental concerns, cement producers are under pressure to reduce their fossil fuel consumption. Precisely because of this reason numerical investigations of various industrial processes have become a major area of the current research. Particularly the CFD simulations have shown to be a useful tool for the investigation of different operating conditions for coal firing. The aim of this paper was to demonstrate that CFD can be used for the analysis and investigation of the coal firing process inside the cement rotary kiln. A pulverized coal combustion numerical model was developed and implemented into a commercial CFD code AVL FIRE®, taking into account the effects of drying, devolatilisation, char oxidation, and volatile combustion. The model is detail enough to contain the relevant physical and chemical processes, yet simple enough to run on the big meshes needed for detailed CFD simulations of cement furnaces. The paper gives some preliminary results on coal firing numerical modelling. The results gained by this study can be used for the investigation and better understanding of particle kinetics and distribution and operating conditions of a particular cement rotary kiln. By improving the understanding of the coal firing process inside the rotary kiln, the operating process can be improved, and consequently a reduction in fuel consumption can be achieved. Thus, the presented numerical model can be used for an ex-ante investigation of the complex thermo-chemical phenomena occurring inside the rotary kiln, e.g., regarding temperature distribution, flame position and burnout rate of the utilized fuels.

Acknowledgements

Authors would also wish to thank Dr. P. Priesching and Dr. R. Tatschl, from the CFD Development group at AVL-AST, Graz, Austria, for their continuous support and useful discussions during the development of numerical models used in this study.

References

Ariyaratne W.K.H., Malagalage A., Melaaen M.C., Tokheim L.A., 2015, CFD modelling of meat and bone meal combustion in a cement rotary kiln – Investigation of fuel particle size and fuel feeding position impacts. Chemical Engineering Science, 123, 596–608.

Elattar H.F., Stanev R., Specht E., Fouda A., 2014, CFD simulation of confined non-premixed jet flames in rotary kilns for gaseous fuels, Computers and Fluids, 102, 62–73.

Granados D.A., Chejne F., Mejia J.M., Gómez C.A., Berrio A., Jurado W.J., 2014, Effect of flue gas recirculation during oxy-fuel combustion in a rotary cement kiln, Energy, 64, 615–625.

Juřena T., Bělohradsky P., Skryja P., Hudák I., 2016, Numerical modelling of combustion in 1.5 MW Low-NOx burner, Chemical Engineering Transactions, 52, 1261-1266.

Mikulčić H., Vujanović M., Duić N., 2015a, Improving the sustainability of cement production by using numerical simulation of limestone thermal degradation and pulverized coal combustion in a cement calciner, Journal of Cleaner Production, 88, 262-271.

Mikulčić H., Wang X., Vujanović M., Tan H., Duić N., 2015b, Mitigation of climate change by reducing carbon dioxide emissions in cement industry, Chemical Engineering Transactions, 45, 649-654.

Mikulčić, Hrvoje, Jiří Jaromír Klemeš, Vujanović M., Urbaniec K., Duić N., 2016a, Reducing greenhouse gasses emissions by fostering the deployment of alternative raw materials and energy sources in the cleaner cement manufacturing process, Journal of Cleaner Production, 136, 119-132.

Mikulčić H., von Berg E., Vujanović M., Wang X., Tan H., Duić N., 2016b, Numerical evaluation of different pulverized coal and solid recovered fuel co-firing modes inside a large-scale cement calciner, Applied Energy, 184, 1292-1305.

Niu Y., Liu X., Zhu Y., Tan H., Wang X., 2015, Combustion characteristics of a four-wall tangential firing pulverized coal furnace, Applied Thermal Engineering, 90, 471-477.

Wasilewski M., 2017, Analysis of the effect of counter-cone location on cyclone separator efficiency, Separation and Purification Technology, DOI:10.1016/j.seppur.2017.02.012

Wasilewski M., Brar LS., 2017, Optimization of the geometry of cyclone separators used in clinker burning process: A Case Study, Powder Technology, DOI: 10.1016/j.powtec.2017.03.025

Williams A., Jones J.M., Ma L., Pourkashanian M., 2012, Pollutants from the combustion of solid biomass fuels, Progress in Energy and Combustion Science, 38, 113–137.

Yong J.Y., Klemeš J.J., Varbanov P.S., Huisingh D., 2016, Cleaner energy for cleaner production: Modelling, simulation, optimisation and waste management, Journal of Cleaner Production, 111, 1-16.