Modelling of heat transfer in a combustion chamber and multiphase flow in a pilot-scale CFB

Agata Katarzyna Widuch
agataa.widuch@gmail.com
Instituto Superior Tecnico, Universidade de Lisboa, Portugal
September 2019

Abstract

Fluidization process progressively gains interest. It is mainly due to high heat transfer and mixing rate of solid material. It is used for drying, cooling, mixing, and converting fuel energy into heat. The last of mentioned application is intensively developed in the energy sector. In case of building new units or improving the existing one, an extensive engineering work is required. In order to overcome necessity of performing on-site or laboratory test a numerical techniques can be used. Computational methods give great opportunity to predict and understand complex physical mechanisms. Industrial boilers, where fluidization technology is used, have high efficiency, leads to reduce emissions of harmful substances such as SOx and NOx along with the use of larger range of burnt fuel, including wastes. Nevertheless application of numerical techniques for predicting complex particle transport in dense solid flow is not a trivial task and require special treatment and attention. This situation is mostly caused by intensive mixing, mutual particle interaction and strong coupling between phases. Modelling of multiphase flows can be done using Euler-Euler, Euler-Lagrange, hybrid Euler-Lagrange approaches as well as Discrete Element Method (DEM). The DEM techniques take into account collisions between particles applying direct collision model while remaining approaches model the collision using the idea of kinetic theory of granular flow. In contrary to Euler-Euler technique, DEM is not well tested yet for fluidization processes, mainly due to the cost of simulations. In this work, the potential of using the DEM for modelling combustion and heat transfer in fluidized bed was tested. Also, a comparison between data resulting from experiments on pilot-scale Circulating Fluidized Bed unit, Euler-Lagrange approach and DEM will be provided.
Key words: Computational Fluid Dynamics, Discrete Element Method, Fluidized Bed, Combustion, Heat Transfer

1. Introduction

1.1. Fluidization

Fluidization is a multiphase process, where solid material is transported by flowing gas. The process in the last few decades has gained more and more interest. The technology finds application in many different industries, i.e. chemical for re-drying of chemicals after mechanical drying, pharmaceutical (e.g. medication mixing fluidization gives higher efficiency and more uniform distribution of material in comparison to mechanical mixing), food processing (popcorn popper) and many applications in the energy sector. The last mentioned application is intensively developed. Intensive heat transfer, mixing of the solid material, long residence time of the fuel, or the uniform temperature distribution over the combustion chamber are the main factors that characterize this technology. The industrial boilers that use fluidization techniques have higher efficiency, even 10% higher in comparison with pulverized coal boilers. This is caused mainly by the high mixing rate which affect all thermo-chemical processes that occur during coal combustion, for instance this helps in reduction of SOx emission by direct limestone injection. The combustion temperature with fluidized bed ranges between $800-950^{\circ}C$, which accounts for lower NOx emissions, allows to burn fuel (as wastes) with lower heating value [1]. Lower temperature results in no softening of an ash, that further does not create problems with slag or coating formation on a chamber walls that will lead to decrease of heat transfer rate between the heating surfaces and the heated medium [2]. Another advantage of the usage of this type of boilers is their low sensitivity to fuel quality.

A large range of fuel types, including wastes can be easily burned. Looking to the proportions of the material transported in the combustion chamber, the coal fraction is only 3% of the total material in the combustion chamber, the rest is the inert material. The purpose of the inert material is t transfer the heat released during combustion process [3]. As it was mentioned the fluidization is a process where due to gas flow through the chamber, the suspension of solid particles in the fluid switches them into a fluid-like state. The process begins putting material on the porous plate. Then gas with low flow rate starts passing through the porous plate and material. The minimal fluidization velocity of a gas (u_{mf}) is reached once the gravitational and other forces affecting particles are balanced and they start to move (Fig. 1a). Continuing the increase of velocity above the minimal fluidization velocity rises smaller particles to the top of the boiler and gas bubbles appear (Fig. 1b). Exceeding of so called terminal velocity (u_{tf}) bubbles vanish, instead turbulent motion of particles occur (Fig. 1c). Further increase of gas velocity transforms the turbulent bed into a circulating fluidized bed. Particles are removed through the top part of the chamber and after passing over the solid separator return to the chamber (Fig. 1d).

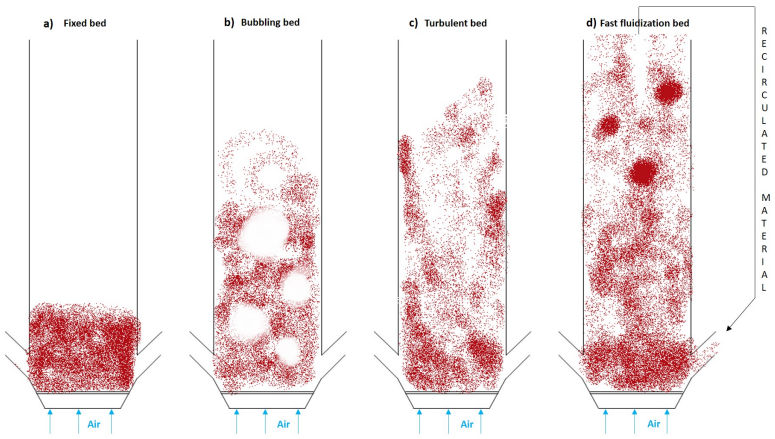


Figure 1: Fluidization regimes

1.2. Circulating Fluidized Bed

Among the fluidization technology, nowadays the most popular is the Circulating Fluidized Bed (CFB). Lower temperature of combustion allows to burn fuel of low quality e.g. wastes or biomass. Due to relatively low combustion temperature fuel with higher moisture content, lower heating value fuels can be burnt in the CFB boiler. Those properties characterize biomass or RDF (Refuse Derived Fuel). This is of a great importance where challenge with waste management is constantly increasing. The CFB boilers are mainly dedicated for units of high generating power. Already installed, example units, have electrical power of 460 MWe (Łagisza, Poland), 330 MWe (Novocherkasskaya, Russia) and 550 MWe (Samcheok, South Korea) [4]. The particle resistance time, due to the close loop is much longer in comparison to the traditional pulverized coal boilers. Instead of removal of unburnt fuel particles, they are separated from flue gases and returned to the combustion chamber through separator and loop seal. High conversion rate of fuel results in higher boiler efficiency. It has to be taken into account that co-combustion of different types of fuels is possible and also function properly. The amount of oxidizer provided to the combustion chamber, within different zones need to be adjusted for the type of fuel, taking into account its size, density and moisture content. One of the operating conditions is the velocity, which needs to be maintained over the entrainment velocity ranging from 4.5 to 6.7 m/s) [5]. This allows to uplift particles and redistribute them uniformly in the chamber - lighter particles are raised and heavier are occupying the bottom part of the chamber until fragment into the smaller ones during collision, particle swell, and breakage processes.

A typical CFB boiler scheme, shown in Fig. 2, consists of the combustion chamber, loop-seal and drain section.

2. Mathematical model

The numerical modeling of the fluidization processes in large scale installations as well as in the laboratory devices is computationally complex and very expensive. As it was mentioned earlier, different models are available to simplify and speed up numerical simulations. Nevertheless, all of them suffer from some deficiency, mainly concerning the accuracy of prediction of the particle-particle collisions. In order to accurately resolve this problem, the Discreet Element Model (DEM) soft sphere model may be used. Therefore, approach which gives possibility to reduce calculation time without losing accuracy is very desirable. In order to deal with the aforementioned difficulties new approach for predicting particles collision has to be developed in future. The presented thesis investigates the possible application of a complex collision model based on the DEM approach for modeling laboratory scale CFB installation. Moreover, the heat and mass transfer aspects in case of use the DEM technique will be investigated. The core of the DEM approach for modeling fluidization process where strong interaction between gaseous and solid phase need to be taken into account is the coupling between phases. In this model the same strategy as in application of Dense Discreet Phase Model (DDPM), named also hybrid Euler-Lagrange (HEL) technique is used. For accurate prediction of interaction between particles and particles with fluid as well as Particle Size Distribution (PSD) Discrete Element Method can be used. DEM approach defines collisions using two models: soft-sphere and hard-sphere. In this, based on the work of Cundall and Stack [6] implemented in ANSYS Fluent takes into consideration the spring deformation and shape of particle resulted from collision. Additionally in this work combustion process was investigated. Combustion is a rapid chemical process of oxidizing, releasing great amounts of heat. Substrates of combustion are substances supplied to the domain (fuel and air) while products are substances released (flue gas, gaseous and solid substances) [7]. The solid fuel combustion process consists of few steps: heating, evaporation, devolatization, oxidation. In the first stage of the combustion process the injected particle is heated up to a defined evaporation temperature. Over the evaporation temperature the evaporation process proceeds until moisture is removed from the particle. After evaporation the particle is again heated to devolatilization temperature, above which the gaseous fraction is released from the particle to the continuous phase, where the volatile matters are combusted. After devolatilization, the char combustion process starts. When the entire char in the particle is consumed, the heating process is again activated and remaining hot ash is used as the inert material for heat transfer processes.

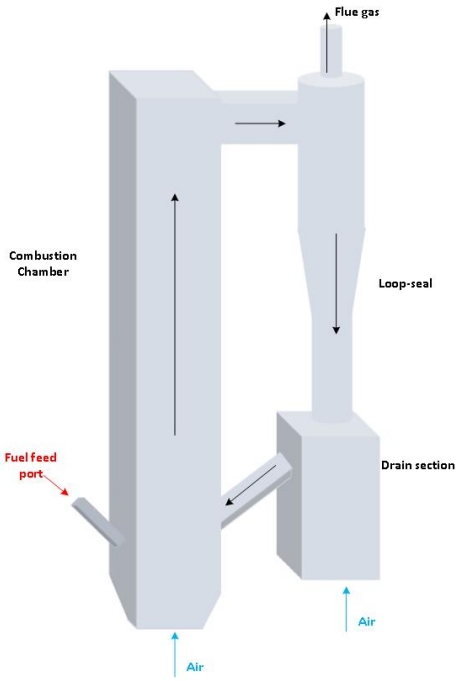


Figure 2: General scheme of the CFB boiler with higlighted injection ports

3. Numerical simulations using simplified geometrical models

Geometry and a mesh of simplified combustion chamber were made in Ansys environment. Used geometry is depicted in Fig. 4. Geometry consists of 4,718 hexahedral elements with size of 0.03 m. The dimensions of a domain are $0.2x0.3x2$ m. Solid material is injected through the port located at the left side of the computational domain, 0.15 m above the bottom air inlet, see (Fig. 3). For modeling coal combustion the solid material injection ports have been slightly reconfigured as it can be seen in Fig. 5b. Coal injection ports have been located at the side walls. This operation was done in order to ensure better mixing of the fuel and inert material at the bottom section of the combustion chamber.

Figure 3: Part of a domain used for injecting sand particles

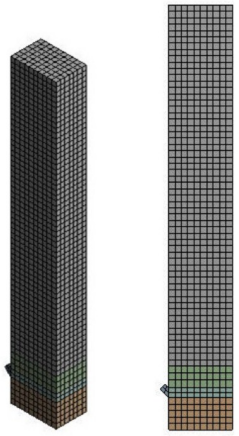
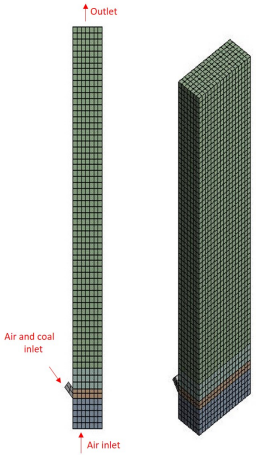
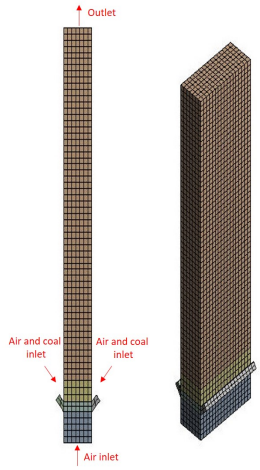


Figure 4: Geometry used for heat transfer modelling



(a) Port of fuel on one side of the domain



(b) Ports of fuel on both sides of the domain

Figure 5: Geometries used for combustion modelling

The calculations were initialized by providing 100,000 particle parcels (named sand) into the bottom part of the geometry. Using the volume injection method it is possible to prescribe defined amount of material to the selected zones of the domain, highlighted at Fig. 3. The initial velocity of the sand was set to zero. In case of modeling only mixing with heat transfer the initial sand temperature was set to 383 K while modelling combustion the initial sand temperature was set to 1173 K. A mixing process was organized by injecting solid material (named here as coal) through the port located at the side ports of the computational domain. For heating purpose the hot air was provided to the domain through bottom inlet with temperature equal to 573 K. The simulation time was set to three seconds while the time step was set to 0.001s. The effect of heat transfer between gaseous and solid phases was resolved in terms of the convection and conduction. The radiation was not taken into account due to high loads of solid material within the domain. For collecting information of unburnt coal removed from the domain through outlet the user defined function (UDF) was created. The main objective was to collect the mass of char and ash at the outlet for calculating the energy leaving through the outlet.

3.1. Results

A set of numerical calculations has been carried out for two configuration. The results have been divided between heat transfer and mixing processes and combustion. Firstly the heat transfer between gaseous and solid phases was investigated together with mixing of the solid material. In the second stage the combustion of coal together with mixing of sand has been taken into account. Results have been illustrated using parcels

distribution colored by solid phase temperature. The changes of physical property of mentioned variables were illustrated after 0.01 s, 1.5 s and 3 s from beginning of simulation.

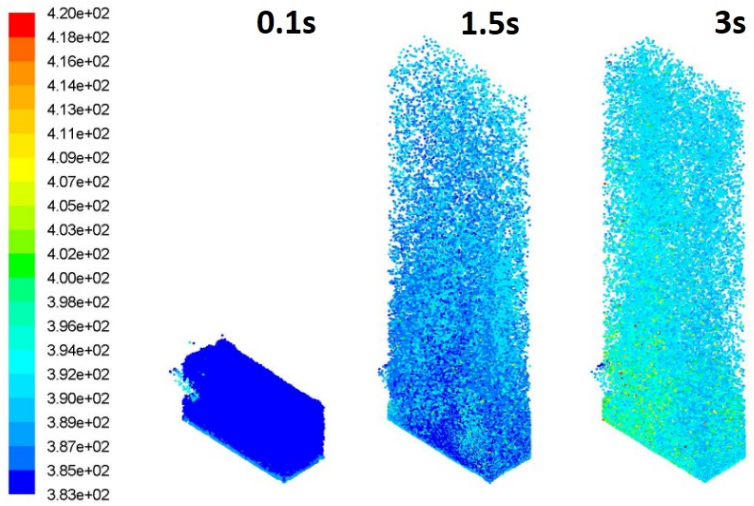
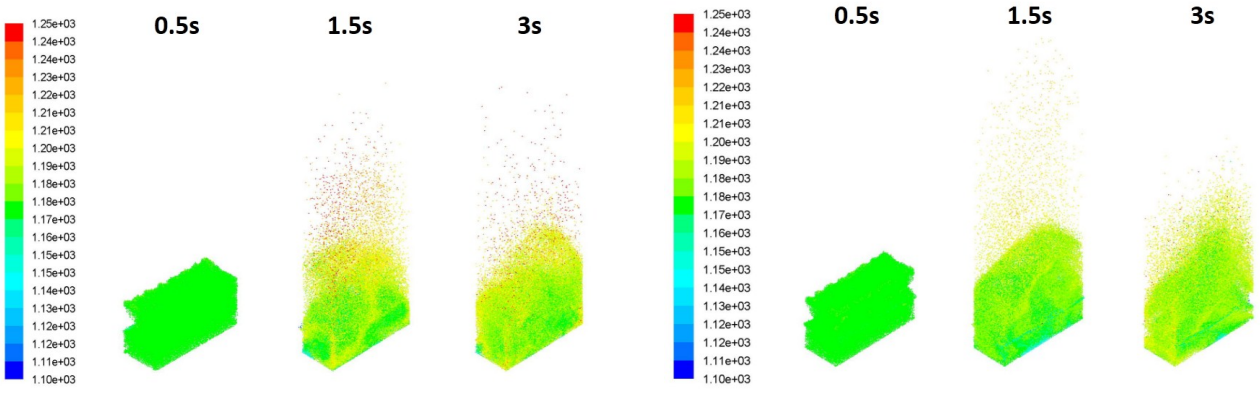


Figure 6: Particle temperature in 0.1s, 1.5s and 3s of simulation respectively

The combustion process is discussed based on the particle distribution is shown in Fig. 7. The combustion chamber was assumed to be adiabatic, which means it does not exchange heat through the walls, so the whole energy is absorbed and accumulated by the inert material. Initial values of CO_2, H_2, O_2 were set as 0.1, of a N_2 as 0.6 and the initial temperature was 1173 K. The effect of non uniform distribution of solid material within the riser for both of the considered cases can be seen in Fig. 7 where small fraction of solid material is entrained by passing gas. This is caused by the properties of the inert material which is quite heavy and parcels are relatively large. Intensive mixing helps in ensuring the uniform combustion of solid material and distribution of combustion products as well as gaseous phase temperature.



(a) Injection ports located at one side

(b) Injection ports located at both sides

Figure 7: Temperature of sand in 0.1s, 1.5s and 3s of simulation respectively

The composition of flue gases obtained from numerical simulations was monitored in every time step during the solution procedure. To obtain the final value, all gathered data was averaged in the time range where the solution was stabilized. This procedure was done both for the case with single and two solid injection ports. Comparison of the numerical and analytical results is presented in table 1. For both investigated configurations of the numerical model, the calculated flow parameters are similar. Some differences can be observed between numerical and analytical results. This can be caused by incomplete combustion of the coal in the considered riser geometry, which means that particles leaving the domain contain a non zero fraction of unburnt char. In the case of total combustion of char, the composition of flue gases calculated analytically and numerically will agree.

Table 1: Mass fraction of flue gas components.

Flue gas components	Analythical results, %	Computational results, %	
		1 inlet	2 inlets
(CO2)	18.55	16.89	15.29
(O2)	4.48	5.34	7.05
(N2)	65.35	64.92	66.12
(SO2)	0.13	0.48	0.08
(H2O)	11.49	12.38	11.47

4. Numerical simulations of the laboratory scale test-rig

4.1. Geometrical model

Similarly to the heat transfer modelling, a three-dimensional computational domain and mesh of the experimental rig were made in Ansys environment. The model was created based on the laboratory scale CFB test unit, located at the Lappeenranta University of Technology at Laboratory of Modelling of Energy Systems [8]. The same stand has been already used for modelling fluidization of the solid material using DDPM-HEL approach in [9]. The general scheme of the installation is depicted in Fig (8). The complete test rig, working as a real CFB unit, consists of a riser, loop-seal and drain section. The loop-seal itself is composed of the cyclone (device separating gas and solid particles) and a downcomer.

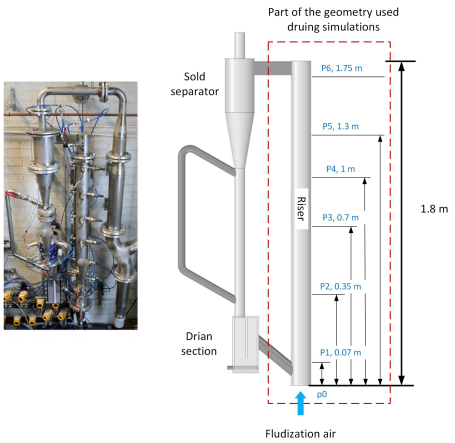


Figure 8: The pilot installation located at the Lappeenranta University of Technology [8] (left) and numerical mode with highlighted simplification and locations of the measurement ports

In order to simplify calculations, the geometry was modified. The following elements have been removed: loop-seal and drain section. As result the geometry used for calculations consists of the riser and two short pipes at the bottom and at the top as shown in a Fig. 9. The height of the riser is 1.8 m and its internal diameter

equals to 0.11 m. The mesh consists of 38,450 hexahedral and tetrahedral elements with size of 0.01 m (Fig. 11). Measurement points of pressure are situated along the riser, starting from ground level at 0 m, 0.07 m, 0.35 m, 0.7 m, 1.0 m, 1.3 m, and 1.75 m.

The experimental tests, together with DDPM simulations, were carried out in ambient conditions (cold). The mass of particle parcels in the domain amounts to 1 kg and gas velocity, introduced from the bottom of the riser, equals to 3.75 $\frac{\mathrm{m}}{\mathrm{s}}$. In the experiments glass particles, with the density of 2450 $\frac{\mathrm{kg}}{\mathrm{m}^3}$ were used. On the basis of 2d analysis particle diameters were determined. Particle diameter was ranging from 100 μm to 713 μm, the mean diameter was 513 μm, and the spread factor was determined to be equal to 9.13 based on Rossin-Rammler distribution [9]. To replace removed elements of a geometry a user defined function (UDF) was developed and implemented into the solution procedure. The general idea is to save, at the end of every time step, information about the number of particles and their mass which is leaving through the outlet of the domain. At the beginning of next time step, particle parcels with a total mass trapped previously are injected back to the riser through the recirculation surface (see Fig. 11). In general terms the number of streams injected back to the domain depends on number of faces composing a particular surface.

Figure 9: Simplified mesh

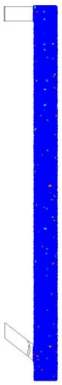


Figure 10: Part of pilot-scale rig used for injecting material

The close up of a surface is also presented at Fig. 11. For this specific case the number of faces at the recirculation inlet surface is 80. Procedure of saving informations about the number of particle parcels and their mass leaving through the outlet, in order to inject back the same mass of solid phase, ensured the constant mass of the material in the domain. Results from the simulation of the flow in the riser using the DEM model will be validated with experiments, as well as with results obtained using the DDPM (Dense Discrete Phase Model) [9].

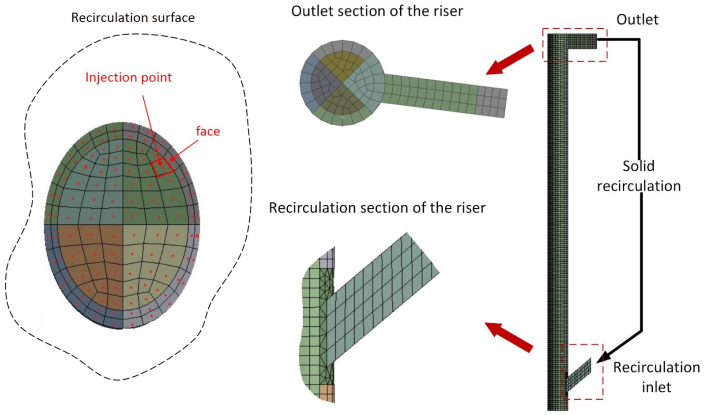


Figure 11: Recirculation scheme with close-up for recirculation surface

The number of injection points is defined by the number of faces composing the surface and points are located in the middle of each face. The location of the particle parcel injection point at the beginning of the time step cannot be changed, therefore only one particle parcel diameter could be simulated at one time. Otherwise, to keep diversity in particle sizes few injection files would have to be assigned to one surface. It would result in the same position of injection points

for different particles which would lead to unnatural overlapping (δ). Thus DEM simulations were conducted for three different cases, each with different particle diameter but constant mass of material in the riser (1.0 kg). Calculations were initialized by providing particle parcels into the riser. Three particle diameters ware tested, 275 μm, 325 μm, and 513 μm. For all cases the Gidaspow drag model was used to calculate interphase exchange coefficient (K). The number of parcels injected into the computational domain were 230,000, 135,000, and 75,000, respectively. To account for differences in particle diameters the different particle parcels diameters were set: 0.0015 m, 0.0018 m, 0.0022 m for particle parcel diameter equal to 275 μm, 325 μm, and 531 μm respectively.

4.2. Results

The results have been illustrated using the contours of solid volume fraction and pressure distribution in a pilot-scale rig. Pressure values, at different heights of the riser were gathered in table 2 together with results obtained in the experiments and results of modelling multiphase flow using DDPM. The changes of pressure and volume fractions are illustrated after 20 s of simulations. From the tenth second of simulations data sampling for time statistics was enabled to obtain values averaged in time.

Making use of gathered values of pressure, for visualizing results the figure 12 was created. Pressure values obtained from simulations using DEM are higher than experimental and DDPM data as shown at figure 12. The greatest difference can be noticed for the particle diameter 531 μm, for the smaller diameters the difference slightly decreases. For cases with smaller particle diameters (325 μm and 231 μm) dissimilarities in results are smaller because the particle parcel diameter is approaching the particle diameter. The forces resulting from collisions are weakened since the mass of the parcel is decreased. To obtain results similar to experimental values, the parcel diameter should be equal to the particle diameter in such a way one parcel would contain only one particle. However, the calculation time would be extended due to increased number of particle parcels. Hence the probability of interparticle collisions would also be greater.

Due to wrongly predicted, exaggerated collision forces, the uniform distribution of the material in the riser occurs in a case with the biggest particle parcel diameter which can be seen in Fig. 13. Due to larger mass, parcels are reflecting from walls and each other constantly increasing energy. The obtained results are opposite to expected, whereas in a case with bigger particles they should accumulate at the bottom of the riser and only part of the material should occupy the top. Forces resulting from collisions are overpredicted, since the momentum is not calculated for the particle but for the parcel, which diameter is bigger. The decrease of the size of the parcel results in similar pressure values as the experimental data.

Table 2: Pressure measurements points

Measurement points, -	Height, m	Pressure, Pa				
		Experiment	DDPM	DEM		
				d=513um	d=325um	d=231um
1	0	1002.08	1076.45	1278.95	1281.91	1262.94
2	0.07	562.61	917.91	1209.33	1110.63	1078.90
3	0.35	348.93	518.16	991.80	677.58	668.26
4	0.7	216.77	317.80	735.62	497.83	488.05
5	1	141.12	210.19	519.72	373.65	363.71
6	1.3	75.98	122.14	307.72	246.80	237.26
7	1.75	0.00	0.00	0.00	0.00	0.00

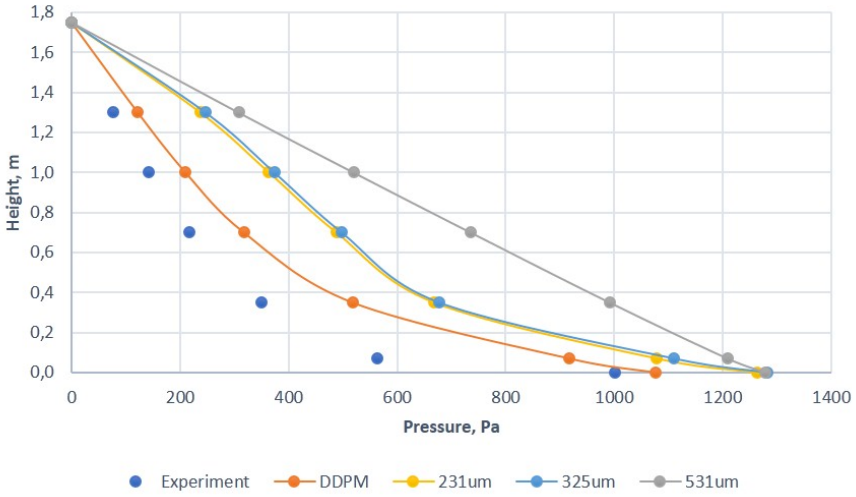


Figure 12: Time averaged pressures comparison for different cases

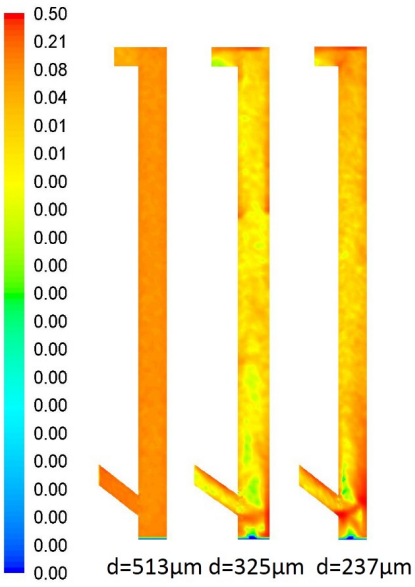


Figure 13: Averaged volume fraction distribution of phase 2 in pilot-scale rig

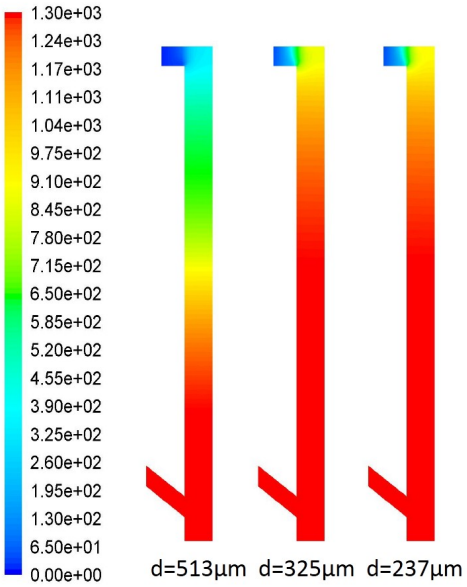


Figure 14: Pressure averaged distribution of phase 2 in pilot-scale rig

5. Conclusions

The work was divided into two parts: simulations of heat transfer together with combustion and modelling multiphase flow in a pilot scale CFB testing rig. For this purpose the commercial code Ansys Fluent was used. For predicting interparticle collisions DEM was applied, as the main goal of the thesis was to check and investigate if it is applicable to such problems. The temperature and particle distribution in a domain with changes

in its temperature were discussed. In the first part, by not having experimental results, analythical calculations were made to check global mass and energy conservation from simulations. Comparing temperature of injected sand particles, while only hot gas was flowing through domain, the values obtained both from Ansys Fluent and external source gave similar values (temperature rise of 10 K). Small value called into question if DEM model do not taint results. After further investigations and additional calculations it was found that heat transfer is not calculated for single particle but for a parcel composed of particles (in order to decrease computational cost). If the parcel had the diameter/mass of a single particle, the increase in temperature would be higher than 410 K. Furthermore, the results obtained from simulating combustion of coal in a combustion chamber almost covered with external calculations. At the outlet concentration of flue gas compounds was slightly lower in domain with two side ports. Taking advantage of applying UDF, that allowed nipping mass of char and ash off at the outlet of a domain the energy balance was calculated. For both cases similar value of energy absorbed by a sand to increase its temperature was obtained. Even though two variants can be treated as good solution, it leans towards applying two inlets of fuel to a chamber since results stabilize faster. In the second part, a trial to simulate multiphase flow through a pilot-scale test rig. Results of calculations diverged from experimental and DDPM values. Pressure distribution did not followed volume fraction of the solid phase in domain. Pressure rised from the bottom through the top while the volume fraction of the solid phase was biggest at the bottom and on the top, diluted in the middle of the riser. Assumed at the beginning that only for simulations particle parcels are created in order to simplify calculations, not influencing the results of collisions between particles. Throughout additional simulations, it was found out that similarly to heat transfer between particles also momentum is calculated for the particle parcel. To obtain exact results of interparticle interactions the particle parcel should contain only one particle inside, so is have the same diameter as particle itself.

References

[1] A. Szlek, R. Wilk, S. Werle, N. Schaffel, Czyste technologie pozyskiwania energii z wegla oraz prespektywy bezplomieniowego spalania, Rynek spalania 25 (2009) 100–110 (2009).

[2] P. Palamarczuk, J. Konieczynski, Ekologiczne walory stosowania kotlow fluidalnych w sektorze komunalnym, Prace Naukowe GiG Gornictwo i Srodowisko 3 (2005) 99–118 (2005).

[3] Y. Zhang, L. Qinghai, Z. Hui, Theory and calculation of heat transfer in furnaces, 1st Edition, Academic Press, 2016 (2016).

[4] K. Nuortimo, State of the art cfb technology for flexible large scale utility power production (March 2015). URL `http://www.amecfw.fr/documents/downloads/technical-papers/state-of-the-art\ \-cfb-technology-utility-power-production.pdf`

[5] H. Topal, Y. Altinsoy, O. Erbas, Des ign and application of automatic control system in a laboratory scale fluidized bed coal combustion system, Journal of Science 5(2) (2018) 49–63 (2018).

[6] P. Cundall, O. Strack, A discrete element model for granular assemblies, Géotechnique, 1979 (1979).

[7] J. Szargut, Termodynamika techniczna, Wydawnictwo Politechniki Slaskiej, 2003 (2003).

[8] M. Nikku, P. Jalali, T. Hyppänen, Comparison of Ansys Fluent and OpenFoam in simulation of circulating fluidized bed riser., 12th International Conference on Fluidized Bed Technology 62 (2016) 100–111 (May 2016).

[9] A. Bartoszewicz, Hybrid Euler-Lagrange approach for modelling transport phenomena in a laboratory CFB unit, Wydawnictwo Politechniki Slaskiej 2018.