CFD modeling of the gas–particle flow behaviour in kiln burner pipe.

Z.NGADI[1], **M.L.LAHLAOUTI**[2]

1. Laboratory: Energy, University Abdelmalek Essaadi, Faculty of sciences of Tetouan. E-mail: zakia.ngadi@gmail.com
2. Laboratory: Energy, University Abdelmalek Essaadi, Faculty of sciences of Tetouan. E-mail: hlahlaouti@hotmail.com

Abstract

Because of high environmental impact and energy costs, many cement plants tend to substitute waste-derived fuel for fossil fuels. The combustion of waste-derived fuels in the rotary kiln burner is more challenging than in the precalciner where there is a less strict temperature requirement and where gravity facilitated feeding is possible. This paper present a 2D computational fluid dynamic conducted for a cement rotary kiln burner. Simulations are performed using commercial COMSOL multiphysics and are conducted for coal and olive pomace using Euler-Euler model and the k-ε model for swirl and jet air transport in the burner inlet. The predict result will after that using in the combustion model.

Key words: *Euler-Euler model, K-ε model, kiln burner pipe, alternative fuel***.**

1. Introduction

The range of waste-derived fuel types using in cement kiln is extremely wide. Nevertheless, one cannot expect a combustion performance similar to that of conventional fuels when burning such waste fuel [1]. However, knowing the impact of using such waste fuels is important for improving the process economy. Even though these types of studies have been carried out to model coal combustion in rotary kilns, Kaantee et al used the Aspen plus process modelling tool to investigate the influence of the fuel change on the combustion [2].

3D-CFD modelling for full scale rotary cement kiln with multi-channel coal burner can be found in [3]. The calculation of the fuel particle velocity and charge in the burner end is one of the main ingredients in the simulation of fuel combustion. The particle velocity plays a major role in the interphase processes that are the driving force of fuel combustion, such as particle drag and turbulent dispersion, heat-up and heterogeneous combustion. The fuel particles velocity has also a clear influence on the convective transfer of heat within the kiln. The waste-derived fuel is transported under the burner pipe by air, the particle velocity and charge in the burner end is calculated using the Euler-Euler model in COMSOL software. The results obtained will be used as an inlet condition and values for the combustion model.

2. Euler-Euler model

Eulerian-Eulerian models use Eulerian conservation equations to describe the behaviour of the gas and particulate phases in a multiphase flow [4]. The local share of space occupied by each phase is given by the phase volume-fraction r, (x, y, z, and t), which is obtained from the conservation equation:

$$\frac{d}{dt}\left(r_i \rho_i\right) + \nabla\left(r_i \rho_i v_i\right) = 0 \tag{1}$$

Where i=gas (g); i=particle(s) / $r_g+r_s=1$

The transport equation is given by:

$$\frac{\partial}{\partial t}\left(r_i \rho_i v_i\right) + \nabla\left(r_i \rho_i v_i v_i\right) = -r_i \nabla p + \nabla.\left(r_i \tau_i\right) + r_i \rho_i g - \beta v \tag{2}$$

The stress tensor of both phases is often modelled using the Newtonian strain-stress relation:

$$\tau_i = \xi_i \left(\nabla v_i\right) I + \mu_i \left[\nabla v_i + \left(\nabla v_i\right)^T\right] - \frac{2}{3}\mu_i \left(\nabla v_i\right) I \tag{3}$$

Non-Newtonian properties of the particulate phase can be taken into account by modelling the particle viscosity. The bulk viscosity of phase ξi is commonly set to zero in both phases, in accordance with Stokes' assumption [5]. In this work, Gidspaw approach is used [6].

3. K-ε Model

The turbulence model has proven, over the years, to be a useful engineering approach for the prediction of the mean velocity profiles of turbulence flows.

The steady-state continuity equation can be written as:

$$\vec{\nabla}.\vec{v} = 0 \tag{4}$$

The velocity components are given by the eq. (5), wich is the steady-state momentum equation. Here, P is the static pressure. T is the shear forces and is called a stress tensor.

$$(\vec{v}.\vec{\nabla})\vec{v} = -\frac{1}{\rho}\vec{\nabla}p + \nu\,\vec{\Delta}\vec{v} + \vec{f} \tag{5}$$

The steady-state equation for the turbulence kinetic energy and turbulence kinetic energy dissipation rate are as follows:

$$\rho(\bar{U}.\nabla)K = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla K\right] + P_k - \rho\varepsilon \tag{8}$$

$$\rho(\bar{U}.\nabla)\varepsilon = \nabla\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right] + C_{\varepsilon1}\frac{\varepsilon}{K}P_K - C_{\varepsilon2}\rho\frac{\varepsilon^2}{K} \quad (7)$$

4. Computational details

4.1. The geometry

A two-dimensional pipe burner rotary kiln was created in COMSOL. Modified dimensions and shapes were used for the burner inlets compared with the real system to facilitate the modelling (figure.1). However, care was taken to use a flow area that would conserve not only mass but also momentum; the total cross-section area of each of those in the real system.

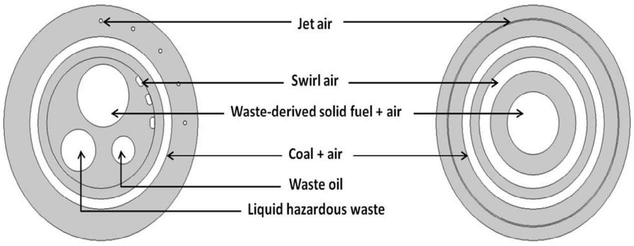

Figure.1: The real (left) and simplified (right) kiln burner

4.2. Boundary conditions:

The air inlets are treated as velocity inlets. The velocity is calculated from known mass flow rate from the real kiln system. Fuel conveying air mass flow rates are fuel-type dependent (table1). The outlet of the system is treated as a pressure outlet. All of the walls are treated as adiabatic and no-slip walls.

Table.1: Momentum and thermal conditions at air and fuels inlets.

Boundary	Mass flow rate (kg/s)	Temperature (K)	Velocity (m/s)	Density (Kg/m3)
Central tube for fuel	AF : 0,83	373	28	600
	Air : 3	373	148	0,295
Swirl air	1,18	373	126	0,256
Fuel annulus	Coal :1,8	373	65	900
	Air : 3	373	148	0,295
Jet air	0,38	373	123	0,256

5. Results and discussion

The velocity of gases and both fuels are analysed and discussed below, and also the charge and pressure will be given by simulation.

Two models have been used in this work. The Euler-Euler model is used to compute fuels transport, and K-ε model for the swirl and jet air.

The Euler-Euler model is defined by [6] with the drag function model of Gibilaro [7]. However, the Newtonian phasic stress model [5], is used only for the gas phase while, for the particulate phase, the kinetic theory of granular flow is introduced.

For the particle phase, the stress tensor is given by [8] and the granular temperature is computed with [8] where gas phase turbulence is neglected. A time sequence of particles charge rising up through the bed, calculated from the two-dimensional form of the constant viscosity model is presented in figure.2.b for coal and figure.3.b for Olive Pomace. Instantaneous gas and particle velocities are shown in figure.2/3.a for fuels particles and figure.2/3.c for the continuous phases. As shown in the figures, the flow is stochastic, with strong particles interactions including both coalescence and split-up.

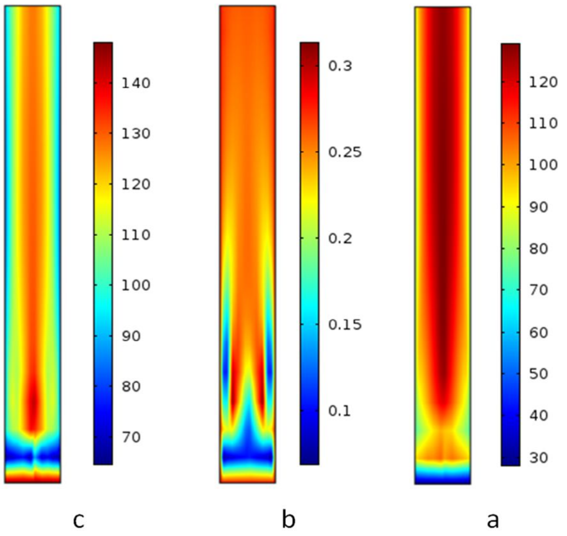

Figure.2: central tube for fuel simulated results. a) AF particles velocity (m/s), b) AF charge and c) transport air velocity (m/s)

The difference between two solids fuels velocity and charge is due to the density and the mass flow rate at the inlet of each fuel. The choice of the mass flow rate is depending on the thermal characteristics of fuel [1].

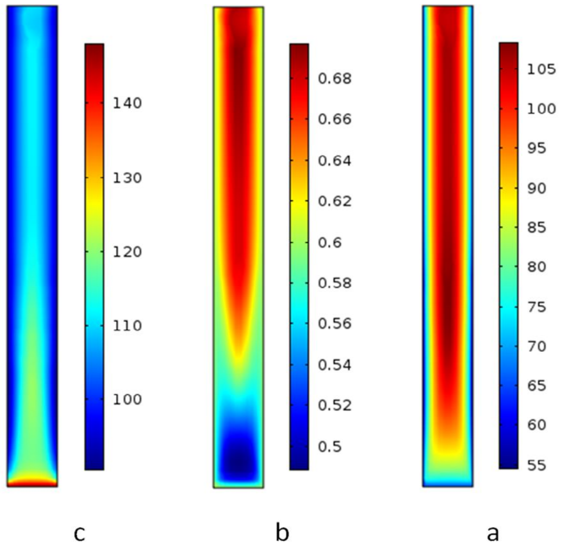

Figure.3: fuel annulus simulated results. a) Coal particles velocity, b) Coal charge and c) transport air velocity

Figure 4 and 5 give The analysis provided pressure, velocity and turbulent kinetic energy distribution within the flow domain and at boundaries for the swirl and jet air using the K-ε model. As observed from Figure.4/5.c, it is seen that turbulent kinetic energy increases from inlet to outlet. This may be due to boundary layer effect. Figure.4/5.b shows variation of pressure from inlet to outlet which decreases gradually.

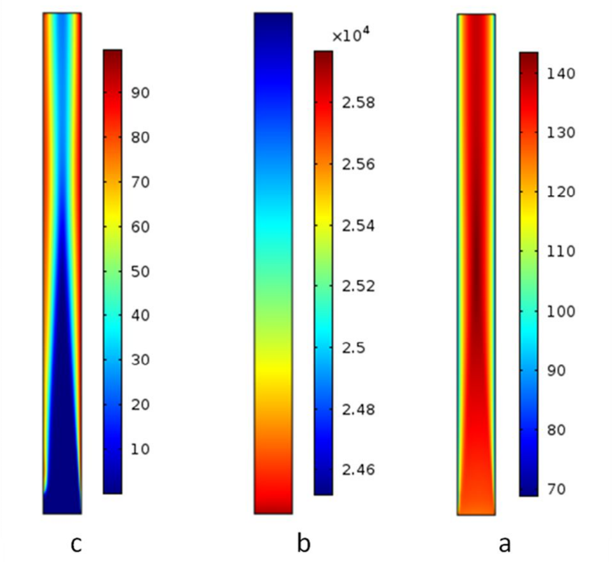

Figure.4: jet air simulated results. a) Jet air velocity (m/s), b) pressure distribution (Pa), c) kinetic energy variation (m^2/s^2)

Air velocity distribution in Figure.4/5.a indicates highest velocity at centre of pipe and least at the surface of pipe. Velocity at boundary also increases from inlet to outlet.

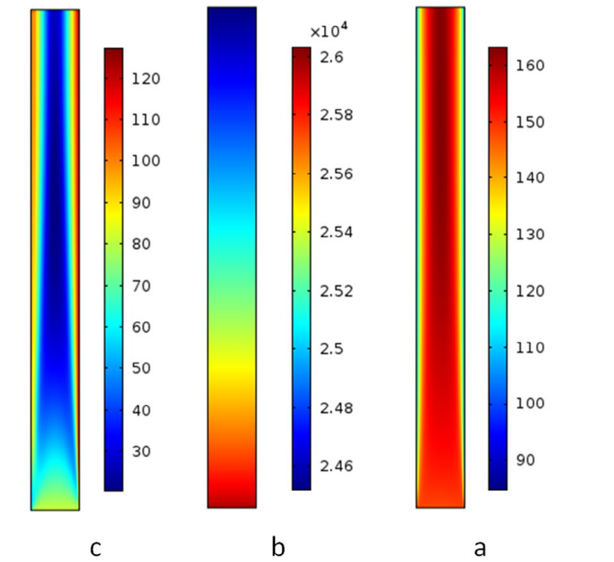

Figure.5: swirl air simulated results. a) swirl air velocity (m/s), b) pressure distribution (Pa), c) kinetic energy variation (m^2/s^2)

The results obtained from both Euler-Euler and K-ε model agree with the results obtained in the literature [9-10].

6. Conclusion

A computational fluid dynamics model was developed successfully to describe the hydrodynamics of fuels transported in kiln burner pipe. This model is based on the Eulere–Eulere two-fluid modeling approach, incorporating a kinetic-frictional constitutive model for dense assemblies of the particulate solid, a drag model for gas–particles interaction. The typical flow patterns of kiln burner pipe were obtained and compared favorably with the reported literatures results.

It may also be concluded that CFD is good tool to predict the performance of pipes in less time. The results obtained at the outlet systems will be used as inlet variables for a 2D combustion model.

References

[1] Z.Ngadi, M.L.Lahlaouti, *Alternative fuels impact of burning pomace on cement quality and process progressing,* International conference on Material and Energy(ICOM), Prof.Rachid Bennacer, Prof Mohamed el ganaoui, Prof.Abdelaziz Mimet, Tetouan, Maroc, 2015, 183

[2] U.Kääntee, R.Zevenhoven, R.Backman,M.Hupa, *Cement manufacturing using alternative fuels and the advantages of process modeling,* fuel process technol 85 (2004) 293-301.

[3] Bhad, T.P.Sarkar, S.Kaushik, A.Herwadkar, *CFD modeling of a cement kiln with multi channl burner for optimization of flame profile,* the Seventh International Conference on CFD in the Minerals and Process Industries, Australia, 2009, 9-11.

[4] N.Fueyo,J.Ballester, C.Dopazo, *An Eulerian Eulerian model of coal combustion, NO, formation and reburning*, ed. S.-H. Chiang, 12th Annual International Pittsburgh Coal Conference, Pittsburgh, USA, 1995, 1113-1118

[5] R.L.Panton, *Incompressible Flow*, Wiley, New York. 1984

[6] D.Gidaspow, *Multiphase Flow and Fluidization.* Academic Press, Boston, 1994.

[7] A. L.Gibilaro, Di Felice, S.P.Waldram, *Generalized friction factor and drag coefficient correlations for fluid-particle interactions*, Chem. Engng Sci, 40(1985), 1817-1823.

[8] G.Balzer, A.Boelle, O.Simonin, *Eulerian gas-solid flow modelling of dense fluidized bed*, Fluidization VIII Int. Symp Of the Engineering Foundation, Tours, France, 1995, 1125-1134.

[9] W.Zhonghua ,A.S. Mujumdar, *CFD modeling of the gas–particle flow behavior in spouted beds,* Powder Technology 183 (2008) 260–272

[10] A.Sharma, V.Gupta, A.K.Jain, S.K.Singh, *Numerical Simulation for Visualising Effect of Surface Roughness on Flow in Simple Pipe,* Ultra Engineer 3 (2015).