

Computing fluid dynamics on burner air flow in natural draft furnaces

Asya Alabdalah, Hassan Al-haj Ibrahim, Ethat Aloosh

Al-Baath University, Homs, Syria

Abstract— The mean key variable to control the burner performance for safe and efficient operations is the amount of excess air or oxygen which flow to react with fuel. In accordance with **Typical Draft Profile in a Natural Draft furnace** the numerical results of computing fluid dynamics model of the air flow from the environment to the burner by adjusting the area of air box prototype openings will give the necessary information to adjust the performance of burner as optimum. The model of air box prototype is useful tool in applying similarity theory on another burners' designs which depends the air box tools to calibrate the quantity of combustion air flow to the burner.

Keywords— excess air, air box design, computing fluid dynamics, natural draft furnaces.

I. INTRODUCTION

In many burners of natural furnaces today there are insufficient means to adjusting the ratio of fuel to air, the key variable to control the burner performance for safe and efficient operations is the amount of excess air or oxygen which flow to react with fuel. While some excess oxygen is necessary to have safe operation, too much oxygen wastes fuel, energy, and heat. The Air box is one component of burner which is used to control the amount of air excess by modifying its air registers. So the numerical results of computing fluid dynamics of the air flow from the environment to the burner by the air box will give the necessary information to adjust the performance of burner as optimum.

In result; the process engineer will not depends on the laboratory analysis about the flue gas data from all sides of the combustion chamber of furnace at each burner. That is because the numerical results give a lots of information about the air flow through the air box to burner and its quantities from a design point to the operating case as a result of coupling the average velocity of combustion air which enter from the gats of air box into burner with the pressure arise across the burner or the air box.

The pressure arise across combustion equipment like a burner etc., can be calculated from the following equation [1]

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

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 [1]

But from operation point, the air flow rate in burner can calculated from [1]

$$V_{air} = B \cdot V^{\circ}(n_{air}) \cdot \frac{273 + T_{air}}{273}$$

V_{air} The flow of air m³/s

V° Theoretical air flow Nm³/kg

B Fuel feed rate kg/s

n_{air} Excess air coefficient at furnace outlet (there is not leakage into furnace)

T_{air} Air temperature °C

So the numerical results information which are gotten from the prototype air box can be used to predict the quantity that may be enter from the doors of the air box by using the theory of dynamics similarity which related by the Reynolds number of the air flow into the burner at all times with its Reynolds number at design case.

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 [2]

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

B_{max} Maximum amount fuel consumption

l_a Amount of air providing for burning the fuel

j Area of activated gate = 1 in this case study

k Discharge consumption coefficient 0.6-0.75

w_g Air speed through gate 0.5 m/s

Here the process engineer thinks that in fact the excess air does not related only by adjusting the air box registers because it is important to maintain the correct furnace draft. So too little draft can damage the metal structure and snuff

out the burners or flame and too much draft can pull excessive amounts of air into the furnace. So the operation of fired furnaces is complex process and there are different variables on both the combustion and process sides of the furnace. In result; there is just enough air to assure complete combustion. Once, the air registers are adjusted for proper combustion, the damper should be adjusted for proper draft. So the engineer can make natural draft Heater adjustment according to the following flow chart showed in fig (1).

TO benefit from the numerical results information which are gotten from the prototype air box from a design point, the process engineer must begin to ultimate the pressure arise across the burner (draft available at burner level), and he must follow these general rule which are mentioned for any duct in furnace not only for burner .

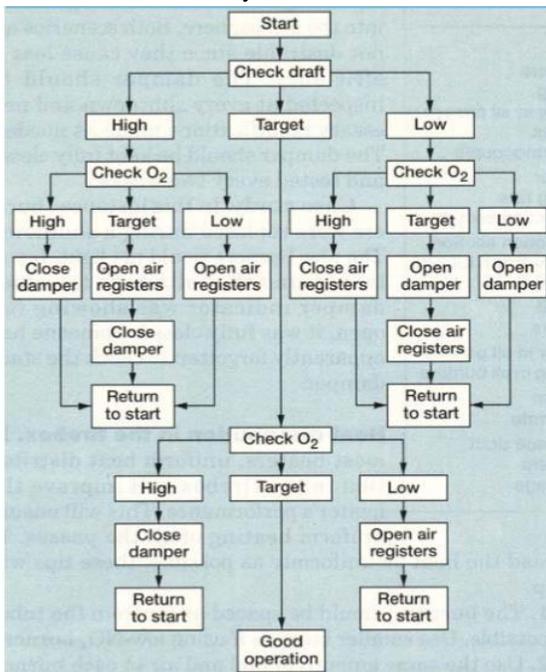


Fig. 1: Natural Draft Heater Adjustment Flow Chart [3]

1- General Rules to ultimate the pressure arise across any air duct in furnace

By 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 [1]

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

ΔP_{air} Is the total pressure drop in the air ducts in the furnace, Fig.(2) Shows typical draft Profile in a natural draft furnace

ΔP_r Is the total pressure drop after correction of elevation.

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

The sum of resistance of all parts of the air duct gives the total resistance of the air duct $\sum \Delta P$.

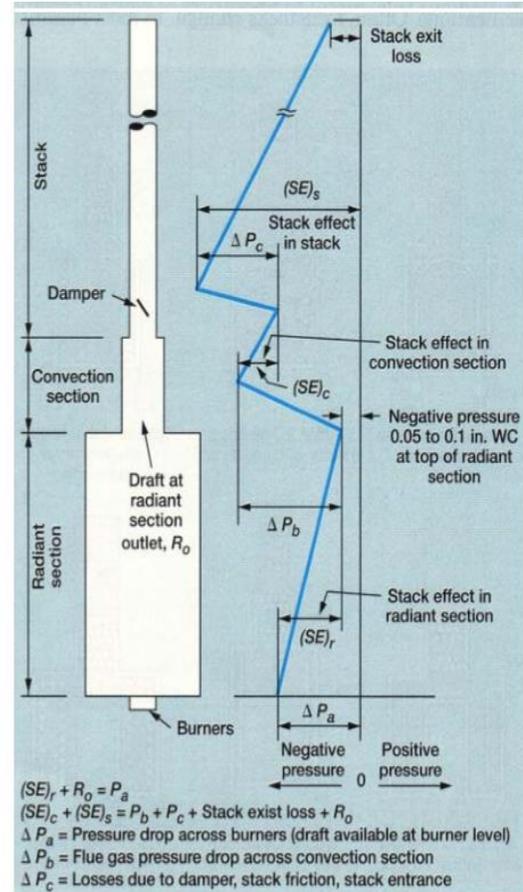


Fig.2: Typical Draft Profile in a Natural Draft furnace [3]

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

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

$$P_{nd} = H \cdot g \cdot (\rho''_a - \rho'_a) \text{ Pa}$$

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 9.8 kg/s^2

ρ''_a Is the density of air at the lowest point of duct kg/m^3

ρ'_a Is the density of air at the highest point of duct kg/m^3

2- Mathematical model on air flow into swirl burner as case study

The burner used in this study consists of two concentric tube with a tangential and radial swirler placed in annular arrangement. Fig. (4) and (5, B) illustrates the important aerodynamics feature of the vertical low NO_x swirl burner.

The combustion air is introduced through two concentric annular nozzles of which the positions of the primary, secondary air, and fuel tube are adjusted to produce the

desired internally staged flame, the burner utilizes swirl induces centrifugal body forces to dampen turbulent exchange between the fuel and air stream allowing an extended residence for fuel conversion chemistry in locally fuel –rich environment prior to burn out [4].The NO_x

emission is at its peak at stoichiometric conditions when other pollutants, like CO and VOC, is at its minimum (see Fig.(3)).This is mainly because of the strong relation between temperature and thermal NO_x production [5]

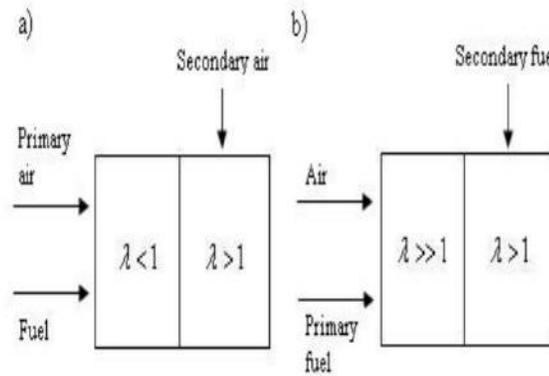
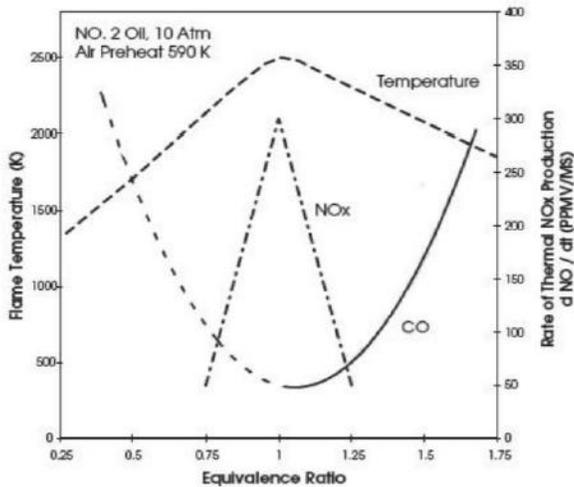


Fig.3: NO_x formation rate driven by temperature at equivalence ratio λ and Schematic drawing of (a) air staging, and (b) fuel staging [5]

NO_x reduction needs to apply correct operating parameters and used as a design basis, so in accordance with NO_x reduction work, it is essential that excess air levels be reduced to close to 2–4% O₂ as all NO_x emissions are corrected to 3% O₂ (see fig. (4))[3]. The most significant useful phenomenon of the swirling flows is the existence of internal recirculation zone IRZ which aids

in flame stabilization by providing a storage of hot flow flue gas, so its flow field is thank for its effect in internal flue gas recirculated to the combustion zone to reduce the NO_x emissions, by reducing the oxygen concentration as well as lowering the flame temperature. In internal FGR, the fuel gas is recycled into the flame zone due to burner aerodynamics [5]

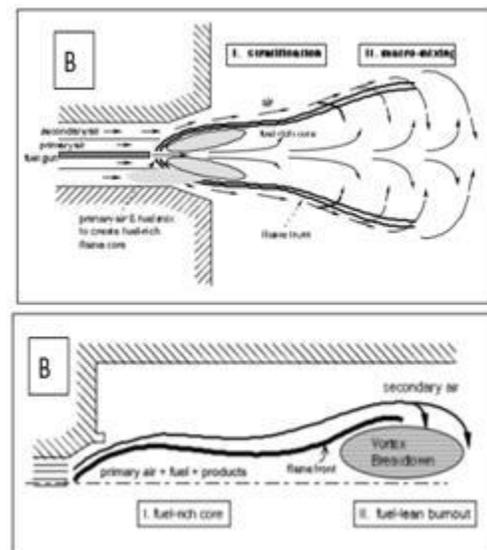
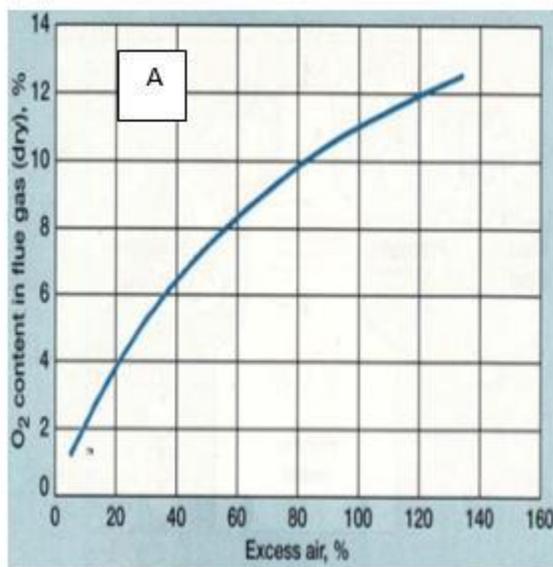


Fig.4: Excess Air vs. Oxygen Content in Flue Gas (A) [3] and typical low NO_x swirl burners' flow field with internal staging schematic (B) [4]

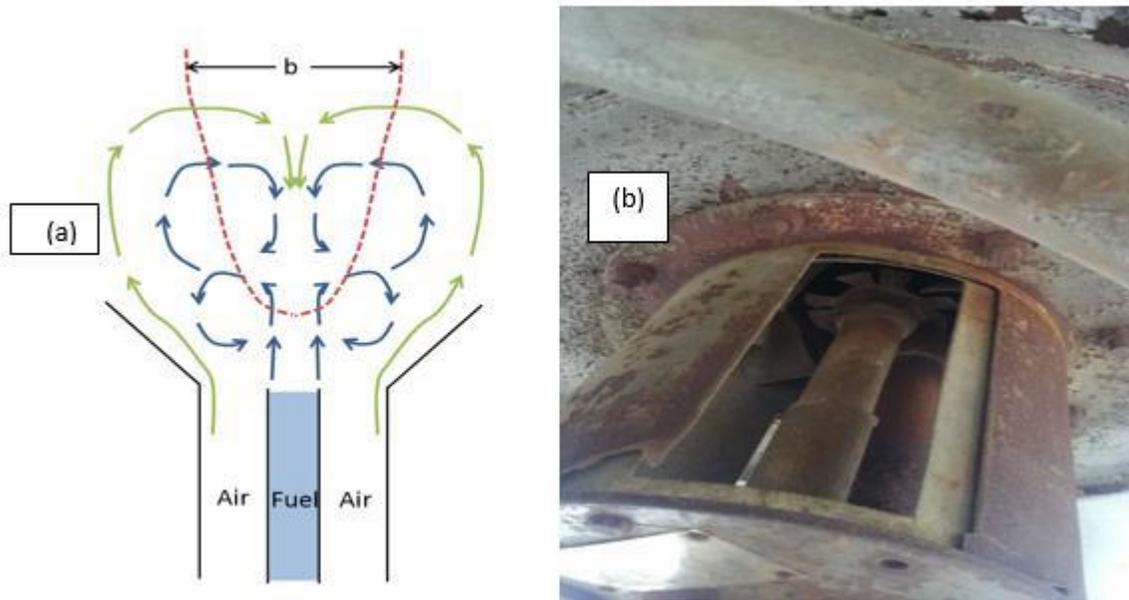


Fig.5: Shows a schematic diagram of the air and fuel flow velocities of a swirled jet burner (a) [6] and burner geometry (b).

In result, combustion product are recirculated and by this phenomena reduced velocity region is created, the boundary of the recirculation eddy is determined by the radial points at which the forward mass flow equals the reverse mass flow at the axial station so the region which is enclosed where the axial velocity reaches zero magnitude in each axial plane [6].

3- Computing fluid dynamics for the air box of burner:

-air box geometry:

Air box Consists of Two cylinders, the internal one is fixed with thickness 2 mm and dimensions $804^1 \times 370$ mm and diameter $258^{0.5}$ mm, and the outside is movable one with thickness 2 mm, dimensions $823^{+2} \times 268$ mm and diameter 264^{+1} mm. These two cylinders have the same gates and by moving the outer cylinder against the fixed cylinder, changes the air flow area. See (fig. (6) Air box fluid domain)

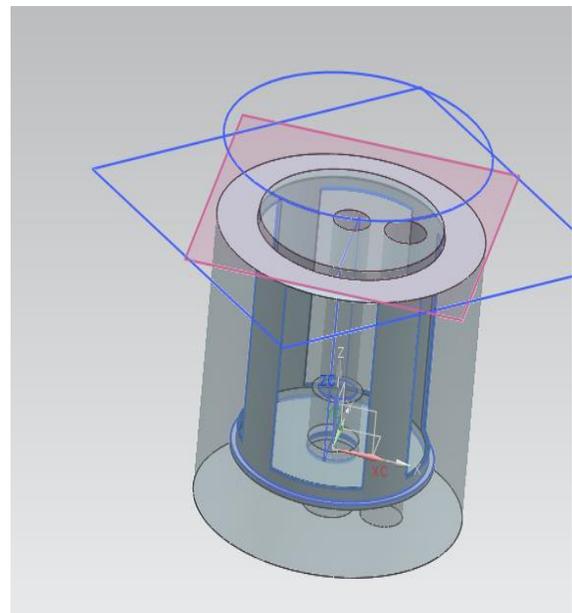


Fig.6: Air box fluid domain (full opening)

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 vapor 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 [7]

$$m_{air} = \sqrt{\frac{2g\rho_{air}A_b\Delta p_b}{k_p}}$$

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 in pascal (natural draft is available at air box level), k_p Pressure loss coefficient through air box and can be estimated as equal to 0.6-0.75

- Cold mathematical model flow of vertical air box:

As in the case of any problem in flow field this particular type of flow described by the mathematical expressions of continuity and momentum in Cartesian coordinates:

Continuity equations:

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0$$

Momentum Equations:

In this situation the body force with constant density equals to the balance between the driving force of buoyancy and the initial hydrostatic pressure, so at any point in the fluid domain the equation of motion is:

$$\rho \left[\frac{\partial \bar{u}_i}{\partial \tau} + \frac{\partial (\bar{u}_j \bar{u}_i)}{\partial x_j} \right] = - \frac{\partial p}{\partial x_i} + \frac{\partial T_{ij}}{\partial x_j} + B_T \cdot g \cdot (T - T_\infty)$$

Where: T is the actual temperature applicable locally. T_∞ : is the undisturbed constant temperature. B_T : is the coefficient of the thermal expansion of air and by using the equation of stat for the perfect gas

$$\rho = \frac{p}{RT}$$

Where R is universal gas constant, p is the local static pressure, and ρ is the local density so ($B_T = 1/T$), T_{ij} is the Reynolds stress tensor divided by the density and it is computed by using the Boussinesq model equation

$$T_{ij} = -\bar{u}_i \bar{u}_j = \nu_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} K \delta_{ij}$$

Where K is the turbulent kinetic energy, δ_{ij} is Kronecker Delta Function, Then the letters i, j, k ...can be used as variables, running from 1 to 3 so Instead of using x, y, and z to label the components of a vector, we use 1, 2, 3 and ν_T is the turbulent eddy viscosity can be calculated by K-Epsilon model so

$$\nu_T = C_\mu \frac{K^2}{\epsilon}$$

Where C_μ is constant equals to 0.09

These models are not valid near wall to model wall effect so wall function have to be employed.

- Wall y^+ Strategy for Dealing with Wall-bounded Turbulent Flows for air box geometry:

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, so for the existence of deferent boundary layers in air box geometry enhanced wall function is used

- **Boundary condition:**

Here the air box was opened to the environment from surrounding sides of the gates at the maximum fuel gas consumption which it was $B_{max} = 0.02 Nm^3/s$ and

($\lambda = \left(\frac{12 Nm_{air}^3}{1 Nm_{fuel, \lambda=1}^3} \right)$). In result from design point, air mass flow will be equal to ($\rho_{air} \cdot B_{max} \cdot \lambda$) at $\lambda = 1.15$

So calculating the air mass flow yields

($\lambda = \left(\frac{14 Nm_{air}^3}{1 Nm_{fuel, \lambda=1.15}^3} \right)$), and the following are the technical data of burner:

- Burner axis: vertical
- Regulating range of capacity of burner 20%-60%
- Excess of air nair $\lambda = 1.15$
- 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/Nm³
- * Net calorific value: 11000kcal/kg
- * Density kg/Nm³ 0.924
- * Flammability limits% 3.4-16.5
- Temperature in the spot of the burner inlet 30°C
- Heating gas consumption of I burner Nm³/h;
 For min. output 23.2, for nominal output 58.0 and for max. Output 69.6

II. RESULTS AND CONCLUSIONS

The wall y^+ is a non-dimensional number similar to local Reynolds number, determining whether the influences in the wall-adjacent cells are laminar or turbulent, hence indicating the part of the turbulent boundary layer that they resolve [8], and it can be seen below as well as Velocity vector plots, can be seen below in the photos of Fig. (a), (b) and (c). These plots give a lot of information about the flow field of the air box of the burner. The recirculation at the corners of air gates and air flow by the area of gates is also obvious from these figures. The air combustion tends to move into the air box by natural draft of the furnace at its level.

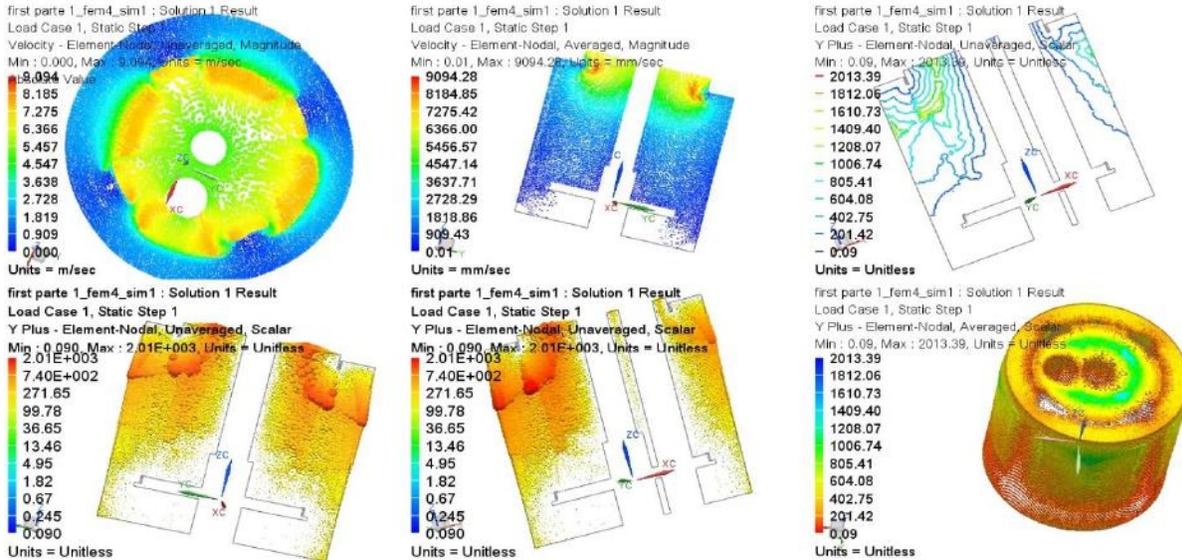


Fig.a: y^+ and velocity vector plots in the field of air box for solution 1.

Solution 1 by iteration method at a steady state-physical relaxation step equals to 0.05 sec, the convergence criteria of RMS residuals is 0.0002, first order is used for advection scheme, distorted fluid elements are 6927elements , and the following table 1 ultimate the data of this solution

Table.1: Ultimates the data of solution 1

Computing Reynold number	Number of element	y+ information on adiabatic faces			Computing cost
		min y^+	max y^+	ave. y^+	
1.8×10^5	3810849	1.59×10^{-1}	3.28×10^1	5.72	52min 24sec

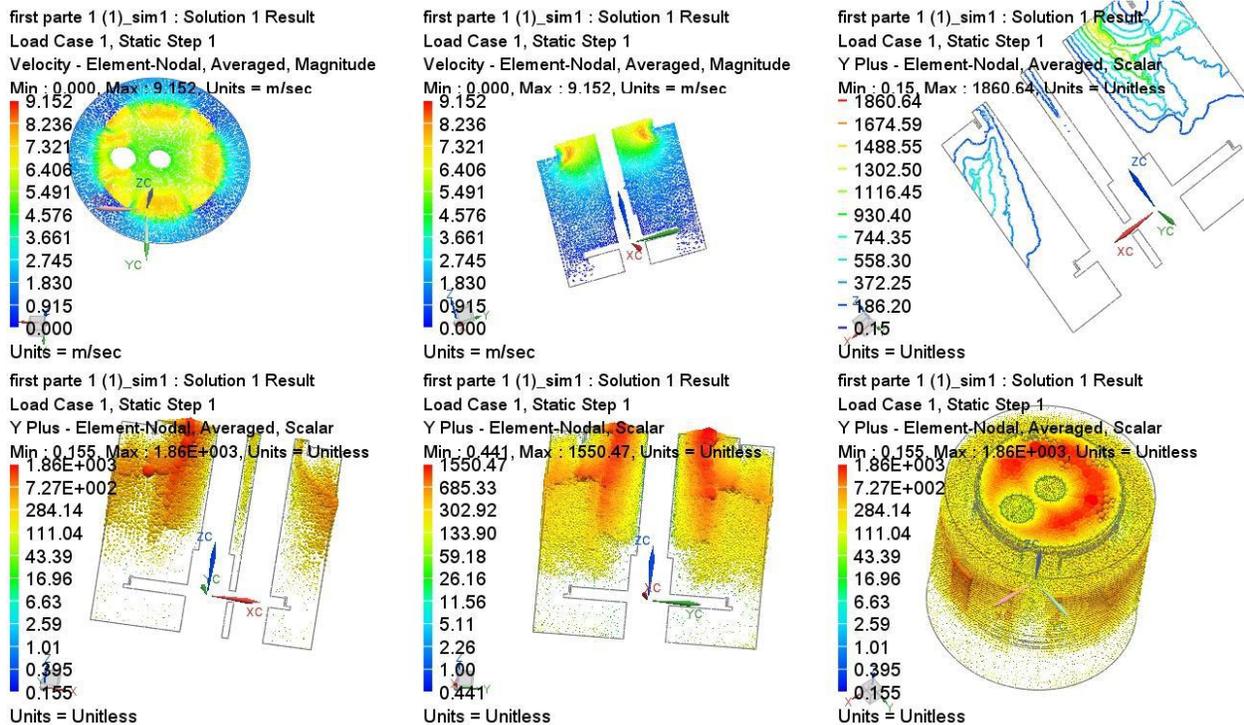


Fig.b: y^+ and velocity vector plots in the field of air box for solution 2.

Solution 2 by iteration method at a steady state-physical relaxation step equals to 0.05 sec, the convergence criteria of RMS residuals is 0.0002, first order is used for advection scheme, distorted fluid elements are 22432 elements , and the following table 2 ultimate the data of this solution

Table.2: Ultimates the data of solution2

Computing Reynold number	Number of element	y+ information on adiabatic faces			Computing cost
		min y+	max y+	ave. y ⁺	
1.8×10 ⁵	1236619	min y+	max y+	ave. y ⁺	12min 20sec
		3.70×10 ⁻¹	4.08×10 ¹	6.26	

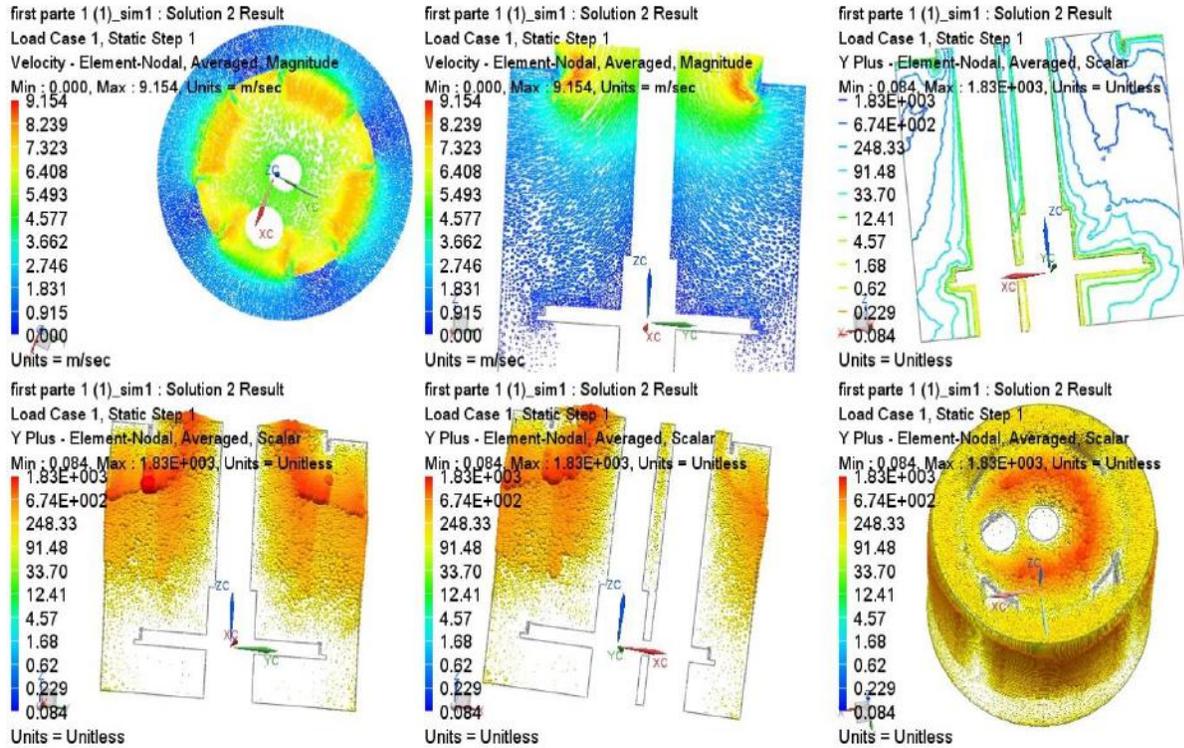


Fig.c: y+ and velocity vector plots in the field of air box for solution 3.

Solution 3 by iteration method at a steady state-physical relaxation step equals to 0.05 sec, the convergence criteria of RMS residuals is 0.0002, first order is used for advection scheme, distorted fluid elements are 331 elements , and the following table 3 ultimate the data of this solution

Table.3: Ultimates the data of solution3

Computing Reynold number	Number of element	y+ information on adiabatic faces			Computing cost
		min y+	max y+	ave. y ⁺	
1.8×10 ⁵	1922514	min y+	max y+	ave. y ⁺	23min59sec
		2.78×10 ⁻¹	3.24×10 ¹	5.44	

The comparison between these three solutions emphasis that solution 2 is preferred. That is because it gives the necessary information by the best computing cost as it is clear from fig. (c) Which shows the same velocity profiles for the three solutions at the same path.

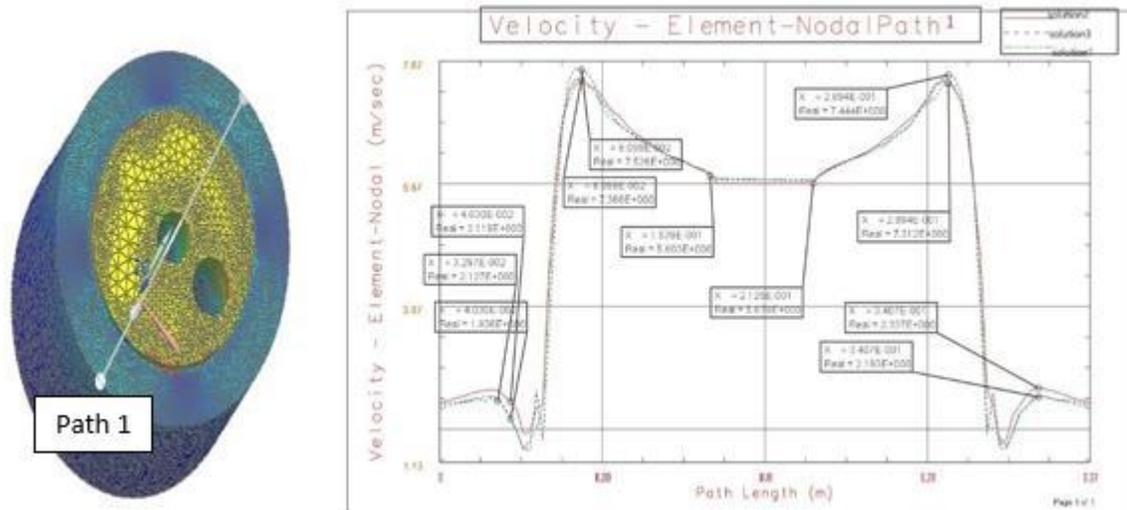


Fig.d: Shows the same velocity profiles for the three solutions at the same path.



Fig.e: Which shows low NOx burner with two air boxes

In fact, by industry these kinds of air boxes are used as tools to adjust the fuel /air ratio in many burners' designs which work on natural draft as it can be seen from fig. (e) Which shows low NOx burner with physical separate between two stages (two air stages and two fuel stages). Here burner depends on two air boxes for natural draft of the combustion air. So in result this model for air box in the low NOx swirl burner which is studied is too useful to get the important information and calibrate another burners performances by using the similarity theory as dynamics and geometry parameters, or to build different air boxes models by the same methodology to get the best from the system as large efficiency and less pollution especially for adjusting NOX emissions.

REFERENCES

- [1] Sasu, P. Kefa, C. Jestin, L. Boilers and Burners Design and Theory .Springer ISBN0-387-98703-7
- [2] Hay, N.Researching On Burner, Construction Equation Of Air Flow Velocity And Defining Appropriate Drayer Wall for Tobacco Dryer In Vietnam. Nong Lam University
- [3] Electronically reproduced by special permission from HYDROCARBON PROCESSING (June 1997). Optimized Fired Heater Operation Saves Money. Copyright © 1997, Gulf Publishing, Texas
- [4] Shihadeh,A.L. Toqan, M.A. Beér, J.M. Lewis P.F, Teare, J.D. Jiménez, J.L. and Barta, L. 1994. LOW NOx EMISSION FROM AERODYNAMICALLY

- STAGED OIL-AIR TURBULENT DIFFUSION FLAMES. ASME FACT-18, Combustion Modeling, Scaling and Air Toxins
- [5] Fiskum,A. 2008. Calculation of NO_x Formation in a Swirl Burner. Department of Energy and Process Engineering, Norwegian University of Science and Technology.
- [6] Ashworth,D.A.2017.An Analytical Model to Predict the Length of Oxygen-Assisted, Swirled, Coal and Biomass Flames. Department of Mechanical Engineering Brigham Young University
- [7] Masoumi,M.E.Izakmehri,z. 2011.Improving Of Refinery Furnaces Efficiency Using Mathematical Modeling. International Journal of modeling and optimization .volume1:pages74-79
- [8] Salim, M.Cheah, S.C.2009. Wall y+ Strategy for Dealing with Wall-bounded Turbulent Flows. Proceedings of the International MultiConference of Engineers and Computer Scientists, Vol II IMECS 2009, March 18 - 20, Hong Kong