# Wall $Y^+$ Approach for Dealing with Flow into Low NOx Swirler Burner in Industrial Furnaces

Asya Alabdalah[1], Hassan Al-haj Ibrahim[1], Ethat Aloosh[1], Dr. Sergio Hoyas Calvo[2]

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

***Abstract***—*This paper investigates the accuracy of CFD simulation in predicting the nature of swirl flow field interactions in a Low NOx Swirler Burner s' environment. RANS-based turbulence models; k-ε were used to predict the turbulence phenomena. 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 . Here the internal recirculation zone is created to stabilize the flame and to adjust the flow field flow as optimum to reduce NOx emissions by using internal staging schematic.*

***Keywords— Low NOx Swirler Burner, turbulence phenomena, viscous sublayer turbulence model, internal recirculation zone, NOx emissions.***

## I. INTRODUCTION

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 extending the residence time. In practice industrial burners cannot rely on an exact metering of the fuel and air to exactly match the flame speed but it depends on the two phenomenon the turbulent flame speed and flame holders. The shape of flame holder, its size and blockage ratio affect the flame stability characteristics which influences the size and shape of the wake region. The simulation by using computing fluid dynamics shows one type of Radially Stratified Flame Core Burner RSFC and this shows the important aerodynamics features of a typical low Nitrogen Oxides NOx of internal staging schematic. As the fuel and air flow by the ledge at high velocity, the ledge or step forces some gas to recirculate, often in chemical processes turbulent flow at these interfaces (walls) increases the rates of mass transfer by increasing gradients because close to walls molecular viscosity and diffusivity dominate but away from the walls turbulent diffusivity dominates, so law of the wall describes length –time scales for turbulence near walls. The log law formula can be used to compute the velocity boundary condition at the outer edge of the buffer layer, or even farther from the wall if extremely coarse gridding in employed. The simple law of the wall described here is not valid for non-similar boundary layers, and furthermore cannot be used accurately in the presence of flow separation and/or not fully–developed flows.

## II. METHODOLOGY

Consider fully turbulent flow at high Reynolds number ($Re=UL/\nu$) where U, L and ν are consequently the characteristic velocity, length and kinematic viscosity of fluid flow. Turbulence can be considered to have different sizes of eddies. Eddies of size (l) have a characteristic velocity u (l) and timescale ($\tau(l) \equiv l/u(l)$). Eddies in the largest size range are characterized by the length scale ($l_o$) which can be estimated from the characteristic length scale (L) of flow, and characteristic velocity ($u_o \equiv u(l_o)$) is on the order of the r.m.s. where turbulence intensity $u' \equiv (2k/3)^{1/2}$ is comparable to U where k is the kinetic energy of turbulence .Here energy cascade process happens in which the energy successively transfers to smaller and smaller eddies until the Reynolds number ($Re(l) \equiv u(l)\, l / \nu$) is sufficiently small that the eddy motion is stable, and molecular viscosity is effective in dissipating the kinetic energy of turbulence into heat fig.(1).shows the energy cascade idea from large to small eddies [1]

These eddies have energy of order $u_o^2$and timescale $\tau_o=l_o/u_o$, and rate of energy transfer can be scaled as $u_o^2/\tau_o= u_o^3/l_o$ so the energy cascade indicates that the dissipation ε can be estimated from the large-scale motions because it is proportional to $u_o^3/l_o$ and independent of ν at high Reynolds numbers [2].

https://dx.doi.org/10.22161/ijcmes.4.1.2 ISSN: 2455-5304

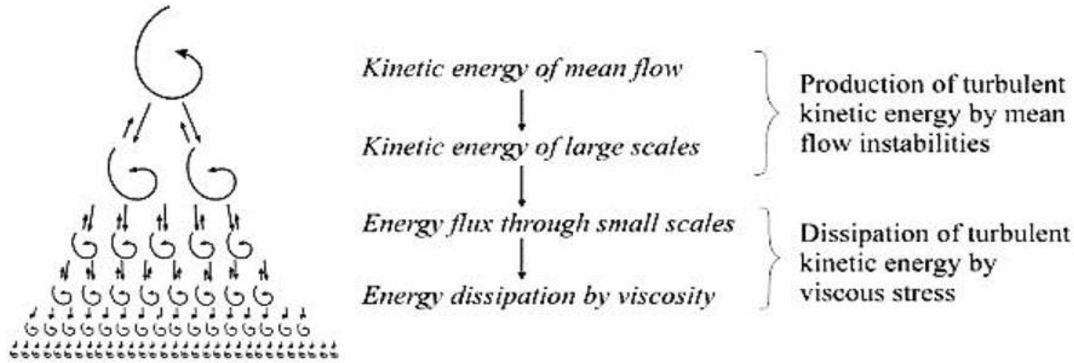


*Fig.1: The energy cascade idea from large to small eddies [1]*

In direct numerical simulation DNS one solves the complete Navier-Stokes equations on a computational grid without any approximations or averaging on turbulence, so the grid resolution should be fine enough to capture the smallest scales of motion in the flow. The computational cost associated with computing a flow that resolves all of the scales of flow structure in a DNS is equal to $Re^{17/4} \sim Re^4$ [2]

A DNS is not practical for high Reynolds-number flows because the range of space and time scales is too large. Therefore, the spatio-temporal motions must be filtered In order to have acceptable computation speeds. This issue can be passed in industry models by using Reynolds' averaged Navier Stokes (RANS) turbulence approach, which applies Reynolds' decomposition to the full Navier-Stokes equation, where the exact flow solution through time and space is averaged to produce an average and fluctuating component [2]. However aerodynamics of swirling flow is very complex a strong coupling with aerodynamic effect and reaction effect occurs in flow and these must be included in the model to have acceptable numerical solutions.

## III. SWIRL BURNER GEOMETRY AND ITS TECHNICAL DATA

in this combustion the presence of the internal recirculation zone plays important role in flame stabilization and introducing rotation in the stream can be achieved by the following three principle Methods: tangential entry of the fluid stream (Axial plus tangential entry swirl generator), and guided vanes [3]. So the vanes are mounted on a central gaseous fuel tube by the air deflector geometry, and another vanes occupy the space in the annular around it by a vane-type swirler, and consists of a fixed set of vanes at a certain angle to the main stream direction which deflect the stream into rotation the fig. (2) 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 , so the following are the technical data of burner:

- 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/$Nm^3$

* Net calorific value: 11000kcal/kg

* Density kg/$Nm^3$ 0.924

*Fig.2: The burner geometry*

* Flammability limits% 3.4-16.5

https://dx.doi.org/10.22161/ijcmes.4.1.2 **ISSN: 2455-5304**

- Temperature in the spot of the burner inlet 30°C
- 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
- Nozzle placing Design 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 α=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
- Air box consists of two cylinders, the internal one is fixed with thickness 2 mm and dimensions $804^{-1}$×370 mm and diameter $258^{-0.5}$mm, and the outside is movable one with thickness 2 mm, dimensions $823^{+2}$×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.

So the swirling flow is produced by tangential jet injectors and vane swirlers in addition to the tip of burner which works as bluff body. The size and the strength of the recirculation zone and most flame properties are determined by the swirl angle, the propagation of flame to other regions is done by the dynamics of internal recirculation zone where heat and radicals transport from the boundaries of the recirculation zone to the adjacent fresh mixture, the burnt gases transfer heat to the recirculation zone to balance the heat lost in igniting the combustible gas. 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 conbustor and consequently the mixing pattern might be affected to some extent [4]

By using three dimensional prototype multi –annular swirl burner in line with the actual situation, the important aerodynamics features effect the NOx emissions was studied. In result NOx emissions are not more than $120ppm_v$ at 15% excess air and the implement internal staging schematic is effective and promise way to have optimum flow pattern field along with high furnace efficiency at low NOx emissions. So by using swirl means the structure of flame is adjusted to locally fuel-rich environment prior to burn out. Numerical simulation of a turbulent gas burner of diffusion flame were carried out using the commercially available software package NX Siemens.

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 fluid flow namely the set of average Navier stocks Models RANS which are the three equations for conservation of mas, momentum and energy. Fig. (3) Shows the prototype multi –annular swirl burner in accordance with design data (a) and typical low $NO_X$ RSFC flow field with internal staging schematic in accordance with ASME 1994(b).

The flow was solved in steady-state frame. All walls of the components in the model were smooth, buoyancy 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.

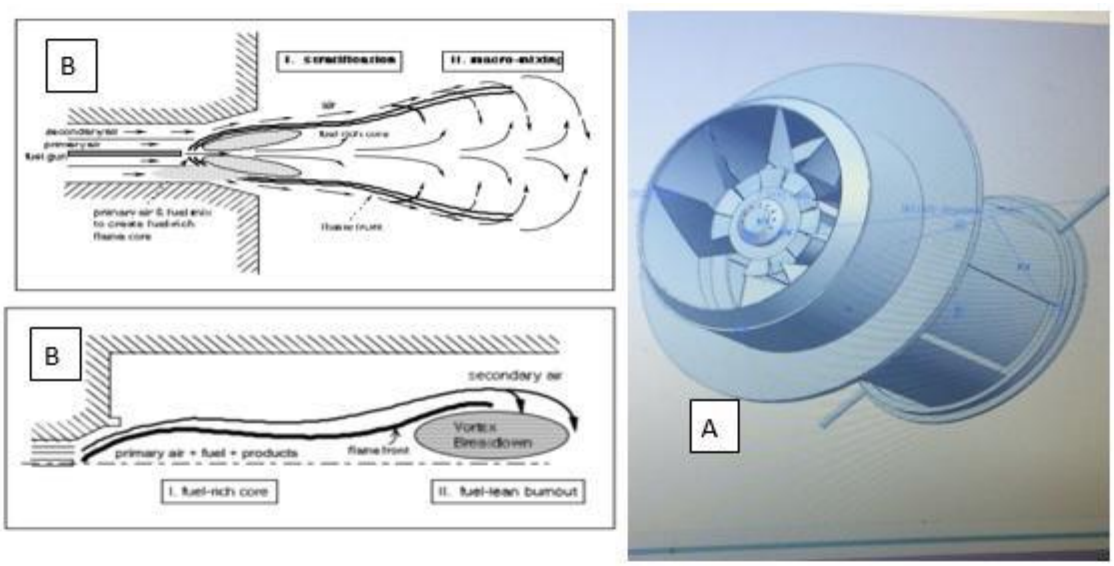


*Fig.3: The prototype multi –annular swirl burner in accordance with burner s'design data (A), and typical low $NO_X$ RSFC flow field with internal staging schematic (B) [5].*

https://dx.doi.org/10.22161/ijcmes.4.1.2 ISSN: 2455-5304

## IV. COLD MATHEMATICAL MODEL FLOW OF SWIRLER BURNER

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:

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:

$$\rho\left[\frac{\partial \bar{u}_i}{\partial \tau} + \frac{\partial\left(\overline{u_j u_i}\right)}{\partial x_j}\right] = -\frac{\partial p}{\partial x_i} + \frac{\partial \mathrm{T}_{ij}}{\partial \mathrm{x}_j} + B_T . g . (T - T_\infty)$$

Where: $T$ is the actual temperature applicable locally. $T_\infty$ : 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 $\rho$

Is the local density so ( $B_T = 1/T$), $\mathrm{T}_{ij}$ is the Reynolds stress tensor divided by the density and it is computed by using the Boussinesq model equation

$$\mathrm{T}_{ij} = -u_i^{'} u_j^{'} = \nu_\mathrm{T}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3}\frac{\partial \bar{u}_k}{\partial x_k}\delta_{ij}\right) - \frac{2}{3}\mathrm{K}\delta_{ij}$$

Where K is the turbulent kinetic energy, $\delta_{ij}$ 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 $\nu_\mathrm{T}$ is the turbulent eddy viscosity can be calculated by K-Epsilon model so

$$\nu_\mathrm{T} = C_\mu \frac{\mathrm{K}^2}{\varepsilon}$$

Where $C_\mu$ is constant equals to 0.09

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

## V. WALL BOUNDED TURBULENT FLOWS

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 [6] .So for wall bounded turbulent flows (boundary layers, channels etc..) it is important that mesh near the wall is properly sized to ensure accurate simulation of the flow field. In NX Siemens there are near wall modeling assumptions for each turbulent model, that applying wall function because in 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 must be 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 must obey this condition

$\mathbf{y^+ \le 5}$ or $\boldsymbol{y^+ \ge 20}$ .That's because there is separation flow in the flow field especially for the internal recirculation zone which is the result of the positive and adverse pressure gradient occurs in the direction of flow in the middle of the swirl burner , this flow field for internal recirculation zone will be clear enough for wall-adjacent mesh when it obeys the condition $\mathbf{y^+ \ge 20}$ on the burner walls components of nozzles , burner tip and air deflector walls although this choice is acceptable for another various geometry of the swirl burner s walls where there are different boundary layers that is because for these walls it is good to avoid having the wall-adjacent mesh in the buffer region since neither wall functions nor near wall modelling approach accounts for it accurately , and another choice for these different boundary layers on the walls of burner is to apply $\mathbf{y^+ \ge 60}$ **,** and this method is more effective way to have less expensive computing cost. So for all $\boldsymbol{y^+}$ treatment it must have been a hybrid or adaptive treatment for both coarse and fine meshes which is necessary to provide a reasonable solution for cells lying in buffer regions as well as the cost of the computing fig. (4) Shows the low of wall function modeling approach

https://dx.doi.org/10.22161/ijcmes.4.1.2 ISSN: 2455-5304

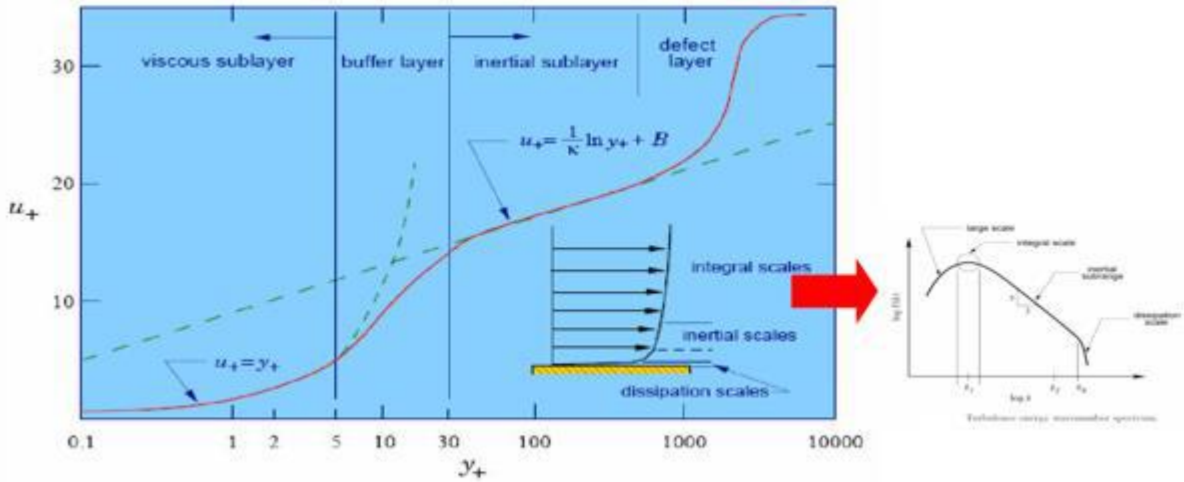


Fig.4: The law of wall function modeling approach [6].

## VI. MESHING

The geometry features of this burner is very complex so it is necessary to use tetrahedral elements because the effort required to model hexahedral element is too excessive, and it will be result in poor quality while tetra mesh allows to fill an arbitrary volume defined by its surface using Tera/quad elements with tetrahedral mesh elements . Refining the meshes would reduce the truncation error.

When the mesh is sufficiently fine, mesh type has little influence on the accuracy of simulation results fig (5) shows the typical neighboring cells and parameters of two adjacent cells for different mesh grid types: (A) tetrahedral mesh and (B) hybrid mesh [7].

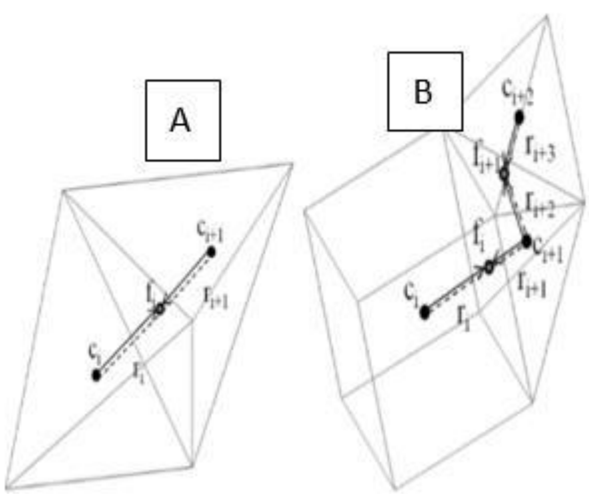


Fig.5: (A) tetrahedral mesh and (B) hybrid mesh [7]

## VII. 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 ($l = 0.07L$).

## VIII. RESULTS AND CONCLUSION

Velocity vector plots can be seen below in Fig. 6 (a) and (b). 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 portion of air combustion equals to 30% tends to move around the air deflector as primary air so an initial fuel –rich flame core is created by the mixing process between the central fuel jets and with the 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 Figure 6.

*https://dx.doi.org/10.22161/ijcmes.4.1.2* *ISSN: 2455-5304*

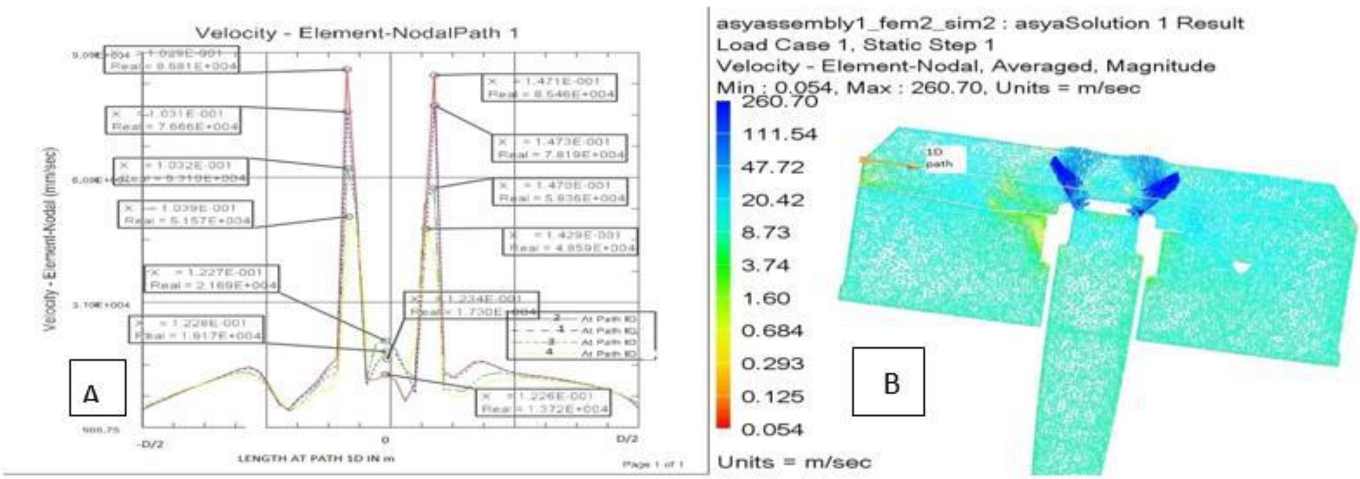


*Fig.6: Velocity vector plots can be seen at path 1D at deferent $y^+$ (a) and internal recirculation zone in the middle of the burner (b)*

From the flowing photos it can be seen the differences in $y^+$ and the velocity plot 1,2,3,and 4 at path 1D at the same model for the mention $y^+$

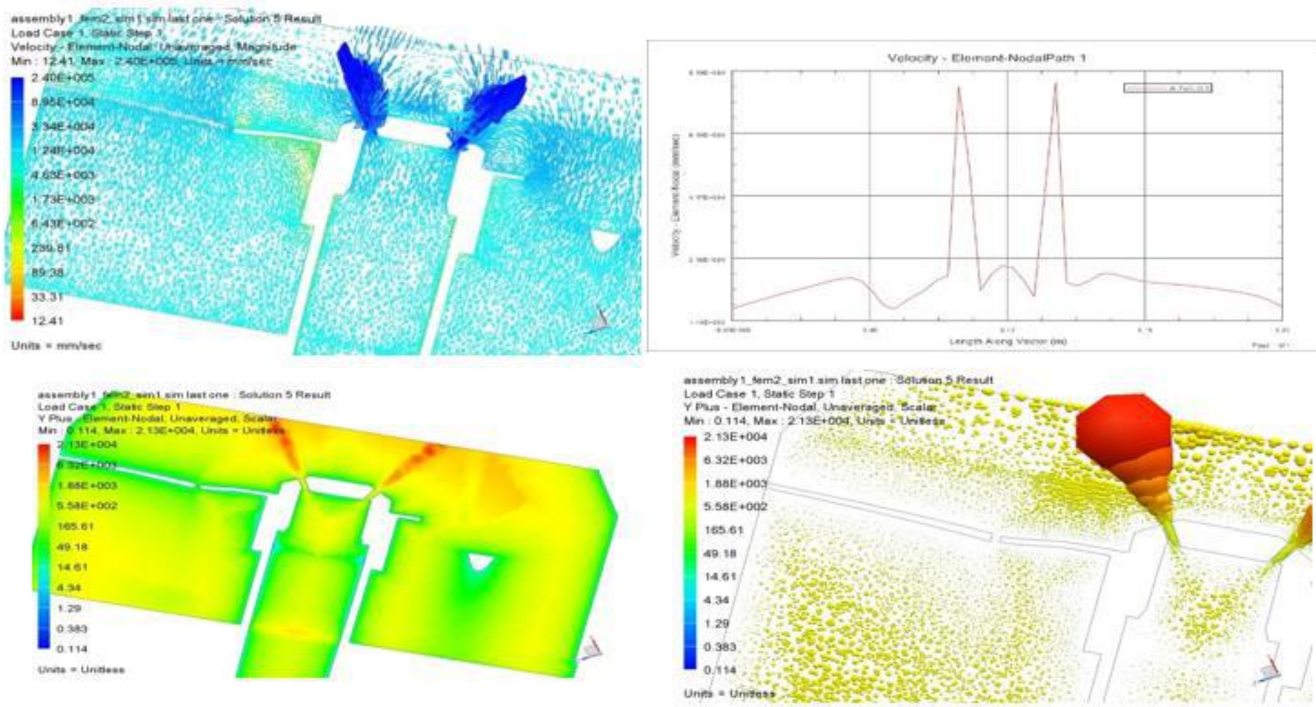


*Photo.1: solution 1 by iteration method at a steady state-physical relaxation step equals to 0.0005 sec, the convergence criteria of RMS residuals is $10^{-5}$, and the following table 1 ultimates the data of this solution*

*Table.1: ultimates data of solution 1*

| **Computing Reynold number** | **Number of element** | **$y^+$ information on adiabatic faces** | | | **Computing cost** |
|---|---|---|---|---|---|
| $1.2\times10^5$ | 5458580 | min $y^+$ | max $y^+$ | ave. $y^+$ | **8hr 05min 48sec** |
| | | $1.29\times10^{-1}$ | $8.50\times10^1$ | $1.18\times10^1$ | |

*https://dx.doi.org/10.22161/ijcmes.4.1.2* *ISSN: 2455-5304*

Photo (2) solution 2 by iteration method at a steady state-physical relaxation step equals to 0.0005 sec , the convergence criteria of RMS residuals is $10^{-6}$ , and the following table 2 ultimates the data of this solution.

*Table.2: ultimates data of solution 2*

| **Computing Reynold number** | **Number of element** | **$Y^+$ information on adiabatic faces** | | | **Computing cost** |
|---|---|---|---|---|---|
| $1.2\times10^5$ | 5613826 | min $y^+$ | max $y^+$ | ave. $y^+$ | **21 hr 04min 41sec** |
| | | $9.79\times10^{-1}$ | $8.46\times10^1$ | $1.41\times10^1$ | |

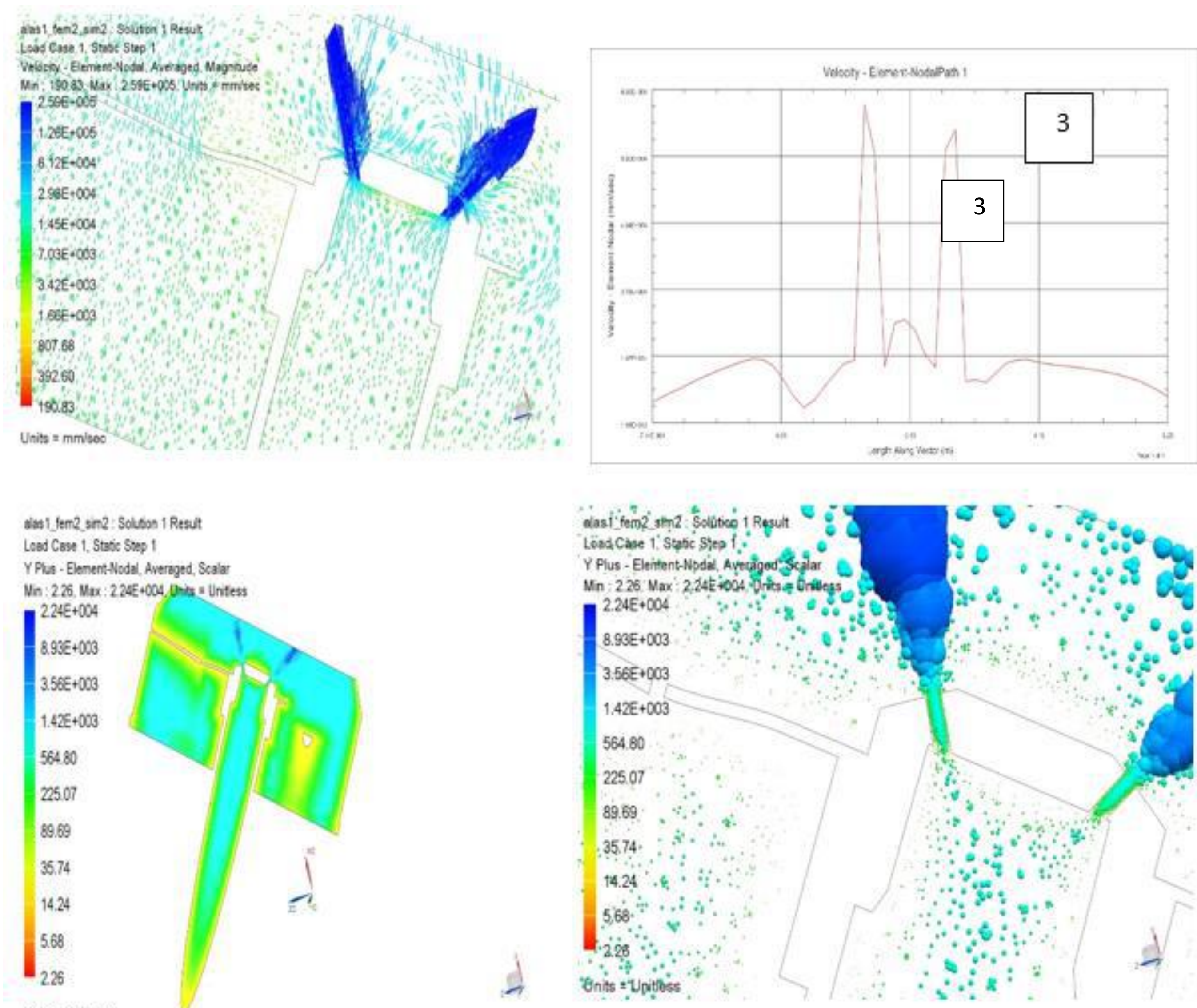


*Photo.3: solution 3 by iteration method at a steady state-physical relaxation step equals to 0.005 sec , the convergence criteria of RMS residuals is $10^{-5}$, and the following table 3 ultimates the data of this solution*

*Table.3: Ultimates the data of this solution*

| **Computing Reynold number** | **Number of element** | **$Y^+$ information on adiabatic faces** | | | **Computing cost** |
|---|---|---|---|---|---|
| $1.2\times10^5$ | 1095305 | min $y^+$ | max $y^+$ | ave. $y^+$ | **1hr 33 min 56 sec** |
| | | 4.09 | $1.14\times10^2$ | $2.01\times10^1$ | |

https://dx.doi.org/10.22161/ijcmes.4.1.2 ISSN: 2455-5304

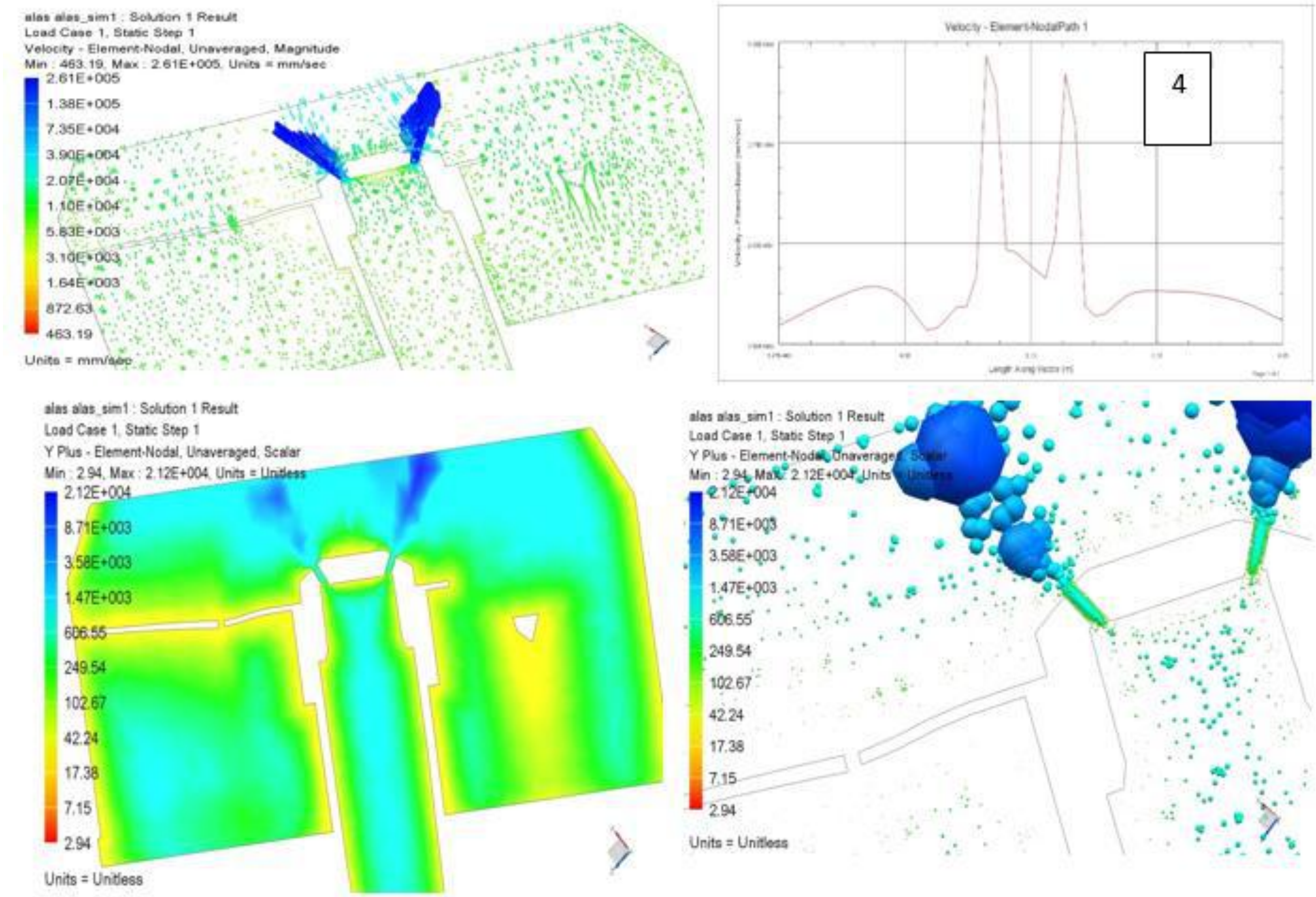


*Photo.4: solution 4 by iteration method at a steady state-physical relaxation step equals to 0.005 sec, the convergence criteria of RMS residuals is $10^{-5,}$ and the following table 4 ultimates the data of this solution*

*Table.4: Ultimates the data of solution 4*

| **Computing Reynold number** | **Number of element** | **$Y^+$ information on adiabatic faces** | | | **Computing cost** |
|---|---|---|---|---|---|
| $1.2\times10^5$ | 271022 | min $y^+$ | max $y^+$ | ave. $y^+$ | **11 min 18 sec** |
| | | 3.84 | $1.66\times10^2$ | $3.38\times10^1$ | |

The internal recirculation zone is a major reason of pressure drop due to impingement on the air deflector. At the outlet, boundary layer separation takes place and the flow from the air box tends to mix with each fuel jet. The internal staging schematic adjusts the flame flow field to control the $NO_x$ 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.

As the swirler burner has a complex geometry which involves the turn in the sharp corners by very rapid acceleration causing separation at the edge and recirculation in the wake region of the backward facing step it is a condition to use hybrid or adaptive treatment for both coarse and fine meshes which is necessary to provide a reasonable solution for flow field in burner and in this research the solution 1 is preferred.

## REFERENCES

[1] Andersson, B. Andersson, R. Hakansson, L. Mortensen, M. Sudiyo, R. Van Wachem, B. Hellström, L. 2011. Computational Fluid Dynamics for Engineers. Cambridge University Press, Published in the United States of America by Cambridge University Press, New York.

[2] Versteeg, H. K. Malalasekera, W. 2007. An Introduction to Computational Fluid Dynamics, The finite volume method

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

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

[5] 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

[6] 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

[7] Duan,R.Liu,W. Xu,L. Huang,Y. Shen,X. Lin,C.H. Liu,J. Chen,Q. and Sasanapuri,B.2015.MESH TYPE AND NUMBER FOR CFD SIMULATIONS OF AIR 1 DISTRIBUTION IN AN AIRCRAFT CABIN. Numerical Heat Transfer, Part B: Fundamentals, 67(6), 489-506