ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 11, May 2014

Design and analysis of a sampling probe at high temperature under a CO_2 atmosphere susceptible to oxidative damage. Part 1: Virtual design and dimensional optimization

Héctor Alfredo López Aguilar[1], Jorge Alberto Gómez[2], Abraham Hernández[3], Marco Antonio Merino[3], Carolina Prieto-Gómez[4], Antonino Pérez Hernández[1✉].
[1] Centro de investigación en materiales avanzados (CIMAV unidad Chihuahua).
[2] Universidad Autónoma de Ciudad Juárez. UACJ.
[3] Universidad Autónoma de Chihuahua. UACH.
[4] Grupo Cementos de Chihuahua. GCC

***Abstract*—This paper presents the design and modelling of a sampling probe at high temperatures and CO_2-rich atmospheres applications and when samples are susceptible to deterioration by contact with oxygen from air. Applying the simulation tool CFD (Computational Fluid Dynamics)-Fluent ANSYS 15.0, this study verifies and optimizes a first dimensional configuration of the probe. The optimized device considers the cooling rate and the internal gases velocity for negative pressure generation (Bernoulli effect) and its suitable for sampling in cement industry.**

***Index Terms*—Bernoulli, cement, CFD, sampling probe, oxidative damage.**

I. INTRODUCTION

Any industry requires the use of devices for process control and quality assurance. In cement industry, the extraction of samples at high temperatures and inert atmospheresturns the extraction process into a complex activity by the potential degradation of the specimen in contact with oxygen from air. Xue [1] performed the analysis of a jet type pump, Yimer [2] through Computational Fluid Dynamics(CFD) software, which principle of operation is based on the Venturi effect [3]. This effect is widely used in the automotive, aviation and flow measurement industries [4]-[7] and many researchers have modelled this phenomenon by finite volume element [8]-[13]. In cement industry it has been used for the design and optimization of calciners [14] and to simulate the main transport processes in rotary kilns [15]; there are also patents focused on sampling systems for combustion gases from the rotary kiln. Some patents deal with volatile gases, chlorine and sulfur compounds, and for the removal of lead in the sample [16]-[20].

This paper proposes the design and modelling of a sampling probe for sample extraction from high temperatures and CO_2-rich atmospheres, especially in cement industry, when phase assemblage may be altered by the slow cooling process and the presence of oxygen from air during sampling.

II. METHODS

A. Design considerations of specimen extraction probe.

Considering the need of a procedure to extract specimens during the formation process of cement clinker and keeping the structure in present phases through a quick cooling in a protective atmosphere, an extraction Jet probe type is proposed. This design was based on fluid dynamics, especially on Bernoulli's principle.

$$p = k - \frac{1}{2}\rho v^2$$

Where k is a constant.

In this equation derived from Bernoulli's principle, when density ρ is kept constant and disregarding the elevation, at speed $v > \sqrt{2k/\rho}$, a drop pressure is generated, going from a constant value "*k*" until negative values of pressure. Moreover, considering a basic structure of two tubes connected in "T" shape, by passing a non-reactive high velocity fluid by one of the tubes, a drop pressure is generated and safe extraction of samples subjected to certain atmospheres, velocity and high temperatures is allowed.

On the other hand, in order to maintain the microstructural and chemical characteristics of the extracted samples under the mentioned extreme conditions, it is necessary a fast cooling and special-atmosphere device to avoid a reversible reaction in clinker phases or a reaction with oxygen fastened by the high temperature environment. The materials inside the cyclone (Fig.1) are at temperatures close to 1090 K in a CO_2-rich atmosphere. In this invention, two streams of $CO_2$are injected tangentially. The first stream is used to generate a low pressure for the extraction of materials, and the second for quick cooling and mixing. For an optimal cooling with the minimal dimensions of this invention, a strategic design was used to drive the particles in a helical path increasing the residence time.

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 11, May 2014

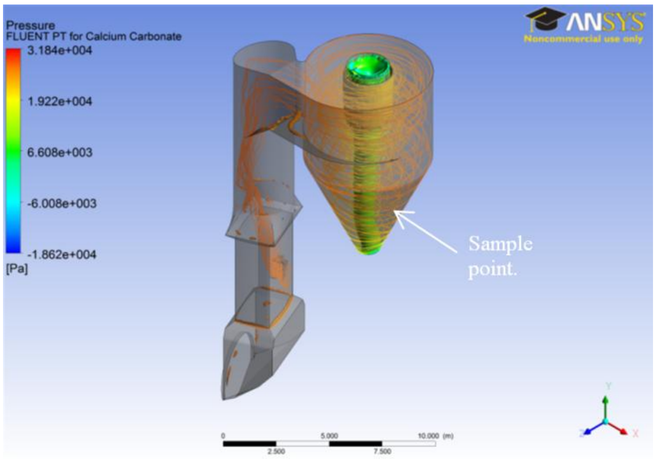

Fig. 1 Isometric view and CFD simulation inside cyclone process in cement industry.

For this purpose, parametric equations of the spiral were considered:

$$x(\theta) = r sen(\theta)$$
$$y(\theta) = r cos(\theta)$$
$$z(\theta) = h\theta$$

Getting:

$$x^2(\theta) + y^2(\theta) = r^2 \quad (1)$$
$$z(\theta) = h\theta \quad (2)$$

Equation 1 is associated to the confinement of the particles in the system, a cylinder and its conical regions; θindicates time. Equation 2 establishes that the injection of the fluid should be done with a slight angle to the plane transverse to the circular tube. The initial condition of these equations is established from the CO_2 entry pressures and angles; the first injection generates a negative pressure and is used in the extraction of samples; the second propitiates the mixing effect.

B. Setup and operation of the sampling probe. Practical case: cement industry.

For quality control in the production of cement, analysis of the samples before and after each process in the manufacture of cement clinker is necessary. In this process, the raw material in transformation is kept in CO_2-rich atmosphere and temperatures close to 1090 K, which hinders the extraction of samples.

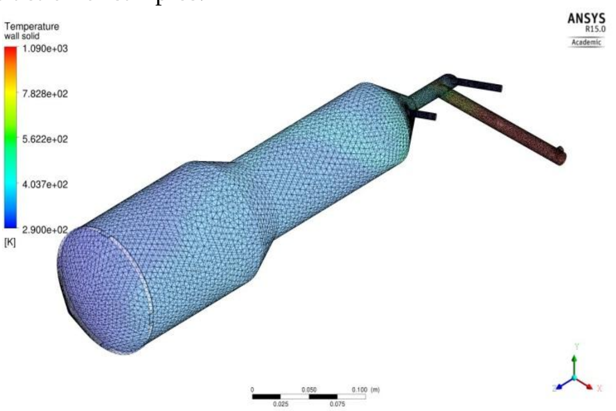

Fig. 2Isometric view of the mesh of the sampling probe

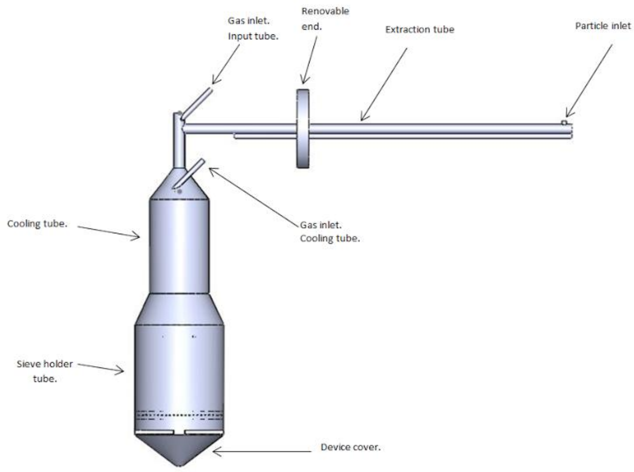

Fig.3.Full frontal view of the configuration of the sampling probe

To achieve the extraction and fast cooling in a CO_2-rich atmosphere, the device shown in Fig. 2 and Fig. 3 is proposed.The device consists of an extraction tube that is introduced into the specimens' extraction gate in the cyclones (Fig. 2). A flow of CO_2(used as carrier fluid) is injected in both, the extraction tube and cooling inlet tube. The tangential position (Fig. 4) of each pivot promotes a helical flow of CO_2. Inside the cyclone, the sample rotates and enters through the hole of the collecting tube.This hole is placed facing the direction of the particles flow within the cyclone, slowing the particlesby the impact with the internal wall of the collecting tube.Once the particle sample is stopped, lays under the action of the low pressure (Bernoulli's effect), and it helps to generate a suction effect that takes the captured particles from the collection inlet tube into the cooling tube.

A second helical effect on the cooling tube is generated by providing a CO_2 flow in the cooling inlet tube; the primary objective is to force the particles to follow a helical path toward the same cooling tube to increase the residence time, strengtheningat the same time, the generation of low pressure.

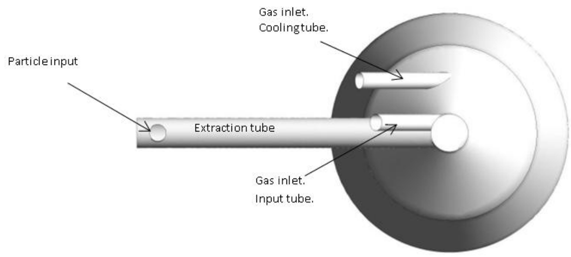

Fig. 4.Lateral view of the device with the details of the extraction tube and the gas inlets of the inputs tubes and the cone cooling tube.

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 11, May 2014

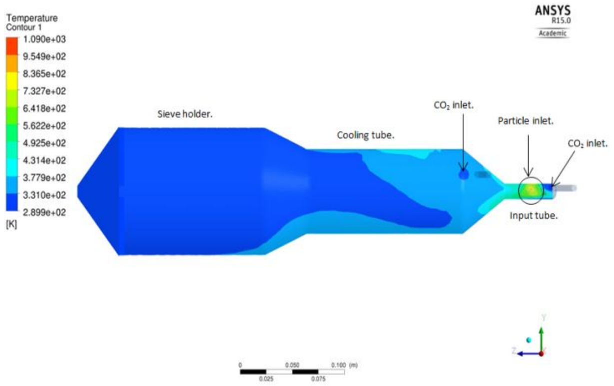

Fig. 5.Longitudinal section of the device showing temperature contours inlet tubes, and carries cooling sieves.

In this way, *i)* by the second cold CO_2flow in the cooling cone, *ii)*the sudden change in volume of the mixture of the particles with the cold carrier gas and *iii)*the helical path, the original structure of the samples is maintained inside cyclones.Therefore, the chemical and crystallographic analyses performed to the particles extracted with this invention, is representative of the particlesinside the extreme atmospheres in the kiln.

From the process mentioned above, the particles experiment a new volumetric expansion from the second conical section to the sieves-holdertube, where are captured by the sieves and CO_2 gases leaving the sieves-holder tube through the venting slots. After the venting slots the conical lid closes the system; the lid is removable in order to place or remove the sieves.

C. Simulation and dimensional optimization of the device by ANSYS - FLUENT CFD

For the development of the present computational study, finite volume CFD package, Fluent ANSYS 15.1 was used. To verify the operation, the diffusive behavior of gases flowing into the proposed device was modelled using the Reynolds stress model turbulence model, based on the Navier-Stokes equations which describe the motion of fluids [21].

The operating conditions consideredwere: a stable state, g = -9.81 ms^{-2} and the boundary condition at the entrance of the particles of 2.3 kPa (into the cyclone in the extraction point). To obtain this boundary condition into the cyclone, this region was modelled with the same software (Fig.1).

The inlet gas pressure was 10 atmospheres for both, input pivot and the cooling tube pivot. The mesh used to solve the conservation equations and the final configuration of the device is illustrated in Fig. 2 and Fig. 3 respectively, this design is under Mexican patent registration with number mx/a/2014/002336.

The simulation tool is alternated with the Design of experiments (DOE) and theResponse Surface (RS) modelling; the first one to minimize calculation costs and the second to optimize the dimensional configuration of the design of the proposed device.

The DOE method is used to analyze experimental data and build empirical models to obtain the approximate representation of the physical situation, creating a table with the values of the variablesto optimize. The RS methodology may be defined as a method to construct global approximations of the behavior of the system on the calculated results at various points in the design space. [22]

III. RESULTS AND DISCUSSION

In Fig. 5, the isotherms of the simulation are illustrated. Note that in the inlet tube, the temperature of the gases is close to 700 K. Likewise, in the outlet of the cooling tube the temperature drop near to 290K can be observed.

The detail of the flow gases drawings is illustrated in Fig. 6. This picture perfectly defines the helical paths of the gases, and shows the temperature variations with more detail.

Dimensional optimization

This design optimization is based on the construction of a response surface based on the design of experiments listed in Table I.

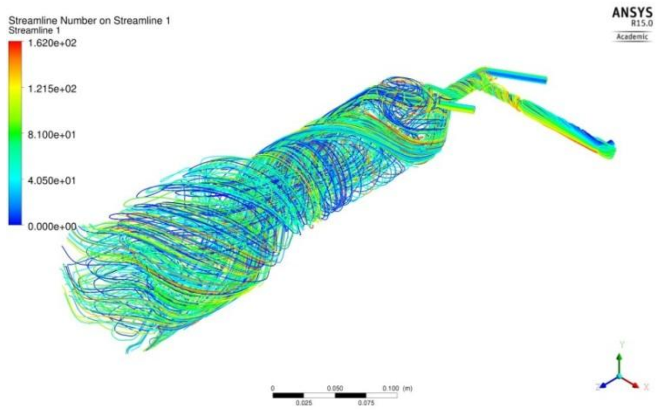

Fig. 6. Detail 3D flow lines where gas temperatures shown from the inlet to the outlet of the device.

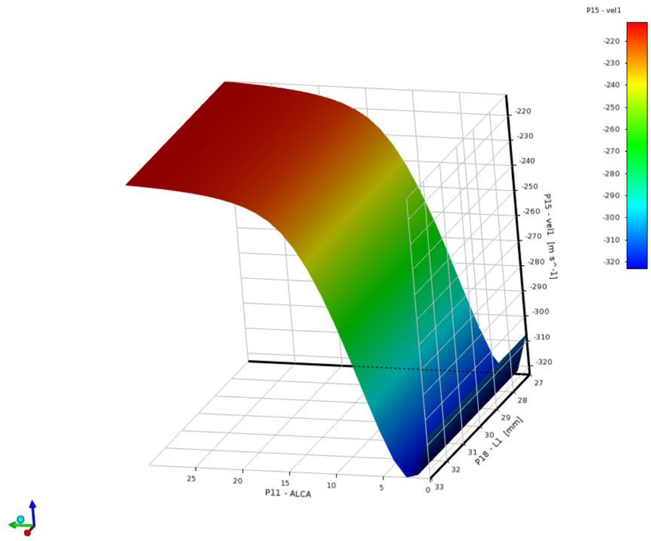

Fig. 7. 3D graph of the angle between the extraction and the input tube (P11-ALCA) and the intake lengthtube (P18-L1) in function to velocity extraction (P15-vel2).

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 3, Issue 11, May 2014

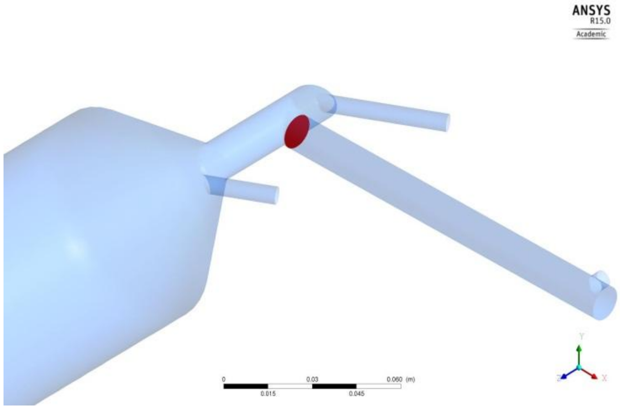

Fig. 8 Isometric probe (in red) where the surface taken as a reference for the calculation of gas velocity.

The parameterized variables were: the main angle between the manifold tube and the main tube, the inlet length tube and the cooling length tube.

The response variables simulated were the temperature at the output of the device and the gases flow rate used to generate the Bernoulli's effect.

Fig. 7 illustrates the gas rate behavior (shown in red in Fig. 8) in function of the angle between the inlet tube and the extraction tube. In Table I, this behavior is shown in a discrete way. The optimal design angle between the two tubes is 0° (90° among themselves) and there is no considerable effect of the length inlet tube in the gas rate.

In Fig.9, the behavior of the temperature at the exit gases is shown with respect to the length of the cooling and inlet tubes. Differently from the cooling tube, no critical dependence is observed for the inlet tube. The discrete value shown in Table Iindicates an optimal distance for a 162.2 mm to improve the heat dissipation rate.

Finally, regarding Fig. 10 for simulation purposes, 228 hypothetical particles with a uniform size of 5 microns were considered to observe the path from the extraction tube inlet until their exit into the sieve holder. The trajectory rates of the particles which completed the full course within the simulated device was about 43%.

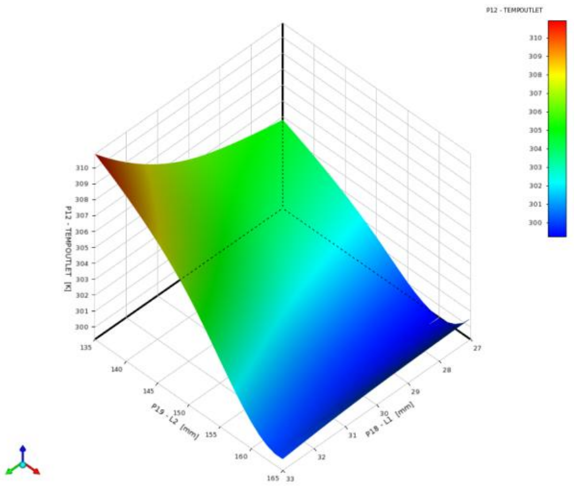

Fig. 9. 3D graph of the cooling length tube (P19-L2), the length of the inlet tube (P18-L1) in function of gases temperature outlet (P12-TEMPOUTLET).

Name	ALCA	L1 (mm)	L2 (mm)	Temp (k)	Vel1 (ms^{-1})
1	15	30	150	303.97	-205.86
7	15	30	165	300.48	-234.53
2	**0**	**30**	**150**	**302.2**	**-306.65**
3	30	30	150	302.59	-204.97
4	15	27	150	301.01	-213.32
5	15	33	150	306.1	-255.52
6	15	30	135	305.94	-248.89
8	2.8045	27.561	137.8	303.05	-326.26
9	27.196	27.561	137.8	306.49	-170.53
10	2.8045	32.439	137.8	309.22	-295.54
11	27.196	32.439	137.8	309.05	-189.99
12	2.8045	27.561	162.2	299.58	-321.53
13	27.196	27.561	162.2	299.3	-245.97
14	2.8045	32.439	162.2	300.17	-346.64
15	27.196	32.439	162.2	299.8	-254.26

Table I. DOE

The mean residence time obtained was about 5.4x10^{-02} seconds and a thermal rate variation was close to14810 Ks^{-1}. For the rest of the particles the mean residence time was 3.0 seconds.

IV. CONCLUSIONS

Through the analysis of finite volume CFD it has been possible to verify the correct operation of the cement clinker probe sampler at high temperatures and velocity, in a CO_2-rich atmosphere within a cyclone, used in cement industry.

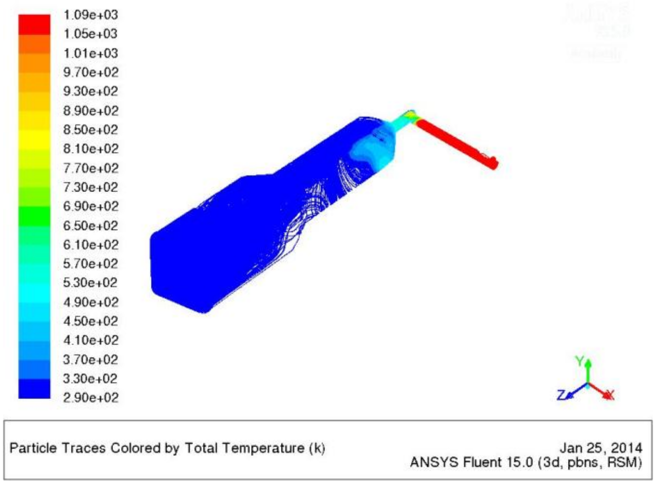

Fig. 10.3D image of the trace of uniform particle size of 5 microns.

The accelerated cooling rate and the generation of negative pressure, this last used as a condition that facilitates the extraction of the particles, are among the parameters considered in this study. Likewise, the optimal parameters and the correlation of dimensional parameters studied were defined with ANSYS FLUENT simulations.

The angle between the inlet tube and the extraction tube was 90°, the length of both, the inlet and cooling tubes, was 162.2 mm. It was found that the length of the inlet tube has not a dependency to the cooling rate.

Finally, as a supplementary work, the functioning optimization and the parameters for the economizing construction and implementation of sampling device shall be regarded to include the actual particle size distribution in the cyclone and to intensify the particle analysis, to obtain a closereffect of the cooling rate on the particle distribution.

V. Acknowledgment

Authors thanks: CONACyT, CIMAV, UACH, UACJ, GCC, Omar Rodríguez GrupoSSC.

BIBLIOGRAPHY

[1] S. Xue, P. Joon, K. Seung, and P. Young, "Performance comparison and erosion prediction of jet pumps by using a numerical method", Mathematical and Computer Modeling, vol. 57, pp. 245–253, June 2011.

[2] I. Yimer, H.A. Becker, and E.W. Grandmaison, "The Strong-jet/Weak-jet Problem: New Experiments and CFD. COMBUSTION AND FLAME", vol. 124, pp. 481–502, 2001.

[3] A.M. Abdulaziz, "Performance and image analysis of a cavitating process in a small type venturi," Experimental Thermal and Fluid Science, vol. 53, pp 40–48, February 2014.

[4] P. Kumar, and M.W. Ming, "A CFD study of low pressure wet gas metering using slotted orifice meters", Flow Measurement and Instrumentation, vol. 22 pp. 33–42, December 2010.

[5] J.A. Cruz, F. Sánchez, and P. Quinto, "A new correlation to determine the discharge coefficient of a critical Venturi nozzle with turbulent boundary layer", Flow Measurement and Instrumentation, vol. 17, pp. 258–266, June 2006.

[6] H. Ghassemi, and H. Farshi, "Application of small size cavitatingventuri as flow controller and flow meter",Flow Measurement and Instrumentation, vol. 22, pp. 406–412, May 2011.

[7] R.K. Singh, S.N. Singh, and V. Seshadri, "Study on the effect of vertex angle and upstream swirl on the performance characteristics of cone flow meter using CFD", Flow Measurement and Instrumentation, vol. 20, pp. 69-74, December 2008.

[8] M.T. Kandakure, V.C. Patkar, and A.W. Patwardhan, "Characteristics of turbulent confined jets", Chemical Engineering and Processing. vol. 47. Pp. 1234–1245, April 2007.

[9] A.M. Silva, J.C.F. Teixeira and S.F.C.F. Teixeira, "Experiments in a large-scale venturi scrubber Part I: Pressure drop", Chemical Engineering and Processing, vol. 48, pp. 59–67, February 2008.

[10] P. A.B. de SampaioJ. L.H. Faccini and J. Su, "Modelling of stratified gas–liquid two-phase flow in horizontal circular pipes", International Journal of Heat and Mass Transfer,vol 51, pp. 2752–2761, December 2007.

[11] D. He, and B. Bai, "Numerical investigation of wet gas flow in Venturi meter", Flow Measurement and Instrumentation, vol 28, pp. 1–6, August 2012.

[12] A.N. Johnson, J.D. Wright, S. Nakao, C.L. Merkle, and M.R. Mold over, "The effect of vibrational relaxation on the discharge coefficient of critical flow venturis", Flow Measurement and Instrumentation, vol. 11, pp. 315–327, December 1999.

[13] K.A. Ibrahim, Mofreh H. Hamed, W.A. El-Askary, and S. M. El-Behery, "Swirling gas–solid flow through pneumatic conveying dryer", Powder Technology, vol. 235, pp. 500–515, October 2012.

[14] K.S. Mujumdar, V. V. Ranade,"CFD modeling of rotary cement kilns", Asia-Pacific Journal of Chemical Engineering, vol 3, pp.106–118, 2008.

[15] H.Mikul, M.Vujanovi, D.K. Fidaros, P.Priesching, I.Mini, R. Tatschl, N. Dui and G.Stefanovi,"The application of CFD modelling to support the reduction of CO2 emissions in cement industry", Energy, vol 45, pp. 464-473,2012.

[16] U.S. Patent: 7,789,944 B2.

[17] U.S. Patent: 4,059,019.

[18] U.S. Patent 4, 276,092.

[19] Requestpatent 2008/0092739.

[20] Requestpatent 2011/0083745 A1.

[21] D. Montgomery. Diseño y análisis de experimentos. Segunda edición, LimusaWiley, ISBN 13: 978-968-18-6156-8.

[22] User Manual ANSYS-FLUENT 15.1

AUTHOR BIOGRAPHY

Héctor Alfredo López Aguilarborn January 24th 1980 in Uruapan Michoacán, México.Professional education: Master and PhD student in Environmental Science and Technology, Advanced Materials Research Center Chihuahua (CIMAV). Actually, the research activities are focused in combustion simulation CFD and life cycle assessment methodology.

Jorge Alberto Gómez, PhD in materials science. Born in Chihuahua city, state of Chihuahua Mexico on July 3th 1974. Professional education: Industrial Engineer 2000, Technological Institute of Chihuahua. Master's and doctorate degree in materials science 2001 and 2008 respectively, CIMAV, unidad Chihuahua. The relevant contributions in this subject is with the first paper "A Method to Evaluate the Tensile Strength and Stress–Strain Relationship of Carbon Nanofibers, Carbon Nanotubes, and C-chains" published in 2005 on SMALL, as several patents of nanomaterials devices. Actually, the research activities are focused in the lithography construction of the device, test "in situ" of carbon nanotubes employing the new device and study the mechanical properties of different types of carbons nanotubes as a different metallic catalyst on them.

Antonino Pérez Hernández, PhD. Born in Gutierrez Zamora Veracruz, Mexico on January the 9th, 1963. Professional education: B. Sc. Physics 1987, Faculty of Physical and Mathematical Sciences, Autonomous University of Nuevo León, San Nicolás de los Garza Nuevo

León, México. M Sci. in Mechanical Engineering with Specialization in Materials 1992, FIME-UANL, PhD Materials Engineering 1994, FIME-UANL. Dr. Pérez is actually member of the CIMAV

The relevant contributions;"Simulation of Flow Field Pattern Influence on the Hydrogen Consumption in A PEMFC" published in 2013 on Journal of New Materials for Electrochemical Systems. "The mathematical modeling of biomethane production and the growth of methanogenic bacteria in batch reactor systems fed with organic municipal solid waste" published in 2009 on Int. J. Global Warming. "A Method to Evaluate the Tensile Strength and Stress–Strain Relationship of Carbon Nanofibers, Carbon Nanotubes, and C-chains" published in 2005 on SMALL.

Actually, the research activities: Applied Mathematics: Models of Physical Phenomena, Chemicals and Transfer. Process Simulation: Combustion, Filtration. Mass Transport Phenomena and Energy. Alternative Energy: Biomass, Bioelectricity, Hazardous Waste. Life Cycle Assessment (LCA).Human Resources Training.