CFD Investigation Of The Effect Of Tilt Angles To Flow And Combustion In A 120 Mw Gas-Fired Full Scale Industrial Boiler

Wisam Saad Azeez, Hasril Hasini, Zamri Yusoff

Abstract: Extensive Computational Fluid Dynamic simulations on combustion process in a 120 MW natural gas-fired industrial boiler have been performed. The boiler is of tangential system by which, natural gas and air are fired into the furnace from its four corners at three different elevation levels. Fuel burners and air nozzles can be tilted in the range of ± 30º.The simulation results show good prediction of temperature profile and distribution as compared to practical observation. The flow in the furnace is highly swirling with intense mixing of natural gas and air. The turbulence level is extremely high and combustion gas flows in helical shape in anti-clockwise direction towards the furnace exit. The results revealed the importance of the effect of heat transfer to the water wall, as the temperature distribution and flow pattern is significantly difference with heat transfer and without heat transfer cases. Temperature distribution prior to the entry to the re-heater elevation is non-uniform with lower velocity at vortex core, where the fireball is located. The average temperature at re-heater level reduces when the burner is negatively tilted up to -10º. A slight change in firing angle will interrupt the temperature and flow contour at furnace-re-heater. However, this is only true for air nozzles where the overall flow pattern inside the furnace is dominated by these sources..

Keywords: CFD , Natural Gas-Fired , 120 MW Gas-Fired , Turbulent flow , nozzles, heat transfer, firing angle.

1 INTRODUCTION

Over the past 20 years, digital technologies have gained the reputation of being an effective tool in identifying and solving problems related to combustion. This resulted in combustion models which are able to predict the behavior of complex flows, offering great potential for use in improving the performance of energy conversion systems. Numerical calculations generally require the application of Computational Fluid Dynamics (CFD). The use of CFD codes for modeling of combustion, heat and fluid flow is a useful tool to predict boilers performance in the scientific community and industry [1]. Many applications of CFD in modeling the flow and temperature distribution of combustion process in a boiler have been published in recent years. He et al. [2] utilized the CFD package FLUENT to numerically diagnose the metal surface overheating issues of the re-heater pendants that exist in the full scale No. 3 boiler of Dagang Power Station, Tianjing, China. This numerical study included nine cases and the first case baseline case with no reformation. Anand and Jenny [3] developed a study of a unified PDF modeling framework and a new hybrid solution algorithm to simulate turbulent evaporating sprays by using Eulerian-Lagrangian approach. Two ways coupling between the droplet and gas phases was implemented and an infinite thermal conductivity (ITC) evaporation sub-model was used. They concluded that to enhance the computational efficiency, a local particle time-stepping algorithm needs to be implemented and a particle time averaging technique to be employed to reduce statistical and bias errors.

- *Wisam Saad Azeez, Hasril Hasini, Zamri Yusoff*
- *Email: wisam.saad@ymail.com*
- *Department of Mechanical Engineering,Universiti Tenaga Nasional , Jalan Ikram-Uniten , 43000 Kajang, Selangor, Malaysia*

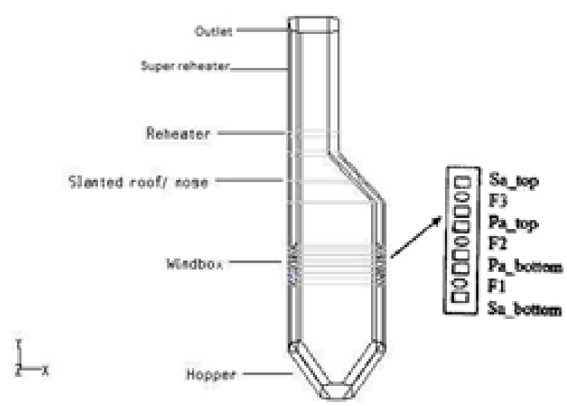
The PDF algorithm was validated with the experimental data of a turbulent evaporating Iso-propyl alcohol spray. Overall, reasonable agreement could be observed. Choi and Kim [4] instigated the use Air staging combustion technology (over fire air -OFA) and its effects to reduce NOx emission in 500 MWe tangentially fired pulverized-coal boiler using CFD . The present simulation was used the non-premixed model, DO (discrete ordinates) radiation model, the SIMPLE algorithm and a mixture-fraction equation. The RNG k– ε model used to consider the effect of swirling turbulent flow in the furnace as well as the weighted-sum-of-gray- gases model (WSGGM) for Absorption coefficients. Experimental and CFD investigation of gas phase freeboard combustion was studied by Andersen et al. [5], Reliable and accurate modeling capabilities for combustion systems are valuable tools for optimization of the combustion process. This work concerns primary precautions for reducing NO emissions, thereby abating the detrimental effects known as “acid rain”, and minimizing cost for flue gas treatment. Nikolopoulos et al. [6] studied effects of the oxy-fuel technology in 330MWe tangentially fired pulverized-coal boiler used the commercial code CFD. In this simulation, The SIMPLE algorithm, standard k– ε turbulence model, standard wall functions, and finite rate/eddy dissipation model for reaction occurring during combustion were adopted. Yin et al. [7] study the role of combustion chemistry and radiation heat transfer in oxy-fuel combustion modeling, a computational fluid dynamics (CFD) modeling performed for two different oxy-fuel furnaces. The CFD simulation results are also compared against the available in-flame measurement data. Different combustion mechanisms were also implemented and compared in the CFD simulations, from which significant difference in the predicted flame temperature and species was observed. Zhang [8] represented Numerical modeling of biomass combustion in a stoker boiler, Biomass fuel is considered a promising substitute for traditional fossil fuels. Combining a standard CFD model describing the turbulent dynamics and combustion with several sub-models specifically developed for this study to model the fuel bed, fuel particle movement, and fuel gasification. To verify and baseline these sub-models, a series of experiments are performed, including a temperature measurement campaign for coal combustion in the boiler, a chemical lab analysis of oat hull chemical characteristics, an experiment measuring oat hull particles' physical properties, and a high-heating-rate gasification test of oat hulls. Karampinis et al. [9] studied the combustion processes and simulation difference of co-firing ratios and biomass particle sizes (biomass and lignite) in 300 MWe pulverized-fuel, tangentially fired boiler. Using CFD (fluent).This model accounted for the non-spherical form of the biomass particle which has effects on drag coefficient, combustion efficiency and its devolatilization. The SIMPLE algorithm, standard k– ε turbulence model, standard wall functions, and finite rate/eddy dissipation model for reaction occurring during combustion were used in the present simulation. Al-Abbas et al. [11] [12] performed CFD simulation for the combustion of the brown coal in a large-scale tangentially-fired furnace (550 MW) under different operating conditions. The mathematical models of coal combustion with the appropriate kinetic parameters were written and incorporated to the code as user defined functions. The numerical results showed that the turned off burners in the opposite direction are better than those in the adjacent direction under full load operation, and this can help to provide an improved distribution of the flames in the furnace. A slight increase in the carbon monoxide concentrations was evident under the high and low load operations. Most of the works are mainly concentrated on boiler, which used natural gas as the main source of fuel [1, 2, 13]. Yin et al.[7] investigated the performance of a 609MW tangentially fired pulverized-coal boiler, with emphasized on formation mechanism of gas flow deviation and uneven wall temperature in crossover pass and on NOx emission. It was showed that the residual airflow swirling at the furnace exit that essentially caused the gas flow deviation and therefore, the uneven wall temperature in the crossover pass in tangentially fired boilers. Zhang et al. [14] studied numerically the characteristics of the combustion, stoichiometry, temperature, species concentration and NOx emissions in a 200 MWe tangentially-fired pulverized-coal boiler. It was worth highlighting that the the horizontal bias combustion HBC leaded to a higher stoichiometry in the area close to the furnace wall when combined with the OFA, which would reduce the risk of slagging when the boiler adopted the low NOx technology. Chen and Liu [15] Numerical investigation on the flow, combustion and NOx emission characteristics in a 10 MW Premixed Gas Burner was studied , The characteristics of the combustion temperature, flow velocity, CO distribution and NOx emissions of a 10 MW gas burner at different primary to secondary air ratios are numerically studied using Computational Fluid Dynamics CFD software Fluent. Belosevic et al. [16] studied numerically the effects of different operation conditions such as turn off the burners, change air/fuel ratio and boiler load within utility scale pulverized coal-fired furnace. Hbib et al. [17] investigated numerically the problem of NOx pollution using a model furnace of an industrial boiler utilizing fuel gas. The results of this simulation could help in improving the predictive emission monitoring techniques, more effective instrumentation and monitoring of the combustion conditions and better control of the boiler. Only a limited number of works on natural gas and oil fired boiler are available in the literature [18, 19]. The present study examines 3D turbulent flow convective and combustion process in a 120 Mw Gas-Fired Full Scale Industrial Boiler. This investigation covers fuel burners and air nozzles angle in the range of +30º to -30, four corners at three different elevation levels. Results of interests such as temperature distribution, flow contour at furnace-re-heater, heat transfer rate, in tangential furnace are reported to illustrate the effect of flow pattern and combustion process on these parameters.

2 Mathematical Modeling

2.1 Physical Model and Assumptions

The schematic diagram of the investigation domain is shown in Fig.1(a). All dimensions required to create the model is obtained from the plant boiler manual. The furnace is divided into y-cutting planes throughout the elevation of the furnace. The location of three fuel burners and four air nozzles that located in each corner as shown in figure 1(b). A detailed location of the planes is listed in Table 1. Figure 2 and 3 show the mesh generation within the furnace and at a y-plane. Intensive mesh is applied at the windbox region, nose and reheater section based on flow properties.

a)



b)

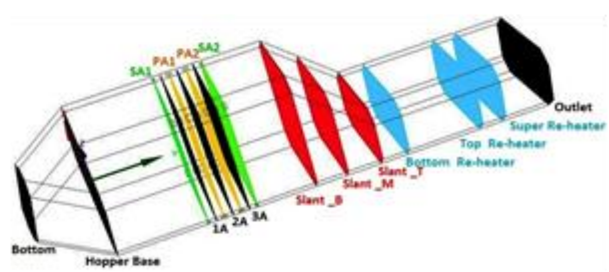


Fig.1. *Schematic Diagram of the furnace with air and fuel arrangement: a) investigation domain, b) 3D view for location of corresponding y-plane in the furnace.*

Table 1.1 *Location of the y-planes*

Y-planes		Elevation(m)
Hopper		
1.	Bottom	-3.84
2.	Hopper Base	0
Wind Box		
3.	Bottom Secondary Air Inlet (SA1)	5.6
4.	Bottom Fuel Inlet (1A)	6.093
5.	Bottom Primary Air Inlet (PA1)	6.59
6.	Middle Fuel Inlet (2A)	7.12
7.	Top Primary Air Inlet (PA2)	7.65
8.	Top Fuel Inlet (3A)	8.145
9.	Top Secondary Air Inlet (SA2)	8.63
Slant Region		
10.	Slant Bottom (Slant _B)	12.39
11.	Slant Middle (Slant _M)	14.35
12.	Slant top (Slant _T)	16.55
Re-heater and Super Reheater		
13.	Bottom Re-heater	18.25
14.	Top Re-heater	23
15.	Super Re-heater	24.5
16.	Outlet	27.6066

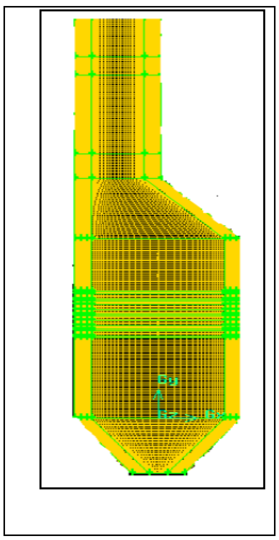


Figure 2 *Structured mesh of the boiler furnace*

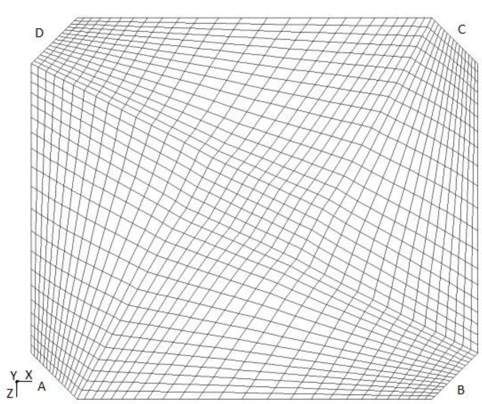


Figure 3 *Grid scheme at the furnace cross section*

2.2 Governing Equations

By considering the geometry and physical problem as shown in Fig.1, the k-ε standard turbulence model [20] is used to simulate the turbulent flow and combustion process. In this model, the turbulent viscosity is expressed as:

$$v_t = \frac{C_{\mu} k}{\varepsilon} \tag{1}$$

The transport equation for k and ε are:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho u\ k) = \rho P - \rho\varepsilon + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] \tag{2}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_j}(\rho u_j \varepsilon) = C_{\varepsilon_1}\frac{\rho P\varepsilon}{k} - C_{\varepsilon_2}\frac{\rho\varepsilon^2}{k} + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] \tag{3}$$

The production term P is defined as:

$$P = v_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_m}{\partial x_m}\delta_{ij}\right)\frac{\partial u_i}{\partial x_j} - \frac{2}{3}k\frac{\partial u_m}{\partial x_m} \tag{4}$$

In the above equations, σ_k , σ_ε and are effective Prandtl numbers for turbulent kinetic energy and rate of dissipation, respectively; $C_{\varepsilon 1}$and $C_{\varepsilon 2}$ are constants. The empirical constants for the turbulence model are arrived by comprehensive data fitting for a wide range of turbulent flow [21]:

$C\mu = 0.09$, $C\varepsilon 1 = 1.44$, $C\varepsilon 2 = 1.92$, $\sigma k = 1.0$, $\sigma\varepsilon = 1.3$

2.3 Boundary Conditions

The boundary condition for air and fuel for simulation of combustion natural gas is listed in table 2. Simulation is done under design condition (full load operation condition). The fuel and air properties are obtained from the boiler operation. For the combustion with heat transfer case, the wall is assumed to be absorbing heat at a rate of 10,000 W/m2. The flow rate of air at primary nozzle is 9.8 kg/s and secondary air nozzle 4.9 kg/s. The mass flow inlet of fuel is set 0.57 kg/s at each burner. The air temperature is set to 300 K and fuel temperature 300 K. The value of turbulence kinetic energy and dissipation rate (k and ε) are 2.0 m2s-2 and 0.1 m for the air inlet and 1.0 m2s-2 and 0.5 m for fuel.

Table 2 *Boundary condition for gas simulation*

Fuel	
Parameters	Combustion for Full load and heat transfer toWater wall
Mass flow rate (kg/s) Temperature (K) k, fuel (m2/s2) epsilon, fuel (m)	0.57 300 1 0.5
Air	
Air mass, primary (kg/s) Air mass, secondary (kg/s) Air Temperature (K) k, air m2/s2) epsilon, air (m)	9.8 4.9 300 2 0.1
Tilt angle	0

2.4 Numerical Computation

Numerical analysis using commercial software ANASYS, Fluent 14.5 is performed in order to understand the flow pattern and combustion process in the tangential furnace. The governing equations (1) – (4) are a set of convection equation. It is based on the control volume method, SIMPLEC algorithm of Versteeg and Malalasekera [21] is used to deal with the problem of velocity and combustion process. Second – order upwind scheme and structure uniform grid system are employed to discretize the main governing equation as shown in Fig.2.

2.5 Code Validations and Grid Testing

This study involves the combustion process with swirly flow from the beginning of the process, and heat transfer between the gas and furnace walls. The gas from the combustion process will be experiencing few transitions in geometry from fuel or nozzle entering the furnace and flowing through the nose region. It is also important to confirm whether or not the CFD code FLUENT is able to simulate data such as heat transfer, fluid flow through a non-uniform region, and wall effect for boilers. Therefore, test cases are used to test the suitability of the CFD code. Figure 4 shows the comparison between the reported experimental values and two predicted values obtained from FLUENT. The agreement between the two values is excellent Davis and Gessner, [22], despite the fact that the pressure recovery downstream transition being underestimated. The RNG *k-ε* model resulted in better values compared to the produced in the standard *k-ε* model.

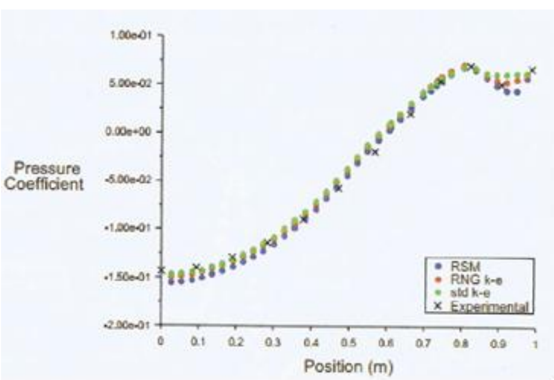


Fig.4. *Comparison of wall pressure coefficient.*

3 Results and Discussion

3.1 Temperature Distribution

Figure 5 illustrates the temperature distribution on two planes located at the vertical center plane of the furnace in direction of y- axis. These planes are located diagonally and in contact with the fuel and air nozzles, thus the gas temperature distribution can be clearly analysis and study at full load operation condition. The fuel and air that introduce through the nozzles in the windbox region do not combust instantaneously as they enter into the furnace. This can be shown by the temperature gradually increases as it flows toward the center of furnace. The lower and uniform temperature region is located at the area below the combustion region near the hopper region. Then, gas temperature increases rapidly with increase elevation specially in the wind box region (combustion region) where the combustion more active. Combustion process occurs extensively within this region and results quickly increase in the temperature of the gas.

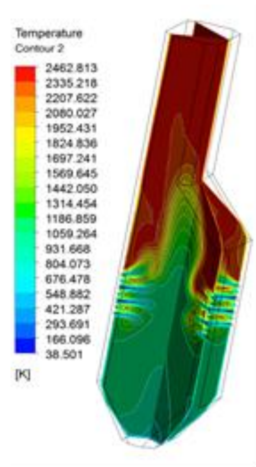
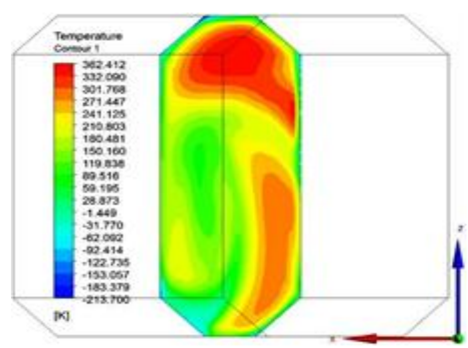


Figure 5 *Temperature contour at the diagonal planes*

The temperature distribution for y-cutting planes within the furnace is also analysis and they are shown clearly in figure 6. Uniform temperature scale is applied for natural gas to assure the analysis is achieved consistently. Temperature distribution at the elevations in this region becoming non-uniform with scatted region of high and low temperature due to combustion process which is resulted from mixing of fuel and air in the windbox region.

a – Bottom y = -3.84 m



b – Hopper y = 0

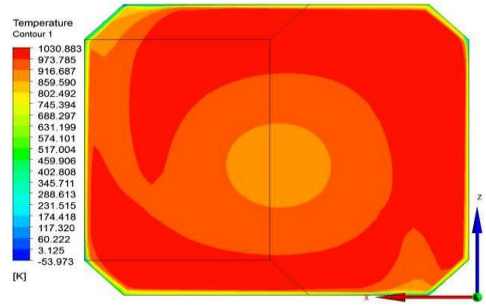


Figure 6 *Elevation of temperature distribution at y planes for combustion with heat transfer to water wall.*

3. 2 Velocity Distribution

Figure 7 shows the general flow pattern where natural gas and air undergo combustion process inside the boiler furnace. The flow is highly swirling in anti-clockwise direction. The swirling pattern remains until the furnace exit. Mix of fuel and air is intense in the windbox region. Intense velocity appears at all the corners of the furnace where the fuel and air are injected. The intensity of the combustion gas reduces slightly as the gas flows upward towards the slanted region. However, at the entrance of slanted region, the flow intensity increases where most of it concentrated at the left wall. Beyond the slant region the gas moves towards the right and front wall, where the flow deviation is corrected.

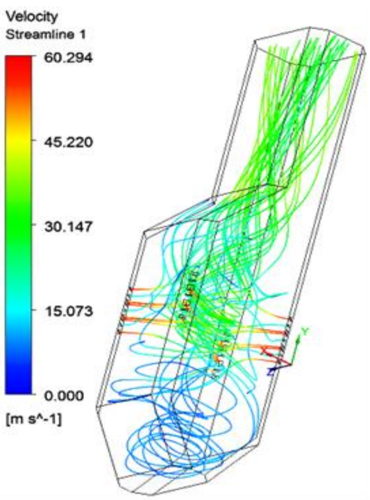


***Figure 7** Flow pattern inside the furnace simulation on the effect of tilt angle 0 degree.*

Figure 8 illustrate the velocity distribution in term of x-,y-,and z-component at several y planes. At first, in x direction dominates the flow with sinusoidal and symmetrical sharp while the y-velocity component tended to have ‘w’ sharp distribution. The y- and z- velocity distribution are plotted at a lower magnitude. Since the combustion has just begun, the momentum of the flow was only determined by the momentum brought by fuel and air. This can be notice with greater x-velocity at bottom fuel plane and top secondary air, However, as elevation increase (the hotter combustion gas moves upward) velocity in y direction dominates the flow. At bottom slant plane, the y-velocity has bigger velocity magnitude than x-velocity. This is due to additional momentum brought by the flow from lower planes, flowing upwards thus increasing the magnitude of axial velocity. Besides that, as the combustion started, the combustion gas temperature lowers the density of gas, which in turn flow upwards toward the exist at a faster velocity. Started from bottom slant plane, it can be noticed an existence of velocity deviation, where greater velocity is predicted at left wall. Reducing in the left cross-sectional area at the nose region has increases the velocity of the flow since the pressure at the area is increases. This condition produces velocity deviation between the right and left wall. Velocity in z-component remains small all over the furnace.

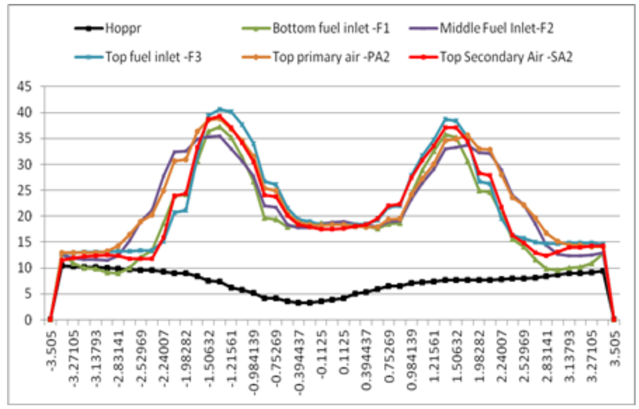


***Figure 8** Velocity distribution at hopper plane to the top secondary air plane (Left to right furnace side)*

.

3. 3 Effects of Burner Tilt Angles

Figure 9 shows the temperature profile for various burner tilt configurations along the furnace height. The analyzed data were along the y-axis, parallel to the furnace height from the bottom elevation to the furnace exit. A clear trend could be observed, in which a lower temperature was found at the furnace bottom, and high temperature reached 2250 K at the combustion region. Rapid gas temperature increased at the combustion region as combustion took place. As the combustion gas flows passed the re-heater, a slight temperature increase occurred at approximately 2260 K. This value remained constant with a slight decrease as the gas flows exited the furnace. At the furnace bottom, +10° burner tilt displayed the highest temperature, because the direction of the burner allows combustion to occur at the lower furnace elevation, thereby increasing the temperature at this level. A similar profile is obtained from other tilt configurations, i.e+20° and +30°. However, a positive burner tilt configuration displayed an opposite trend. A -10° burner tilt yielded the lowest temperature at the furnace bottom, followed by 0 and -20° and -30° burner tilts. As the combustion gas flowed through the slanted roof, the temperature profile tended to remain constant because of the reduction in the flow area. This trend is true for all burner tilt configurations.

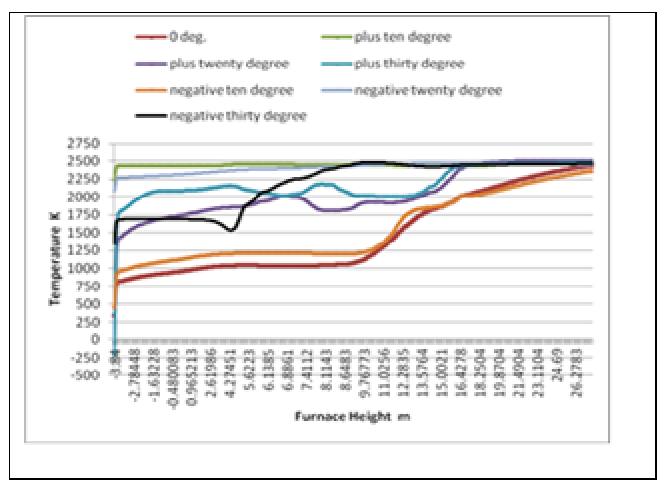


***Figure 9** Temperature profiles along furnace height for various tilts at furnace center*

4. Conclusions

Numerical investigations to study the fluid flow and combustion process in 3D- a tangential furnace of 120 MW natural gas-fired industrial boilers were carried out using CFD. The effects of various parameters and configurations on the thermal and hydraulic behavior inside the boiler were examined. Temperature profiles and distributions, velocity vectors, and combustion gas flow patterns and the prediction process are analyzed and discussed. Results show that the temperature profile along the furnace height exhibited a lower temperature at the bottom furnace, a drastic increase at the combustion region, and remained approximately constant at the upper furnace.

- The +10° burner tilt displayed the highest average temperature at the re-heater entrance among all tilt configurations. The configuration likewise provides good symmetrical temperature distribution at the furnace re-heater.
- The -10° burner tilt indicated the lowest average temperature at the re-heater entrance among all tilt configurations.
- In general, increasing the burner tilt angle in a positive and negative direction increases the average temperature at the furnace re-heater.
- All negative burner tilts displayed reasonably good symmetrical temperature distribution at the entrance of the furnace re-heater.

References

[1] Hasini, H.M., N. CFD Analysis of Flow and Combustion Process in an Oil-fired Boiler at Design Condition. in Proceeding of the Eleventh Asian Congress of Fluids Mechanics, K. 2006. Kuala Lumpur , Malaysia.

[2] He, B., Zhu,L. , J. Wang, S. Liu, B. Liu, Y. Cui, L. Wang , Wei G., Computational fluid dynamics based retrofits to reheater panel overheating of No. 3 boiler of Dagang Power Plant. Computers & fluids, 2007. 36(2): p. 435-444.

[3] Anand, G., Stochastic modeling of turbulent evaporating sprays. Diss., Eidgenössische Technische Hochschule ETH Zürich, Nr. 19186, 2010.

[4] Choi, C.R. and C.N. Kim, Numerical investigation on the flow, combustion and NOx emission characteristics in a 500MW e tangentially fired pulverized-coal boiler. Fuel, 2009. 88(9): p. 1720-1731.

[5] Andersen, J, P. Glarborg, P. A. Jensen Lovmand Hvid , S. Experimental and CFD investigation of gas phase freeboard combustion. 2009, DONG Energy A/SDONG Energy A/S.

[6] Nikolopoulos, N., N., A., E. Karampinis, P. Grammelis and E. Kakaras Numerical investigation of the oxy-fuel combustion in large scale boilers adopting the ECO-Scrub technology. Fuel, 2011. 90(1): p.214-198 .

[7] Yin, C., L.A. Rosendahl, and S.K. Kær, Chemistry and radiation in oxy-fuel combustion: a computational fluid dynamics modeling study. Fuel, 2011. 90(7): p. 2519-2529.

[8] Zhang, X., Numerical modeling of biomass combustion in a stoker boiler.2011 .

[9] Karampinis, E., N. Nikolopoulos, A. Nikolopoulos, P. Grammelis and E. Kakaras., Numerical investigation Greek lignite/cardoon co-firing in a tangentially fired furnace. Applied Energy, 2012. 97: p. 514-524.

[10] Chen, L., S.Z. Yong, and A.F. Ghoniem, Oxy-fuel combustion of pulverized coal: Characterization, fundamentals, stabilization and CFD modeling. Progress in Energy and Combustion Science, 2012. 38(2): p. 156-214.

[11] Al-Abbas, A.H., J. Naser, and D. Dodds, CFD modelling of air-fired and oxy-fuel combustion in a large-scale furnace at Loy Yang A brown coal power station. Fuel, 2012. 102: p. 646-665.

[12] Al-Abbas, A.H., J. Naser, and E.K. Hussein, Numerical simulation of brown coal combustion in a 550MW tangentially-fired furnace under different operating conditions. Fuel, 2013. 107: p. 688-698.

[13] Fan, J., et al., Computational modeling of pulverized coal combustion processes in tangentially fired furnaces. Chemical Engineering Journal, 2001. 81(1): p. 261-269.

[14] Zhang, X., X., J. Zhou, S. Sun, R. Sun , Qin, M., Numerical investigation of low NOx combustion strategies in tangentially-fired coal boilers. Fuel, 2015. 142: p. 215-221.

[15] Chen, W. and G. Liu, Numerical Investigation on the Flow, Combustion and NOx Emission Characteristics in a 10 MW Premixed Gas Burner. The Open Fuels & Energy Science Journal 2015. 8 p. 1-13.

[16] Belosevic, S., S., M. Sijercic, D. Tucakovic , Crnomarkovic, N.., A numerical study of a utility boiler tangentially-fired furnace under different operating conditions. Fuel, 2008. 87(15): p. 3331-3338.

[17] Habib, M., M. Elshafei, and M. Dajani, Influence of combustion parameters on NO x production in an industrial boiler. Computers & Fluids, 2008. 37(1): p. 12-23.

[18] Coelho, P. and M. Carvalho, Evaluation of a three-dimensional mathematical model of a power station boiler. Journal of engineering for gas turbines and power, 1996. 118(4): p. 8.895-87

[19] Hasini, H., Yusoff, M. Z., Ahmad, Z., Majid, K. A., Ramli, M. R. Modeling of Combustion in a 120 MW Gas Fired Full Scale Industrial Boiler: the Effect of Burner Tilt Angle. in Proceeding of the 3rd International Conference on Numerical Analysis in En. 2003.

[20] Launder, B.E. and D. Spalding, The numerical

computation of turbulent flows. Computer methods in applied mechanics and engineering, 1974. 3(2): p. 269-289.

[21] Versteeg, H.K. and W. Malalasekera, An introduction to computational fluid dynamics: the finite volume method. 2007: Pearson Education.

[22] Davis, D.O. and F. Gessner, Experimental investigation of turbulent flow through a circular-to-rectangular transition duct. AIAA journal, 1992. 30(2): p. 367-375