Energy 75 (2014) 89–96

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 14 May 2014

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:

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

where $\overline{u_i}$ and $\overline{u_j}$ are the filtered velocity vectors, ρ is the density, $\overline{p}$ is the filtered pressure field, and ν is the viscosity. The turbulent stress tensor, grouping all unclosed terms, is defined as:

$$\tau_{ij} = -\left(\overline{u_i u_j} - \overline{u}_i \overline{u}_j\right) \tag{3}$$

The sub-grid scale stress tensor is modelled using the Smagorinski model:

$$\tau_{ij} - \frac{1}{3}\tau_{kk} = -2\nu_{SGS} S_{ij} \tag{4}$$

where the sub-grid scale viscosity is written as:

$$\nu_{SGS} = (C_s f \Delta)^2 |S|, \tag{5}$$

the Smagorinsky constant is $C_s = 0.1$, whilst the resolved rate-of-strain tensor is:

$$|S| = \left(2S_{ij}S_{ij}\right)^{1/2}. \tag{6}$$

The filter width Δ, is defined as $\Delta = (Vol)^{1/3}$ where *Vol* is the volume of the computational cell. Near a wall, a wall-damping function is required to perish the eddy viscosity on the wall:

$$f = 1 - \exp\left(\frac{y^+}{25}\right) \tag{7}$$

2.3. Particle dynamics

The motion and transport of the solid particles are described by using the Lagrangian modelling approach. The trajectory of each particle within the flow field is tracked by using a set of equations that describe their dynamic behaviour as they move through the calculated flow field.

The particle trajectories are calculated from their corresponding momentum differential equation:

$$m_p \frac{du_{ip}}{dt} = F_{idr} + F_{ig} + F_{ip} + F_{ib}, \tag{8}$$

where m_p is the particle mass, u_{ip} is the particle velocity vector, F_{ig} is a force including the effects of gravity and buoyancy, F_{ip} is the pressure force, F_{ib} summarises other external forces, and F_{idr} is the drag force.

2.4. Chemical reaction model

The main chemical reaction occurring in cement cyclones, in which the temperature is between 850 °C and 950 °C, is the thermal degradation of limestone, widely known as the calcination process. Calcination is a strong endothermic reaction, where limestone $CaCO_3$ thermally decomposes into lime CaO and carbon dioxide CO_2. This process is widely used in cement, chemical, pharmaceutical, sugar industry, and some other industries. In the cement industry the calcination process is extensively used since lime is a key ingredient of the final product cement. 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 the calcination process is predominately a temperature driven process.

A previously developed model was used in order to correctly describe the reactive side of the flow inside the cement cyclone [18]. This model was thoroughly tested and validated by simulating an experimental pipe reactor, for which limestone decomposition measurements exist. The developed model combines an Arrhenius type chemical reaction rate with limestone's physical limitations, e.g. it combines the decomposition pressure and temperature effects with the limestone's diffusion and pore efficiency effects.

3. Simulation conditions

A scroll type cement cyclone used at the Lukavac cement plant, Bosnia and Herzegovina, was simulated and analysed. The simulated cyclone is the fourth cyclone of a five stage cyclone preheating system used within this cement plant. This cyclone is particularly interesting as from it the raw material, e.g. limestone, goes to the cement calciner, where it is calcinated. Around 60% of the thermal energy used during the cement manufacturing process is used for the calcination process. Due to this fact and in order to lower the fuel consumption, it is of great importance to have a certain degree of calcinated limestone entering the cement calciner.

Fig. 1 shows the calculated cyclone's three dimensional geometry and the boundary conditions used in the CFD simulation. The cyclone is 13 m high in total. It consists of a trapeze-shaped inlet which is 3.8 m high and 2.3 m width, a cylindrical region that is 4.3 m high and has a diameter of 6 m, and a conical region that is 7.9 m high and has a bottom outlet diameter of 0.6 m. The cyclone has a very short inner vortex finder that is 0.2 m long and has a diameter of 3.3 m. The gas outlet at the top of the cyclone is used to direct the outgoing gas flow to the third cyclone and the solid outlet at the bottom of the cyclone is used to collect limestone particles and direct them to the cement calciner.

A three dimensional geometry was precisely constructed using the cement plant's drawings, and FIRE's automatic mesher was used to generate the computational mesh needed for the numerical simulation. As the accuracy of the simulation highly depends of the

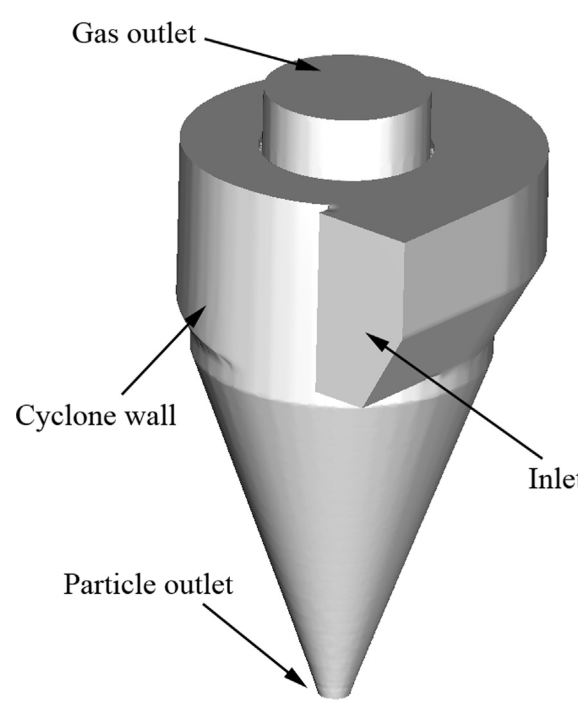

Fig. 1. Cement cyclone's geometry and boundary conditions.

mesh, and due to the reason that the flow field inside the cyclone is highly swirled, mesh quality needs to be especially taken into account. For that reason the unstructured mesh with local refinements, for outlets and near wall region, with 852,012 cells was generated. A transient simulation was performed with a time-step of $5 \cdot 10^{-4}$ s. The pressure velocity coupling of the momentum and continuity equations was obtained using the SIMPLE algorithm. The Central difference discretisation scheme was used for momentum, continuity and enthalpy balances, and the Upwind differencing scheme was used for turbulence and scalar transport equations. The P-1 radiation model was employed to model the radiative heat transfer, due to the fact that it takes into account the radiative heat exchanges between the gas and the solid particles. It has some disadvantages but it is advantageous in such a way that it is easily applicable to the any type of geometry, and is not computationally demanding. The boundary conditions used for the simulation are given in Table 1. The values given in Table 1 were the input data that were provided to the authors. As it is difficult to have a steady state solution in highly swirled flows, the criterion was set up for achieving this solution. The criteria was that the temperature, pressure and velocity field together with the particle size distribution does not change significantly in several time-steps. To satisfy that criteria 7 s of simulation time was needed, which corresponds to 3 weeks of actual computation time.

The limestone used at the Lukavac cement plant, comes from a raw material quarry located close to the cement plant. Its particle size distribution was also the input data that was provided to the authors. The raw material particle size was analysed at the location of the cement plant by using sieves for different particles size. The limestone mass-weighted particle size distribution is given in Table 2.

4. Result and discussion

In the following, the results of the CFD simulation of a real industrial cement cyclone are given. The results showed some interesting features of the flow, which help to understand the operating conditions of the simulated cyclone.

Fig. 2 shows the flow streamlines inside the calculated cyclone. The characteristic cyclone gas flow can be observed from this figure. As can be seen, the entering gas firstly swirls downwards in the outer cyclone part, forming an outer vortex, and in the lower part of the cyclone, when the axial velocity reverses the gas starts to swirl upwards in the inner cyclone region, forming an inner vortex. What can also be observed from this figure is that due to the short length of the inner vortex finder, some of the flow goes directly from the inlet to the upper gas outlet. Understanding of the flow characteristics inside the cyclone is of crucial importance for practical engineers in cement plants, as the flow characteristics provide a good estimation of the particle residence time. The particle residence time is important, since limestone particles need several seconds to heat up and start to decompose. Furthermore, it should be mentioned that flow characteristics also provide a good estimation of the cyclone's pressure drop. Exact determination of the pressure drop, one of the key parameters that determine the cyclone's efficiency, is of great importance as it has direct impact on the operational expenses.

Fig. 3 shows the absolute pressure inside the calculated cyclone. Here it can be seen that along the cyclone's axis a central low pressure core exists. It goes from the upper gas outlet to the bottom particle outlet. It has its minimum close to the vortex finder's entrance, in the centre of the cyclone. When compared to the previous figure, the central low pressure core is at the same position as the inner vortex. It can also be seen that the pressure grows in the radial direction, and that the higher pressure values are in the outer cyclone region. This effect is due to the higher concentration of solid particles in the outer cyclone region.

Table 1
Boundary conditions.

Notation		Mass flow rate [kg/h]	T [°C]	ρ [kg/m^3]	O_2 [vol. %]	N_2 [vol. %]	CO_2 [vol. %]
Inlet	Air	192,920	905	1.190	21.00	78.98	0.02
	Limestone	120,000	690	3100			
Wall			50				
Gas outlet	Static pressure	10^5 Pa		1.190			
Particle outlet	Static pressure	10^5 Pa		1.190			

Table 2
The limestone mass-weighted particle size distribution.

Particle size [μm]	Mass-weighted percentage [%]
10	5
30	19
50	41
70	21
90	14

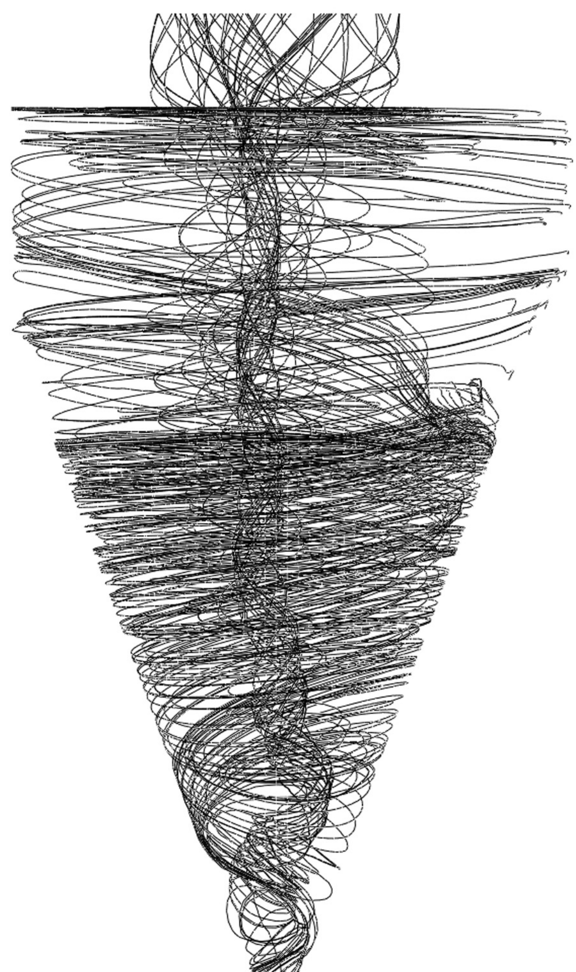

Fig. 2. Flow streamlines inside the calculated cyclone.

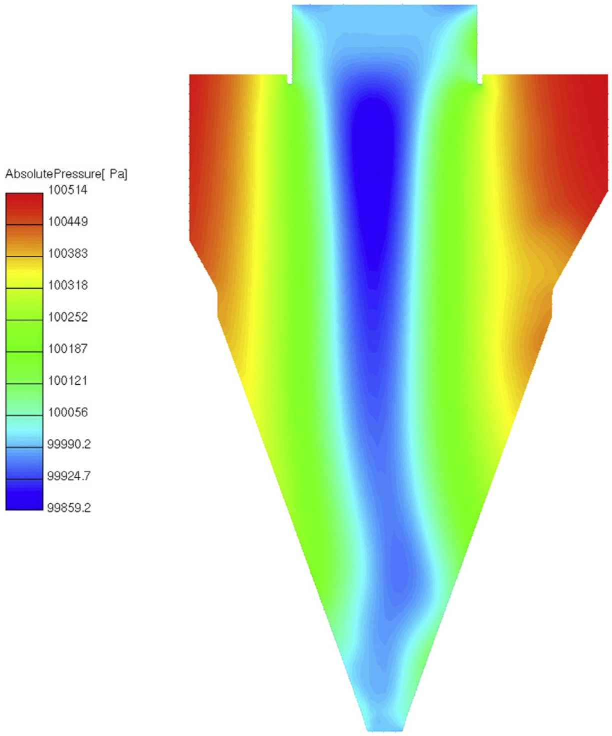

Fig. 3. Absolute pressure inside the calculated cyclone.

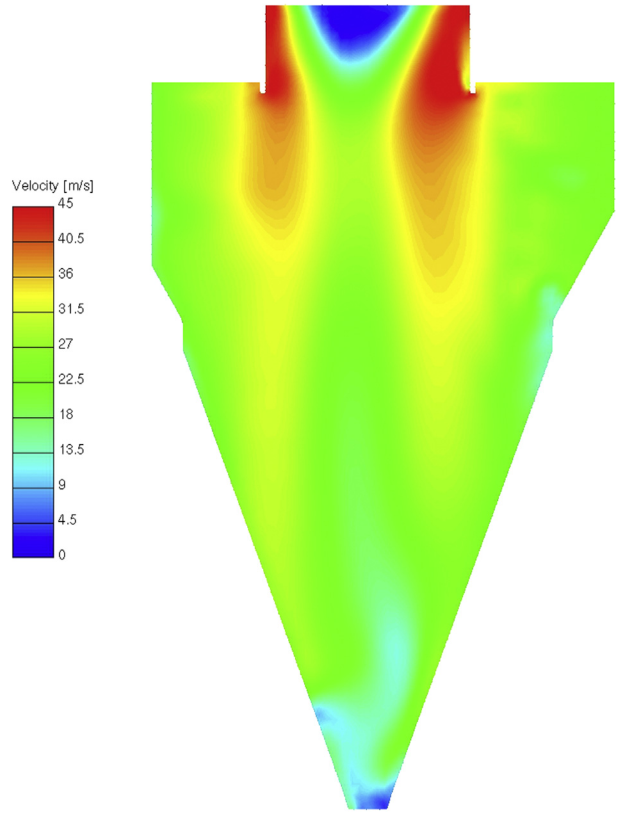

Fig. 4. Velocity field inside the calculated cyclone.

Fig. 4 shows the velocity field inside the calculated cyclone. As can be observed, the highest velocities are at the vortex finder entrance. This local high speed effect is due to the short length of the inner vortex finder. In that part of the cyclone a stream coming directly from the inlet collides with the inner vortex forming a local high speed effect. In the rest of the cyclone the velocity is around 20 m/s, except in the centre of the upper gas outlet where a recirculation zone is present due to the surrounding high velocities and at the bottom particle outlet where a slow gas outflow with a small recirculation zone is present.

During the calcination process a significant amount of CO_2 is emitted. Fig. 5 shows the CO_2 mole fraction inside the calculated cyclone. As can be seen, the local highest concentrations of CO_2 are at the outer parts of cyclone where most of the particles are located due to the centrifugal force acting on them. Except for the inlet and outlet regions where there is less CO_2 present, throughout the cyclone the CO_2 concentration is uniform and is around 5%. In the inlet region there is no CO_2 present as limestone needs some time to heat up, and start to decompose. Within the outlet regions there is a smaller CO_2 concentration due to the numerical procedure. Furthermore, it should be mentioned that the simulation results showed that from approximately 100 kg of limestone, around 7 kg of CO_2 was released.

The calcination process is predominately a temperature driven process. As mentioned, the cyclone preheating systems have been developed to enhance the heat exchange process between the raw material and the flue gases. It is of great importance for a cement plant operator to have a certain degree of decomposed limestone at the cyclone's outlet, in order to lower the cement plant's overall fuel consumption, and in that way to lower the operating expenses. Fig. 6 shows the particle temperature inside the calculated cyclone. As can be seen, the particles with the lowest temperature are located at the inlet. As the particles move through the cyclone, they are heated up due to the convective and radiative heat transfer. The particles with the highest temperature are located at the inner part

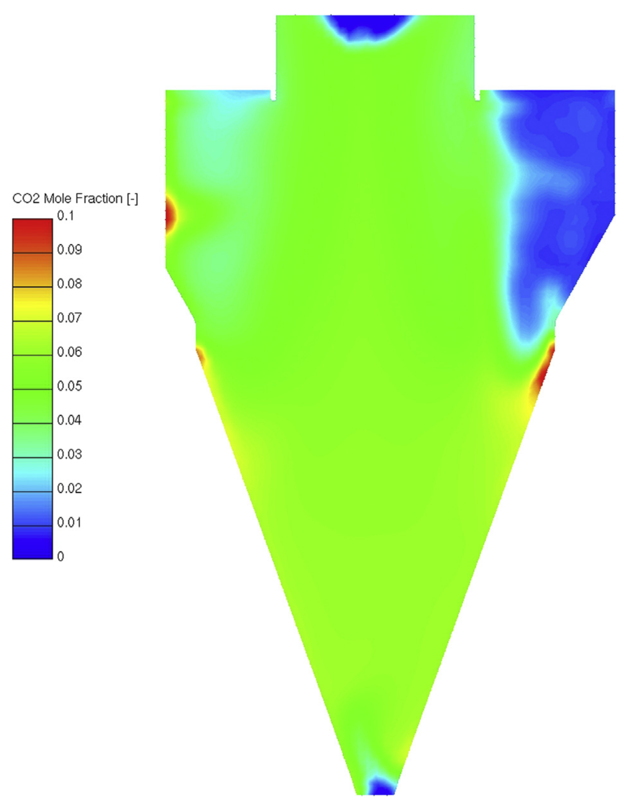

Fig. 5. CO_2 mole fraction inside the calculated cyclone.

of cyclone and at the gas outlet. These are predominately particles with smaller diameters that heat up quicker than particles with bigger diameters. Also what can be observed is that in the cyclone's conical part, the particles tend to agglomerate as they move to the particle outlet located at the cyclone's bottom.

Fig. 7 shows the calcination process inside the calculated cyclone. As can be observed, there are only few limestone particles that decomposed inside the cyclone. The reason for this is due to the fact that the thermal decomposition of limestone is a strong endothermic process that needs high temperatures and several seconds of residence time for limestone to decompose. In the present case the residence time of the limestone particles inside the cyclone is too short for them to start to react. However here it should be mentioned that the simulation showed that at the particle outlet, the degree of limestone decomposition was around 2%. This showed that some degree of calcination was still achieved inside the calculated cyclone.

Fig. 8 shows the surface absolute pressure distribution. As can be observed, the highest pressure is at the inlet, and higher pressure can be seen in the outer cyclone regions. The pressure gradually decreases when approaching the bottom of the cyclone and to its inner part, having the lowest value at the gas outlet. The pressure drop effect due to the characteristic flow inside the cyclone can be observed from this figure.

Fig. 9 shows the temperature field for six cut planes of different height inside the calculated cyclone. As can be seen, the lower temperatures are at the outer parts of cyclone where most of the particles are located, and where the endothermic calcination process is occurring. Due to smaller particle concentration in the inner cyclone part, higher temperatures can be observed at the position of the inner vortex.

A comparison between the numerically predicted results and the measurement data is essential for the validation of the used numerical model. The measurement equipment of this fully

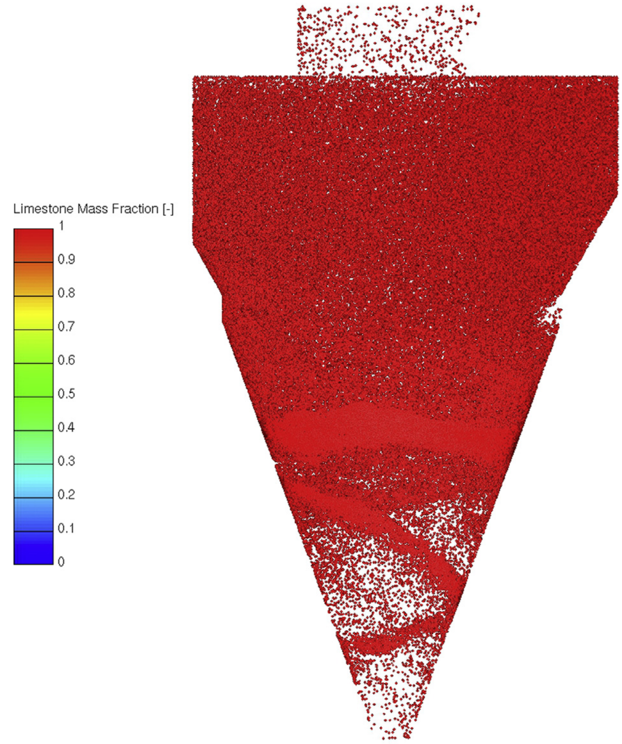

Fig. 7. Calcination process inside the calculated cyclone.

operating industrial cement cyclone was placed on its inlet and gas outlet. Pressure was the only quantity that was measured. Table 3 shows the comparison of the measurement data with the numerical prediction. As can be seen, the numerical prediction of the

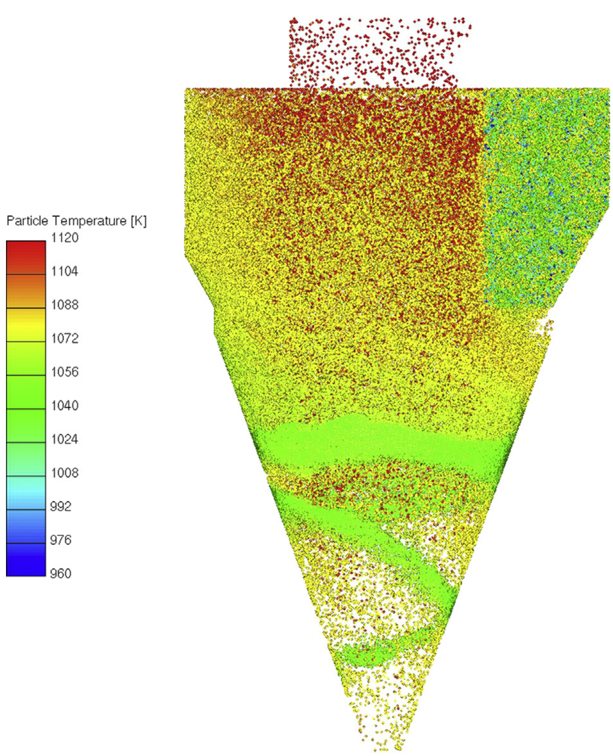

Fig. 6. Particle temperatures.

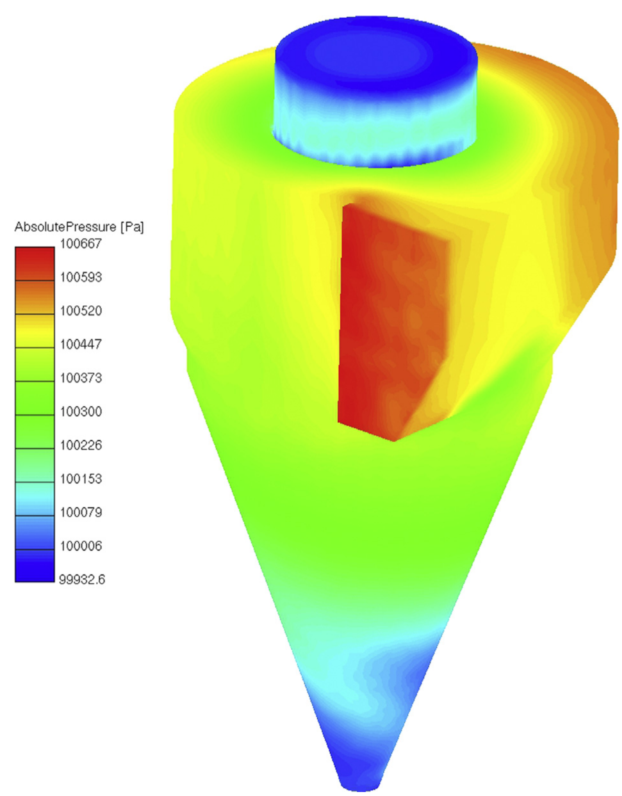

Fig. 8. 3-D view of the cyclone's absolute pressure distribution.

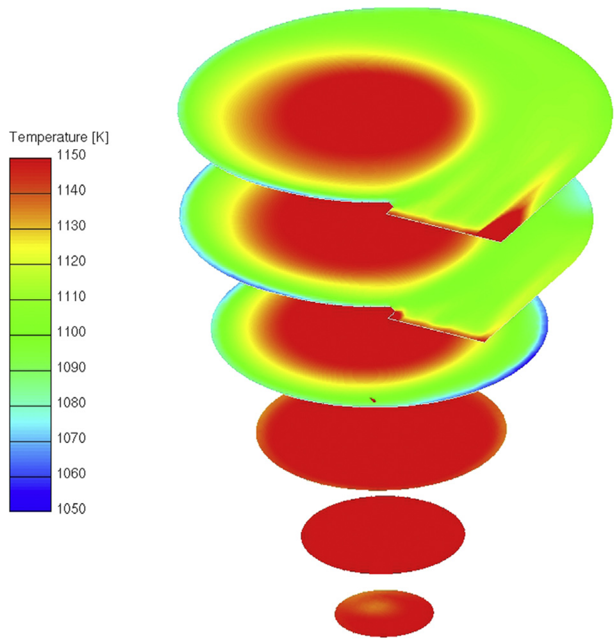

Fig. 9. Temperature field inside the calculated cyclone.

pressure drop is in good correlation with the measured data, and the outlet temperature is in fairly good agreement. Here it should be noted that numerically predicted results given in Table 3 are outlet average values, as it is not known where exactly on the outlet the measurements were taken. The results presented herein show that numerical modelling can be used with high degree of accuracy for the analysis and the improvement of the understanding of the complex two-phase reactive swirl flows inside real industrial cement cyclones. The proposed models and methods can assist plant operators and practical engineers in the optimisation of cement cyclone's operating conditions. By optimising operating conditions a better overall cement plant efficiency can to be achieved.

Table 3
Comparison of measurement data and numerical predictions.

	Measurement data	Numerical predictions
Pressure drop [Pa]	670	674
Outlet temperature [K]	1061	1136

5. Conclusion

Due to the increased environmental concerns regarding global climate changes, and as the cement industry sector represents a significant threat to the long term sustainable development, various environmental regulations impose ever more stringent requirements on the cement industry as a part of the solution. Thus, various techniques that can assist in the improvement of the cement manufacturing's energy efficiency now represent a major area of the current research. Computer modelling of the highly swirled reactive multi-phase flows provides a valuable tool that can be used for the investigation and better understanding of thermo-chemical reactions, particle kinetics and pollutant emissions. This work has presented a numerical model for the prediction of the highly turbulent flow, particle kinetics, and chemical reaction occurring in the cement cyclone. The model uses the Eulearian–Lagrangian approach for the numerical computation of the gas–solid flow and the LES approach for obtaining the turbulent fluctuations of the gas phase. The numerical model for the calcination process is treated in the Lagrangian module, and was implemented into a commercial CFD code FIRE. The model takes into account the effects that dominate the reaction, such as decomposition pressure, temperature, diffusion, and pore efficiency. A detailed three-dimensional numerical simulation with actual plant data was performed in order to obtain comprehensive understanding of the complex highly swirled reactive gas–solid flow inside a cement cyclone. The numerically obtained results were compared with available measurement data, and are in good correlation with it. From the results shown it can be concluded that the presented model predicts physically reasonable results. Thus, it can be used for further investigation of the reactive multiphase flows and the optimisation of different cyclones found throughout the cement industry.

Acknowledgements

Financial support from the European Union's seventh Programme managed by REA – Research Executive Agency (FP7/2007-2013) under Grant agreement PIRSES-GA-2011-294933 (DISKNET project) is gratefully acknowledged.

Authors would also wish to thank Mr. E. von Berg, 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.

Furthermore, the authors wish to thank Mr. Dž. Omerdić, of the cement plant Lukavac, Bosnia and Herzegovina, for providing the cement cyclone's geometry for a CFD calculation, and input data needed for the calculation. Finally, the authors wish to thank Prof. Z. Osmanović from University of Tuzla for useful discussions.

References

[1] IPCC (Intergovernmental Panel on Climate Change). IPCC fourth assessment report. Cambridge, United Kingdom and New York, NY, USA: Cambridge University Press; 2007. Available from: www.ipcc.ch/publications_and_data/publications_and_data_reports.shtml#1 [accessed 06.11.13].
[2] Čuček L, Klemeš JJ, Kravanja Z. A review of footprint analysis tools for monitoring impacts on sustainability. J Clean Prod 2012;34:9–20.
[3] Klemeš JJ, Pistikopoulos EN, Georgiadis MC, Lund H. Energy systems engineering. Energy 2012;44:2–5.
[4] Atmaca A, Kanoglu M. Reducing energy consumption of a raw mill in cement industry. Energy 2012;42:261–9.
[5] Karellas S, Leontaritis A-D, Panousis G, Bellos E, Kakaras E. Energetic and exergetic analysis of waste heat recovery systems in the cement industry. Energy 2013;58:147–56.
[6] Mikulčić H, Vujanović M, Markovska N, Filkoski RV, Ban M, Duić. CO_2 emission reduction in the cement industry. Chem Eng Trans 2013;35:703–8.
[7] Zhang H, Dewil R, Degrève J, Baeyens J. The design of cyclonic pre-heaters in suspension cement kilns. Int J Sustain Eng; 2013. http://dx.doi.org/10.1080/19397038.2013.843605.
[8] Mikulčić H, Vujanović M, Fidaros DK, Priesching P, Minić I, Tatschl R, et al. The application of CFD modelling to support the reduction of CO_2 emissions in cement industry. Energy 2012;45:464–73.
[9] Cortés C, Gil A. Modeling the gas and particle flow inside cyclone separators. Prog Energy Combust Sci 2007;33:409–52.
[10] Bernardo S, Mori M, Peres AP, Dionísio RP. 3-D computational fluid dynamics for gas and gas-particle flows in a cyclone with different inlet angles. Powder Technol 2006;162:190–200.
[11] Wang B, Xu DL, Chu KW, Yu AB. Numerical study of gas-solid flow in a cyclone separator. Appl Math Model 2006;30:1326–42.
[12] Karagoz I, Kaya F. CFD investigation of the flow and heat transfer characteristics in a tangential inlet cyclone. Int Commun Heat Mass Transf 2007;34:1119–26.
[13] Wan G, Sun G, Xue X, Shi M. Solids concentration simulation of different size particles in a cyclone separator. Powder Technol 2008;183:94–104.
[14] Gronald G, Derksen JJ. Simulating turbulent swirling flow in a gas cyclone: a comparison of various modelling approaches. Powder Technol 2011;205:160–71.
[15] Chu KW, Wang B, Xu DL, Chen YX, Yu AB. CDF-DEM simulation of the gas-solid flow in a cyclone separator. Chem Eng Sci 2011;66:834–47.

[16] Costa KK, Sgrott Jr OL, Decker RK, Reinehr EL, Martignoni WP, Meier HF. Effects of phases' numbers and solid-solid interactions on the numerical simulations of cyclones. Chem Eng Trans 2013;32:1567–72.
[17] Sgrott Jr OL, Costa KK, Noriler D, Wiggers VR, Martignoni WP, Meier HF. Cyclones' project optimization by combination of an inequality constrained problem and computational fluid dynamics techniques (CFD). Chem Eng Trans 2013;32:2011–6.
[18] Mikulčić H, von Berg E, Vujanović M, Priesching P, Perković L, Tatschl R, et al. Numerical modelling of calcination reaction mechanism for cement production. Chem Eng Sci 2012;69:607–15.
[19] Mikulčić H, von Berg E, Vujanović M, Priesching P, Tatschl R, Duić N. Numerical analysis of cement calciner fuel efficiency and pollutant emissions. Clean Technol Environ Policy 2013;15:489–99.
[20] FIRE v2011 manuals; 2011. Graz, Austria.