Revista Internacional de Investigación e Innovación Tecnológica

Página principal: www.riiit.com.mx

CFD model for the simulation of the combustion of alternative fuels in a rotary kiln

Modelo CFD para la simulación de la combustión de combustibles alternativos en un horno rotatorio

López-Aguilar H.A.[a,b], Gómez J.[a]., Morales-Castillo J.[c], Pérez-Hernández A.[a*]

[a] Centro de Investigación en Materiales Avanzados, Chihuahua, México.
[b] Universidad la Salle de Chihuahua, Chihuahua, México.
[c] Universidad Autónoma de Nuevo León, Nuevo León, México.
hlopez@ulsachihuahua.edu.mx; joralgomez74@gmail.com; tequilaydiamante@yahoo.com.mx; antonino.perez@cimav.edu.mx

Technological innovation: CFD model simulation that describes the combustion process in the Rotary kilns in specific industrial conditions, which represents the coal combustion, also let it simulate the thermal effect use of fuel alternative to the rotary kilns.

Industrial application area: Concrete and construction industry.

Received: april 06th, 2021
Accepted: october 04th, 2021

Resumen

Los hornos rotatorios son ampliamente utilizados en la industria del cemento, dentro de este tipo de equipos industriales ocurren reacciones químicas mientras la materia prima se mueve en el interior del horno. El perfil de temperatura en el interior del horno y el tiempo de permanencia son dos parámetros importantes para controlar la calidad del cemento, así como el tipo y la calidad de las materias primas utilizadas para formular el cemento.

La fuente de energía es la mezcla de gases de combustión que viajan en contracorriente. En este estudio se empleó el software de CFD Ansys Fluent® para analizar el proceso de combustión dentro del horno rotatorio y su interconexión con la cámara de humo. El dominio virtual del horno se dividió en pequeños volúmenes de control interconectados; para cada uno se aplicaron las ecuaciones de conservación de materia y energía y las reacciones químicas de combustión para estimar el desempeño del horno rotatorio de cemento. En este trabajo se estudia la fluidodinámica dentro del horno cementero considerando el perfil de temperatura y velocidades de los gases, así como la pérdida de energía a través de las paredes del horno. El modelo Euleriano-lagrangeano de

combustión en estado estable desarrollado se utilizó para simular el uso de lodos de depuradora y biogás como combustibles alternativos para la producción de cemento, resultando ser una herramienta con potencial para encontrar el efecto térmico de la implementación de mezclas de combustibles alternativos con baja y alta liberación de energía en el horno rotatorio en estudio.
Se presenta una comparación entre productos de combustión estimados por el modelo y valores reales obtenidos del proceso. Los resultados se limitan a las condiciones del horno cementero en estudio. El uso de herramientas de modelado y simulación computacional son elementos clave para mejorar el desempeño de la industria cementera.

Palabras clave: cemento, CFD, combustión de carbón, horno rotatorio, lodos depuradora, biogas.

Abstract

Rotary kilns are widely used for the cement industry; into this kind of industrial ovens occurs chemical reactions while raw material moves through the kiln shell. Temperature profile inside the Kiln shell and permanence time are two critical parameters to control the cement quality, as well as the type and quality of raw materials to formulate the cement.
The energy source is from the combustion gases traveled counter flow. In this study a CFD software Ansys Fluent was used to analyze the combustion process inside the rotatory kiln and its interconnection with the smoke chamber. The virtual domain of the kiln was divided into small interconnected volumes as "control volumes"; the conservation equations for matter and energy and the combustion chemical reactions were applied to each “control volumes” to estimate the performance of the rotatory cement kiln. In this work the fluid dynamics inside the kiln is studied considering the temperature and gases velocity profile, as well as the loss energy trough the kiln walls. The steady state Eulerian-Lagrangean combustion model developed was used to simulate the use of sewage sludge and biogas as alternative fuels for cement production and proved to be a tool with the potential to find the thermal effect of the implementation of alternative fuel blends with low and high energy release in a rotary kiln.
A comparison is presented between combustion products estimated and real values obtained from the process. The results are limited to the conditions of the kiln studied. The use of modeling tools and computational simulation are crucial elements to improve the performance of the cement industry.

Keywords: cement, CFD, coal combustion, kiln, sewage, biogas.

1. Introduction

Portland cement is a hydraulic binder composed of clinker as a result of the calcination of a mixture of limestone and clay. The cement industry emits about 5 to 7 % of the global anthropogenic CO_2 emissions, and it is responsible for 12 to 15% of industrial energy use worldwide. Today Portland cement is the most important material for buildings globally, with an annual production of about 10 billion tons [1-3]. Into the Rotatory Cement Kiln (RCK), the firing stage significantly impacts the final product quality; this stage has a higher energetic consumption [4, 5]. It is for this reason that the efficiency of the process through alternative fuels is of great interest today. The use of alternative fuels in cement manufacturing has also gained wide attention due to its effectiveness in substituting fossil fuel's thermal energy requirement and reducing pollutant emission [6].

Research and technologies have been carried out to optimize energy consumption and CO_2 emissions in the cement process worldwide; using alternative fuels called low carbon fuels [7], material recovery facilities primarily composed of high-energy-content non-recycled plastics and fiber [8, 9], solid recovered fuel from landfills [10], sewage sludge [11, 12], food residue biomass [13]. these alternatives are part of a strategy to reduce the cement sector's impact on the environment from a life cycle perspective.

Many chemical reactions and complex phase transformations occur inside the RCK due to simultaneous processes not well understood [14,15]. Over time, various models have been used to describe this behavior [4, 16-21].

Most of the RCK is closed on the tubular section, and it is impossible to take samples to analyze each stage; this is a reason to study the process inside using tools as the Computational Fluid Dynamics (CFD). These tools have been used for calcining cyclones design and optimization through flow analysis and the chemical reactions as a movement of a mixture of gas and particles [22, 23]. CFD is widely used for the transport phenomena and combustion into rotary kiln [24-27].

It is essential to understand the phenomena inside the RCK and the relationships between its operating parameters. This will permit improving and proposing new designs adapted for different kinds of raw material, thus reducing the energy consumed and reducing the CO_2 emissions [28].

In this work, the Eulerian–Lagrangean model presented considers the combustion parameters as combustion air input, the cooling air input, and the heat involved in the chemical reaction to get the cement as the heat transference between the oven walls and the environment as boundary conditions. This model can predict changes in the thermal profile and air velocity for different fuel and production conditions.

2. Materials and methods

2.1 Domaine Module

A virtual domain was made by using the software Design Modeler, this software is native to ANSYS and fluent compliant. Real data were taken from an industrial Rotatory Cement Clinker. The generic rotary kiln simulated is 70-meter-long, 4.5 meter in diameter and inclined by 2 degrees and capacity of 71 tons per hour of clinker. The rotary kiln is provided with a typical burner Pillar KGO that consumes 2.2 kg/s of coal., Figure 1.

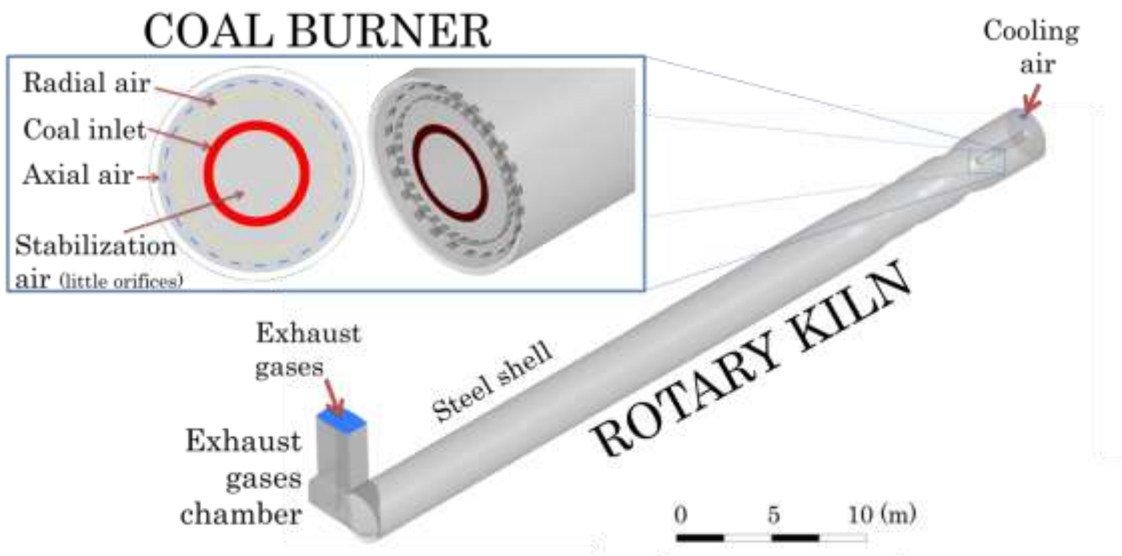

Figure 1. CFD domain: cement rotary kiln.

Domain was discretized by the ANSYS-FLUENT software's mesh module by 5,628,397 polyhedral elements with 33,105,218 nodes and an orthogonal quality mean of 0.543, this metric allowed to ensure the quality of the mesh, Figure 2.

Previously, several meshes of different sizes and shapes were made and it was proved with the mass transport of these cases that the solution is not dependent on the mesh. It was chosen to use polyhedric elements in the selected mesh. As shown in the Figure 2.

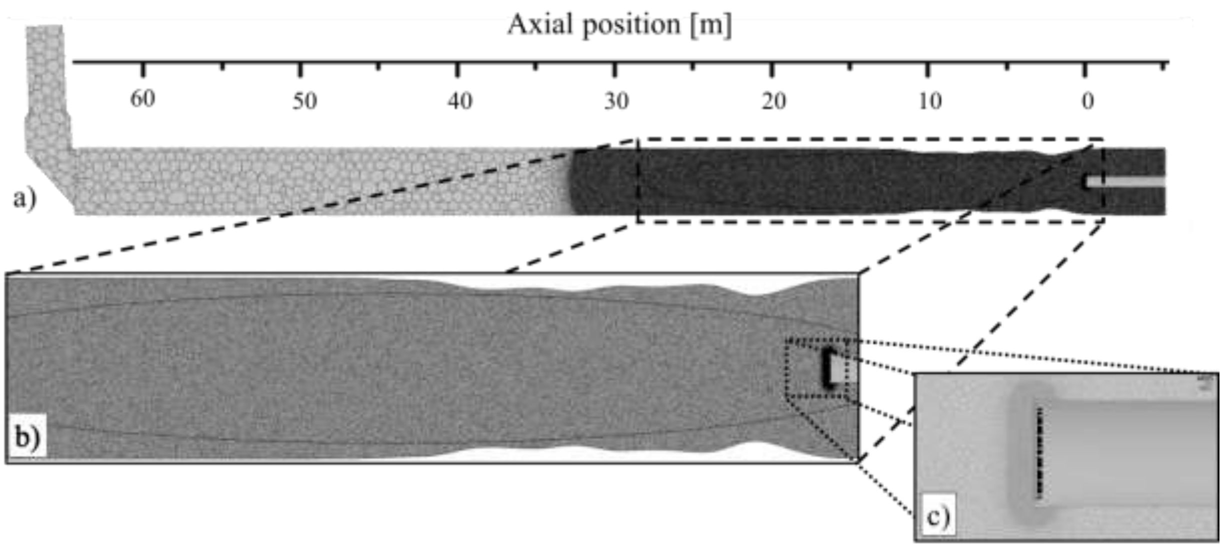

Figure 2. Polyhedral mesh of the furnace-smoke chamber system domain a) cross-section, b) zoom to the combustion zone, c) zoon to the burner region.

2.2 Solution

A volume code from an academic ANSYS-FLUENT version was employed [26]. There were considered three criteria: Residuals minimization (energy below $1x10^6$, the rest considered below $1x10^3$ since the energy equation is of great importance to describe the combustion phenomenon), mass and energy balance (lost below 10 % of the total) and stability of the volumetric integral for key variables as velocity, temperature and mix fraction.

Cluster Linux Centos with 8 Processors Intel Xeon 6 cores, 2,6 GHz, RAM 96 GB, was used to process model data, while the post-processing of the data was done in a Workstation Linux Processor Intel Core i7, 6 cores. 3,6 GHz, RAM 64 GB a graphic card AMD FirePro 4 GB, an average data processing time for each simulation is 20 to 25 hours with the computing capabilities mentioned above.

2.3 Theory/Calculation

Mass, moment, and energy conservation equations were considered and modeled into Navier Stokes equations [29-31]. Model include turbulent effects and multiple species transport and its chemical reactions of the combustion.

Turbulence simulation on industrial scales demands a robust model with reasonable precision and low numerical calculus cost as the k-ε realizable model [32]. This model is based on transport equations considering the kinetics energy k of the turbulence and its dissipation ε. The k-ε realizable model includes an alternative formulation for eddy viscosity and one modified equation for transport condition that satisfied mathematical limitations on the Reynolds Stresses according with the turbulent flow physics. Furthermore, other physical phenomena are included as buoyancy and compressibility [31]. The k-ε model shows advantages to analyze highly turbulent flow as jets, in our case air and fuel injection into the burner studied here. Equations 1 to 3 represent the, k-ε realizable model.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho k u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{Pr_k}\right)\frac{\partial k}{\partial x_j}\right]-\rho\overline{u_i'u_j'}\frac{\partial u_j}{\partial x_i}-\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p g_i\frac{\mu_t}{Pr_t}\frac{\partial T}{\partial x_i}-\rho\varepsilon \quad \textbf{(Eq. 1)}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_j}(\rho \varepsilon u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{Pr_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]-\rho C_2\frac{\varepsilon^2}{k+\sqrt{\upsilon\varepsilon}}+C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}-\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p g_i\frac{\mu_t}{Pr_t}\frac{\partial T}{\partial x_i} \quad \textbf{(Eq. 2)}$$

Where μ_t is the eddy viscosity and is computed from

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad \textbf{(Eq. 3)}$$

C_μ is a constant, and its value depends on rotation and deformation rate, the angular velocity of the system, and the turbulence field k y ε [31]. $C_{3\varepsilon} = tanh|\nu/u|$, constant $C_{1\varepsilon}$, C_2, Pr_ε y Pr_k, are 1.44, 1.9, 1.2 y 1.0, respectively [29].

A Non-Premix Combustion and a Probabilistic Density Function (PDF) model were used to simulate the chemical reaction occurring in the combustion of bituminous carbon. In this model, the only parameter considered was the mixed fraction, f. It considers both, fuel and oxidant go into the combustion chamber through different paths [31].

$$f=\frac{Z_i-Z_{i,ox}}{Z_{i,fuel}-Z_{i,ox}} \quad \textbf{(Eq. 4)}$$

Where Z_i is the mass fraction for the element i. Subscript ox and $fuel$ represent the value of the oxidant and fuel, respectively. The model permits the calculation of the intermediate species, radicals, dissociation effects, and couple rigorous turbulence chemistry. It assumes an instantaneous state of chemical equilibrium where the spices' thermochemical parameters such as a mass fraction, density, and temperature are directly related to the mix fraction.

Transport equation for mix fraction.

$$\frac{\partial}{\partial t}(\rho\bar{f}) + \nabla\cdot(\rho\vec{v}\bar{f}) = \nabla\cdot\left[\left(\frac{\mu_l+\mu_t}{\sigma_t}\right)\nabla\bar{f}\right] + S_m$$ **(Eq. 5)**

$$\frac{\partial}{\partial t}(\rho\overline{f^2}) + \nabla\cdot(\rho\vec{v}\overline{f^2}) = \nabla\cdot\left[\left(\frac{\mu_l+\mu_t}{\sigma_t}\right)\nabla\overline{f^2}\right] + C_g\mu_t\cdot(\nabla\bar{f})^2 - C_d\rho\frac{\epsilon}{k}\overline{f^2}$$ **(Eq. 6)**

Term S_m means the mass transfer in a gaseous phase by the chemical reaction, it is necessary to know the mass ratio, specific and caloric power heat for each element that makes up the fuel [31].

The Discrete Phase Model (DMP) was used to simulate bituminous coal injection as particles through a Rosin-Rambler distribution. This model is based on the Euler-Lagrange approach, where the fluid phase is considered as a continuous phase, solving the Navier-Stokes equations, and the discrete or dispersed phase is solved by tracking a finite number of particles through the calculated continuous phase. Considering that the dispersed phase interchanges mass, momentum, and energy with the continuous phase, Equation 7 represents the particle's force balance.

$$\frac{d\vec{u}_p}{dt} = \frac{18\mu}{\rho_p d_p^2}\frac{C_D\,Re}{24}(\vec{u}-\vec{u}_p) + \frac{\vec{g}(\rho_p-\rho)}{\rho_p} + \vec{F}$$ **Eq. 7)**

Where Reynolds number is defined as

$$Re = \frac{\rho d_p|\vec{u}_p-\vec{u}|}{\mu}$$ **(Eq. 8)**

2.4 Boundary conditions

Elemental chromatography analysis was performed on bituminous coal and sewage sludge using a CE Instruments EA1110. The solid samples' caloric value was determined in a PARR 1341 constant volume calorimetric bomb. An ORSAT equipment measured the combustion exhaust gases of the industrial rotary kiln, and the Steel Shell temperature was monitored with a SEMSCANNER infrared thermal scanner. Tables 1 and 2 show the experimental boundary conditions for both models.

Table 1. Operation and boundary conditions for the Rotary Kiln-Smoke Chamber and Burner mathematical model.

Solver time	**Stationary**		
Steel shell	Rotational velocity	0.21 rad/s.	
	Heat flux losses profile:	Chemical reactions (Supplementary material) and environmental measure	
Primary Air	Axial air	Pressure inlet condition	22400 Pa , 353 K.
	Radial air:	Pressure inlet condition	19600 Pa , 353 K.
	Stabilization air (60 orifices with a diameter of 0.01m)	Pressure inlet condition	16800 Pa , 353 K.
Cooling air	Pressure inlet condition	25.18 Pa , 1100 K	
Coal inlet	Pressure inlet condition	4500 Pa, 353 K	
	DPM injection	See Table 2	
Exhaust gases outlet	Velocity condition: 30 m/s		

Table 2. Coal particle properties.

Temperature	353 K	
Velocity	30 m/s	
Total flow rate	2.2 kg/s	
Diameter by Rosin-Rammler distribution,	Minimum	$1x10^{-6}$ m
	Maximum	1x10-4 m
	Mean	$5.5x10^{-5}$ m
	Number of points	10

3. Results

Figure 3 shows a transversal view of the gas velocity profile through the oven. Velocity is decreasing radially from the center to the surface and decreases from the burner along the z-axis to the kiln's input well. Figure 4 shows flow lines for each type of air independently in the burner and together. High velocities and turbulent effects near the burner help the mix of the air end fuel. The multiple air injections assure morphology and flame control.

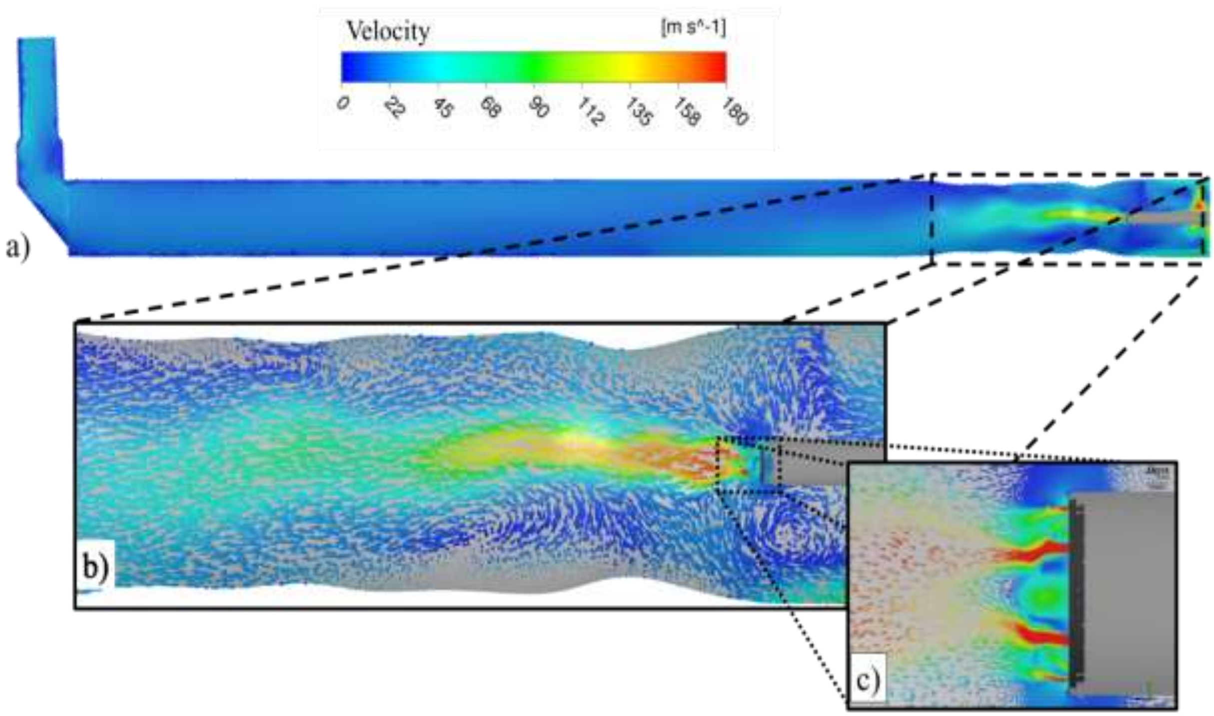

Figure 3. Speed profile of the cross-sectional system: a) furnace-smoke chamber, b) flame, c) burner.

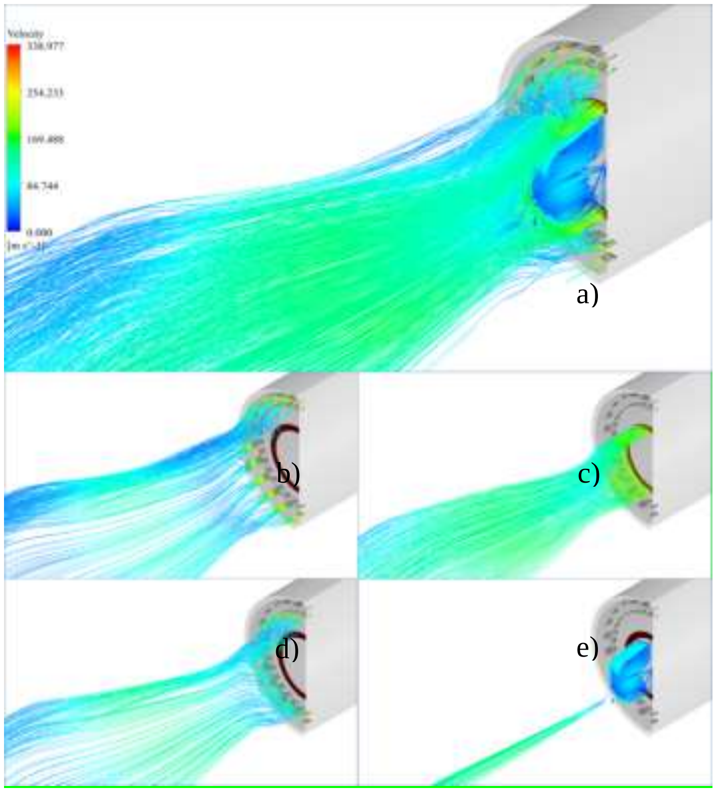

Figure 4. a) Pathlines: b) axial air, c) carbon injection, d) radial air, and e) stabilization air.

Axial profiles in Figure 5 show the most significant amount of turbulent kinetic energy near the burner and the cooling air inlet. This profile shows the nodal values of the variable demonstrating the usefulness of the k-ε model in the simulation of fluid dynamics for the burner.

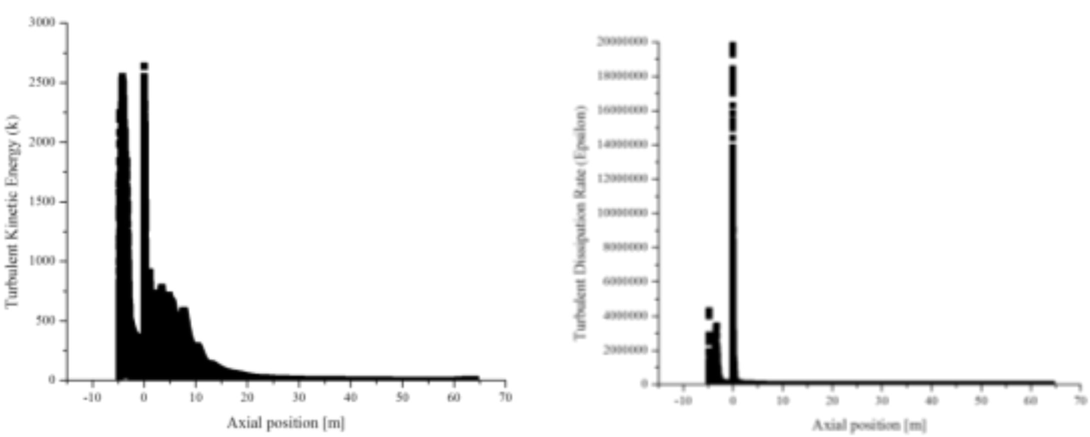

Figure 5. Axial dispersion of the generation of turbulent kinetic energy (k) and turbulent dissipation rate (ε).

Figure 6 exhibits temperature of gases, CO, and CO_2 mass fraction along with the cement kiln,

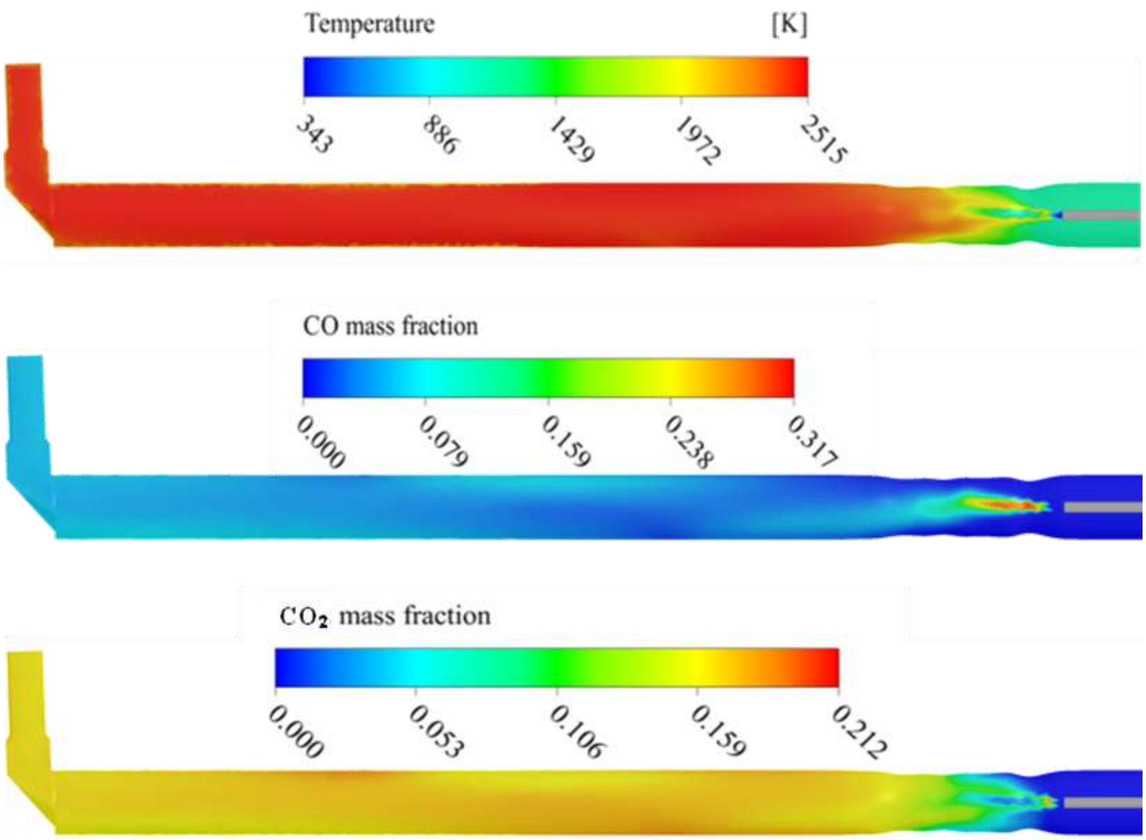

Figure 6. Temperature, CO, and CO_2 mass fraction profiles as efficiency combustion indicators.

Regarding the relationship of the CO and CO_2 mass fractions near the burner, incomplete combustion is observed because the CO is present in a higher proportion. As CO comes into contact with cooling air or excess air, it oxidizes to form CO_2.

Figure 7 shows the nodal values for the bituminous coal fraction (C(s)), carbon monoxide (CO), carbon dioxide (CO_2), and mixed fraction. It is observed that the coal is practically burned in the first ten-meter from the burner; this is favorable because here is the place inside the clinker where the energy demand is highest because of the clinkerization process. However, a specific analysis indicates that carbon particles persist up to 30 meters from the burner along with the oven.

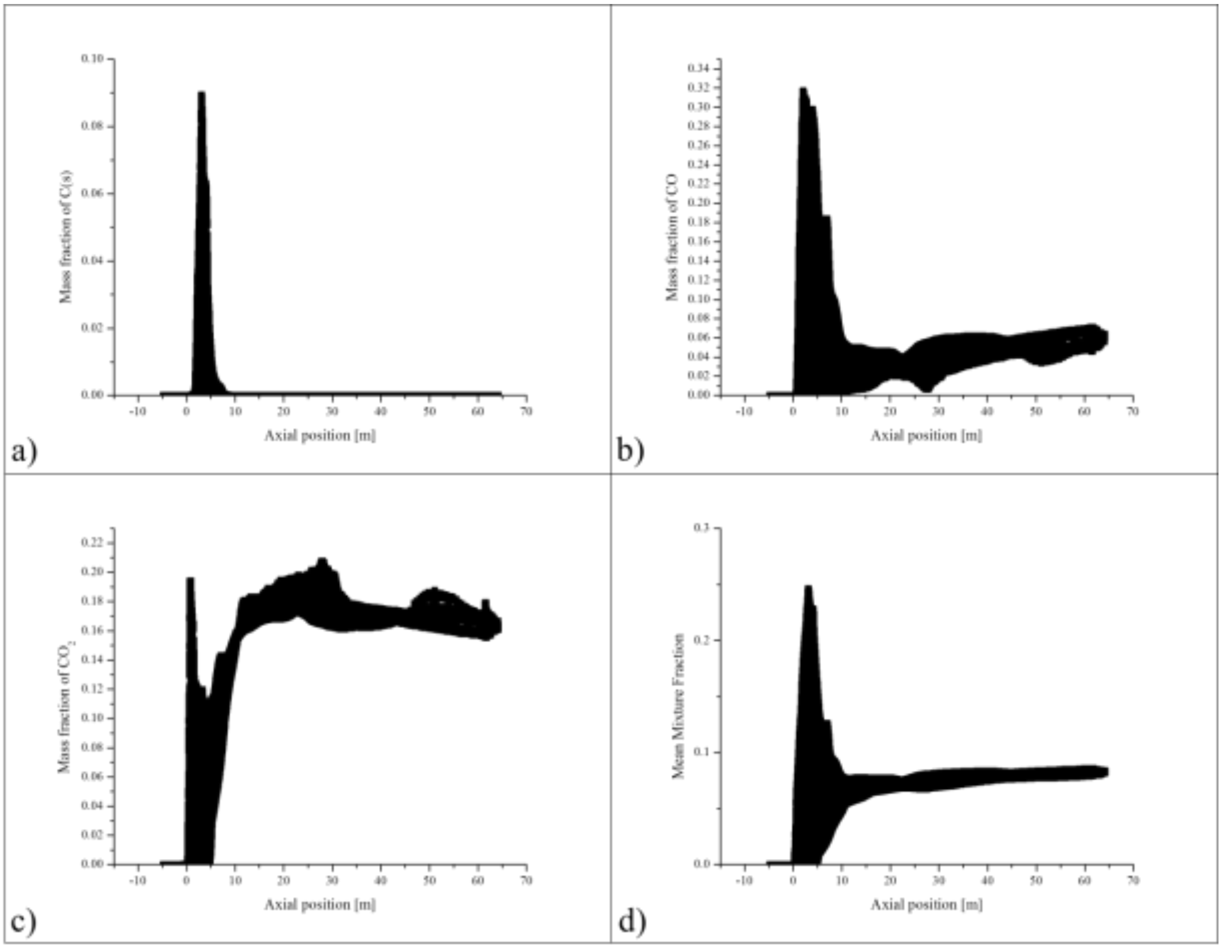

Figure 7. Nodal values of solid carbon (a), CO (b), CO_2 (c), and a mixed fraction (d) in an axial profile.

Emission gases simulated and measured in a real process are shown in Table 3. The model considers only the extraction of the necessary energy for decarbonization, not the generation of this process's carbon dioxide. So, the industrial measured CO_2 value is higher than the modeled one.

Table 3. Simulated emission gas values and those measured in an industrial plant using bituminous coal as an energy source.

Parameter	Measurement	Simulation
O_2	0.042	0.0578
CO	0.0005	0.003
CO_2	0.252	0.165
N_2	0.702	0.7742

The Eulerian-Lagrangean model was used for the simulation of the use of sewage sludge and biogas as alternative fuels, all injected at 2,2 kg/s in the rotary kiln, conserving the particle size distribution of the carbon case and considering the characterization of the elements present (CHON) and the calorific value of the materials. Figure 8 shows the nodal values of the temperature profile obtained.

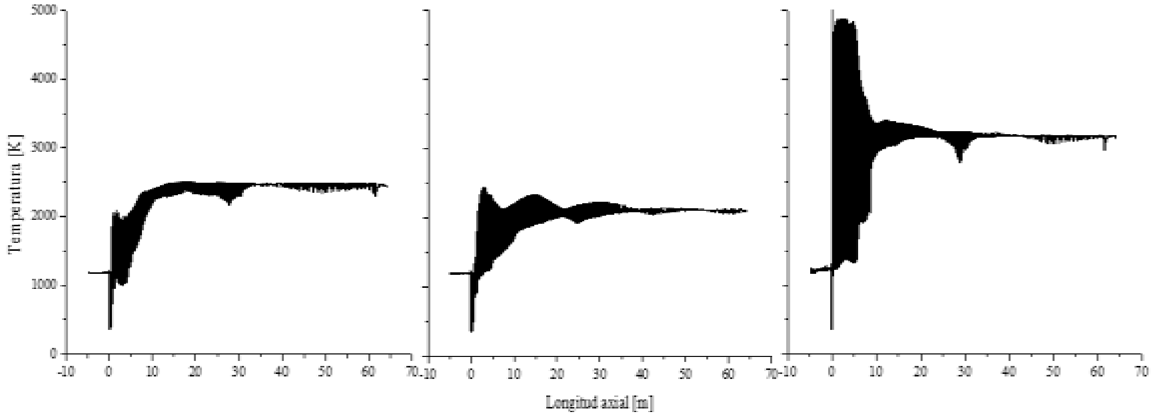

Figure 8. Nodal values of temperature in an axial profile combusting: a) carbon fuel, b) sewage sludge, c) biogas.

As shown in Figure 8, the model results developed representing the use of alternative fuels allowed the identification of the temperature profile that will be produced inside the furnace and the temperature gradient in the area near the burner.

4. Conclusions

The mathematical model developed in the ANSYS-Fluent Academic Virtual Laboratory applied to the corresponding domain of the system: Oven Smoke Chamber and Burner, demonstrate that it can establish optimal control of the flow of gases involved based on their elemental analysis and caloric value and opens the opportunity to use alternative residual fuels.

Discretizing the domain required creating modular domains, separating the turbulent zones with the most significant gradient to obtain the convergence or solution.

The simulation of the system shows the turbulent behavior of the mass of gases inside a rotatory kiln and its interconnection with the smoke chamber.

The energy extracted value for the formation of the cement phases is a significant parameter for the model, such as the effects of the kiln's turning, injection of the primary and secondary air on the combustion gases, and flame. The CO_2 emission shows a significant difference since the simulator only considers the emission due to coal combustion but not due to the CO_2 release from decarbonation of limestone.

With the Eulerian-Lagrangean model, it was identified that using sludge exclusively as fuel does not allow reaching the temperature that occurs with coal. On the other hand, the use of biogas exceeds the required temperature and generates important temperature gradients in the region near the burner, which could generate a change in the crust formation within the rotary kiln, compromising the integrity of the kiln.

The model developed proved to be a tool with the potential to find the thermal effect of alternative fuel mixtures with low energy releases such as sludge and high energy release such as biogas and promote the substitution of fossil fuels within the rotary kiln.

The challenges facing the cement industry require a new vision focused on a circular economy. The use of modeling tools and computational simulation are key elements in this transformation.

5. Acknowledgments

The authors thank to Consejo Nacional de Ciencia y Tecnología (CONACyT) and to the Secretary of Energy (SENER) for the infrastructure generated through project 243715. Cement industry Grupo Cementos Chihuahua GCC and the Dr. Carolina Prieto for the support.

6. Bibliography

[1] Mikulčić, H., M. Vujanović, and N. Duić, "Improving the sustainability of cement production by using numerical simulation of limestone thermal degradation and pulverized coal combustion in a cement calciner", Journal of Cleaner Production, 88, 2015, 262-271. https://doi.org/10.1016/j.jclepro.2014.04.011

[2] Rodrigues, F. and I. Joekes, "Cement industry: sustainability, challenges and perspectives", Environmental Chemistry Letters, 9(2), 2011, 151-16. https://doi.org/10.1007/s10311-010-0302-2

[3] Hache, E., Simoën, M., Seck, G. S., Bonnet, C., Jabberi, A., and Carcanague, S. "The impact of future power generation on cement demand: an international and regional assessment based on climate scenarios", International Economics, 163, 2020, 114-133. https://doi.org/10.1016/j.inteco.2020.05.002.

[4] Goshayeshi, H. R., and Poor, F. K. "Modeling of Rotary Kiln in Cement Industry". Energy and Power Engineering, 8(01), 2016, 23. doi: 10.4236/epe.2016.81003.

[5] Supino, S., Malandrino, O., Testa, M., and Sica, D. "Sustainability in the EU cement industry: the Italian and German experiences". Journal of Cleaner Production, 112, 2016, 430–442. https://doi.org/10.1016/j.jclepro.2015.09.022.

[6] Rahman, A., Rasul, M. G., Khan, M. M. K., and Sharma, S. "Recent development on the uses of alternative fuels in cement manufacturing process". Fuel, 145, 2015, 84–99. Doi:10.1016/j.fuel.2014.12.029.

[7] Zhang, L., and Mabee, W. E. "Comparative study on the life-cycle greenhouse gas emissions of the utilization of potential low carbon fuels for the cement industry". Journal of Cleaner Production, 122, 2016, 102-112. https://doi.org/10.1016/j.jclepro.2016.02.019.

[8] Fyffe, J. R., Breckel, A. C., Townsend, A. K., and Webber, M. E. "Use of MRF residue as alternative fuel in cement production", Waste management, 47, 2016, 276-284. https://doi.org/10.1016/j.wasman.2015.05.038.

[9] Bourtsalas, A. T., Zhang, J., Castaldi, M. J., and Themelis, N. J. "Use of non-recycled plastics and paper as alternative fuel in cement production". Journal of cleaner production, 181, 2018, 8-16. https://doi.org/10.1016/j.jclepro.2018.01.214.

[10] Reza, B., Soltani, A., Ruparathna, R., Sadiq, R., and Hewage, K. "Environmental and economic aspects of production and utilization of RDF as alternative fuel in cement plants: A case study of Metro Vancouver Waste Management". Resources, Conservation and Recycling, 81, 2013, 105-114. https://doi.org/10.1016/j.resconrec.2013.10.009.

[11] Zabaniotou, A., and Theofilou, C. "Green energy at cement kiln in Cyprus—Use of sewage sludge as a conventional fuel substitute". Renewable and Sustainable Energy Reviews, 12(2), 2008, 531-541. https://doi.org/10.1016/j.rser.2006.07.017.

[12] Rodríguez, N. H., Martínez-Ramírez, S., Blanco-Varela, M. T., Donatello, S., Guillem, M., Puig, J., ... and Flores, J. "The effect of using thermally dried sewage sludge as an alternative fuel on Portland cement clinker production". Journal of Cleaner Production, 52, 2013, 94-102. https://doi.org/10.1016/j.jclepro.2013.02.026.
[13] Papanikola, K., Papadopoulou, K., Tsiliyannis, C., Fotinopoulou, I., Katsiampoulas, A., Chalarakis, E., ... and Lyberatos, G. "Food residue biomass product as an alternative fuel for the cement industry". Environmental Science and Pollution Research, 26(35), 2019, 35555-35564. https://doi.org/10.1007/s11356-019-05318-4.
[14] Mujumdar, K.S. and V.V. Ranade, "CFD modeling of rotary cement kilns". Asia-Pacific Journal of Chemical Engineering, 3(2), 2008, 106-118. https://doi.org/10.1002/apj.123.
[15] Wang, W., Wang, W., Luo, Z., Shi, Z., and Cen, K. "Experimental study on cement clinker co-generation in pulverized coal combustion boilers of power plants", Waste management & research, 24(3), 2006, 207-214. https://doi.org/10.1177/0734242X06063892.
[16] Spang III, H., "A dynamic model of a cement kiln", Automatica, 8(3), 1972, 309-323. https://doi.org/10.1016/0005-1098(72)90050-7.
[17] Saeman, W. C. "Passage of solids through rotary kilns-factors affecting time of passage." Chemical Engineering Progress, 47(10), 1951, 508-514.
[18] Bui, Rung T., Stanislaw Tarasiewicz, and Andre Charette. "A computer model for the cement kiln."IEEE Transactions on Industry Applications, 4, 1982, 424-430. DOI: 10.1109/TIA.1982.4504103.
[19] Pieper, C., Liedmann, B., Wirtz, S., Scherer, V., Bodendiek, N., & Schaefer, S. (2020). Interaction of the combustion of refuse derived fuel with the clinker bed in rotary cement kilns: A numerical study. Fuel, 266, 117048. https://doi.org/10.1016/j.fuel.2020.117048
[20] Liedmann, B., Wirtz, S., Scherer, V., & Krüger, B. (2017). Numerical study on the influence of operational settings on refuse derived fuel co-firing in cement rotary kilns. Energy Procedia, 120, 254-261. https://doi.org/10.1016/j.egypro.2017.07.176
[21] Pieper, C., Wirtz, S., Schaefer, S., & Scherer, V. (2021). Numerical investigation of the impact of coating layers on RDF combustion and clinker properties in rotary cement kilns. Fuel, 283, 118951. https://doi.org/10.1016/j.fuel.2020.118951
[22] Vegini, A. A., Meier, H. F., Iess, J. J., and Mori, M. "Computational fluid dynamics (CFD) analysis of cyclone separators connected in series." Industrial & engineering chemistry research 47(1), 2008, 192-200. https://doi.org/10.1021/ie061501h..
[23] Mikulčić, Hrvoje, Milan Vujanović, and Neven Duić. "Large eddy simulation of a two-phase reacting swirl flow inside a cement cyclone." Energy 75, 2014, 89-96. https://doi.org/10.1016/j.energy.2014.04.064.
[24] Elattar, H. F., Stanev, R., Specht, E., and Fouda, A. "CFD simulation of confined non-premixed jet flames in rotary kilns for gaseous fuels." Computers & Fluids, 102, 2014, 62-73. https://doi.org/10.1016/j.compfluid.2014.05.033.
[25] Mikulčić, H., Vujanović, M., Fidaros, D. K., Priesching, P., Minić, I., Tatschl, R., and Stefanović, G, "The application of CFD

modelling to support the reduction of CO2 emissions in cement industry." Energy 45.1, 2012, 464-473. https://doi.org/10.1016/j.energy.2012.04.030.
[26] Mastorakos, E., Massias, A., Tsakiroglou, C. D., Goussis, D. A., Burganos, V. N., and Payatakes, A. C. "CFD predictions for cement kilns including flame modelling, heat transfer and clinker chemistry." Applied Mathematical Modelling 23.1, 1999, 55-76. https://doi.org/10.1016/S0307-904X(98)10053-7.
[27] Ortiz Muñoz, Alejandro, Santiago Builes Toro, and Diego Andrés Acosta Maya. Cement in-line calciner kiln modeling for heat optimization using a design of computer experiments. Diss. Universidad EAFIT, 2020. http://hdl.handle.net/10784/25460.
[28] Wang, S., Lu, J., Li, W., Li, J., and Hu, Z. "Modeling of pulverized coal combustion in cement rotary kiln" Energy & Fuels 20.6, 2006, 2350-2356. https://doi.org/10.1021/ef060027p.
[29] Ansys-Inc. Computational Fluid Dynamics (CFD) Software. http://www.ansys.com/.
[30] Valle, M.R., Numerical Modeling of Granular Flows in Rotary kilns. Master of Science Thesis, Faculty of Electrical Engineering, Mathematical and Computer Science Delft University of Technology, 2012.
[31] Ansys-Fluent, 15.0 Theory Guide. ANSYS inc, 2013.
[32] Shih, T. H., Liou, W. W., Shabbir, A., Yang, Z., and Zhu, J. "A new k-epsilon eddy viscosity model for high Reynolds number turbulent flows: Model development and validation." NASA Sti/recon Technical Report N 95, 1994, 11442.

Abbreviations

$\vec{u}_p$	Particle velocity (m/s)
μ, μ_l, μ_t	Molecular, laminar, and turbulent viscosity of the fluid, respectively (Pa-s)
ρ	Fluid density (kg/m^3)
d_p	Particle diameter (m)
t	Time (s)
k	Kinetic energy per unit mass (J/kg)
ε	Turbulent dissipation rate (m^2/m^3)
σ	Surface tension (kg/m)
u, v	Velocity magnitude (m/s)
T	Temperature (K)
$\vec{g}$	Gravitational acceleration (m/s^2)
p	Pressure (Pa)
Pr	Prandtl number