Combustion adjustment in a natural-draught furnace using Computing Fluid Dynamics

Asya Alabdalah[1], Sergio Hoyas Calvo[2], Ethat Aloosh[3]

[1]Homs Oil Refinery , Homs, Syria
[2]Aerospace engineering in UPV, Valencia, Spain
[3]Al-Baath University, Homs, Syria

Abstract— *Shortage of detailed and accurate experimental data on fuel-air mixing in furnaces is due to the difficulty and complexity of measurements in flames. Although it may be possible with infra-Red camera to obtain an indication of what happens in the furnace by graphical image resolution this is not expected to be sufficiently detailed because it contains only the temperature gradient. More detailed information, however, may be obtained from the simulated resolution using Computational Fluid Dynamics (CFD) technique where the total number of elements/points defines the detailed level that can be displayed or captured in graphical image.*

Simulation resolution studies two aspects of the momentum effects on flame which are the forward momentum normally associated with the average outlet velocity of the combustion products and the lateral momentum caused by swirl. Following the American Petroleum Institute guidelines (API 560) for combustion adjustment in furnaces, it may be possible to have less emission and a maximum efficiency, but the potential interaction between the several operation and design factors are not thereby considered as in a mathematical model of CFD.

Keywords— *Cold flow simulation, CFD, Natural draught, swirl burner, NOx emission.*

I. INTRODUCTION

The purpose of this research is to consider the API 560 standard in the adjustment of combustion in furnaces along with consideration of the application of CFD, this should give emphasis to the reader of the good idea of the general behaviour of different components of swirl burners (case study) as a basic key in the furnace work adjustment. The information of the isothermal cold flow simulation from this research should be useful upon consideration of new applications or re-evaluation of existing technologies as a case study by using NX Siemens tool to have image resolutions which are made up of individual picture elements making up a grid of individual elements. As a result, the image resolution is based on two factors: the total number of points or elements and the number of such points/elements per unit value for any given image resolution, the more points are applied the more detail will be displayed so the question is: How many points or elements are really needed? There is a single answer to this question, it is that the information from a graphic picture which may be obtained from CFD simulation must not vary by more than 5%, for three kinds of mesh, fine, middle and coarse. For computing cost it is generally preferable to use the coarse grid. The current API 560 guideline for combustion adjustment does not take into consideration the potential detailed interaction between operation and design factors especially for the flame which tends to closely follow the flue gas pattern inside the firebox. So the complexity of the flame structure is dependent on the combustor configuration. The momentum effects on the flames' structure in swirl burners have an impact on the heat flux distribution from the flame jet. When the jet is confined, entrained fluid must be replaced from location further downstream of the jet and an annular zone of reverse flow will be formed around the jet [1]. So the swirl burner in this research serves as low NO_x burner relay on high rates of internal flue gas recirculation to lower flame temperature and thermal NO_x emissions, this is typically achieved by maximizing the number of tangential fuel jets and fuel gas pressure in spite of flame holder with set of tangential vanes, and tangential impeller in the annulus around the central nozzle. By using CFD simulation quantitative imaging resolutions can be obtained which represent macroscopic properties reported as numerical results. So computer programming becomes useful because it gives a researcher the ability to interpret the analysis results because images contain information about the object which is under study, although it is less information than the object itself but it is very useful. Moreover measurements conducted on isothermal models with ease and high accuracy can be used to test the validity of the theories and

hypotheses encountered in the prediction methods [2]. The difference between the mixing pattern with chemical reactions and that without chemical reactions may arise from the fact that the combustion process is accompanied by high temperature and density gradients across and along furnace and consequently the mixing pattern might be affected to some extent [1]. Isothermal conditions with no combustion were used to calibrate the system, thus the results are of an indicative nature.

Technical Data of the Furnace under Study and Fluid Dynamics Concepts Used in Industrial Furnaces and Burners (Case Study)
Unit: U210-Penex

- Heater: vertical, cylindrical 210-H1
- Generated heat in the furnace 0.59×10^6 kcal/h
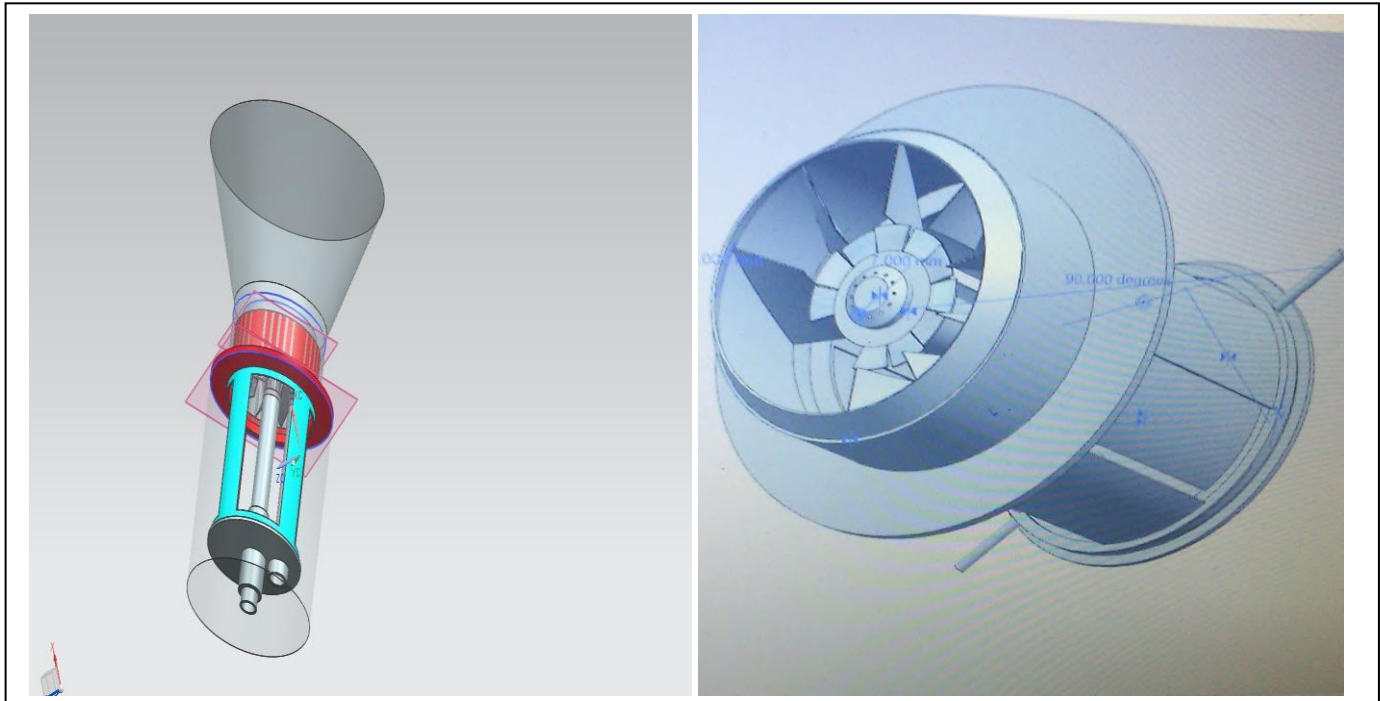- Number of burners: 1 Fig. 1 shows the components of this Swirl burner in CAD environment.

Fig.1: Shows the components of swirl burner and its confusor

- Burner axis: vertical
- Regulating range of capacity of burner 20%-60%
- Excess of air $n_{air}=1.15$
- Required type of the burner: gas type with natural draft of air fitted with a gas injector stabilization torch; Hydrogen content in the heating gas must not exceed 60% volume, and the fuel doesn't contain any nitrogen compounds
- Chemical composition %-vol for gaseous fuel :
  H_2-28,C_1- 34 ,C_2-17,C_3-17 ,C_4-3 ,C_5- 1,H_2O-1.5 ,H_2S-max.5mg/Nm^3
  1- Net calorific value: 11000kcal/kg
  2- density kg/Nm^3 0.924
  3- Flammability limits% 3.4-16.5
- Temperature in the spot of the burner inlet 30℃
- Heating gas consumption of I burner Nm^3/h

For min. output 23.2, for nominal output 58.0 and for max.
Output 69.6

- Heating gas overpressure of the burner kg/cm^2 (g)
  For min. output 0.065, for nominal output 0.405 and for max, output 0.60
- Nozzle placing
  Design 2 of gas head
  Diameter of the head D=38mm, number of nozzles n=12, nozzle diameter d=$2.8^{+0.05}$mm, tangent of the nozzle $\alpha=30°$
- Air deflector
  Outer Diameter of deflector d_1=120mm, fan angles 130×130, thickness of deflector 2 mm
- Confusor with blade
  Plate jacket of thickness 1mm, and dimensions as plate 90×788 mm and its diameter Ø=251mm

Con with thickness 1 mm and dimensions 150×370 mm

Blade with thickness 1 mm and dimensions 103×65 mm

- Air box Consists of Two cylinders, the internal one is fixed with thickness 2 mm and dimensions 804⁻¹×370 mm and diameter 258⁻⁰·⁵mm, and the outside is movable one with thickness 2 mm, dimensions 823⁺²×268 mm and diameter 264⁺¹mm. These two cylinders have the same gates and by moving the outer cylinder against the fixed cylinder, changes the air flow area.

From a design point these gates must provide enough air for fuel's burning process. The correlative with maximum amount fuel consumption B_{MAX} and surface area of all gates F_g is defined by [3]

$$F_g(m^2) = \frac{B_{max} \times l_\alpha}{w_g \times J \times K}$$

B_{max} Maximum amount fuel consumption, in our research = (69.6 ×0.924) kg/h

l_α Amount of air providing for burning fuel in our research = 14Nm³$_{air}$ / $_1$ kg$_{fuel\ gas}$

j Area of activated grate = 1 in this case study

k Discharge consumption coefficient 0.6-0.75

w_g Air speed through grate 0.5m/s

- Ambient air temperature, pressure and humidity affect the air flow rate if the air temperature increases the air flow rate decreases. As definite temperature, pressure and humidity of air increase, water vapour in ambient air is decreased and leads to decrease of the percentage of O_2. So the air flow rate through the air box at definite pressure drop by using corresponding – state is defined by [4]

$$m_{air} = \sqrt{\frac{2g \cdot \rho_{air} \cdot A_b^2 \Delta p_b}{k_b}}$$

m_{air} Air mass flow rate kg/s

g Gravitational constant 9.8kg/s²

A_b Cross section area in internal cylinder m²

Δp_b Air side pressure drop across air box pascal

k_b Pressure loss coefficient through air box

- The computation of hot air resistance ducts is similar to that for cold air ducts; the hot air volume flow rate in furnace often can calculated from [5]

$$V_{hot\ aie} = B.V^\circ(n_{air} + \Delta_{air}).\frac{273 + T_{air}}{273}$$

$V_{hot\ aie}$ The flow of hot air m³/s

V° Theoretical air flow Nm³/kg

B Fuel feed rate kg/s0

n_{air} Excess air coefficient at furnace outlet

Δ_{air} Leakage air coefficient of the furnace

T_{air} Air temperature °C

-The pressure drop across combustion equipment like a burner etc., can be calculated from the following equation [5]

$$\Delta P = K\frac{\rho u^2}{2} \quad Pa$$

U = velocity of air at burner exit m/s

K is the flow resistance factor where K=1.5 for tangential burner and K= 3 for swirl burner of gas and heavy oil [5] so in our case study for the swirl burner K=3 for this CFD model

- The sum of resistance of all parts of the air duct gives the total resistance of the air duct $\sum \Delta P$. If the local elevation above sea level is higher than 200 m, the pressure drop should be corrected for the local atmospheric pressure. The total pressure drop after this correction ΔP_r is [5]

$$\Delta P_r = \sum \Delta P \frac{101325}{P_{av}}$$

P_{av} Is the mean pressure of the air in duct Pa

$\sum \Delta P$ Is the total resistance of the air ducts system Pa

ΔP_r Is the total pressure drop after correction of elevation in this CFD model, this correction was not applied because the general design case is being considered for swirl burner

- When $\sum \Delta P$ is less than 3,000 Pa , P_{av} is taken to be equal to the atmospheric pressure P [5] so this is the case of this swirl burner and it was opened to the atmospheric pressure as a boundary condition on one of its sides .

Otherwise P_{av} should be calculated as $P_{av} = P + \sum \Delta P$ /2

- Natural draft in air ducts in furnace can be calculated from

$$P_{nd} = H.g.(\rho_a'' - \rho_a') \quad Pa$$

P_{nd} Is the natural draft in air duct

H Is the height between the start and end sections of the duct m

g Is the Gravitational constant 9.8kg/s²

ρ_a'' Is the density of air at the lowest point of duct kg/m³

ρ_a' Is the density of air at the highest point of duct kg/m³

- Adding the natural draft components of all parts of the air ducts, the total natural draft $\sum P_{nd}$ is obtained, so the total pressure drop ΔP_{air} in the air ducts in the furnace can be obtained by subtraction.

$$\Delta P_{air} = \Delta P_r - \sum P_{nd} \ \text{Pa}$$

ΔP_{air} Is the total pressure drop ΔP_{air} in the air ducts in the furnace ,

ΔP_r Is the total pressure drop after correction of elevation

$\sum P_{nd}$ Is the total natural draft of all parts of the air duct

II.   THE ANALYSES AND THEIR STANDARD METHODS

-IR camera photos for the furnace

-Testing the furnace according to API 560 standard which is required for these analyses

1- -determination of NOx emissions from stationary source by Environment Protection Agency EPA method 7E [6].

2- Determination of oxygen and carbon dioxide emissions from stationary source by Environment Protection Agency EPA method 3A [6].

3- Gas chromatography for fuel gas and for flue gas.

- Image resolutions by CFD simulation for swirl burner

III.   RESULTS

– By testing the furnace according to API 560 standard it was noted that: There is additional resistance in flue gas flow by partial block in convection section as the result of lining material deposited on tubes in convection section. The gate in this section was opened and air enters the furnace here. It was not possible to get samples for analysis in radiant section to calibrate the amount of this leakage of air .Table 1 shows the test results.

– IR camera photos was taken for the furnace. Fig. 2 shows its details and it is clear that there is lack of refractories in stack flow and the colour was not dark for all the structure. The colours indicate temperature in Celsius degree.

– In this type of non-premixed flame , the fuel and oxidizer are coming in either side of the reaction zone and the product go out of these reaction zone , so the flame is established at allocation where the total enthalpy of the reactants present balance the total enthalpy of the products generated plus any energy losses .

Table.1: shows the data which were from the instruments on 210-H$_1$ furnace for API 560 tests

Fourth test	Third Test	Second test	First test	unit reading	instrument	Furnace 210-H1
17.5	18.25	18	16	m³/h	FRCAL32	The feed in of naphtha
7900	7600	7800	8000	Nm³/h	FRCAL38	The feed in of hydrogen
128	128	130	129	℃	TI34	Temperature of feed in
146	147	148	146	℃	TRCAH48	Temperature of feed out
175	180	180	177	℃	TI274	Temperature under the stack damper
497	482	491	451	℃	TI46	Temperature of convection section
425	420	432	394	℃	TI275	Temperature of radiation section
0.09	0.12	0.17	0.1	Kg/cm²(g)	PIC186	Pressure control of gas fuel for burner(gauge)
-2	-5	-5	-6	mm H₂O	PI45D	Draft gauge under of stuck damper
-11	-13	-12	-13	mm H₂O	PI45A	Draft gauge at the bottom of radiation section
-28	-27	-27	-20	mm H₂O	PI45B	Draft gauge above stuck damper
-1	-1	-1	-1	mm H₂O	PI45C	Draft gauge under convection section
51.2	54.4	80	67.2	Nm³/h	FR 190	Burner line
51	53	50	57	℃	TI191	

0.09	0.14	0.14	0.1	Kg/cm$^2_{(g)}$	P153	
3.4	3.4	3.3	3.7	Kg/cm$^2_{(g)}$	PI150	
0.11	0.12	0.11	0.22	Kg/cm$^2_{(g)}$	PI51	Pilot line
3.6	3.6	3.5	3.8	Kg/cm$^2_{(g)}$	PIAL49	
45+ adjusting burner's gates	60+ adjusting burner's Gates	50	65	%		Stack damper opening ratio
20	21	22	16	℃		Climate temperature
66	66	66	66	%		Relative moisture

Fig.2: shows the lack of refractory
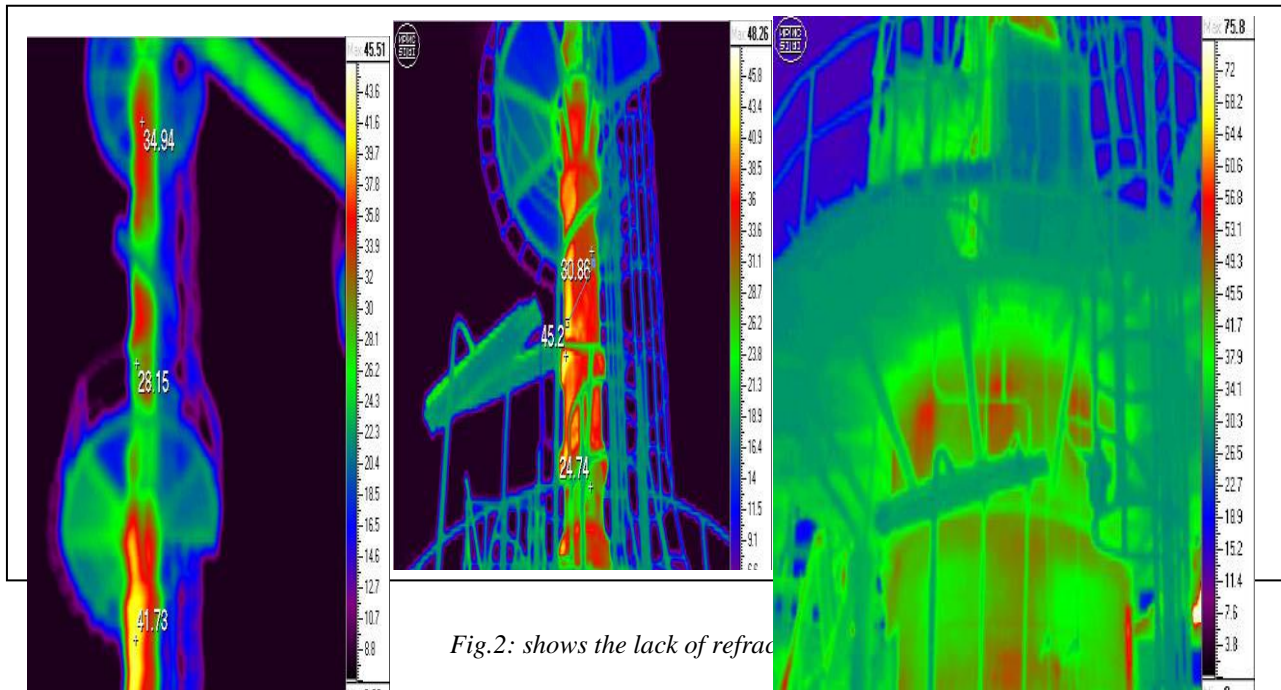
Fig.3: shows the open gate of convection section in furnace,and the destroyed air deflector vanes in axial of burner.

So from the photos in fig.4 and fig.5, it was noted that the flame is rumble or vibration which is a sign of dangerous instability. This was the result of destroyed air deflector vanes. By reaching the critical gas pressure on burner line, the flow type will have been changed from subsonic to sonic at nozzles exits, and this means that thermal stress destroyed the air deflector by detonation phenomena. This was clear from swirl burner data design that when the reading of the fuel gas consumption FR 190 instrument on burner line reaching 80 Nm^3/h the relation between the net calorific value and the gas pressure becomes straight . The flame is lifted off the burner and travelled to the place beyond the flammability limit extinguish so under certain condition the fuel may burn incompletely during one part of the cycle and reignite later causing an explosion and this also destroyed the metal structure but here the cycle of lift-off and burn back occur many times a second causing rumble or vibration such ramble can be a sign of dangerous instability [1].

- Usually the emission quantities depends on pollutants content of the fuel and is related to fuel consumption so adjustment the combustion as clear from red arrows in table 1 decrease the stack temperature and make the fuel gas consumption FR 190 on burner line about50 Nm^3/h , so save the money and reduce the gaseous emission .
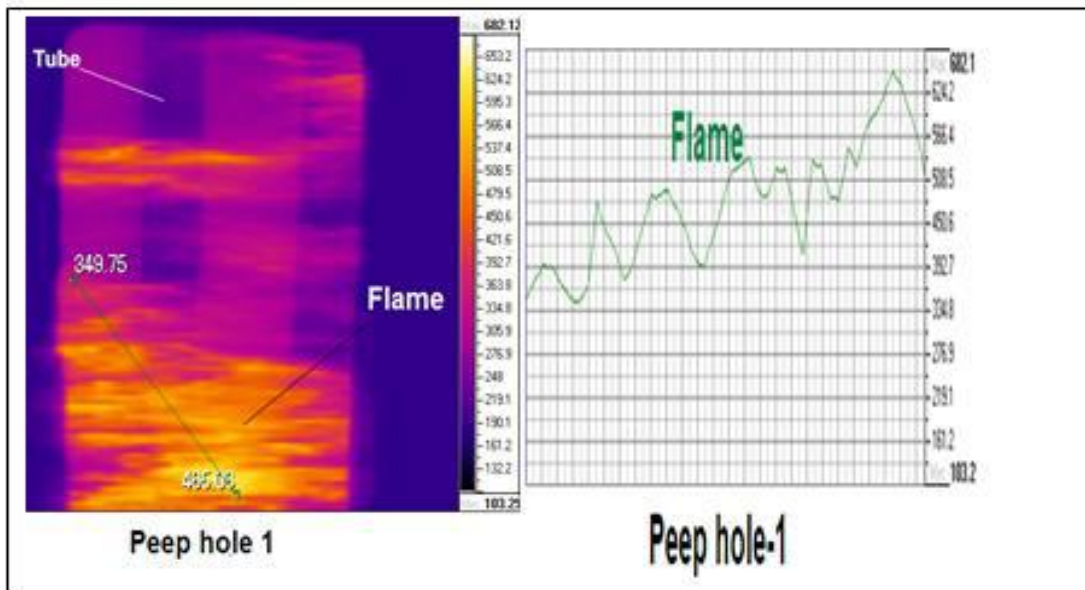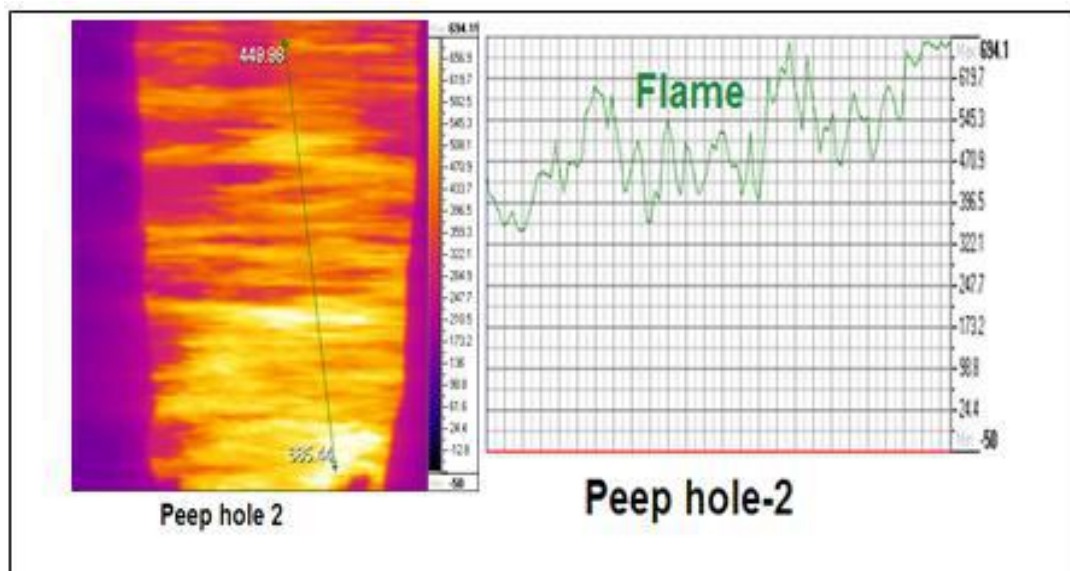

Fig. 4: shows photo of instability flame

Fig. 5 shows flame temperature and its instability

-Here the NO_x is thermal because the fuel doses not have Nitrogen compound. This thermal NO_x is controlled by the swirling dynamics to give high flame stability tacking advantage of structures such as corner and central recirculation zone (CRZ). Coherent structures providing a low velocity region where the flame speed matches the flow velocity thus anchoring the flame whilst by the CRZ saving to recycle heat and active chemical species to the root of the former increasing their residence time [3].Using high swirl number in burner allowing the use of low equivalence ratio thus giving lower flame temperature but for this case study there is a problem in burner which arise from destroyed the air deflector and instability flame.

So according to Zeldovich mechanism it is usual to have little quantities of NO_x emission because by analysis the flame temperature which was obtained it was too far from 1450 ˚C, and it does not easy to have oxygen atoms from their molecules to start thermal NO_x formation and this result is emphasized by used EPA method 7E to evaluated level of NOx and this level was not more than 45 ppm_v for all cases but it is clear that these results for NOx are not real that is because there are a lot of hydrocarbon in flue gas in spite of a large quantities of excess air so the burner needs to calibrate by using CFD .

In engineering practice, analysis is largely performed with the use of finite element computer programs (such as NASTRAN, ANSYS, ADINA, SIMULIA, etc.…) these analysis programs are interfaced with computer-aided design (C A D) programs Catia, SolidWorks, Pro/Engineer, NX, etc. Numerical simulation of a turbulent gas burner of diffusion flame were carried out using the commercially available software package NX Siemens.

IV. COMPUTATION FLUID DYNAMICS MODEL OF SWIRL BURNER

4-1-Physical model of swirl burner
- The cold isothermal flow simulation where numerically simulated in the near burner region where most of the mixing and reaction occur. This is a swirl burner with a cyclone re-combustion chamber. The fluid from the swirl burner is contracted first in the air exit and then it spreads into the furnace by using the expansion cone.

- In order to make the simulation more in line with the actual situation, this paper directly create a three dimensional model for a swirl burner with natural draft in air box. At first air box simulation was obtained for full opening gates upon maximum fuel consumption. By using this simulation results, the average air velocity when inter the second part of the model was obtained. The second part of the model stands the tangent air Swirl plus the swirl burner. It was assumed that isothermal air flow into this swirl burner. As a result it is a signal-phase homogenous flow and therefore the governing equation for the mixture flow can be written according to the three equations for fluid flow namely the conservation of mas, momentum and energy The flow was solved in steady-state frame. All walls of the components in the model were smooth, bouncy effects on the flow were taken into account, and the relative velocity of the fluid at the walls is zero. In this cold flow simulation all working fluid are not compressible.

4-2- Wall y+ Strategy for Dealing with Wall-bounded Turbulent Flows:

A wall y+ in the range of 30 to 60 is determined to be sufficiently accurate. Where the log-law region is resolved. It is also advisable to avoid having the wall-adjacent mesh in the buffer region since neither wall functions nor near-wall modelling approach accounts for it accurately

4-3- K-epsilon model for flow in the model [7]:

The model contains two extra equations to represent the turbulent properties of the flow, the turbulent kinetic energy, and the turbulent dissipation. It is important mention that the model containing five constants, which are determined by data from a wide range of turbulent flows (Versteeg & Malalasekera, 2007). Since it isn't possible to calculate an exact solution it is even impossible to determine the error of the turbulence model. It is significant to mention that CFD analysis is never exact due to approximations and modeling errors, and should only be used as a complement experiments and theories related to fluid dynamics (Wolfshtein, 2009).

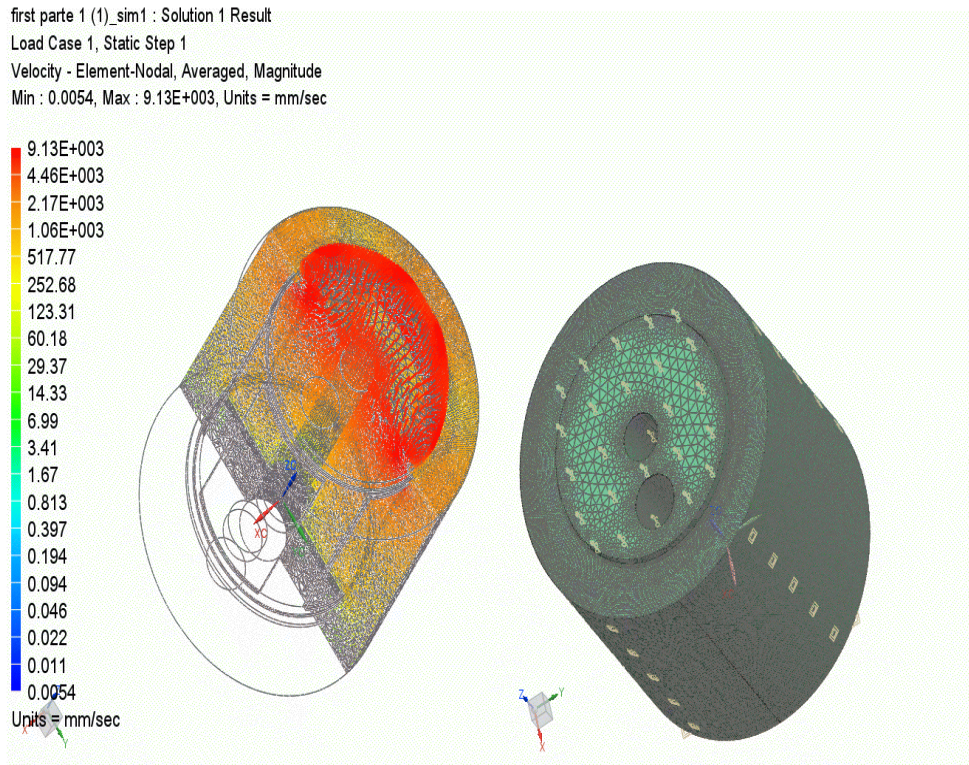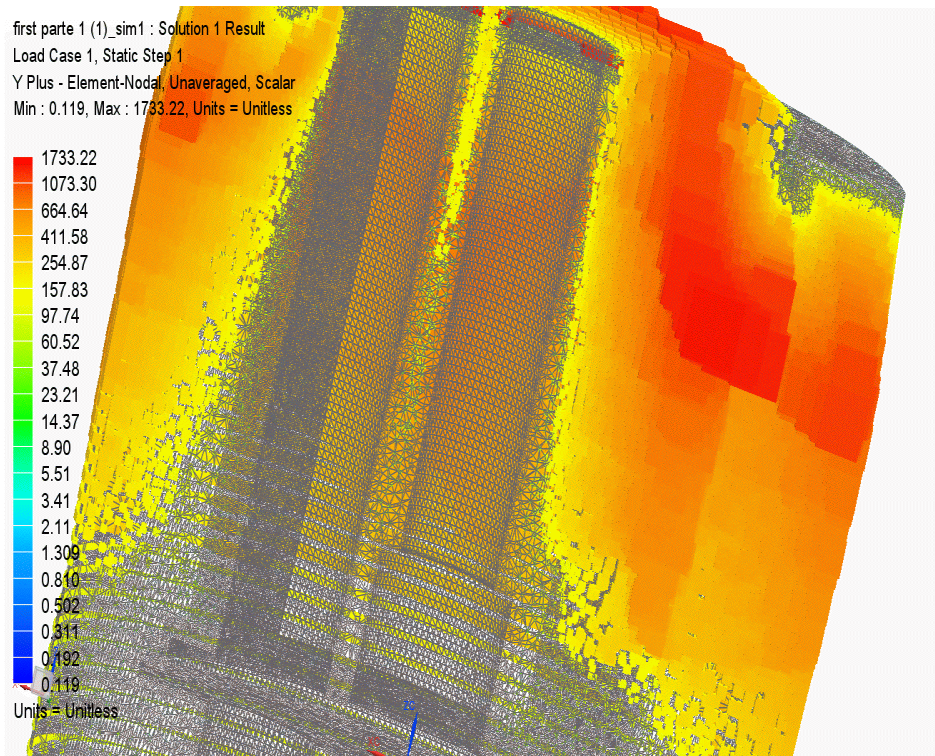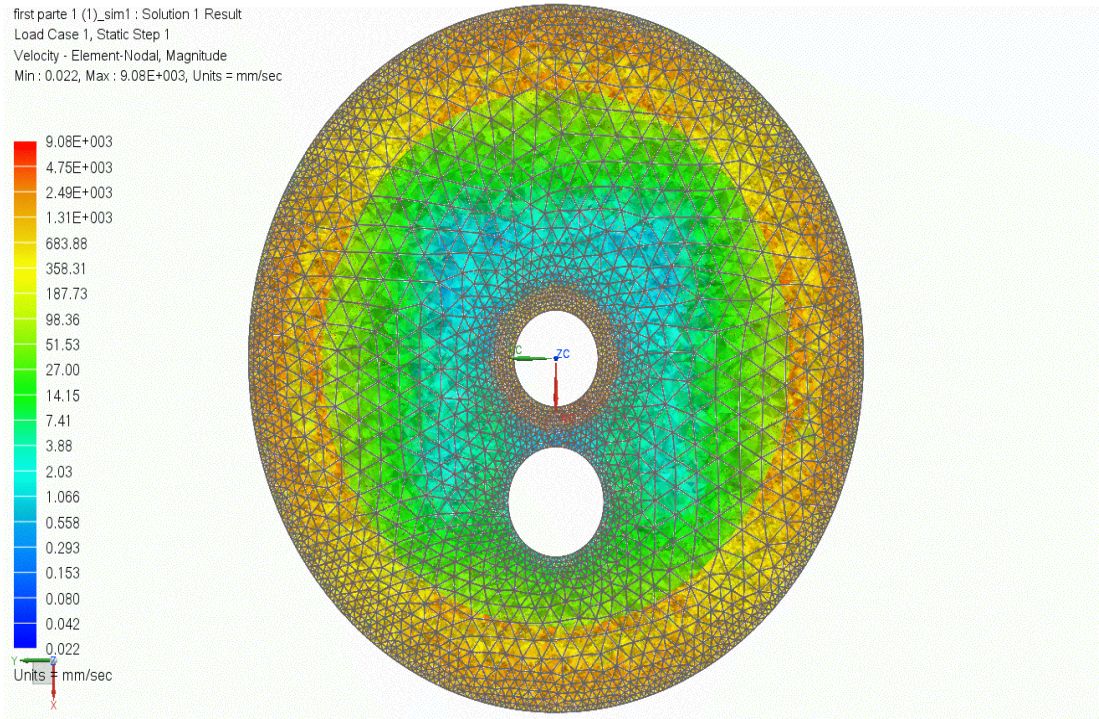4-4- Result of The image resolution of the CFD model
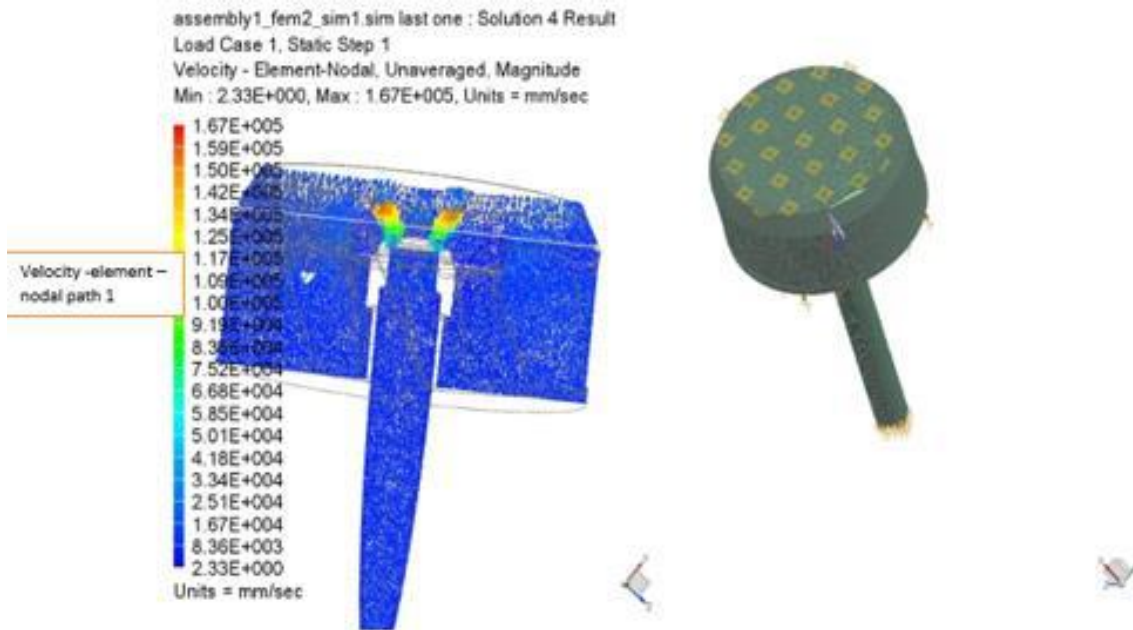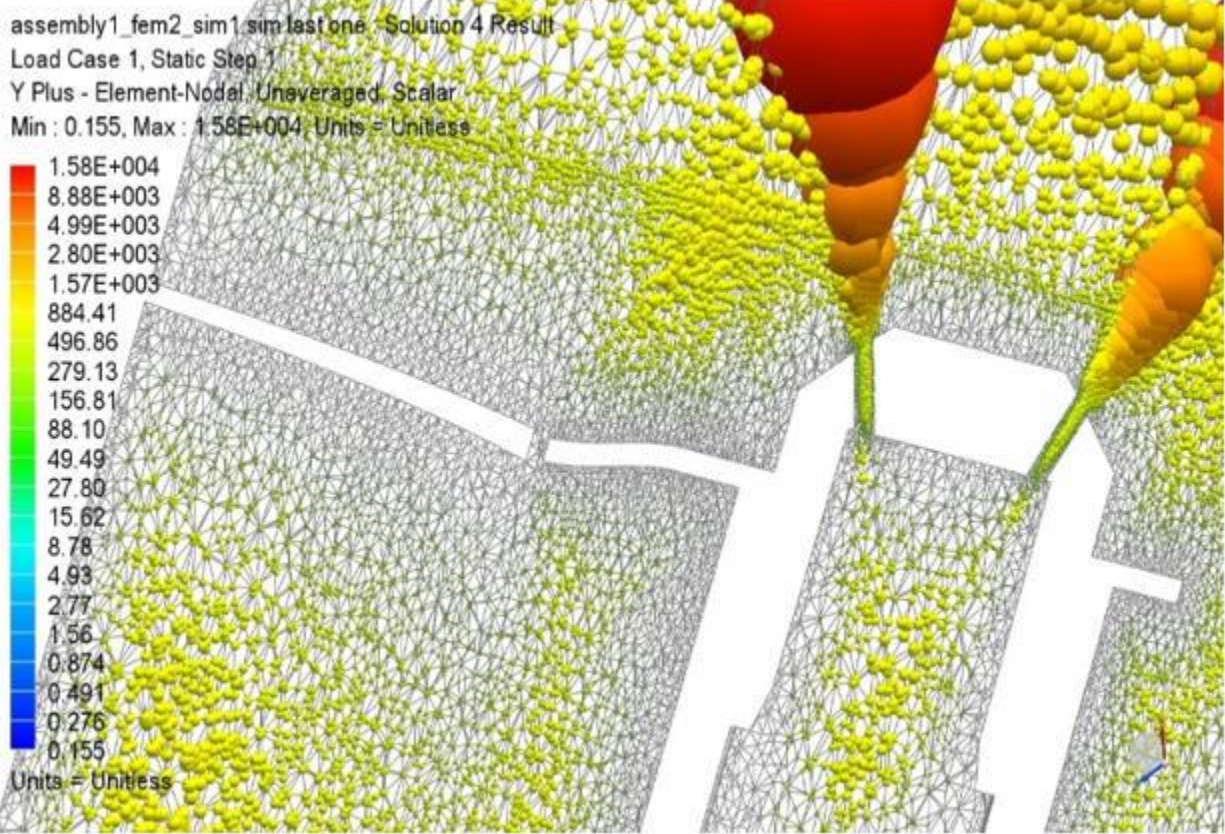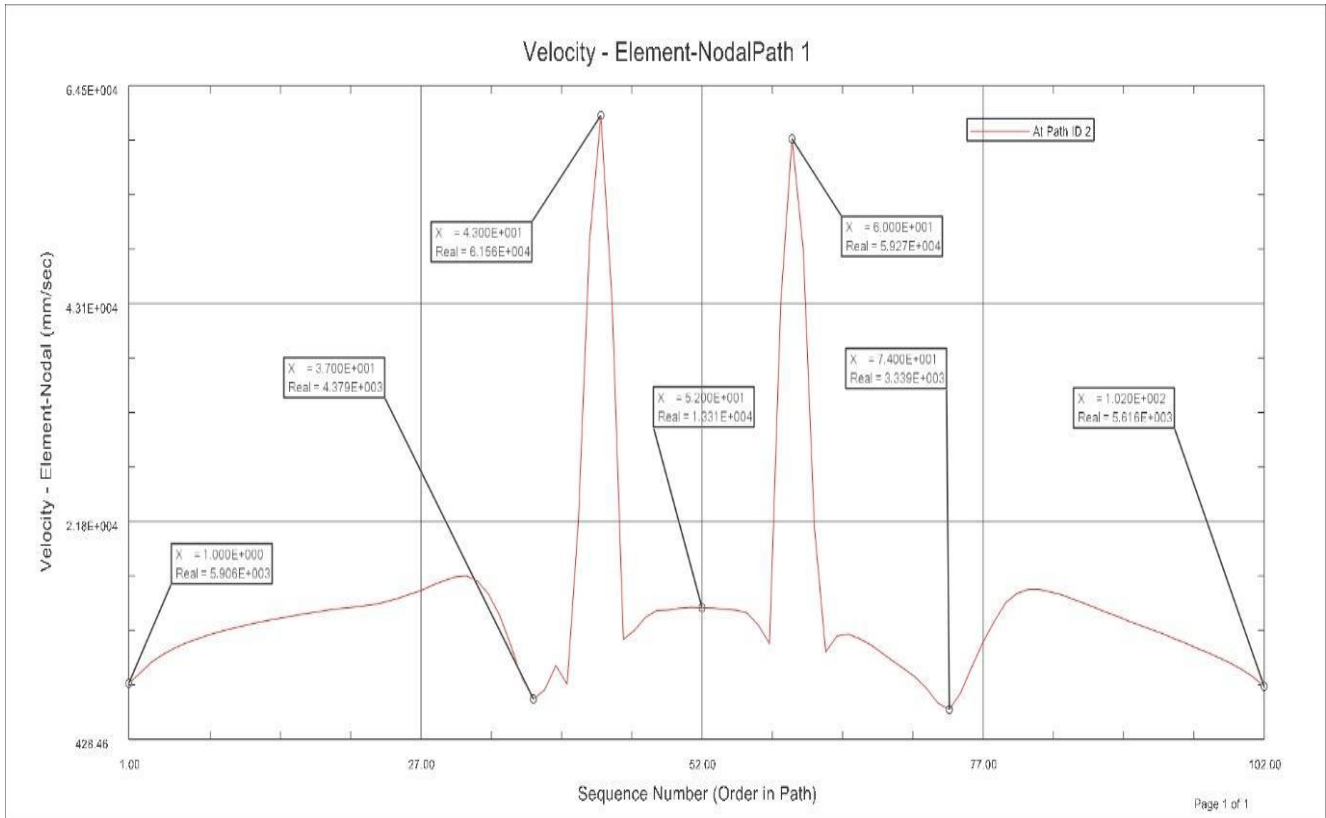
Fig.6 three air box resolution images by CFD simulation

Y+ value of K-epsilon model by use wall function and the resulting air distributed velocity in air box at computing Reynold number equal to 1.8142E+05 . From this simulation of air box we get average value of velocity in the plane cross at the entrance of the next part of the CFD model as it is clear by the last one of numerical results images in fig.6.

- The grid was designed using cold flow solutions. Grid densities were lowered below air exit contracted and along the center axis of nozzles. The residuals used in this study were therefore adjusted appropriately in order to ensure a comparable degree of accuracy between tests. A redesign of the mesh including testing would have taken considerable time away from a swirl burner model development during an advanced stage of the project, thus necessary modification were made to complete with the current grid

Fig.7: five resolution images of the second part CFD simulation for Y+ value of K-epsilon model by use wall function and the velocity distribution in it at the Computing Reynold number equal to 1.2038E+05

- This burner is designed to give high heat release, in short distance within its turn down ratio, so in the numerical results of 1,2,and 3 images of fig.7 , it is clear that fuel velocity exceeds laminar flame, and an ignition ledge is used as air deflector and even if the air flows at very high speed at maximum fuel rate , the air speed very close to the ledge will be small enough , so the flame can establish very close to the ledge ignition and be quit stable even over a wide range of firing rate so it is necessary to clear this idea by CFD

The explanation of 4 and 5 images in fig.7 that is as the tangential momentum is added to flow, radial pressure gradients increase. Essentially the fluid wants to move out from the center of rotation, these radial pressure gradients generate a low axial pressure zone which then draws material back up into a CRZ.

V. CONCLUSION

In this study a tool used to determine the main characteristics of swirl flow quantities of the mixing as represent the turbulent properties of the flow, one handling the turbulent kinetic energy, and the other one the turbulent dissipation, and the CRZ in spite of the wake region of the air deflector which related to its shape, size and blockage ratio. So these cannot be studied by IR camera or API 560 but only studied by CFD. SO the Scientific Laboratory of Researches is nothing without Modern Scientific finite element computer programs analysis.

REFERENCES

[1] Charles, E. Baukal, J.R.2013.Combustion Handbook. The John Zink Hamworthy.volume1:327-387

[2] Surjosato, A. Priambodho, Y.D.2011.Investigation Of Gas Swirl Burner Characteristics On Biomass Gasification System Using Unit Equipment.Jurnal Mekanikal No33,15-3

[3] Hay, N.Researching On Burner, Construction Equation Of Air Flow Velocity And Defining Appropriate Drayer Wall for Tobacco Dryer In Vietnam. Nong Lam University.

[4] Masoumi,M.E.Izakmehri,z.2011.Improving Of Refinery Furnaces Efficiency Using Mathematical Modeling. International Journal of modeling and optimization .volume1:pages74-79

[5] Sasu, P. Kefa,C. Jestin, L. Boilers And Burners Design And Theory .Springer ISBN0-387-98703-7

[6] Central pollution control board .2013Guidolines On Methodologies for Source Emission Monitoring, LATS/80/

[7] Versteeg,H.K.Malasekera,W.2006.An Intriduction To Computational Fluid Dynamics. second edition.ISBN 978-0-13-127489-3

[8] American Petroleum Institute .2013.Fired Heaters for General Refinery Service .API standard 560.third edition.