

Aerodynamically Study by Mathematical model Of Vertical Low NO_x Swirl Burner in Natural Draft Furnace

Hassan Al-haj Ibrahim, Ethat Aloosh, Asya Alabdalah

Al-Baath University, Homs, Syria

Abstract— The flow field and mixing characters of high efficiency vortices in one type of Radially Stratified Flame Core Burner RSFC where analysed through Cold computing fluid dynamics model CFD in accordance with Mathematical model for flame visualization. The model gives good results and clears the important aerodynamics features of a typical low Nitrogen Oxides NO_x of internal staging schematic. The model can be used to evaluate this design performance or to get the necessary information for development its efficiency in accordance with the global demand to clean energy with high efficiency and low NO_x emissions. The mixing of this burner dominated by coherent structures, so it is outperform the molecular diffusivity and the length which fluids will mix is independent of the fluid velocity and can determined only from Taylor macroscale which related by the physical size of the fluid domain or in better understanding with the internal recirculating zone dimensions because it is form a region for axial mixing for species and smear out their residence time distribution.

Keywords— coherent structure, gaseous mixing characters, internal staging schematic flame, cold flow flame model, Taylor macroscale, internal recirculation zone, penetration theory.

I. INTRODUCTION

In the present work, it appears that a statistical analysis of the mixture quality has been sufficient to obtain information which can be used in a relatively fuel/ air flow adjustment scheme to minimize the NO_x emissions. Swirl burners in industrial furnaces utilize powerful vortices to increase the speed of collision between axial and tangential flows, thus speeding up the time for mixing fuel and air and also extending the residence time. Hence, as suggested by Coats (Coats et al. 1996) the mixing of systems dominated by coherent structures cannot be modelled accurately with gradient based techniques. As in most technical applications, the stirring effect of large structures will

namely outperform the molecular diffusivity. The problem can become even more complex in the case of counter gradient diffusion processes which occur during combustion (Veynante et al. 1996).

II. METHODOLOGY

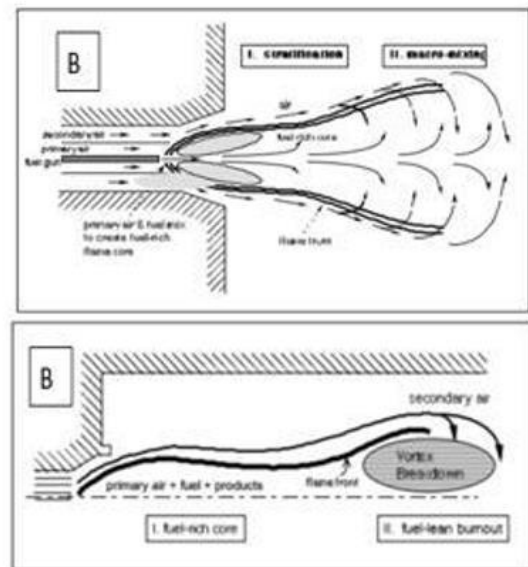


Fig.1: typical low NO_x RSFC flow field with internal staging schematic [3]

The mathematical model for fluid dynamics field shows one type of Radially Stratified Flame Core Burner RSFC which clears the important aerodynamics features of a typical low Nitrogen Oxides NO_x of internal staging schematic see fig(1).The burner used in this study consists of two concentric tube with a tangential and radial swirler placed in annular arrangement .So an initial fuel –rich flame core is created by the mixing process between the central fuel jets with the primary air which is a portion of air combustion equals to 30% tends to move around the air deflector as primary air , and 70% of the air combustion enters through

a radially displaced annulus as secondary air. The fuel jet penetrates through the internal recirculation zone as seen in fig. (2) which clear that there are two mixing regions for the jet in crossflow, the first is the jet momentum dominated near field (MDNF), and the second is the main flow momentum dominated far field (MDFF) as it was mentioned by lee and Chu et al 2003 who also introduced the length scale l_{mv} to separate the two zones:

$$l_{mv} = \sqrt{\rho_j M_{vo} / \rho_a u_a}$$

Where $M_{vo} = u_j^2 \sin(\theta_j) \pi d_j^2 / 4$ is the vertical momentum flux of the jet at $y = 0$, u_a the bulk velocity of

the main flow, ρ_j and ρ_a are the densities in the case of two fluids. For the momentum dominated near field the jet penetration y verifies $y/l_{mv} \leq 1$, and for the momentum-dominated far field, $y/l_{mv} > 1$. In a similar way, they define an integral length scale for a jet in co-flow flow,

$$l_m^* = \sqrt{\rho_j M_{jo} / \rho_a u_a}$$

Where $M_{jo} = u_j(u_j - u_a) \sin(\theta_j) \pi d_j^2 / 4$ and they define for the momentum dominated near field (MDNF) the jet penetration y verifies $x/l_m^* \leq 10$ is valid.

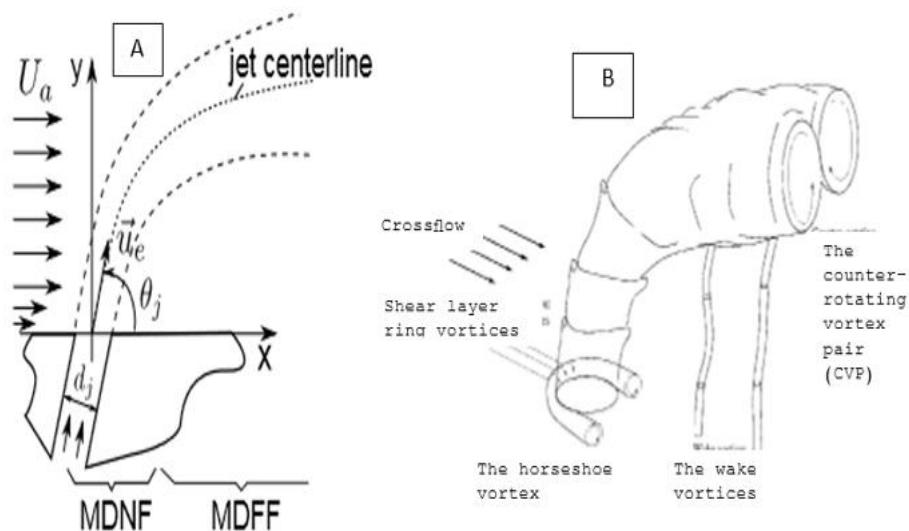
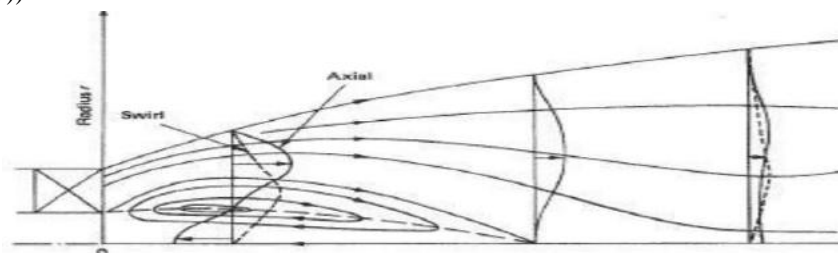


Fig.2: mixing regions for the jet A [4] and structures which are generated by a jet in crossflow B [4], [5].

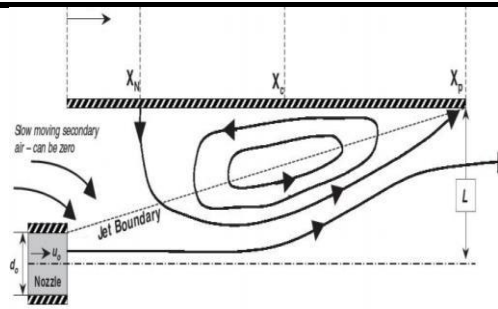
Shortage of detailed and accurate experimental data on fuel-air mixing in furnaces is due to the difficulty and complexity of measurements in flames.

The swirl of fluids in the low NO_x burner forces more mixing process. So its effects is thank to create turbulent operating conditions for mixing fuel /air by inducing high efficiency vortices in addition to benefit from the blades which twist the fluid away or toward in helical movement depends on its position when it flow axially toward the burner

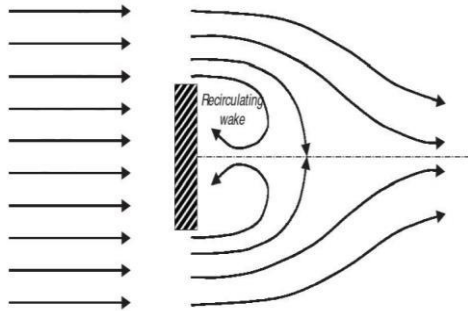
For sufficient turbulence and mixing, the oxidizer try to enter into the fuel rich region by laminar or turbulent diffusion at the boundary of the fuel and oxidizer jet or it take another way in the field of internal recirculation zone which recycle flue gas back into the flame (see fig.(3))



Recirculation caused by wake behind a bluff body



Recirculation caused by wake behind a bluff body



Axial confined jet and secondary recirculation

Fig.3: In internal FGR, the fuel gas is recycled into the flame zone due to burner aerodynamics to reduce NO_x [6].

In all combustion systems, according to Zeldovich mechanism it is usual to have little quantities of thermal NO_x emission at flame temperature which is below 1450 °C, and that is because the oxygen atoms will not be available from their molecules to start thermal NO_x formation so the combustor parameters like the residence time, flame temperature, pressure and heat losses which are relevant in the generation of NO_x must be adjusted for the global demand to clean energy with high efficiency and low emissions.

A minimization of NO_x emissions in combustors is achieved when the fuel/air mixing quality is optimized, as homogeneous as possible but this is not true for mixtures at stoichiometric conditions [7](see fig.(4))

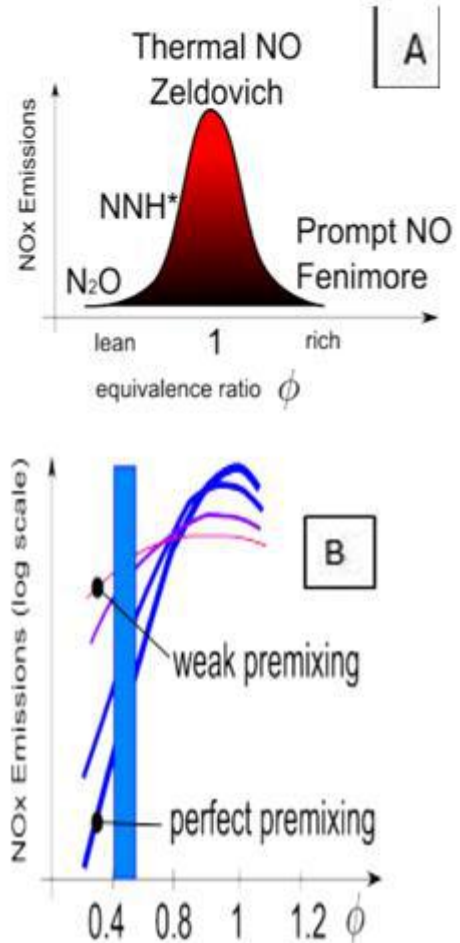


Fig. 4: impact of fuel/air mixing quality on the NO_x emissions for different fuel/air equivalence ratios ϕ , from Joos B [8] and Dominant NO_x formation mechanisms shows Five chemical mechanisms describe how NO_x are produced in combustors A [8]: *thermal NO_x (or Zeldovich) *Fenimore Mechanism (also called prompt NO_x) *Fuel NO by N₂O routes *NNH Radical

III. MATHEMATICAL FLAME VISUALIZATION MODEL

In this combustor the presence of the internal recirculation zone plays important role in flame stabilization and introducing rotation in the stream by the vanes which are mounted on a central gaseous fuel tube by the air deflector geometry has a big role in mixing process in the middle of burner, this effect is done by controlling the aerodynamics of the internal recirculating zone to mix fresh fuel with the primary combustion air. But the vanes which occupy the

space in the annular around it by a vane type - swirler are thank for playing a mazing effects on recycle the secondary combustion air into reaction zone .(see fig. (5))Which shows the geometry of the burner. A swirl burner have convergent-divergent air exit shaped section (also called quaral or burner throat section) , the air from the swirler is contracted first in the air exit and then it spreads into the furnace by using the expansion cone .

Confusor, which for similarity conditions the all were done in line with fact.

The volumetric stoichiometric fuel/air mixture ratio is computed as [9]

$$\frac{1}{\lambda} = \frac{\pi d_F^2 u_F}{4 \pi b l u_c}$$

Where u_F the fuel velocity from the nozzle with diameter d_F and the air mixing with fuel stream is characteristic by mixing velocity u_c within a cylindrical volume its length equal the flame length l and its wide equal b which is the boundary where the average axial velocity is zero.

The characteristic mixing velocity u_c is the sum of two items, one is due to recirculation and another item is due to shearing between fuel and air stream. So it can be computed as [9]:

$$u_c = \pi . b . l . u_{RZ} + constant . d_F . l . \pi . (u_F - u_A)$$

Where u_{RZ} the velocity of characteristics recirculation zone and is computed from the relation [9]:

$$u_{RZ} = \int_0^b -u_z . 2\pi . r . dr / \pi b^2$$

Where u_z the mean axial velocity and b is located where the flow is perpendicular to the jet axis. So by replace the u_c by its two items in the relation which computed the mixture ratio, it can be obtained the following one [9]

$$\frac{l}{d_A} = \frac{c_1 \left(\frac{m_F^0}{m_A^0} \right)}{\left(\frac{u_{RZ} \cdot b}{u_A \cdot d_A} + \frac{(u_A - u_F)}{u_A} \cdot \frac{d_F}{d_A} \cdot c_2 \right)}$$

Where subscript F, A denotes values for fuel and air stream, m , d , u , l are mass flow, diameter, velocity respectively and l flame length while c_1 , c_2 are experimental constants.

So from the last equation it was seen that burners' geometry, viscous shearing, swirl and kind of fuel all of them affect the flame features and their effects can be seen by applying computing fluid dynamics model to notice the flame structure 's effects of on NO_x emissions.

So 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 , $C_5- 1$, $H_2O-1.5$, $H_2S_{max}.5mg/Nm^3$
- * Net calorific value: 11000kcal/kg

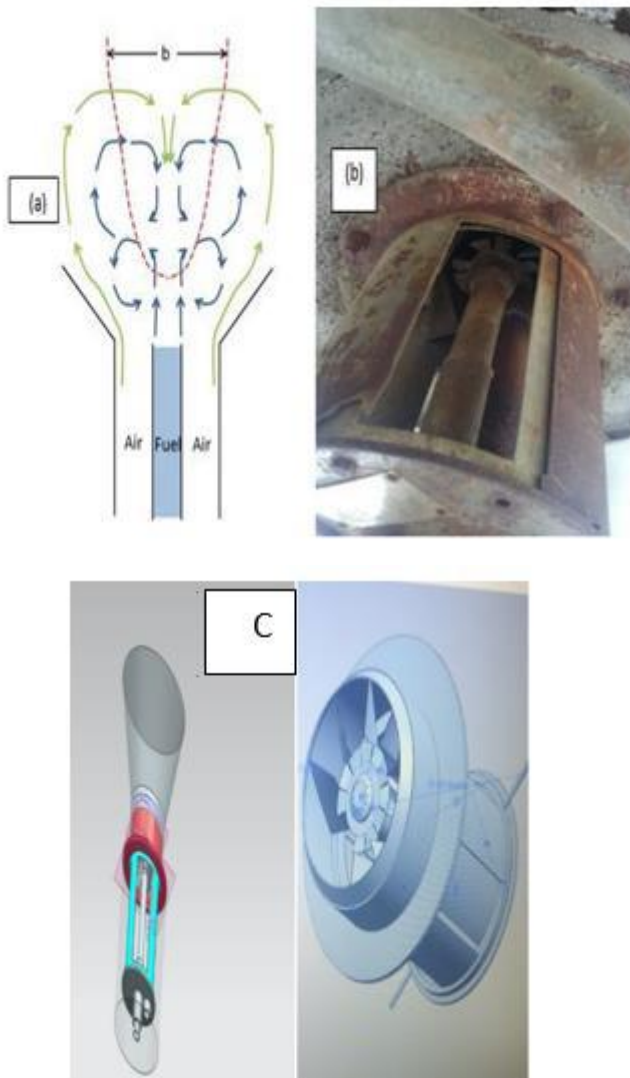


Fig.5: Shows a schematic diagram of the air and fuel flow velocities of a swirled jet burner (a) [9], burner geometry (b), and (c) a proto type of swirl burner, its components and

* 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

IV. COLD COMPUTING FLUID DYNAMICS MODEL CFD

- The interface Γ is between two fluids (air /fuel) but it is movable and unknown. So, it must determine by the solution of CFD model (see fig. (6)).

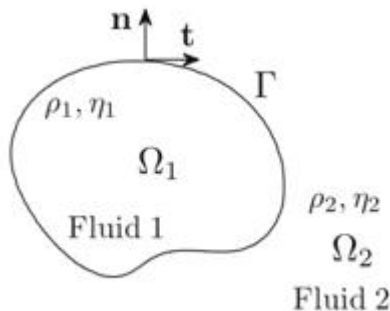


Fig.6: Schematic of a two-fluid domain [10]

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

The continuity equation:

$$\frac{\partial \rho}{\partial \tau} + \nabla \cdot (\rho \vec{v}) = 0$$

Where ρ is mass density for each fluid (fuel/air), τ is the time and $\vec{v} = (u, v, w)$ is the three velocity field, and by writing the local component yields by using the mass fraction w for each fluid [11]

$$\rho \left(\frac{\partial w}{\partial t} + \vec{v} \cdot \nabla w \right) = -\nabla \cdot \vec{J}$$

Where \vec{J} is the Fickian diffusive mass flux of fluid 1 in corresponding fluid:

$$\vec{J} = -\rho D \cdot \nabla w$$

As the mixing process is assumed isothermal, the density is a function of pressure p and composition. so using the derivatives of density in continuity equation yields [11]:

$$\frac{\partial \rho}{\partial p} \left(\frac{\partial p}{\partial t} + \vec{v} \cdot \nabla p \right) + \frac{\partial \rho}{\partial w} \left(\frac{\partial w}{\partial t} + \vec{v} \cdot \nabla w \right) + \rho \cdot \nabla \cdot \vec{v} = 0$$

Neglecting the spatial variation of p on the density and rearranging the last equation by using the equation of spatial variation of w for the fluid 1 yields [11]

$$\nabla \cdot \vec{v} = \frac{-1}{\rho} \cdot \frac{\partial \rho}{\partial p} \cdot \frac{\partial p}{\partial t} + \frac{1}{\rho^2} \cdot \frac{\partial \rho}{\partial w} \nabla \cdot \vec{J}$$

This equation indicates that compressibility and non-ideality (density variations with compositions) can both be source of divergence of the velocity field [11]

For **Computing cold fluid dynamics model some assumptions are used as** assume that the densities of the fluids are uniform at each side of the interface Γ so that the

Atwood number At is very small:

$$At = (\rho_1 - \rho_2) / (\rho_1 + \rho_2) \ll 1$$

So the Boussinesq approximation can be made in this case [10], and it retains density variations in the gravitational term (giving buoyancy forces) but disregards them in the inertial term; i.e. in the vertical momentum equation:

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

Where: T is the actual temperature applicable locally. T_∞ is the undisturbed constant temperature and by using the air as working incompressible perfect gas it is possible to write B_T : is the coefficient of the thermal expansion of fluid 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

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

Where K is the turbulent kinetic energy, δ_{ij} is Kronecker

Delta Function, Then the letters $i, j, k \dots$ 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.

4-1 wall y+ strategy for dealing with wall-bounded - turbulent flows:

It was observed that viscous sublayer turbulence model were in better accord with results while the model results obtained from the wall function bridge the gap with the walls were not suitable choice for the internal recirculation field simulation and this is related by the effect of positive and adverse pressure gradient in the direction of flow [12].

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 burner geometry an enhanced wall function is used.

4-2 Boundary conditions: Boundary conditions are used according to the need of the model. The inlet velocities are used similar to the max fuel gas flow conditions in order to have a comparison. General correlations are used to estimate the turbulence boundary conditions which are specified by estimating the turbulence intensity and length scale ($I = 0.16Re^{-1/8}$) and ($l = 0.07L$).

V. RESULTS AND DISCUSSION

Velocity vector plots can be seen below in Fig. 8 (a), (b) and (c). These plots give an idea of flow separation at region which surrounds the burner tip in the middle of burner. The recirculation at the wake region of air deflector is also obvious from them, the iso surface plots of velocity give the most important huge quantities information about the movement of fluid where its axial velocity is zero in another expression where it is possible to try turn down to the reaction zones (flame) (see Fig. 8 (a))

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 [13] (see fig 7) .So The internal staging schematic adjusts the flame flow field to control the NOx emission that is because the high swirl provides an intense internal recirculating zone which recycles flue gas back into the flame and thus reducing the oxygen concentration as well as lowering flame temperature to prevent the NOx emission.

The simulation was done in steady frame by using NX Siemens code which depends near wall modeling assumptions for each turbulent model because some turbulent cases needs fine mesh close to the walls to capture the flow characteristics field as in the case of presence flow separation and/or not fully– developed flows. So enhanced wall function where applied on the walls of burner components especially for nozzles, burner tip and air deflector walls where to have accurate flow field model their wall-adjacent mesh obeyed this condition $y^+ \leq 5$ or $y^+ \geq 20$ (see Fig. 8 (b))

Images in fig.8 (b) show that 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.

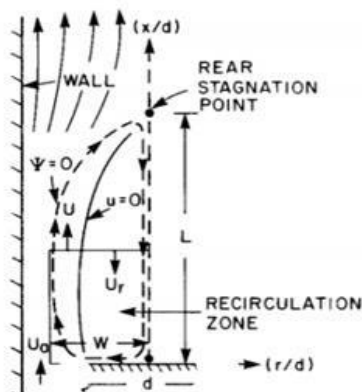


Fig.7: Sketch of the recirculation zone in a confined combustor flow.

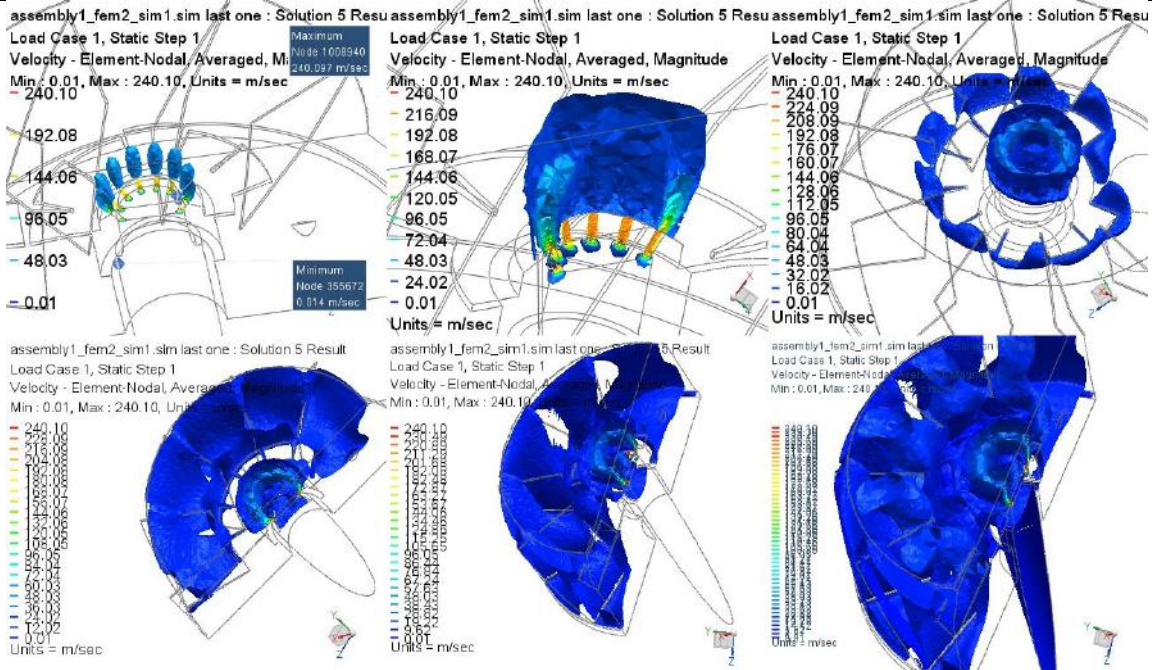


Fig.8: a plots of isothermal surface of velocity

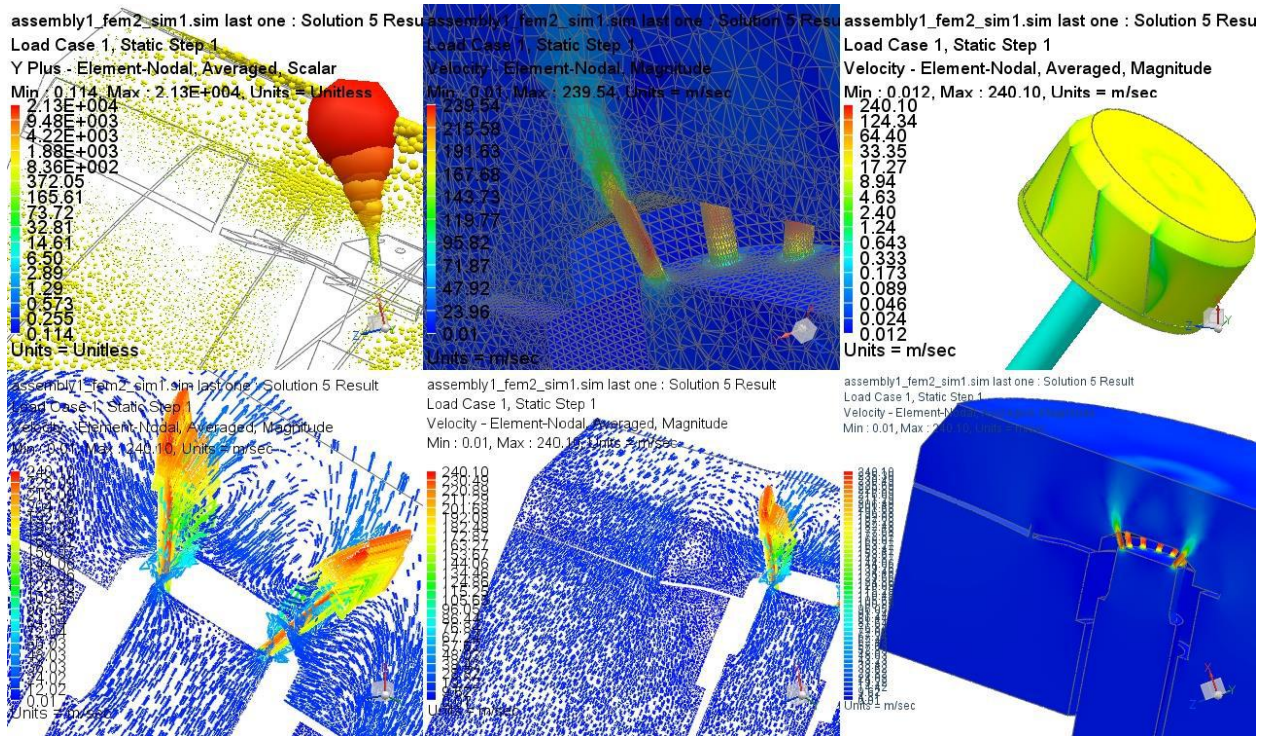


Fig. 8: (b) shows the recycle of fluid due to burner aerodynamics to reduce NO_x in spite of the mesh quality and y⁺.

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 quite stable even over a wide range of firing rate. In other words, the flame speeds and flow velocity are matched in this recirculating zone, and a good mixing of reactants with air occurs due to high turbulence generated by high shear stresses (see fig. 8 (c)).

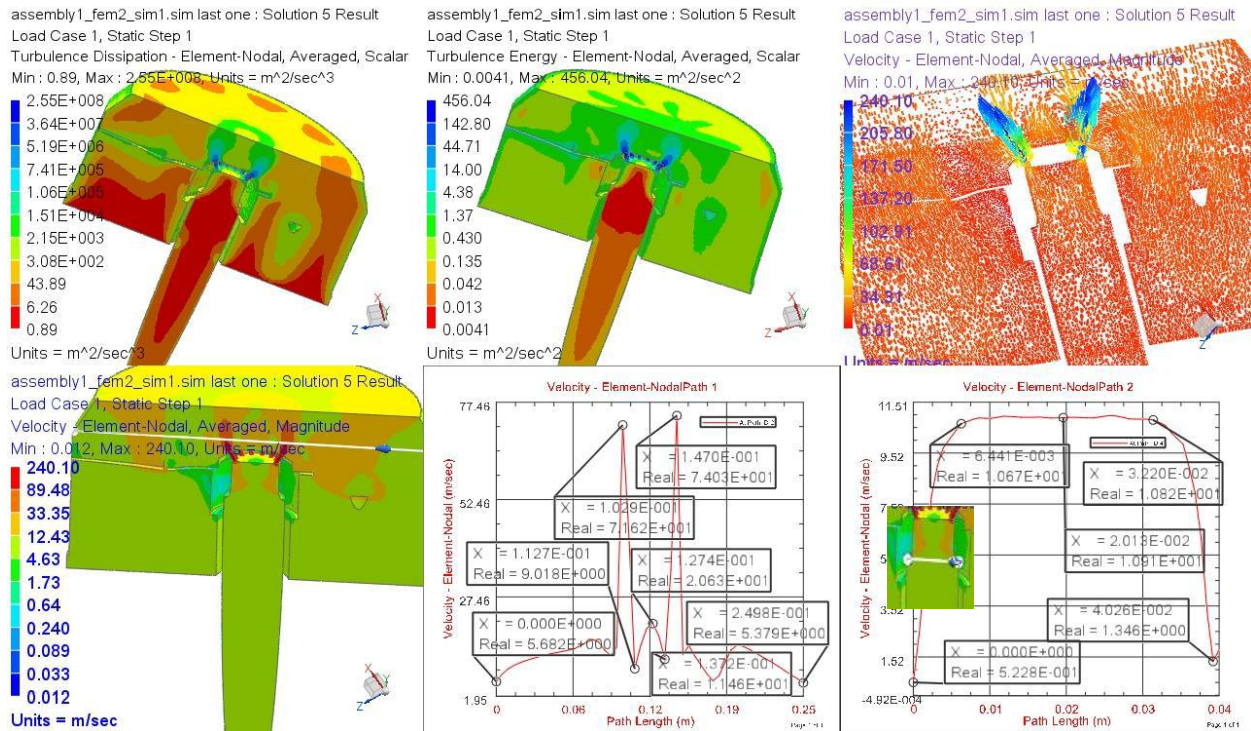


Fig.8: (c) shows that flame speeds and flow velocity are matched in the recirculating zone (note plots on two paths)

The flow fields of jets in crossflow are quite complex and fully three-dimensional (full turbulent), as shown in Fig.(9) They involve four main features [5], [14]: “(1) the horseshoe vortex, due to the adverse pressure gradient in front of the injection hole, (2) the wake vortices, (3) the shear layer ring vortices, which is induced by a Kelvin-Helmholtz instability in the shear layer of the jet, and (4) the counter-rotating vortex pair (CVP), generated by the deflection of the jet, and which seems to be induced by the shear layer vortices. The latter structure dominates the far field mixing, even if it is generated in the near field of the jet” [7].

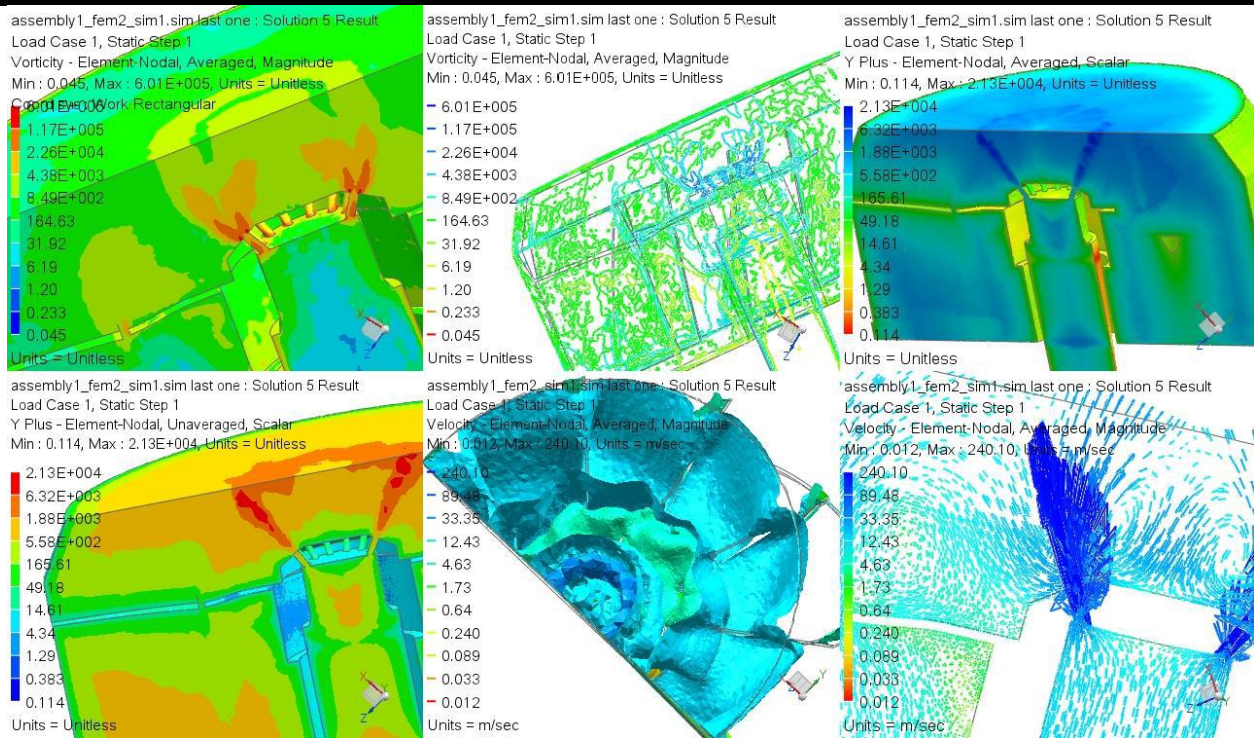


Fig.9: Shows full turbulent flow in burner and the features of jets flow fields

The distance from the inlet at which species will be mixed l_{mix} is given by ($l_{mix} = v \cdot T_{mix}$) where T_{mix} is the mixing time and v is the fluid velocity [15]

($D_T \propto \frac{K^2}{\epsilon}$ and $K \propto (L_T \cdot \epsilon)^{2/3}$) where K is the turbulent energy, L_T is the turbulence macro scale, D_T is the turbulent diffusion coefficient but ϵ is the turbulent energy dissipation rate which is proportional to the multiple of the flow rate and pressure drop ,so in result [15]

$$(\epsilon \propto v^3) \text{ but } (T_{mix} \propto D_T^{-1}) \text{ yields } (\epsilon \cdot D_T \propto L_T^{3/4} \cdot v^4)$$

In the result [15]

$$L_{mix} \propto 1/L_T^{4/3}$$

Thus the length which fluids will mix is independent of the fluid velocity and it can determined from Taylor macroscale [15]

The turbulence intensity is larger in regions of hairpin vorticity which are generated in high shear regions which surrounding the vortices or at the edge of blades (see fig.9 and fig.10). The computing Reynolds number is 1.2×10^5 so the flow field is fully turbulent and the turbulent length scale will not increase more to the restrictions of physical size of the burner so the mixing length in this fully established case will remain constant.

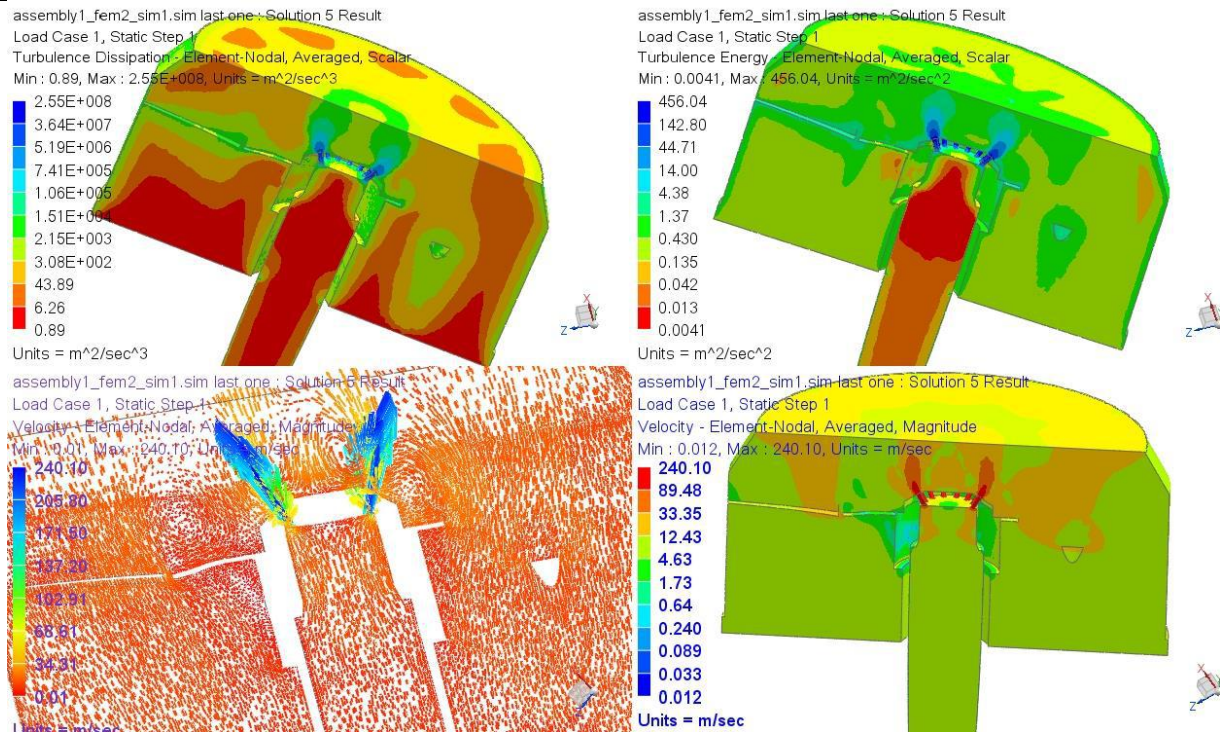


Fig.10: Shows the turbulence intensity of hairpin vorticity which are generated in high shear regions

VI. CONCLUSION

Mathematical model 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. So this Mathematical model shows how cold flow measurements can be combined with flame visualization to model the spatio-temporal response of fuel/air mixing field and the ability to use burner aerodynamics as alternative design concepts for staging the radially stratified flame core burner instead of relying on physically separate zones

REFERENCES

- [1] Coats, C. M.1996. Coherent structures in combustion. Progress in Energy and Combustion Science, 22(5):427–509.
- [2] Veynante,D. Piana,J. Duclos,J. M. and Martel, C. 1996. Experimental analysis of flame surface density models for premixed turbulent combustion. In Symposium International on Combustion, volume 1, pages 413–420. Combustion Institute.
- [3] 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
- [4] Lee ,J. H. W. and Chu ,V. H. 2003. Turbulent jets and plumes - a lagrangian approach. Kluwer Academic Publishers.
- [5] Gutmark, E. J. Ibrahim I. M. and Murugappan, S. Circular and noncircular subsonic jets in cross flow.
- [6] Fiskum,A. 2008. Calculation of NOx Formation in a Swirl Burner. Department of Energy and Process Engineering. Norwegian University of Science and Technology
- [7] Lapacelle,A.2011. Modeling, control, and optimization of fuel/air mixing in a lean premixed swirl combustor using fuel staging to reduce pressure pulsations and NOx emissions.Berlin university of TU ,ISBN on line 9783-9783-2382-7
- [8] Johari, H. 2006. Scaling of fully pulsed jets in crossflow. AIAA Journal, 44(11):2719–2725.
- [9] 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
- [10]Lee,H.G. and Kim,J.2011. A comparison study of the Boussinesq and the variable density models on

- buoyancy-driven flows. *J Eng Math* (2012) 75:15–27
DOI 10.1007/s10665-011-9504-2
- [11] Rongy, L., Haugen, K.B and Firoozabadi, A. 2011. Mixing from Fickian Diffusion and Natural Convection in Binary Non-Equilibrium Fluid Phases. Dept. of Chemical Engineering, Mason Laboratory, Yale University, New Haven, CT 06511. Wiley Online Library, DOI 10.1002/aic.12685.
- [12] Alabdalah, A., Ibrahim, H.A., Aloosh, E., Hoyas, S.C. 2018. Wall Y+ Approach for Dealing with Flow into Low NOx Swirler Burner in Industrial Furnaces. *International Journal of Civil, Mechanical and Energy Science (IJCMES)*, Vol-4, Issue-1, Jan-Feb, 2018. [
- [13] 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
- [14] Coussement, A., Gicquel, O., Schuller, T and Degrez, G. 2008. Large eddy simulation of pulsed jet in crossflow. *AIAA Paper 2010-561*, 2010. *Physics of Fluids*, 20:075110, 2008.
- [15] Bakker A., Cathie N., LaRoche R. (1994) Modeling of the Flow and Mixing in HEV Static Mixers. 8th European Conference on Mixing, September 21-23, 1994, Cambridge, U.K. IChemE Symposium Series No. 136, ISBN 0 85295 329 1, page 533-540 .