International Journal of Advanced Engineering, Management and Science (IJAEMS)
Peer-Reviewed Journal
ISSN: 2454-1311 | Vol-11, Issue-4; Jul-Aug, 2025
Journal Home Page: https://ijaems.com/
DOI: https://dx.doi.org/10.22161/ijaems.114.24

Computation Fluid Dynamics of Aerodynamics Swirl Low NOx Burner in Furnace

Asya Abd Alelah Alabdalah

Department of ITE, SVU University, Syria

Received: 25 Jun 2025; Received in revised form: 21 Jul 2025; Accepted: 26 Jul 2025; Available online: 01 Aug 2025

***Abstract*— *By using a prototype multi –annular swirl burner, the important aerodynamics features of a typical low Nitrogen Oxides NOx of internal staging schematic was studied. So by using swirl means the structure of flame is separated to locally fuel-rich environment prior to burn out. Selecting of the optimal swirl burner design is depending on swirl economy and can estimated using hydraulic resistance parameter. Swirl stabilized flames are lifted a few millimeters above the fuel injector and this liftoff allows air to mix with fuel upstream of the flame base , the positions of the primary and secondary air as well as fuel tube were adjusted to produce a particular flow field which Its name Radially Stratified Flame Core Burner. In result NOx emissions are not more than 120ppmv at 15% excess air and the implement internal staging schematic is effective and promise way to have high furnace efficiency at low NOx emissions*

Keywords*— *computation fluid dynamics, low* NO_x *burner, flame stability, internal recirculation zones, Swirl burn

I. INTRODUCTION

Spiral flow is currently used in modern industrial burners, furnaces, and boilers. This reduces pollution from combustion processes, particularly nitrogen oxides (NOx), both thermal and instantaneous, resulting from fuel.

Adding angular momentum to the flow creates a spiral-shaped core composed of the fuel-air mixture, giving the flame a special shape, making it a vortex. In other words, the molecules in the flame move in a spiral motion along the axis of the burner. This spiral motion has a significant impact on the gas exiting the burner head nozzles, whether in cold flow or hot flow.

This spiral motion is strongly related to the swirl number (S), which is defined as the quotient of the axial flow resulting from the angular momentum divided by the axial flow resulting from the direct momentum. This spiral motion affects the size, shape, and dimensions of the resulting jet or flame. Swirl flow can be classified according to the swirl number (S) as weak (S<0.3), moderate (S<0.6), and strong (S>0.6).

When the torch has a high number of swirl motion, it produces a radial and axial gradient in static pressure. This gradient controls the flame structure, as the reverse pressure gradient on the torch axis produces an internal recirculation zone. This zone makes the flame stable.

The internal recirculation zone is a storehouse of heat and active chemical species that are transferred from it to the blazes (the blazes are a fresh mixture of fuel and air), and also this zone forms an internal recirculation zone for combustion products to the flame, which reduces the local oxygen concentration, lowers the flame temperature and this in turn limits the formation of NOx.

The helical flow characteristics associated with the dimensionless number of helical motion resulting from the use of guide vanes on the burner periphery, and a radial flame regulator at the burner axis on the fuel gas tube have been mathematically simulated using the k-epsilon flow model and the wall function to predict the velocity profiles, and to determine the hydraulic energy loss factor resulting from the use of these flow guides in the burner.

The flame is known as subsonic flow associated with combustion reactions. The angular momentum of the flame structure is added by the guides in addition to the direct momentum of the flow, but the total pressure loss must be minimal to obtain an internal circulation area of suitable dimensions, as well as to ensure good mixing and a stable flame.

Diffused gas flames rely on the use of staged combustion techniques to reduce nitrogen oxides during combustion. They are divided into two categories: The first relies on the physical separation of several stages of the combustion reaction (fuel-rich or fuel-poor stages). Air can be supplied in stages. The theoretical basis for this is to make the initial combustion zone air-poor relative to the fuel, which favors the formation of N2 over NOx. Despite the high flame temperature, the low oxygen concentration limits the thermal formation of NOx [1]. According to Chen (1982), NOx is reduced by 20%–50% when air is used in stages without any negative impact on furnace efficiency [2]. Fuel can also be supplied in stages. This method relies on finding a source of hydrocarbon radicals that reduce nitrogen oxides formed in a previous stage of the combustion reaction to molecular nitrogen [1]. Where 40% to 70% of the fuel is passed to a second stage by a shunt. In the first zone, combustion takes place at a low temperature and in the presence of a high percentage of surplus air, which reduces thermal NOx. In the second zone, the fuel interacts with the air coming out of the first zone, where combustion is complete due to slow mixing, which requires a smaller amount of surplus air compared to rapid mixing, which reduces NOx and improves the thermal efficiency of the furnace [2]. According to (Wented 1991), NOx emissions can be reduced by 50% in this way without causing any negative impact on the efficiency of the furnace [2]. If the flue gases are recirculated without the initial air temperature rising above 1300 K and the combustion reaction is completed with an excess oxygen content of no more than 5%, then, according to Katsuki (1998), fuel efficiency can be increased by 50% at low NOx emissions [2]. However, the disadvantages of using this method of physical separation between the reaction zones include high operating costs, erosion of the heat transfer surfaces, and difficulty in upgrading or modifying the burner [1]. Therefore, the second type of NOx emission control has been resorted to, which is controlling the flame structure using the aerodynamics of the burner. This is done using guide blades that add centrifugal forces to the flow, causing slower mixing of the fuel and air and increasing the residence time of the reactants in the reaction zone. The speed of mixing the fuel with the air determines the flame temperature. However, the optimal burner design in this technology, called internal staging schematic, depends on the amount of energy lost in the flow. When air passes through the torch [1], increasing the intensity of the spiral motion in the torch, expressed by the spiral motion number, is essential for the fuel to ignite and for the flame to be stable. However, the efficiency of the torch must be taken into account when choosing the optimal torch design, by determining the hydraulic loss factor, which expresses the amount of flow energy lost resulting from the spiral motion in the torch [3]. Using mathematical simulation of flow (computing fluid dynamics CFD), the various components of the torch were studied, which control the flame structure with the aim of controlling the aerodynamics of the torch (internal staging schematic).

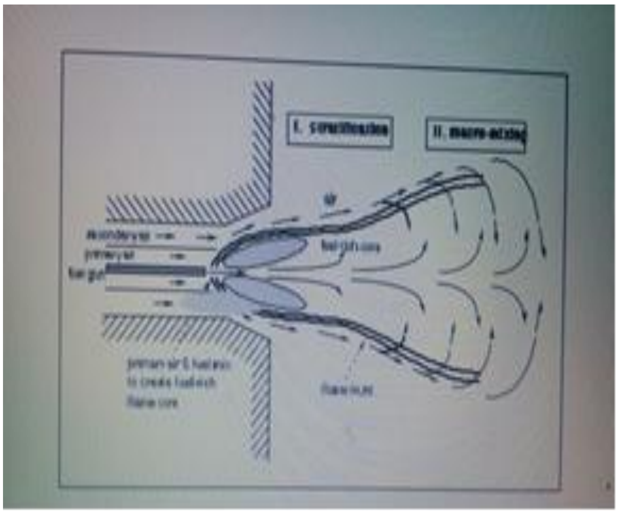

Fig.1: shows the flow field generated by a multi-layered diagonal flame core (internal staging schematic) [1].

All the combustion air is introduced into the torch, but using guide vanes around the torch, the movement and quantity of secondary air are controlled. Using an air deflector in the torch axis on the fuel gas tube, the quantity and movement of the primary air are controlled. That is, by controlling the mixing process in the torch using these guides, we obtain a multi-layered radial flame core (RSFC) Fig.1.

II. RESEARCH OBJECTIVE

The aim of this research is to conduct a simulation of the flow in a torch to determine the internal turbulent flow properties represented by the turbulent kinetic energy (K) and the turbulent dissipation factor (ε).

It also aims to observe various flame regions, such as the internal circulation zone and the wake region downstream of the air deflector. These regions are generally characterized by intense turbulence and high stress and deformation rates resulting from the decay and demolition of the spiral motion in this region. The numerical results of the cold flow model are also used, along with determining the dimensionless spiral motion number (S) for the tangent swirls and the air deflector. This is done to determine their effect on the aerodynamics of the torch, creating fuel-rich and fuel-poor zones in the flame structure and controlling the amount of NOx generated by combustion. We also aim to determine the hydraulic energy loss factor associated with torch efficiency. , which is known as the result of dividing the loss in total pressure (LTP) by the kinetic energy of the flow at the exit from the torch

$$\frac{v_c^2}{2g} \qquad (1)$$

Where: the amount v_c^2 is known as the result of dividing the mass flow by the product of the density of the fluid entering the torch in the maximum flow area within it [3].

III. RESEARCH METHODOLOGY AND RESOURCES

- Determining the dimensionless number of the spiral motion of the flow guides based on the burner's design data.

- Mathematical simulation of cold flow without combustion using the Siemens NX code.

- Determining the amount of nitrogen oxides produced by combustion using the 7E method according to the U.S. Environmental Protection Agency (EPA).

IV. RESULTS AND DISCUSSION:

4.1 Determining the dimensionless swirl number (S) of the secondary and primary air

The detailed diagram of the burner with a multi-layered radial flame core shows that combustion air enters the burner as a result of natural inhalation through an air box consisting of two cylinders: the first, an internal, immovable cylinder, 2 mm thick and 258 mm in diameter, and the second, an external, movable cylinder, 2 mm thick and 264 mm in diameter. These two cylinders have identical windows, and the movement of the external cylinder relative to the internal one changes the area through which the air flows.

The combustion air enters through the air box windows by natural inspiration, occupying the annular space between the concentric channels. The combustion air is initially divided into two parts using tangential air swirlers, which are longitudinal blades 1 mm thick and 103 x 65 mm in dimensions. These blades are positioned around a cylinder concentric with the burner, 1 mm thick, 251 mm in diameter, and 90 x 788 mm in flat plate dimensions. The first section represents the secondary air, which gains spiral motion using the side air blades and constitutes 70% of the total combustion air volume. The second section represents the primary air, which constitutes 30% of the total combustion air volume. It enters through the central annular space and finally exits through the 120 mm diameter, 2 mm thick air deflector. Its blades are angled 170° x 170° from the radial direction. The flame deflector is positioned on the fuel gas tube, which is coaxial to the burner. The fuel gas exits the 38 mm diameter burner head through 12 holes, each angled 30° from the burner axis, to mix with the air. Together, they exit through a gradually narrowing cone, from which the mixture is distributed throughout the furnace after passing through the primary combustor, which takes the shape of a gradually expanding cone.

Determining the spiral motion number of the guides on the torch circumference

Is derived from expressing the spiral motion number mathematically, which is defined as the ratio of the angular momentum of the flow to the direct momentum of the flow [5]:

$$S = \frac{G_{ang}}{G_x . R_b} = \frac{\int_0^{\infty} \rho . U . W . r^2 dr}{R_b \int_0^{\infty} \left(U^2 - \frac{1}{2} W^2\right) r . dr} \quad (2)$$

G_{ang} the direct flow due to the angular momentum

G_x The direct flow due to the direct momentum

R_b The radius of the cylindrical torch

ρ The density of the working fluid, which we assume to be air in the simulation for non-combustion flow

U The average direct velocity in the plane whose area is $2\pi r dr$

w The average angular velocity in the plane whose area is $2\pi r dr$

r The distance from the torch axis.

Since the necessary data about the speed is practically unavailable, in engineering applications we resort to calculating the dimensionless number of the spiral motion using the geometric dimensions of the structure under study, where we express the relationship between w and U using the angle of inclination of the blades from the direct current lines α according to the relationship

$$\tan \alpha = \frac{U}{W} \quad (3)$$

Where $\alpha = 29.5°$ as a result we find this one

$$G_{ang} = 2\pi\rho \int_{R_c}^{R_b} U_a (U_a \tan\alpha) r^2 dr = 2\pi\rho U_a^2 \tan\alpha \left(\frac{R_b^3 - R_c^3}{3}\right) \quad (4)$$

U_a The average value of the direct velocity in the annulus occupied by the secondary air and including the lateral guides on the burner periphery.

R_c The characteristic dimension of the ring occupied by primary air.

R_b The distinctive dimension of the torch

The direct flow due to the direct momentum of the flow in both the primary and secondary airflow regions is calculated from the relationship

$$G_x = 2\pi\rho \left[\int_{R_c}^{R_b} U_a^2 r dr + \int_0^{R_c} U_c^2 r dr\right] = \pi\left[\rho U_a^2 (R_b^2 - R_c^2) + \rho U_c^2 R_c^2\right] \quad (5)$$

Where: U_c the average direct velocity in the ring occupied by the primary air, and R_c the characteristic dimension of the ring occupied by the primary air Thus, the dimensionless number of the helical motion of the guides around the burner circumference is given by the relation

$$S = \frac{2}{3} \tan\alpha \frac{1 - R^3}{1 - R^2 + \left[m^2 R^2 \left(\frac{1}{R^2} - 1\right)\right]^2} \quad (6)$$

R The ratio of the two characteristic dimensions of the space occupied by the primary air to the space occupied by the working fluid in the burner, expressed as

$$R = \frac{R_c}{R_b} \quad (7)$$

m The ratio of the mass flow of the primary air to the secondary air which has the same value as the result of dividing the effective area of the primary air flow at the center of the burner by the effective area of the secondary air flow through the ring occupied by the guides around the burner.

$$S = \frac{2}{3} (\tan 30°) \frac{1 - (0.29)^3}{1 - (0.29)^2 + \left[(0.42)^2 (0.29)^2 \left(\frac{1}{(0.29)^2} - 1\right)\right]^2} \quad (7) \quad S = 0.46$$

Hence, the dimensionless number of the spiral motion of the secondary air has an average value.

Determining the spiral motion number of a flame deflector

In 1996, Shee derived an empirical relationship to determine the dimensionless spiral motion number of flow deflectors such as a flame deflector (a radial-type swirl generator). This spiral motion number (S) strongly depends on both the Reynolds number and the angle of inclination of the deflectors from the radial vector φ , according to the relationship [6]:

$$S = C_1 (Re) . \sigma(\varphi) \quad (8)$$

Where: the value C_1 is a function of the value of the Reynolds number according to the relation

$$C_1 = 0.28[1 - \mathrm{sech}(0.026Re^{0.67})] \quad (9)$$

Hence, when the Reynolds number of the flow is greater than 2200 and the flow is turbulent, the value becomes constant C_1 and equal to 0.28.

The value depends on the geometric structure of the flame σ regulator and is given by the following relationship:

$$\sigma = \frac{1}{1-\xi}\left[\frac{\tan\varphi}{1+\tan\varphi\tan(\pi/Z)}\right] \quad (10)$$

Where ξ the blockage factor resulting from the thickness of the flow guides is given by the relationship

$$\xi = ZT/2\pi R_1 \quad (11)$$

Where Z the number of flame directors, T Flame guide thickness and R_1 the exit radius of the flame regulator and by application we find

$$S = 0.28X\frac{1}{1-(10X2/2\pi X40)}\left[\frac{\tan 10^\circ}{1+\tan 10^\circ\tan(\pi/10)}\right] = 0.05 \quad (12)$$

In this burner, using flow guides, two zones for combustion reactions are created. The first is the zone characterized by the presence of slip layers, where the mixing process between fuel and air is slower, controlled by the density gradient between the slip layers. This zone is relatively cold, as the spiral-moving combustion air surrounds the hot, fuel-rich flame core. The fuel-rich flame core is created by injecting primary air (30% of the combustion air) into the area surrounding the burner head, which is centered on the burner. Due to the centrifugal forces resulting from the spiral motion of the combustion air surrounding the hot flame core, the mixing process resulting from the turbulent mixing between the hot flame core and the surrounding spiral-moving combustion air is dampened. This zone is called the density stratified zone and may extend for a distance equivalent to five times the diameter of the burner in the furnace [1]. The second zone is characterized by microscopic mixing of the combustion products from the first zone with the secondary combustion air, which constitutes 70% of the combustion air. This completes the combustion reaction at a relatively low temperature and with an optimal excess air ratio, which limits the formation of nitrogen oxides. Therefore, the amount of NOx produced by combustion is related to the S-number of the spiral motion generated by the flow guides. To observe how nitrogen oxide emissions are controlled based on the aerodynamics of the turbulent, post-mixed flame, the flow was simulated in the region near the burner head, where all the combustion and mixing reactions take place. Using mathematical simulation of fluid flow (computing fluid dynamics) using the Siemens NX code, we provide a set of numerical modeling results that demonstrate the changes in the macroscopic properties of the fluid at every point in the space within which the fluid moves in the burner mounted on the furnace.

4.2 *The Physical Model of the swirl -Driven Burner*

A mathematical model of cold isothermal flow was developed. To ensure that the numerical results of the flow model match those of the actual burner, a three-dimensional model of the swirl-drive burner was created, identical in geometric structure to the original burner.

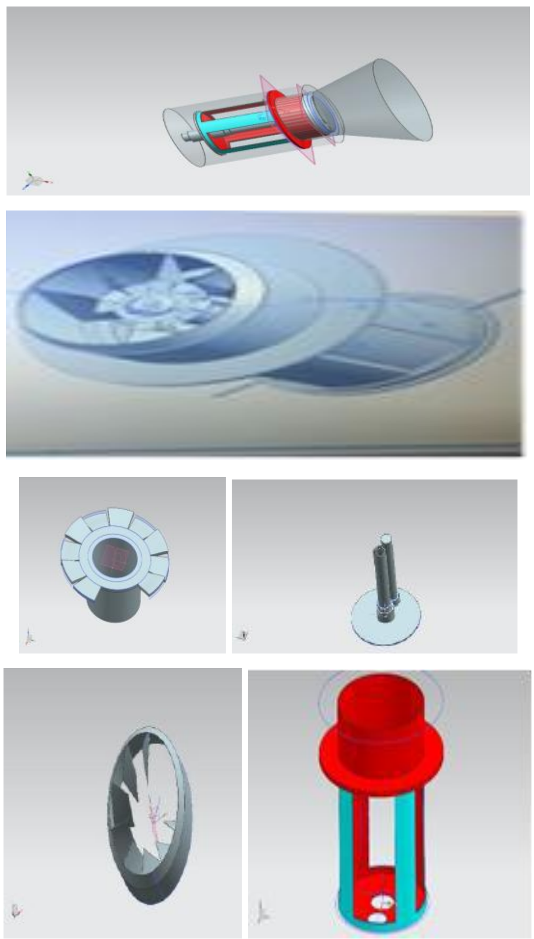

Fig.2: shows the three-dimensional model of the studied torch and its components

The burner under study operates on natural inhalation, with combustion air entering through the fully open air box gates at maximum fuel consumption. Assuming that air is the working fluid and that it is incompressible, and that the relative velocity of the working fluid is zero when in contact with the solid surface, which is assumed to be completely smooth, and taking into account the buoyancy forces, then applying the transition equations for the conservation of mass, momentum, and energy according to the rules for using average values according to Reynolds analysis, we obtain a set of transition equations called the Navier-Stokes equations according to Reynolds analysis, which express the effect of turbulence on the average macroscopic properties at the finite element boundaries of the studied system. The model was then performed in a steady-state frame. Fig. 2 shows the three-dimensional model of the studied torch and its two components: the moving swirl torch, the air box, the flame regulator, and the side guides.

4.2 .1 Flow Model and Determination of the Reynolds Number for Wall Friction

The flow model known as K-Epsilon was used, as it is recommended in industries where most flows occur at relatively high speeds.

Given the diverse geometric structure of the system under study, the value of the Reynolds number for wall friction (Y+) was adjusted according to the wall function to a value less than 5. This avoids entering the buffer region, where no mathematical model or wall function predicts flow in this region.

This model includes two partial differential equations: one expresses the change in kinetic energy (K) of the flow and ϵ the other expresses the turbulent dissipation rate.

Turbulent or Eddy Viscosity μ_T is expressed as a function of both ϵ of K and

This flow model includes five empirical constants.

4.2 .2 Generating a Finite Element Mesh

This stage is considered the most difficult in the simulation, because designing a finite element mesh in the studied geometric structure for the fluid-occupied space requires testing multiple mesh sizes to match the desired degree of accuracy for the approximate solution. The desired adjustments were made by reducing the element density on the axes of the gas flow nozzles and the mixing zone with air in the gradually decreasing cone. Therefore, the solution must be independent of the chosen mesh size and at a reasonable computational cost for the proposed mathematical model. Boundary Conditions of the Model

This burner was designed to generate the required amount of heat over a short distance and within its operational range (turn-down ratio). Therefore, the simulation was performed at maximum fuel consumption and assuming the combustion air intake windows were fully open. The burner burns a gaseous fuel that does not contain any nitrogen compounds, but the hydrogen gas content may reach up to 50%.

The maximum fuel consumption is 69.6 standard cubic meters per hour, where the excess air content is 0.5%. 3%. Since the total pressure drop at the boundaries of a system is less than 3000 Pa, we can consider the average pressure applied to one of its boundaries to be equal to the average atmospheric pressure [4].

This is the case for our spiral-driven torch. Therefore, this is used as one of the boundary conditions applied in the mathematical simulation of flow in the torch. The general design condition was studied without correcting for altitude above sea level.

4.2. 3 The numerical results of the mathematical model and their reliability

Ensuring their consistency with reality by utilizing available similar experiments and studies. Consistency in the results with the study presented by the American Standards Board for Mechanical Engineers in 1994 was observed.

(American Society of Mechanical Engineers ASME)

4.2. 3.1 Numerical Results of Mathematical Simulation of Air flow as a Working Fluid into the Torch

The most important feature accompanying the spiral flow in a torch with a laminated radial flame core is the presence of an internal circulation region resulting from the velocity dissipation of the spiral motion in the direct direction of the flow, shown in Fig.3: This region helps stabilize the flame by providing a hot flow region in which the combustion products are recycled and re-enter the slip layers.

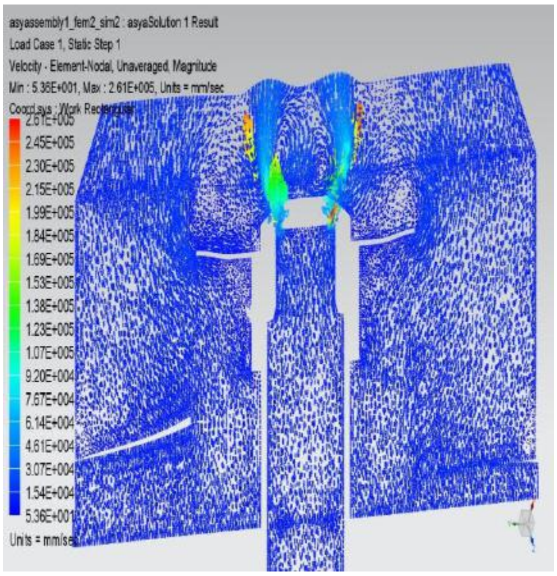

Fig.3: The inner circulation region in the radial laminar flame core.

The boundaries of this region are drawn in the radial plane at points where the mass flow in the direct and reverse directions is equal at any point within the vicinity of the torch axis. In other words, in this region, the flame velocity is equal to the flow velocity, and the reactants are well mixed with the combustion air due to the high turbulence resulting from the high deformation rates. In each axial plate, when the axial velocity is zero at any point, this point falls within the internal circulation region, which results from a reverse pressure gradient along the torch axis. It is observed that ring vortices form within the internal circulation region resulting from the velocity dissipation. The angle in the direct direction of flow, which results from the static pressure gradient in the radial plane as the working fluid exits the nozzles of the burner head, is due to the geometric structure of the studied part of this torch. It is noted that the burner head plays a role similar to a flow-guiding body (BLUFF-BODY), as the nozzles are distributed around the periphery of the burner head at an angle of inclination equal to 30° relative to the direct current lines. Also, the guides, which have a high dimensionless number of spiral motion when S>0.6, cause a radial and axial static pressure gradient sufficient to generate a central toroidal recirculation zone. No central annular circulation region can be observed when S<0.4. Therefore, when S<0.4, the presence of annular circulation regions is due to the geometric structure of the system, which creates a flame with a low swirl motion number, such as the presence of a flow-guiding body.

4.2. 3.2 The relationship between NOx produced by combustion and the aerodynamics of the burner

The amount of NOx produced by dry combustion at 3% excess oxygen was 120 parts per million. This is attributed to the thermal fixation of molecular nitrogen in the air, as the fuel does not contain nitrogen compounds. This low amount of thermal NOx is due to the presence of lateral air guides, which allow the combustion reactions to occur between the bulk of the combustion air and the reaction products of the fuel-rich primary combustion zone (hot radial internal circulation zone + cold radial slip layers).

Despite the high temperature in the fuel-rich zone, this does not stimulate the formation of thermal NOx due to the lack of oxygen. Furthermore, hydrocarbon radicals are formed, which convert nitrogen oxides to molecular nitrogen. Centrifugal forces also prevent combustion air from entering the laminar radial flame core, which is formed by the use of a low-swirl number air deflector. This reduces the intensity of the internal recirculation zone and allows a greater amount of gaseous fuel to enter it. This allows the combustion reaction to occur at a longer residence time, higher temperature, and less oxygen, making the nitrogen oxides' reduction mechanism chemically preferable to their formation in this zone. Meanwhile, in the second stage of combustion, a slow microscopic mixing process takes place between the air and the reaction products of the first zone at a low flame temperature and in the presence of an optimal proportion of excess air, limiting the formation of thermal NOx. The secondary airflow has a greater impact on the amount of nitrogen oxides produced because the greater the angular momentum added to it, the longer and slower the mixing time becomes, increasing turbulence and increasing the efficiency of the combustion process at lower pollutant values than thermal NOx. Note Fig. 4.1: and fig.4.2: which illustrate the basic characteristics of turbulence: one represents the change in the kinetic energy of the turbulence, and the second represents the rate of dissipation of the turbulence.

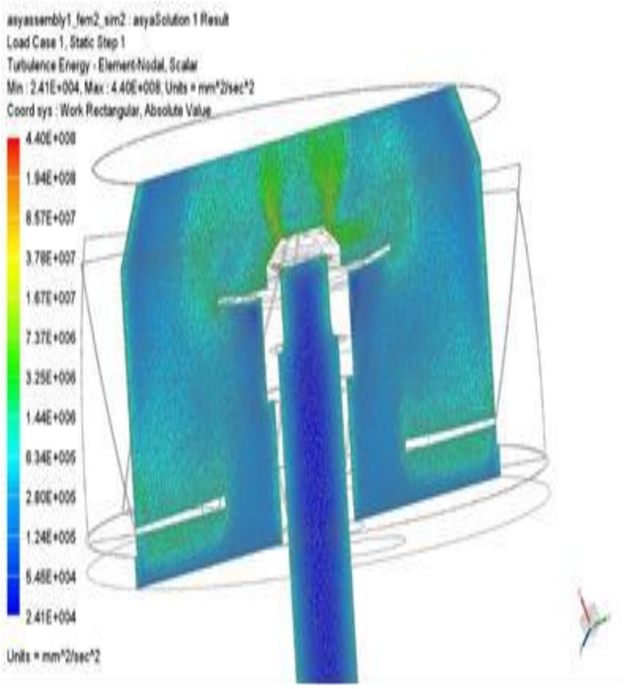

Fig. 4.1: illustrates turbulence.

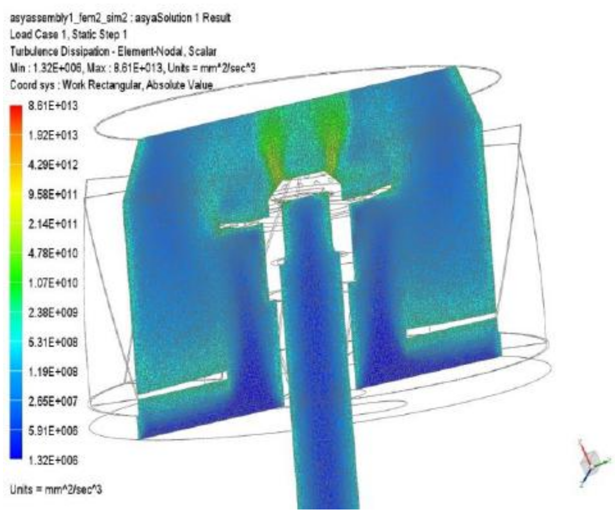

Fig. 4.2: illustrates turbulence.

Weber and Dugue have established a simple relationship to express the amount of change in the value of the dimensionless number of the spiral motion in both the cold and hot flow cases, which is:

$$S_{hot} = S_{cold} \left(\frac{\rho_{hot}}{\rho_{cold}} \right)^{B} \quad (13)$$

The dimensionless number of the spiral S_{hot} motion of the hot flow

The dimensionless number of the helical S_{cold} motion of the cold flow

ρ_{hot} Density when combustion is present

ρ_{cold} Density without combustion

B constant with a value ranging from 0.5 to 1

This change in the value of the dimensionless vortex number in the flow occurs in the presence of combustion reactions due to the large gradient in temperature and density, which alters the size and intensity of the internal recirculation zone due to the greater direct momentum added to the

flow compared to the case of cold flow. The role of the angular momentum added to the flow is noticeable, as it does not change significantly in the presence of combustion. There is no doubt that the use of flow guides causes a specific pressure loss. The hydraulic pressure loss factor expresses the fluid's resistance to flow as it passes through these guides. However, when designing a torch, it is essential to minimize the hydraulic pressure loss factor to achieve an efficient recirculation zone, a stable flame, and good mixing. Therefore, it is usually preferable not to increase the angle of inclination of the guides, as the blockage factor increases with increasing angle, causing an increase in the hydraulic pressure loss factor. Fig. 5: shows the change in static pressure within the torch under study and fig.6 shows the Y+ value in the model..

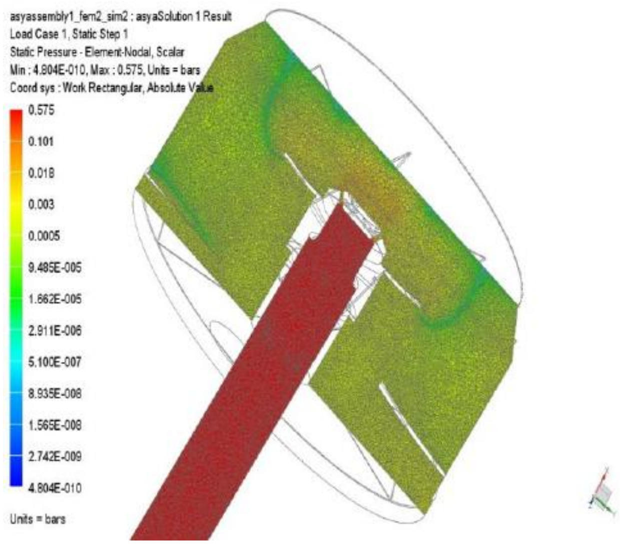

Fig.5 shows the change in static pressure within the torch under study and the Y+ value in the model

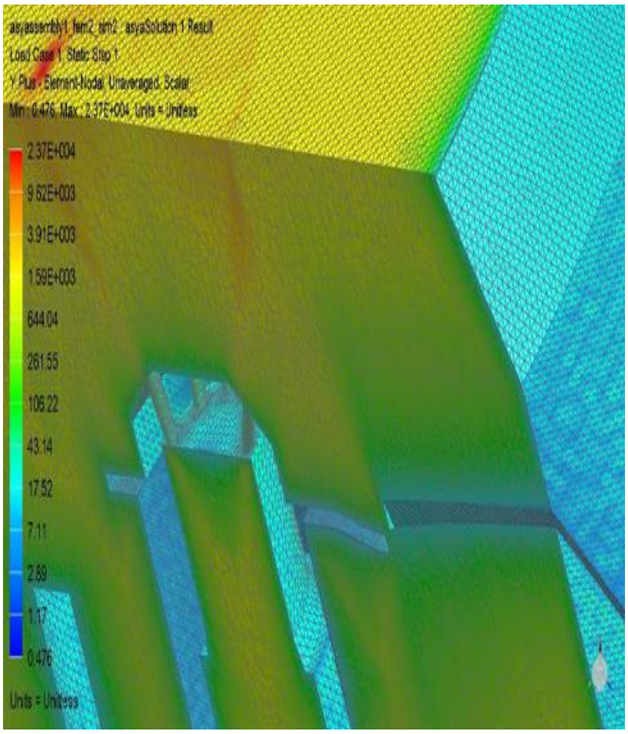

Fig.6: shows the Y+ value in the mode

V. CONCLUSION

From the mathematical simulation of the flow in a spiral burner, we conclude that there are two aspects of momentum that determine the flame structure: angular momentum arising from the geometric structure of the burner and the presence of flow guides, in addition to direct momentum.

This slows the fuel-air mixing process and lengthens the reactant residence time in the multi-structured, layered radial flame core, which includes fuel-rich and fuel-poor regions. The overall combustion reactions are influenced by the quantity and movement of the combustion air available in each region, making the application of dynamic separation to the flame structure a promising and effective method in furnaces for achieving the best combustion efficiency and the lowest nitrogen oxide pollutant levels.

REFERENCES

[1] A.L. 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.

[2] Dong, W.2000. Design of advanced industrial furnaces using numerical modeling method.Royal institute technology.TRIT A-MET062

[3] NIZAM, A. 2008 .SWIRL FLOW IN COMBUSTION CHAMBERS. Faculty of Mechanical Engineering Universiti Teknologi Malaysia

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

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

[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 978-0-13127489-3

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

