Design and Optimization of Flow Field of Tangentially Fired Boiler

Y.J. Wagh[1], V.H.Patil[2]
[1](Mechanical Engineering Department, JIT Nasik, India)
[2](Mechanical Engineering Department, GCOE, Jalgaon, India)

ABSTRACT: *In thermal tangentially fired boiler, the flame stability mainly dependent on the central swirl strength which characterizes the mixing between the air and fuel. In this work, numerical investigation is performed on 600MW pulverized coal tangentially fired dry-bottom boiler and validated with experimental data. The main focus of the work is to study the effects of burner firing angle and mass flow rate on the flow characteristics inside the burner. The important feature of the model is a tangential fired geometry in furnace where four burners are kept at the corners of the furnace for generating swirling vortex in the center tangentially, which decides the flame propagation effectiveness and time to sustain flame for longer time and combustion efficiency. Optimization is performed for different design parameters like burner velocity and firing angle with objective function of enhancement of mixing efficiency in the furnace. From that the designer of optimization and simulation makes it possible to find the optimum design and operating parameters. The literature is reviewed to understand the base case as shown in Figure 1 is simulated using the burners angle& velocity as mentioned in Table 1 and the numerical results for the base parameters are compared with the experimental results.*

Keywords *–Burner, flame, mixing, tangentially fired boiler, vortex.*

1. INTRODUCTION

In this paper investigated that Thermal power plants are one of the most important process industries for engineering. Over the past few decades, the power sector has been facing a number of critical issues. However, the most fundamental challenge is meeting the growing power demand in sustainable and efficient ways. Power plant engineers not only look after operation and maintenance of the plant, but also look after a range of activities in that including research and development, starting from power generation, to environmental assessment of power plants. In thermal power plant, the chemical energy stored in fossil fuels such as coal, fuel oil, natural gas is converted successively into thermal energy, mechanical energy and finally electrical energy. In the Rankine cycle, high pressure and high temperature steam raised in a boiler is expanded through a steam turbine that drives an electric generator.

Pulverized coal tangentially fired furnaces are used extensively in thermal power plants due to a number of their advantages, [1] like uniform heat flux to the furnace walls and NO_x emission lower than in other firing types. Study of the furnaces is done by both experiments and simulations. While full-scale measurements are restricted by considerably high expenses. Comparatively numerical simulation provides a cost-effective and powerful engineering tool, complementing experimental investigations.

Table 1: Geometrical and Flow Parameters of the Burner

Parameters	Values
XL	0.657m
YL	0.741m
Θ_a	45^0
Θ_b	36^0
Inlet Velocity	14.1m/s

2. NUMERICAL MODELLING

The developed comprehensive model extended available submodels by describing fully the 3D flow, combustion and heat transfer in existing geometry, with in details modeling of the interactions between turbulence and particles and by including chemical kinetics of the coals considered and real coal particle size distribution.

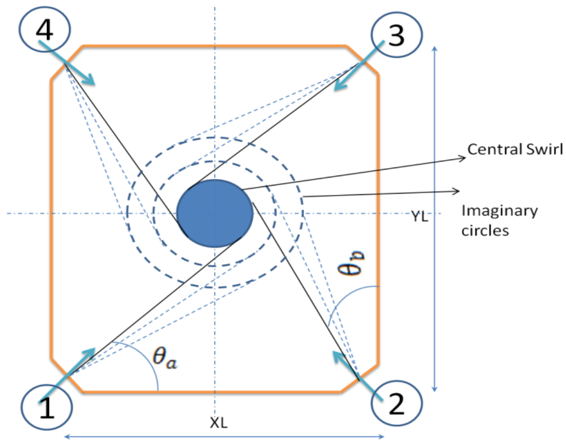


Figure 1- Schematic diagram of power plant burner

2.1 Numerical Model

Turbulent flow of multi component gaseous phase is described by time-averaged Eulerian partial differential conservation equations for mass, momentum, energy, concentrations of gaseous components, as well as the turbulence kinetic energy and its rate of dissipation. For general variable Φ. [1]

$$\frac{\partial}{\partial x_j}\left(\rho U_j \emptyset\right) = \frac{\partial}{\partial x_j}\left(r_\emptyset \frac{\partial \emptyset}{\partial x_j}\right) + S_\emptyset + S_p^\emptyset$$

(1)Standard k–ε gas turbulence model is extended to a 3D case. For coupling of phases PSI Cell method is used, with additional sources due to particles

$S_\rho^\emptyset$ .The effect of particles-to-gas turbulence is modeled by additional sources for k and ε.Dispersed phase is described by differential equations of motion, energy and mass change in Lagrangian field, with diffusion model of particle dispersion by turbulence. Particle velocity is a sum of convective and diffusion velocity:

$$U_p = U_{pc} + U_{pd} \quad (2)$$

The convective velocity is obtained from the equation of motion, by particle tracking along the trajectories with constant particle number density. The dispersion is modeled by introducing the particle diffusion velocity. Inter-particle collisions are neglected and wall-to-particle collisions are supposed to be elastic. Diffusion velocity is given as

$$U_{pd_i} = -r_p \nabla N_p, U_{pdi} = -\frac{1}{N_p} r_p \frac{dN_p}{dx_i} \quad (3)$$

SIMPLE calculation algorithm is used for coupling of the continuity and the momentum equations. The control volume method and hybrid-differencing scheme are used for casting the differential equations into a system of linear algebraic equations. For solving the system, a modification of SIP method is used.

2.2 Turbulence Model

Turbulence modeling is the construction and use of a model to predict the effects of turbulence. Averaging is often used to simplify the solution of the governing equations of turbulence, but models are needed to represent scales of the flow that are not resolved. [2]

Turbulence is that state of fluid motion which is characterized by apparently random and chaotic three-dimensional vorticity. When turbulence is present, it usually dominates all other flow phenomena and results in increased energy dissipation, mixing, heat transfer, and drag. If there is no three-dimensional vorticity, there is no real turbulence. The reasons for this will become clear later; but briefly, it is ability to generate new vorticity from old vorticity that is essential to turbulence. And only in a three-dimensional flow is the necessary stretching and turning of vorticity by the flow itself possible.

There are several subcategories for the linear eddy-viscosity models, depending on the number of (transport) equations solved for to compute the eddy viscosity coefficient.

1. Algebraic models
2. One equation models
3. Two equation models

2.2.1 Algebraic turbulence models

Algebraic turbulence models or zero-equation turbulence models are models that do not require the solution of any additional equations, and are calculated directly from the flow variables. As a consequence, zero equation models may not be able to properly account for history effects on the turbulence, such as convection and diffusion of turbulent energy. These models are often too simple for use in general situations, but can be quite useful for simpler flow geometries or in start-up situations. The two most well-known zero equation models are the

- Baldwin-Lomax model and the
- Cebeci-Smith model

Other even simpler models, such a models written as $\mu_t = f(y^+)$are sometimes used in particular situations (e.g. boundary layers or jets).

2.2.2 One equation turbulence models

One equation turbulence models solve one turbulent transport equation, usually the turbulent kinetic energy. The original one-equation model is Prandtl's one-equation model. Other common one-equation models are:

- Baldwin-Barth model
- Spalart-Allmaras model
- Rahman-Agarwal-Siikonen model

2.2.3 Two equation turbulence models

Two equation turbulence models are one of the most common types of turbulence models. Models like the k-epsilon model and the k-omega model have become industry standard models and are commonly used for most types of engineering problems. [3]Two equation turbulence models are also very much still an active area of research.

By definition, two equation models include two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy.

Most often one of the transported variables is the turbulent kinetic energy k . The second transported variable varies depending on what type of two-equation model it is. Common choices are the turbulent dissipation ϵ , or the specific dissipation ω. The second variable can be thought of as the variable that determines the scale of the turbulence (length-scale or time-scale), whereas the first variable k, determines the energy in the turbulence.

K-epsilon Model:

K-epsilon (k-ε) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate turbulent conditions. It is a two equation model which gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows.

The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy (k).The second transported variable is the turbulent dissipation (ϵ) which determines the rate of dissipation of the turbulent kinetic energy.The K-epsilon model has been shown to be useful for free-shear layer flows with relatively small pressure gradients. Similarly, for wall-bounded and internal flows, the model gives good results only in cases where mean pressure gradients are small; accuracy has been shown experimentally to be reduced for flows containing large adverse pressure gradients.

To calculate boundary conditions for these models the turbulence free-stream boundary conditions are Transport Equations for the model:

For turbulent kinetic energy k

$$\frac{d}{dt}(\rho k) + \frac{d}{dx_i}(\rho k u_i) = \frac{d}{dx_i}[(\mu + \frac{\mu_t}{\sigma_k}) + P_k + P_b - \rho\epsilon - Y_M + S_k$$

For dissipation ϵ

$$\frac{d}{dt}(\rho\epsilon) + \frac{d}{dx_i}(\rho\epsilon u_i) = \frac{d}{dx_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{d}{dx_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(p_k + C_{3\epsilon}P_b) - C_{2\epsilon\rho}\frac{\epsilon^2}{k} + S_\epsilon$$

3. EFFECT OF PARAMETERS ON THE PERFORMANCE OF TANGENTIALLY FIRED FURNACE

The Burner design and analysis is performed for the different parameter with the following objective function

- To investigate the vortex strength of tangentially fired boiler, sustains the flame propagation for efficient combustion
- To investigate the effect of the following important parameters on vortex formation
  - Burner Angle (Base Case 43^0, 39^0)
  - Inlet Velocity (Base Case V=14 m/sec)[4]

3.1Numerical Investigation of burner velocity

A jet is produced when a fluid is discharged through the nozzle. In the jet the velocity of the fluid is accelerated. Free jet is produced when the fluid is discharged in the surrounding with no confinement. A jet is said to be confined when the fluid is discharged in the container. [5] The characteristic feature of the jet (whether free is confined) is that it spreads due to the difference in the density of the jet and the surrounding. A hot jet in the cold surrounding spreads faster than a cold jet in the same surrounding. For any downstream axial distance, the maximum velocity is at the centre and minimum at the periphery such that a parabolic is developed as shown in Fig.2

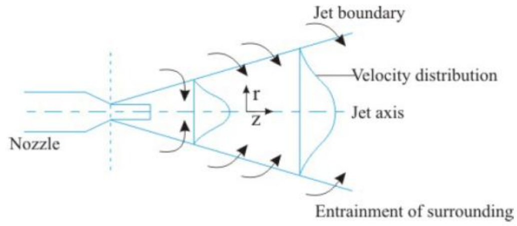


Figure 2. Jet dynamics from the burner as nozzle

Table 2: Burner velocity arrangement configurations

Cases Sr. No.	Parameter-Velocity (m/s) Range
1	10
2	12
3	14 (Base)
4	16

3.1.1 Burner Velocity (V=10 m/sec)

The burner velocity is decreased from the base case i.e. from 14m/sec to 10 m/sec for understanding the effect of Burner velocity on the turbulence dynamics in the Burner. The contours of velocity and turbulence as well as static and dynamic pressure are shown in the Fig. 3&4.

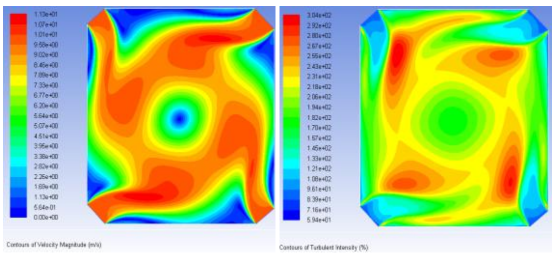


Figure 3. Contours of velocity magnitude and turbulence intensity for Burner Velocity 10 m/sec.

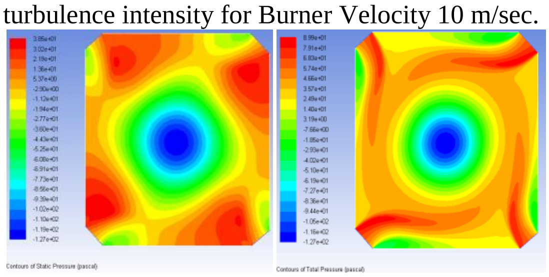


Figure 4. Contours showing static and total pressure for Burner Velocity 10 m/sec.

3.1.2 Burner Velocity (V=12 m/sec)

The burner velocity is further increased from the previous cases to 12 m/sec by less than the base case velocity. [6]The contours of velocity and turbulence as well as static and dynamic pressure are shown in the Fig. 5 & 6.

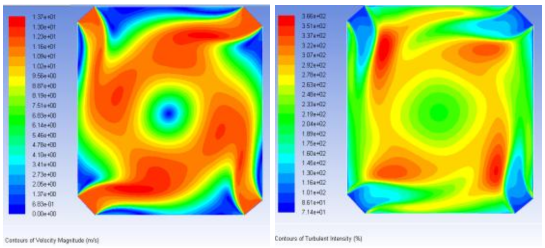


Figure 5. Contours of velocity magnitude and turbulence intensity for Burner Velocity 12 m/sec.

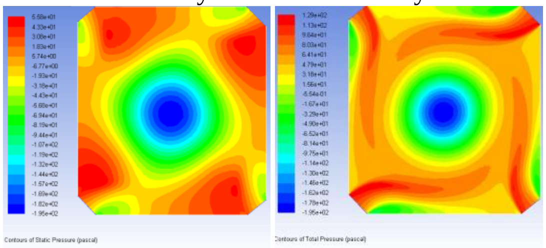


Figure 6. Contours showing static and total pressure for Burner Velocity 12 m/sec.

3.1.3 Burner Velocity (V=14 m/sec)

The base case for the burner is simulated for the given velocity i.e. 14m/sec and effect is checked with other design variables.

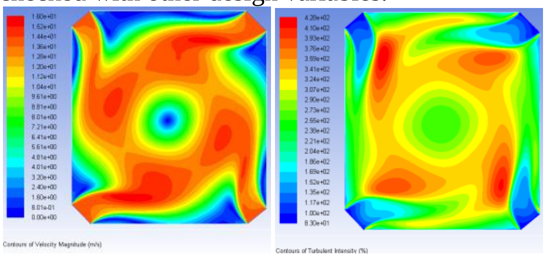


Figure 7. Contours of velocity magnitude and turbulence intensity for Burner Velocity 14 m/sec.

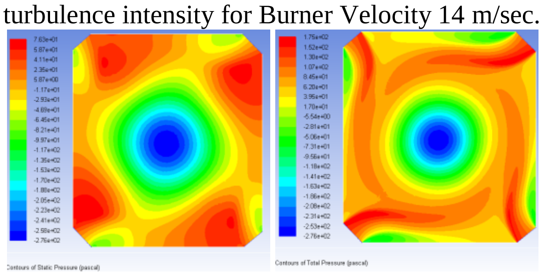


Figure 8. Contours showing static and total pressure for Burner Velocity 14 m/sec.

3.1.4 Burner Velocity (V=16 m/sec)

The burner velocity is increased form 14m/sec to 16 m/sec by increasing the mass flow rate of the pulverized mixture for understanding the behaviors of internal burner flow dynamics.

The velocity and turbulence contour along with pressure contours are shown in the Fig. 9 & 10 respectively.

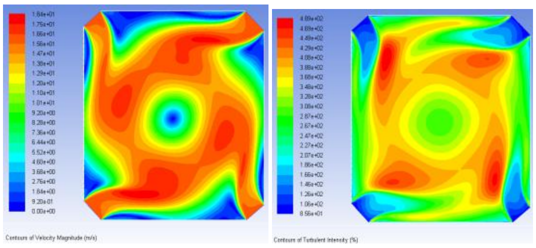


Figure 9. Contours of velocity magnitude and turbulence intensity for Burner Velocity 16 m/sec.

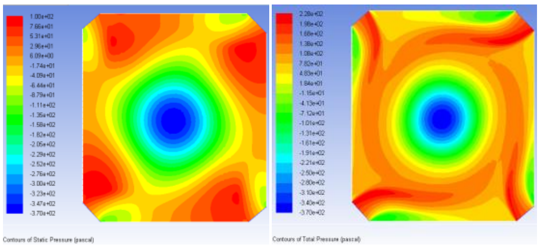


Figure 10. Contours showing static and total pressure for Burner Velocity 16 m/sec.

3.2Numerical Investigation of burner angle

Burner angle decides the direction of flow and the mid-circle diameter, but changing the burner angle is restricted by the size of the furnace, therefore one higher angle set and one lower angle set is selected which is feasible for the current furnace size for understanding the effect of burner angle on the vortex strength.[7]

Table 3: Burner angle arrangement configurations

Parameter-Angle(m/sec) Range	**1**	**2**
1 (Base)	43	39(51)
2	33	29(61)
3	39	35(55)
4	46	42(48)

The results are to be compared with the base case i.e. $\Theta_1=43^0$ and $\Theta_2=51^0$. The velocity and pressure contours are shown in the Fig. 11&12 respectively.

3.2.1 Burner Angle (43^0, 39^0) is BASE CASE

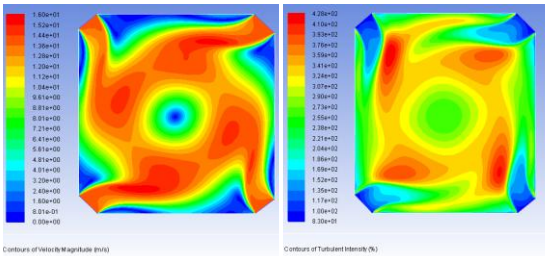


Figure 11. Contours of velocity magnitude and turbulence intensity for Burner angles (43^0, 39^0)

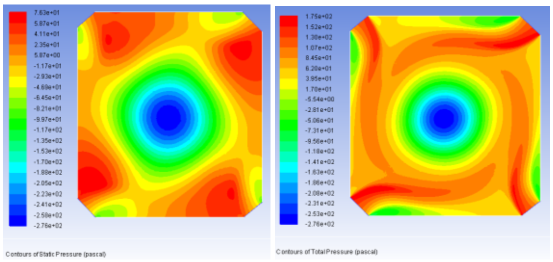


Figure 12. Contours showing static and total pressure for Burner angles (43^0, 39^0)

3.2.2 Burner Angle (33^0, 29^0)

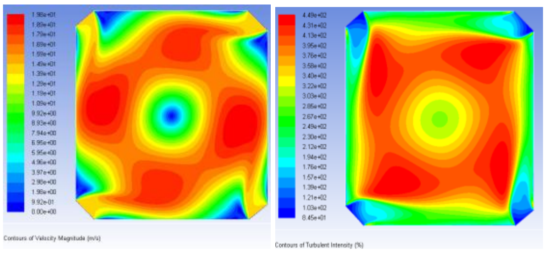


Figure 13. Contours of velocity magnitude and turbulence intensity for Burner angles (33^0, 29^0)

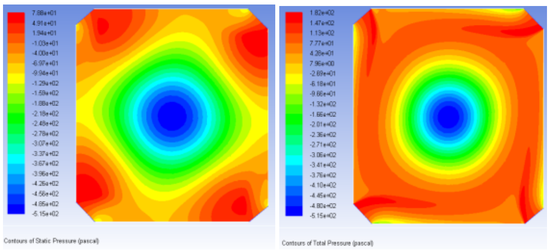


Figure 14. Contours showing static and total pressure for Burner angles (33^0, 29^0)

3.2.3 Burner Angle (39^0, 35^0)

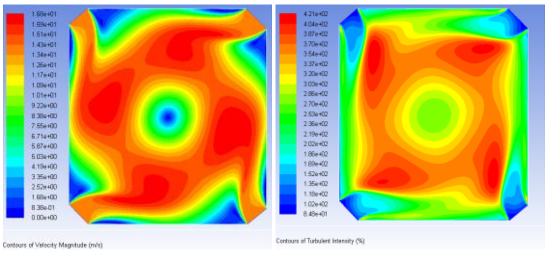


Figure 15. Contours of velocity magnitude and turbulence intensity for Burner angles (39^0, 35^0)

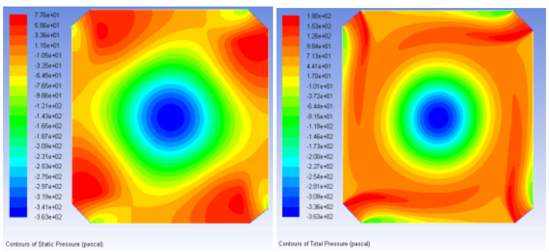


Figure 16. Contours showing static and total pressure for Burner angles (39^0, 35^0)

3.2.4 Burner Angle (46^0, 42^0)

The burner Angle is arranged to check the maximum movement of the burner for the given configuration.

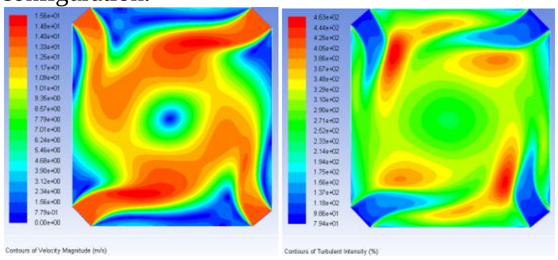


Figure 17. Contours of velocity magnitude and turbulence intensity for Burner angles (46^0, 42^0)

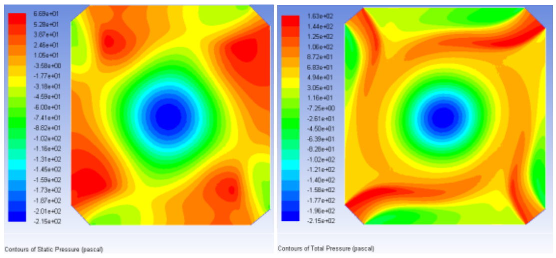


Figure 18. Contours showing static and total pressure for Burner angles (46^0, 42^0)

4. RESULTS AND DISCUSSION

The furnace burner is designed and analysis is performed for different key parameters like burner velocity, burner angle. The analysis of the above parameters is combined for the best combination of parameters with the objective function to maximize mixing efficiency in the burner which ultimately produces efficient combustion and reduces the losses in the mixing stage. The efficient combination of parameters produces cost saving design for better performance of overall plant.

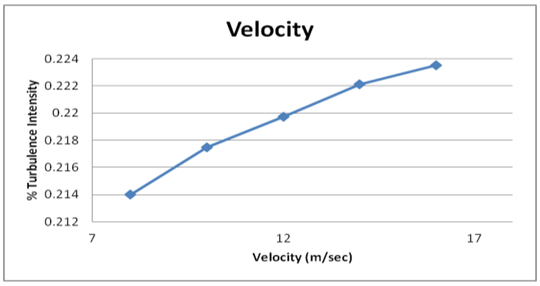
4.1 Effect of Burner Velocity

The turbulence intensity is checked for all the design variables for the velocity i.e. 10, 12, 14, and 16 m/sec. Out of which the results are compared with the Base case results i.e. 14m/sec.

Table 4: Turbulence intensity (%) for different velocity

Velocity(m/sec)	Avg. Turbulence Intensity	Max(Circle region)	Total	%
10	2.07	215.54	991	0.217
12	2.52	562	1206	0.219
14	2.96	315	1418	0.222
16	3.42	365	1633	0.223

As the velocity increases of the flue gases, the momentum is increases which also increasing the pressure loss inside the furnace geometry but the increment in pressure is from 185 Pa to 200 Pa which is < 15% but the increase in the turbulence intensity in the middle core is from 315% to 365% i.e. >20%.[8]



Therefore the higher velocity is selected as the optimum parameter for enhancing the flame stability but beyond this range there is no appreciable change in the turbulence. Optimum Velocity =16 m/sec is selected.

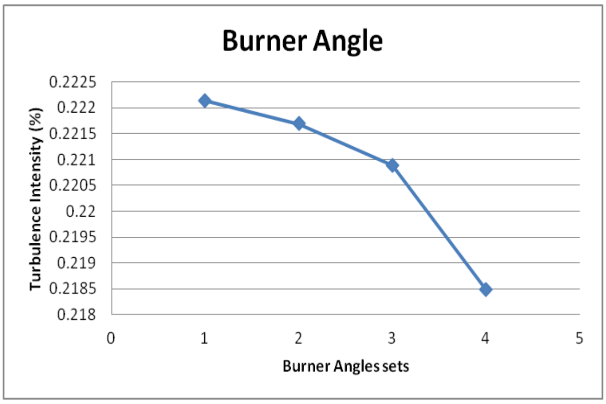
4.2Effect of Burner Angle

The turbulence intensity is checked for all the design variables for the Burner Angle i.e. (43, 39), (33, 29) (39, 35) and (46,42). Out of which the results are compared with the Base case results i.e. (43, 39).

Table 5: Turbulence intensity (%) for different burner angles

SET	Burner Angle	AvgTurb Intensity	Max(Circle region	Total	%
1	43_39	2.96	315	1418	0.222144
2	33_29	3.46	366.9	1655	0.221692
3	39_35	3.12	330	1494	0.220884
4	46_42	2.92	305	1396	0.218481

Increase or decrease in burner angle from the existing set up does not help in increasing in vortex strength as the change in burner angle is restricted due to the size of the furnace; the optimum vortex strength is achieved at set 1 i.e. existing set up which is kept fixed.



5. CONCLUSION

Based on the tangentially fired furnaces in power plant industry, it has been found that four corner burners situated widely used with pulverized coal fuel studied for maximum flame stability by considering the turbulence intensity.

The base design is studied for different design parameters with objective function of increasing the turbulence intensity which directly enhances the flame stability for proper mixing of fuel and air which leads to better combustion efficiency. The effect of different parameters are studied on the vortex strength formed at the middle of circle for tangentially fired boilers. The information presented here would beneficial for presenting in this area of research.

6. ACKNOWLEDGMENT

I would like to thanks Prof. V.H.Patil H.O.D, Mechanical Engineering Department, GCOE Jalgaon for his guidance about the selected topics.

REFRENCES

[1] SrdjanBelosevic, MiroslavSijercic, Simeon Oka, DraganTucakovic, Three-dimensional modeling of utility boiler pulverized coal Tangentially fired furnace, *International Journal of Heat and Mass Transfer 49 (2006) 3371–3378*

[2] Boshu He, Meiqian Chen, Qiumei Yu, Shumin Liu, Lijuan Fan, Shouguang Sun, Jinyuan Xu, Wei-Ping Pan, Numerical study of the optimum counter-flow mode of air jets in a large utility furnace, *Computers & Fluids 33 (2004) 1201–1223*

[3] E.Karampinis, N. Nikolopoulos, A. Nikolopoulos, P. Grammelis, E. Kakaras, Numerical investigation Greek lignite/cardoon co-firing in a tangentially fired furnace, *Applied Energy 97 (2012) 514–524*

[4] Boshu He, Meiqian Chen, Shumin Liu, Lijuan Fan, Jinyuan Xu, Wei-Ping Pan, Measured vorticity distributions in a model of tangentially fired furnace, *Experimental Thermal and Fluid Science 29 (2005) 537–554*

[5] Yuegui Zhou, Tongmo Xu, ShienHui, Mingchuan Zhang, Experimental and numerical study on the flow fields in upper furnace for large scale tangentially fired boilers, *Applied Thermal Engineering 29 (2009) 732–739*

[6] ChoengRyul Choi, Chang Nyung Kim, Numerical investigation on the flow, combustion and NOx emission

characteristics in a 500 MWe tangentially fired pulverized-coal boiler, *Fuel 88 (2009) 1720–1731*

[7] Audai Hussein Al-Abbasa, and Jamal Nasera, Computational fluid dynamic modelling of a 550 MW tangentially-fired furnace under different operating conditions, *Procedia Engineering 56 (2013) 387 – 392*

[8] Jamal Naser,Audai Hussein Al-Abbas, Jamal Naser, David Dodds, CFD modelling of air-fired and oxy-fuel combustion in a large-scale furnace at Loy Yang A brown coal power station, *Fuel 102 (2012) 646–665*