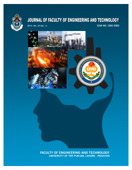
JFET 24(2) (2017) 79-88

Journal of Faculty of Engineering & Technology

journal homepage: www.pu.edu.pk/journals/index.php/jfet/index



CFD SIMULATIONS OF CIRCULATING FLUIDIZED BED COMBUSTOR WITH PROBABILITY DENSITY FUNCTION APPROACH

K. Saleem[1*], R.U. Khan[2], A. Ijaz[2]

[1]NFC Institute of Engineering & Technology, Khanewal road Multan, Pakistan,
[2]Institute of Chemical Engineering & Technology, University of the Punjab, Lahore, Pakistan

Abstract

The focus of this work is to provide a coal combustion simulation in circulating fluidized bed combustor, which is easy and simple and enhances combustion efficiency, reduces operating and investment costs and air pollutant emissions of different fuels or mixture. Also, research data available on Pakistani coal is quite limited, the objective is to increase local coal utilization by using circulating fluidized bed technology. Computational Fluid Dynamics (CFD) is a robust, easy, cost effective and a comprehensive technique through which complex processes like combustion can be studied. It was finally concluded that the air/oxygen flow rate, coal feed rate, type of coal and particle size of coal have clear impact on the overall combustion characteristics. There is less than 5% error when the results of CFD model were compared with the experimental findings.

Keywords: Circulating fluidized bed, Modelling, Low grade coal, Combustion, CFD

1. Introduction

Circulating fluidized bed combustor (CFBC) has gained considerable attention in industrial applications such as drying, combustion, cracking, gasification, catalyst regeneration and pyrolysis and for combustion of low-grade coal, natural waste, biomass and their blends [1,2]. Industrial circulating fluidized bed combustor, have many operational preferences, including the fuel adaptation capacity, high content of sulfur capture efficiency, a low emission of NOx, SOx and the high level of ignition [3,4,5,6]. For multiphase flows, computational fluid dynamics (CFD) simulation is helpful for the analysis of a circulated fluidized bed combustor [7,8,9].

The purpose of the present work is to develop a real-time model to describe the dynamic behavior of the solids inventory in the riser section which being the main focus of this real-time model [10, 11]. Several chemical, physical and dynamic phenomena are involved in the understanding of combustion [12]. These include turbulent mixing, devolatilization, homogeneous and heterogeneous reactions, and heat transfer by convection and radiation. Conservation laws of energy, mass and momentum represented by partial differential equations (PDE) are applied for flow of fluid.

---

* Corresponding Author: kanwarsaleem@nfciet.edu.pk

At present three numerical techniques are used for the understanding of the combustion in fluidized beds, these are Eulerian-Eulerian, Eulerian-Lagrangia and Discrete Element Method (DEM). The Most of studies for dense bed are focused on gasification and Eulerian-Eulerian (E-E) Two Fluid Model (TFM) approach was used for very simple geometries in literature. Few of them [13, 14, 15] used CFD for coal combustion in CFB and overlooked the effects for three-dimensional configuration. However, only two authors [16, 17] considered the three-dimensional or full-scale geometry of the unit to study the combustion in circulating fluidized bed combustor using Eulerian-Eulerian(E-E) approach.

2. Materials and Methods

2.1 Computational Domain Development

The lab scale CFB equipment has already discussed in our earlier work [18] along with experimental conditions. The most important part of the CFB is the fluidized bed column, the 3D model of fluidized bed column has been developed as shown in Figure- 1 using ANSYS Design Modeler®14, an inbuilt application of ANSYS Workbench environment. The diameter of column is 80 mm whereas the height of column is 3900 mm. The dimensions are similar to that of real CFB available at Laboratory.

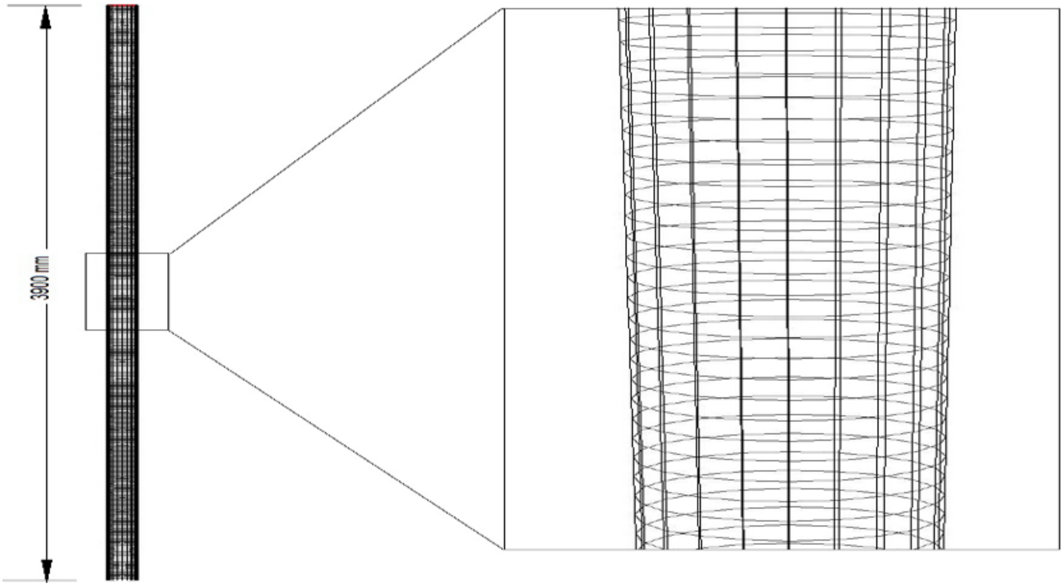


Figure 1: A view of meshed 3D Geometry modeled in Ansys Design Modeler

The Navier-Stokes equations in the steady-state mode along with equations of conservations for mass and energy were solved. In energy equation, the first three terms on right hand side represent the viscous dissipative, diffusive and conductive heat transfer. Enthalpy formation from chemical reactions of the species is represented by $S_h$as a source term. In the present study of modeling, standard $k–\varepsilon$ turbulent model was applied. The turbulence kinetic energy G_k was generated by mean velocity gradients as shown in the equations of turbulent kinetic energy (k) and dissipation rate (ε). For modeling of mass and heat turbulent flux, coefficient of diffusion (D_t) and thermal conductivity (λ_t) were used.The P-1 radiation model was used to calculate the radiated flux at the inner walls of circulating fluidized bed and this is the simplest model applicable to different geometries and is also computationally more economical. The low temperature levels (heat sinks) and the radiated high temperature levels (heat source) were calculated. The radiation flux ($q_{r,w}$)on the walls is affected by the incident radiation (G_w) and was also calculated.

In present research the combustion process was taken as a mixing problem with the help of PDF/mixture fraction technique for modeling the chemistry of gas phase combustion. The chemistry equilibrium model was applied with the assumption of fast chemistry so that chemical equilibrium was present to the molecular level and char and coal volatiles were considered as single fuel stream. The mixture fraction could be expressed in the form of atomic mass fraction. Elemental mass fraction is represented by $Z_i$for element *i*. *fuel* and *ox* subscripts are used to give the values of fuel stream and oxidizer inlets respectively. The value of the mixture fraction which is a scalar was estimated at each control volume by getting the solution of the given transport equation for the (density-averaged) value of *f*. The source term, S_m, is due to the transfer of mass into the gas phase from reacting coal particles. In addition to the solution of the Favre mean mixture fractions, Fluent also solves a conservation equation for the mixture fraction variance, $\bar{f}'^2$.

Values for different constants are tabulated in Table 1 applied in the current modeling

Table 1: Constants used in Turbulence Modeling

Sr. No	Constant	Value
01	$C_{1\varepsilon}$	1.44
02	$C_{2\varepsilon}$	1.92
03	C_μ	0.09
04	σ_k	1.0
05	σ_ε	1.3
06	Pr_t(Prandtl number for Turbulence)	0.85
07	Sc_t(Schmidt number for Turbulence)	0.7

2.2 Boundary Conditions and Case Set Up

At the inlet, all velocities and temperatures are specified. Pressure is not specified at the inlet because of the incompressible gas phase assumption (relatively low pressure drop system). The solid and gas phase initial velocity was specified. In order to analyze best results, fine meshing was carried out after simple meshing for riser at inlet and exit sections. Under relaxation factors were tuned to achieve convergence. The set point of 0.001 was given to convergence tolerance. Iteration procedure of calculation was adopted by ANSYS to check the parameters like flow into the system. Following important steps are involved in iterations like constants, physical conditions, initial guess or initial data values and boundary conditions, while in second step, momentum equation and velocity field are set to be calculated.

Pressure equation and mass balance are then solved.In the next step values of parameters are updated for solid and gas phases. The convergence criterion is then verified to see whether it is fixed or not. Final result is achieved when the convergence criterion is reached. If not, certain correction values are used to adjust the calculated values and the calculation started all over again and finally using these last corrected values of each parameter as initial data.

2.3 Grid Independency Test

As per standard practice the developed meshed computational domain was tested for grid independence. Four meshes were created from coarse grid to fine grid range. The cell sizes are 13684 (coarse), 27368 (medium coarse), 51937(medium fine) and 352428 (fine). The case 1 was converged on all these meshes and the temperature along central axis was plotted along the height of combustor. Figure 2 shows the comparison. It can be observed that the temperature profile for medium coarse grid (27368), medium fine (51937) and fine grid (352428) shows almost similar trend. It shows that the solution become independent from medium coarse size of grid. So that grid was selected for the solution of further cases.

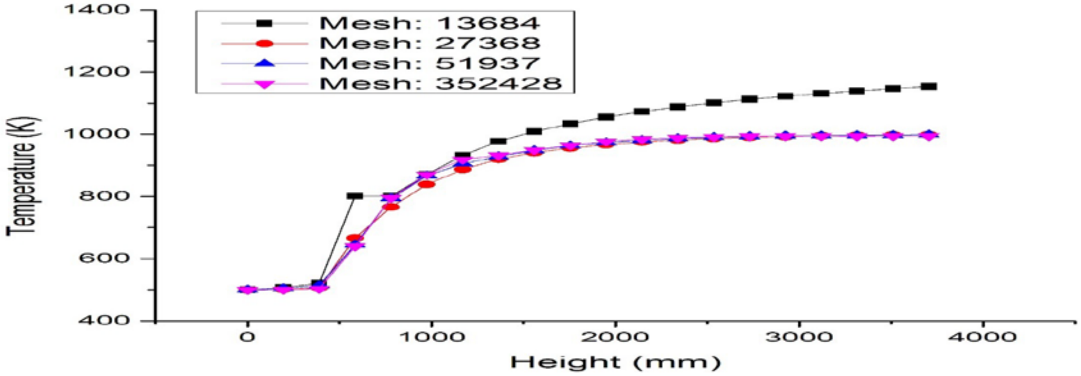


Figure 2: Grid independence test against various numbers of meshes for temperature

3. Results and Discussion

3.1 Model Validation

For validation, five cases were simulated. The description of all the cases is given in Table 2.

Table 2: Simulation Cases Description for Validation

Case	Velocity of Air	Flow rate of air	Flow rate of Coal	Particle Size of Coal
	m/sec	m^3/hr	Kg/hr	µm
1	0.84	15	2.6	250
2	0.93	16.5	2.6	250
3	0.97	17.3	2.6	250
4	1.02	18.2	2.6	250
5	1.11	19.7	2.6	250

As can be seen in the Table 2, the coal feed rate and particle size were kept constant and the air flow rate increased from 15 m^3/hr to 19.7 m^3/hr.

The temperature of exhaust gases is shown in Figure 3. The results show an increase in temperature with increasing air flow rate and the length of riser. This is due to the increase in air to fuel ratio in the combustor as per previous studies [19, 20].

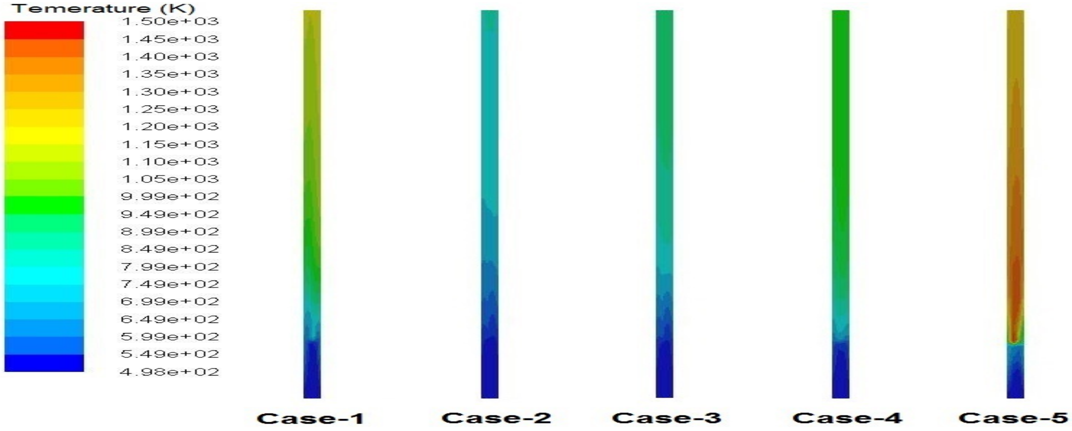


Figure 3: Contours of Flue Gas Exit Temperatures

For validation, the exhaust temperatures measured from simulated cases are compared in Figure 4. The validation curves show a significant resemblance in outlet temperatures for all simulated cases and experimental values. Less than 5% error was observed on comparison of experimental and simulation results. This shows that the selected strategies for CFD modelling and the governing equations were suitable for the cases under consideration. All the numerical settings and discretization schemes fitted well to achieve the results having less than 5% error which is in most of previous cases [21, 22]. The model may therefore regard as validated.

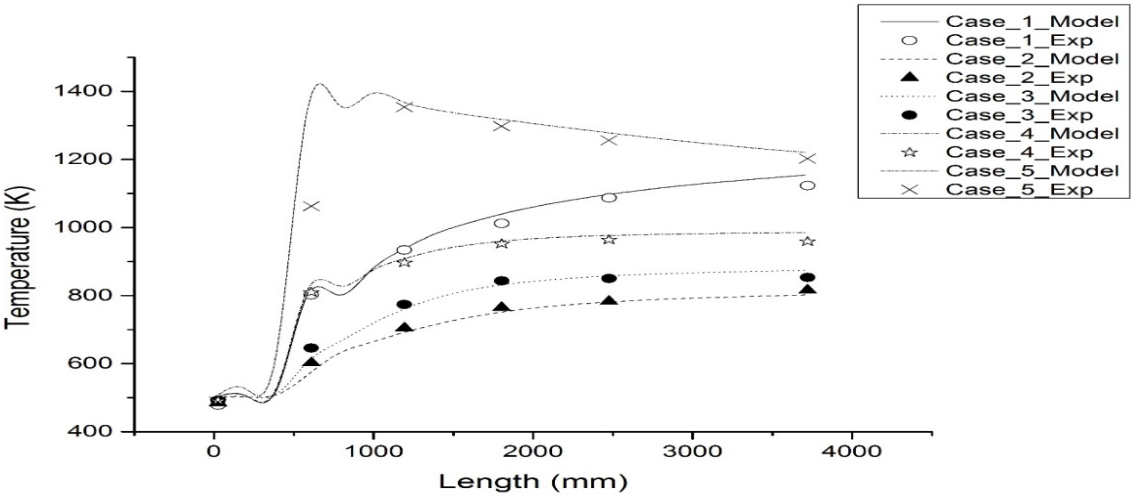


Figure 4: Comparison of flue gases outlet temperatures from experiments and all simulated cases

3.2 Effect of Particle Size

Particle size is an important parameter in combustion. Three different simulations were carried out with 250µm, 500µm and 750µm particle size. The rest of the parameters like coal and oxygen feeding rates, inlet temperatures and type of coal were kept constant. The effect on the conversion of coal, exit temperature and flue gas analysis was monitored. It can be seen from the figure 05 that the conversion of coal is maximum (93%) at 250µm particle size and gradually decreases with the increase in particle size. Similar effect can be observed in the exhaust gases outlet temperature, which is maximum (about 881 °C) for particle size of 250µm. It is a common fact that with the decrease of particle size, the surface area would increase and the chance of reaction occurrence increases. Hence, with the particle size of 250μm, further the effects of particle size on char conversion from the same figure.

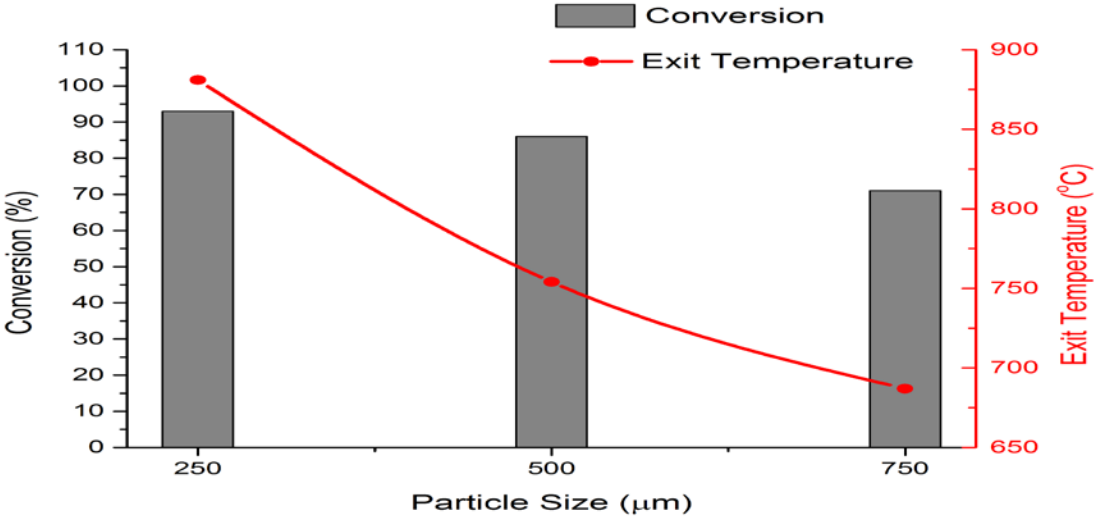


Figure 5: Effect of Particle size on Coal Conversion and Temperature of Exhaust Gases

Moreover, the effect of the particle size variation on temperature inside the combustor can be observed from the contours shown in Figure 06.

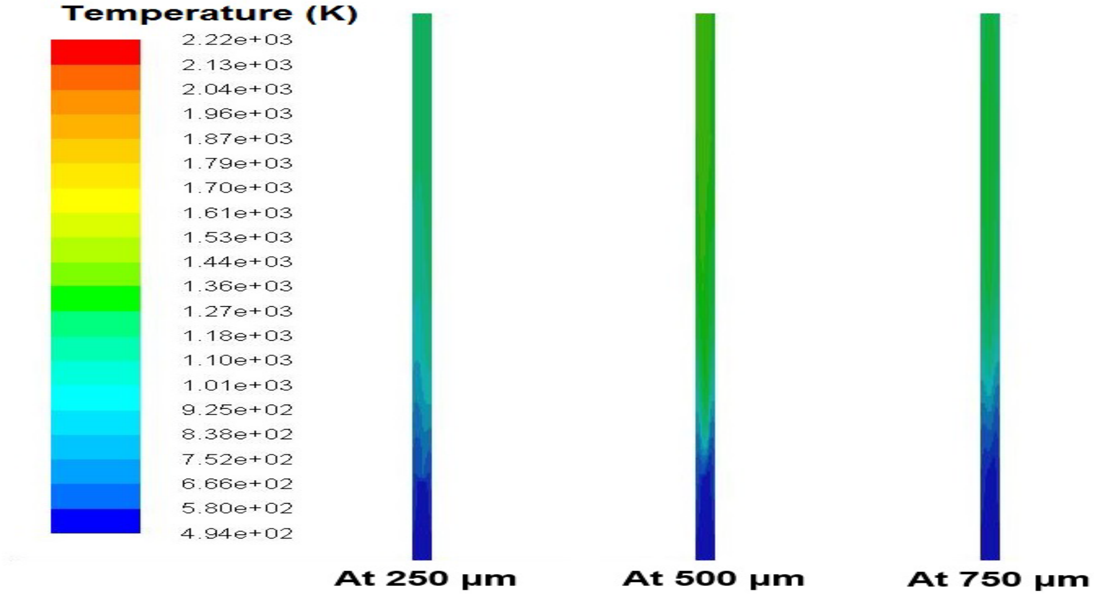


Figure 6: Temperature contours at different particle sizes

The effects of particle size could be verified from the flue gas analysis of all the three cases, as shown in Figure 7. The continuous decrease in CO_2 and increase in O_2 in exhaust gas analysis further confirms the less reactivity of coal on increase of coal particle size. The CO_2 in exhaust gases was maximum 7.9% at 250μm particle size whereas the O_2 was minimum (6.3 %), meaning thereby that maximum O_2 had been utilized in the reaction. The minimum CO_2 (4.2 %) was found at 750μm particle size where O_2 was maximum (13.2%), which meant more O_2 wastage.

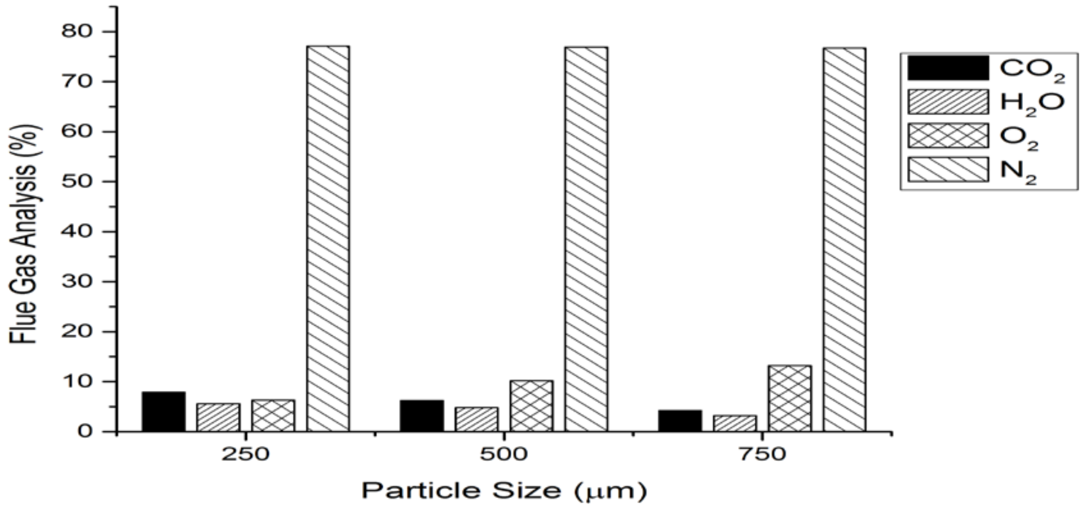


Figure 7: Effect of Particle size on Flue Gas Analysis

3.3 Effect of coal feeding rate

At the time of validation, five different cases were simulated by varying the amount of air in terms of its velocity and volumetric flow rate (Table 3). To investigate the effect of coal feed rate on the performance of combustor, five more cases were simulated keeping the air feed rate and particle size constant and the only variation was that of coal feed rate. As the experimental work was carried out at 2.6 Kg/hr (coal feed rate) [25], so this was taken as the base and further cases were simulated by adding half of its amount, 1.3 Kg/hr, each time. The details are tabulated in Table 3.

Table 3: Cases Description for Variation in Coal Feeding Rate

Case	Velocity of Air	Flow rate of air	Flow rate of Coal	Particle Size of Coal
	m/sec	m^3/hr	Kg/hr	μm
1	0.84	15	2.6	250
2	0.84	15	3.9	250
3	0.84	15	5.2	250
4	0.84	15	6.5	250
5	0.84	15	7.8	250

The amount of coal has direct impact on the stoichiometric relation with the amount of oxygen. If amount of coal would be increased at constant oxygen/air rate then there will be imbalance in the stoichiometric rate of oxygen to the coal.

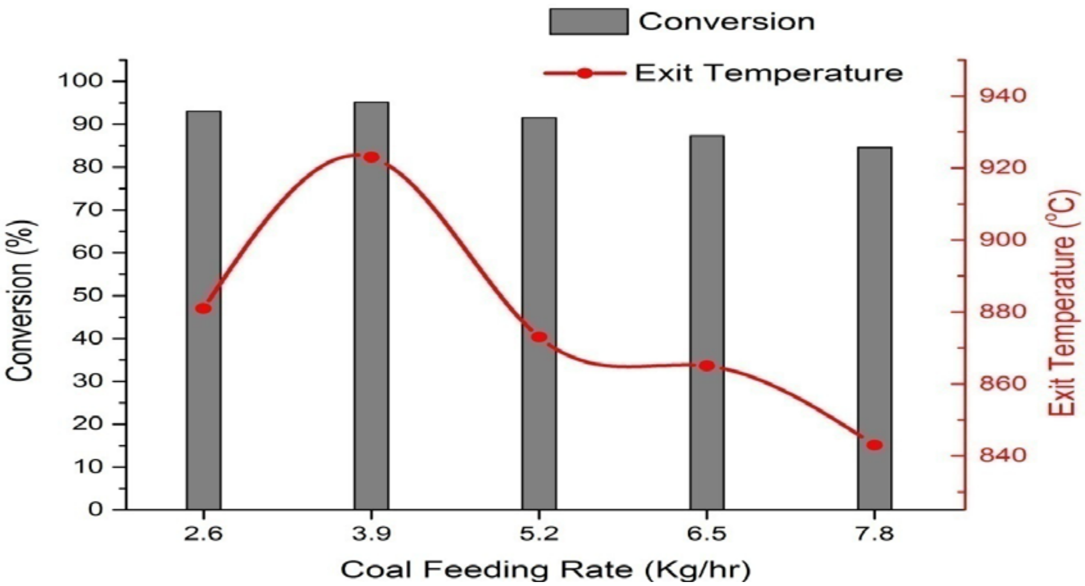


Figure 8: Effect of Coal Feeding Rate on Coal Conversion and Temperature of Exhaust Gases

This effect can be verified from Figure 8 in which conversion is decreased from 93% to 84.6% on increase of coal feeding rate from 2.6 Kg/hr to 7.8 Kg/hr. This effect is replicated in the exhaust temperature, as lower conversion means slightly lower temperature. The conversion goes to a maximum of 95.2% at 3.9 Kg/hr coal feed rate. At this maximum conversion, the temperature reaches 923 °C for the exhaust gases.

4. Conclusions

CFD simulations results provided a fact that particle size has great impact on the overall combustion characteristics. It was observed that the conversion of coal in to gaseous products was maximum (93%) at 250μm particle size and gradually decreased with the increase in particle size. Similar effect was observed in the exhaust gases outlet that had a maximum temperature (881^{o}C) for the same particle size (250 μm).On increasing oxygen, slight increase of CO in ppm was observed whereas the NO_X and SO_X formation decreased due to less reactivity and less temperature in the combustor. It was concluded that CFD is a robust, easy, cost effective and a comprehensive technique through which complex processes like combustion can be studied with acceptable level of confidence and there is less than 5% error when the results of CFD model were compared with the experimental findings.

References

[1] R. Bithi, and S. Bhattacharya. "Oxy-fuel fluidized bed combustion using Victorian brown coal: An experimental investigation." Fuel Processing Technology 117 (2014): 23-29.

[2] R. I. Singh, and K. Ghule. "Design, development, experimental and CFD analysis of a prototype fluidized bed stripper ash cooler." Applied Thermal Engineering 107 (2016): 1077-1090.

[3] M. Bartels, J. Nijenhuis, J. Lensselink, M. Siedlecki, W. de Jong, F. Kapteijn, and J. R.van Ommen. "Detecting and counteracting agglomeration in fluidized bed biomass combustion." Energy and Fuels 23 (2008): 157-169.

[4] G. Afsin, and N. Eskin. "Effects of operational parameters on emission performance and combustion efficiency in small-scale CFBCs." Journal of the Chinese Institute of Chemical Engineers 39 (2008): 541-556.

[5] K. Adam, W. Adamczyk, A. Katelbach-Woźniak, G. Węcel, and A. Szlęk. "Towards a hybrid Eulerian–Lagrangian CFD modeling of coal gasification in a circulating fluidized bed reactor." Fuel 152 (2015): 131-137.

[6] I. Malgarinos, N. Nikolopoulos, P. Grammelis, S. Karrelas, and E. Kakaras."A decoupled approach for NOx–N2O3-D CFD modeling in CFB plants." Fuel 115 (2014): 401-415.

[7] T. Eman, and E. Dahlquist. "CFD Ansys-Fluent simulation of prevention of dioxins formation via controlling homogeneous mass and heat transfer within circulated fluidized bed combustor." Energy Procedia 75 (2015): 130-136.

[8] S. A. Kumar, P. Gupta, and R. K. Saha. "Modeling and experimental investigations on the pyrolysis of large coal particles." Energy & Fuels 25 (2011): 5573-5583.

[9] Y. Cheng, and J. Zhu. "Hydrodynamics and scale-up of liquid–solid circulating fluidized beds: Similitude method vs. CFD." Chemical Engineering Science 63 (2008): 3201-3211

[10] D. Abbas, J. J. Zhu, and C. Zhang."A computational fluid dynamics study on the flow field in a liquid–solid circulating fluidized bed riser." Powder Technology 260 (2014): 52-58.

[11] E. Hartge, L. Ratschow, R. Wischnewski, and J. Werther. "CFD-simulation of a circulating fluidized bed riser." Particuology 7 (2009): 283-296.

[12] W. Zhong, A. Yu, G. Zhou, J. Xie, and H. Zhang. "CFD simulation of dense particulate reaction system: Approaches, recent advances and applications." Chemical Engineering Science 140 (2016): 16-43.

[13] W. Zhou, CS. Zhao, LB. Duan, CR. Qu, and XP. Chen. "Two-dimensional computational fluid dynamics simulation of coal combustion in a circulating fluidized bed combustor." Chemical Engineering Journal 166 (2011): 306-314.

[14] W. Zhou, C. Zhao, L. Duan, D. Liu, and X. Chen. "CFD modeling of oxy-coal combustion in circulating fluidized bed." International Journal of Greenhouse Gas Control 5 (2011): 1489-1497.

[15] W. Zhou, CS. Zhao, LB. Duan, CR. Qu, and JY. Lu. "Numerical simulation on hydrodynamics and combustion in a circulating fluidized bed under O2/CO2 and air atmospheres." Proceedings of the 20th International Conference on Fluidized Bed Combustion. Springer, Berlin, Heidelberg, 2009.

[16] A. Nikolopoulos, I. Rampldls, N. Nlkelopoules, P. Grammells, and E. Kakaras. "Numerical investigation of 3-D transient combusting flow in a 1.2 MWth pilot power plant." Proceedings of the 20th International Conference on Fluidized Bed Combustion. Springer, Berlin, Heidelberg, 2009.

[17] K. Myöhänen, T. Hyppänen, and A. Vepsäläinen. "Modelling of circulating fluidized bed combustion with a semi-empirical three-dimensional model." Proceedings of the 47th conference on simulation and modelling, Helsinki, 2006.

[18] K.S. Akhtar, S. Hussain, RU. Khan, and A. Ijaz. "thermo chemical studies and fluidized bed combustion of low grade pakistani coal blends." Life Science Journal 10 (2013): 3358-62.

[19] D. Lundmark, C. Mueller, and BJ. Skrifvars. "Computational fluid dynamic modeling of combustion and ash deposition in a biomass-cofired bubbling fluidized bed boiler." International Journal of Energy for a Clean Environment 8 (2007): 155-169.

[20] D. Lundmark, C. Mueller, R. Backman, M. Zevenhoven, BJ. Skrifvars, and M. Hupa. "CFD based ash deposition prediction in a BFBC firing mixtures of peat and forest residue." Journal of Energy Resources Technology 132 (2010): 031003.

[21] C. Mueller, D. Lundmark, BJ. Skrifvars,and M. Hupa. ”CFD based ash deposition prediction in a bubbling fluidised bed combustor firing mixtures of biomasses." Proceeding Finnish-Swedish Flame Days, Vassa, Finland, 2002.

[22] M. Rozainee, SP. Ngo, AA. Salema, and KG. Tan. "Computational fluid dynamics modeling of rice husk combustion in a fluidised bed combustor." Powder Technology 203 (2010): 331-347.

[23] C. Mueller, D. Lundmark, BJ. Skrifvars, and M. Hupa. "Numerical simulation of the combustion behavior of different biomasses in a bubbling fluidized bed boiler." Proceeding of the 18th International Conference on Fluidized Bed Combustion, Toronto, Ontario, Canada, 2005.

[24] R.I. Singh, A. Brink, and M. Hupa. "CFD modeling to study fluidized bed combustion and gasification." Applied Thermal Energy 52 (2013): 585-614.

[25] S. Hussain. “Factors Influencing the Formation of Producer Gas from Coal and Biomass in Circulating Fluidized Bed Gasifier." PhD thesis, The University Of the Punjab Lahore- Pakistan, 2015.