Journal of Advanced Science and Engineering Research Vol 3, No 3 September (2013) 286-293

Simulation Of 120mw Industrial Boiler At Design Condition

Article Info

Received: 7/6/2013
Accepted:25/8/ 2013
Published online: 1/9/2013

S. Mohammad [1, a], Esam.Hamd [2,b] , H.Hasini [3,c] and Z.Yusoff [4,d]

[1,2,3, 4] Department of Mechanical Engineering, College of Engineering,University Tenaga Nasional, 43000, Kajang, Selangor, MALAYSIA

[a]sinan_taha83@yahoo.com, [b] esam0070@yahoo.com, [c] Hasril@uniten.edu.my, [d]zamri@uniten.edu.my

ISSN 2231-8844

Abstract

This paper presents an investigation of flow and combustion process in an oil fired, full scale furnace at design condition. The modeling is done using RNG turbulence model based commercial CFD-code FULENT. The simulations are executed in two stages with the effect of heat transfer to the furnace wall for full load and reduced load condition. The results of the simulations show that flow inside the furnace is highly swirling with intense combustion and comparing between full load and reduced load conditions, the temperature magnitude is not much different. On top of that, the predicted flow pattern and temperature distribution inside the furnace show reasonable qualitative agreement with practical observation.

Key words: Computational Fluid Dynamics (CFD), Boiler, power Plant

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. Make the simulation with a comprehensive. Resulted in combustion models and symbols of material to a large extent a new Insight into 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 among performance of the scientific community and industry (Hasini and Maruf, 2006). Many applications of CFD in modeling the flow and temperature distribution of combustion process in a boiler have been published in recent years. Some examples on the application of CFD method in industrial boiler. The boiler under investigation is a 120 MW oil-fired boiler of an electricity generating plant. However, most of the works are mainly concentrated on boiler, which used coal as the main source of fuel (Hasini and Maruf,2006; Fan et al., 2001; Lee et al., 2001) . Only a limited number of works on natural gas and oil fired boiler are available in the literature (Coelho.and Carvalho (1996); Yusoff et al., (2002).This research aims to gets a better understanding on the performance of the combustion using liquid fuel as a guide to any similar type operation.

2 MODEL DESCRIPTIONS

2.1 Boiler Description

The boiler under investigation is a tangential fired boiler, with dual firing system. The boiler is designed for a main steam output of 378 t/hr (108 kg/s) at a pressure of 128 bar and a temperature of 540 C. The boiler has across sectional area of 7010mm (width) 8230mm (length) 27.6 (height).The height of each burner elevation is given in Table 1. All the burners and nozzles are fired from the furnace’s 4 corners A, B, C and D, and are directed to an imaginary circle at the centre of the furnace. It is design to create and imaginary fireball, which swirls in an anticlockwise direction at the furnace centre. The illustration of the firing system is shown in Figure 1. The firing angle is fixed, but the tilt angle is adjustable depending on the load demand and average temperature during operation. In the boiler system, combustion gas flows pass the reheater and several panels of superheater before exiting to the atmosphere via a chimney. The reheater section in the boiler is critical since any variation of more than 20C of temperature distribution between the left and right side of the boiler will result in the alarm being activated, which in turn activate the emergency attemperations by spraying water at the outlet header. This is undesirable since it causes extra thermodynamics loss and hence reduces the overall efficiency of the plant. There are a total of 12 oil burners and 24 air nozzles located at all the corners on 3 different elevations.

Table 1: Elevations level height

Elevation	Height above furnace (m)
Bottom	-3.840
Hopper base	0.000
Bottom Fuel, F1	6.090
Tp Primary Air, PA2	7.650
Top Secondary Air, SA2	8.630
Bottom Slant, Slant_b	12.390
Reheater	18.250

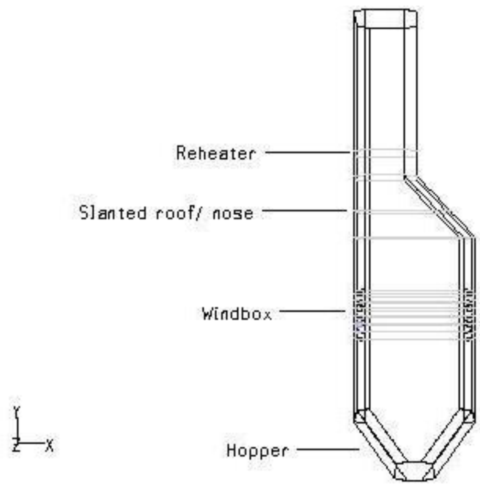


Figure1: Front view of furnace with air and fuel arrangement

2.2 Methodology

The RNG k-ε model performs well in predicting flows with swirling flow within the calculated region. In this paper, we try to use RNG k-ε model [12] as an alternative substitution for standard k-ε model. Generally, k and ε equations are follows:

$$u_j \frac{\partial k}{\partial x_j} = P_k - \varepsilon + \frac{\partial}{\partial x_j}\left[\left(v + \frac{vT}{\sigma T}\right)\frac{\partial k}{\partial x_j}\right] \qquad \text{Eq.1}$$

$$u_j \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon 1}\frac{\varepsilon}{k} P_k - C_{\varepsilon 2}\frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j}\left[\left(v + \frac{vT}{\sigma T}\right)\frac{\partial k}{\partial x_j}\right] \qquad \text{Eq.2}$$

Where

$$P_k = 2vT\overline{S_{ij}S_{ij}}\ ,\ \overline{S_{ij}} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right), vT = C_\mu \frac{k^2}{\varepsilon}\ ,\ k = \frac{\overline{u_i u_i}}{2}, \varepsilon = v\,\overline{\frac{\partial u_i}{\partial x_j}}\ \overline{\frac{\partial u_i}{\partial x_j}} \qquad \text{Eq.3}$$

The RNG *k*-ε model can give better results for swirling flow and sharp change flow within calculated regions.

2.3 Meshing

Good mesh generation is vital in any combustion simulation using CFD in order to get a reliable outcome, at the same time reducing the running time for the simulation. In this analysis, structured mesh is created using the GAMBIT mesh generation module. Fig 2 illustrates the mesh of the boiler furnace, which consists of 413,000 numbers of cells and 436,896 nodes. High mesh concentration is built at the combustion region where air and fuel mixed together and combust. The cross-sectional view of the structured mesh is shown in Fig 3. Visually, the number of cells is adequate to resolve the flow field.

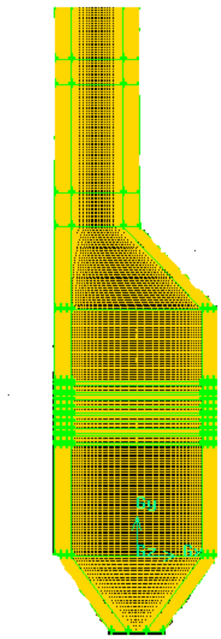


Figure 2: Structured mesh of the boiler furnace

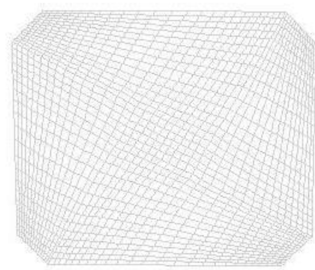


Figure 3: Grid scheme at cross-section

2.4 Boundary condition

Simulation is done at design case full load and reduced load condition .Table 2 summaries the boundary conditions for air and fuel for simulation. Fuel and air properties at design condition are obtained from the boiler operators. For the combustion with heat transfer case, the walls are assumed to be absorbing heat at a rate of 20,000 W/m2. This value of heat transfer rate was found to be adequate to bring down the temperature to approximately 1400° C prior to entry to the reheater. The radiative heat transfer equation was solved using P1 model and the wall temperature and emissivity were taken as 800 and 0.7 respectively.

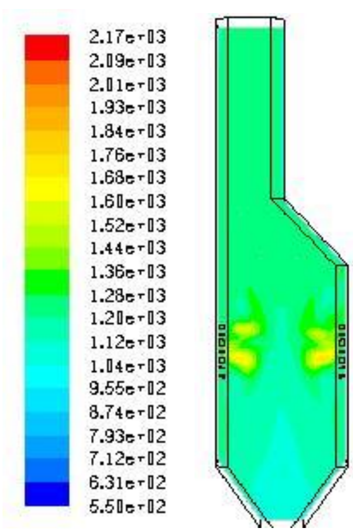
Table2: Boundary condition for Oil simulation

Fuel		
Parameters	Full load	Reduced load
Mass flow rate (kg/s)	0.66	0.33
Temperature (K)	350	350
Internal emissivity, ε(m)	0.7	0.7
Primary and Secondary air		
Parameters	Full load	Reduced load
Primary air mass flow rate (kg/s)	9.8	4.9
Secondary air mass flow rate (kg/s)	4.9	2.45
Temperature (K)	1273	1273
Internal emissivity, ε(m)	0.7	0.7
Turbulence intensity (%)	10	10
Primary hydraulic diameter (m)	0.3767	0.3767
Secondary hydraulic diameter (m)	0.2464	0.2464

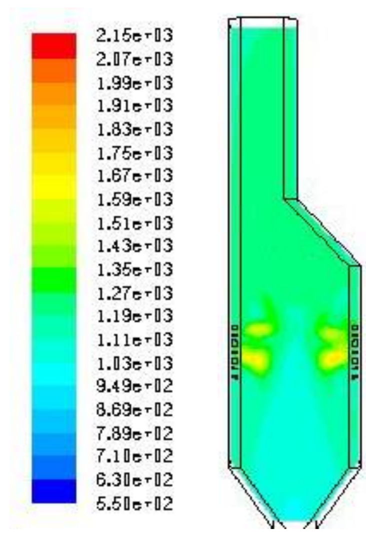
3 Results and Discussions

3.1 Temperature Distribution

Fig 4 illustrates the temperature distribution throughout the furnace for two cases .Appears difference in the temperatures where the figure shows that the temperature be higher when using RNG k-ε turbulence model . The planes are located diagonally and in contact with fuel and air nozzles. It can be seen that fuel and air do not combust instantaneously as it enters the furnace. This can be shown by the temperature gradually increases as it flows towards the center of the furnace. Low temperature region is located at the hopper region. As elevation increases the gas temperature increase as well, especially at the windbox region where the combustion process occurs. High temperature distribution is found in all cases at center of furnace. The designated angles of fuel and air entering the furnace causes the gas to be concentrated at the center of the furnace thus creating higher temperature. However, the distribution observed as ring-type in all cases. The ring-type distribution can be clearly seen with the contour plots at y-plane. Once the gas flows beyond the windbox region. The temperature of the gas decreases as it flows towards the slanted region and further decreases at the reheater region.



(a) Full load RNG k-ε turbulence



(b) Reduced load RNG k-ε turbulence

Figure 4: Temperature contour at diagonal planes

The temperature distribution within the furnace is also analyzed at several y-planes as shown in Fig 5 until Fig 8. Temperature scale is used for each case to ensure the analysis is done consistently. Looking at the planes located at the windbox region, high and non-uniform temperature distribution is found at the center of plane. Since the injection of fuel and air is directed at the center of the furnace, the gas is concentrated at the center of the furnace. Lower temperature reading is found near the walls since the heat transferred to the walls. The ring-type temperature distribution is seen in Fig 5 higher temperature gas surrounding a lower temperature gas. Evaluating between effect of fuel and air, it can be said that injection of air influences the combustion process more as the temperature at secondary plane is higher than the F1 plane as also observed that the temperature be high on the left wall when using RNG k-ε turbulence model at SA as shown in Fig 6. Beyond the windbox region, the temperature distribution is found more uniform and at lower temperature .Heat transferred to the walls making the temperature to go down besides no more combustion process takes place at this stage. The phenomenon remains towards the reheated section. Nevertheless, the temperature deviation is corrected slightly as the area remains constant, between the top slanted region to reheater region as shown Fig 7 and Fig 8.

(a) Full load RNG k-ε turbulence (b) Reduced load RNG k-ε turbulence

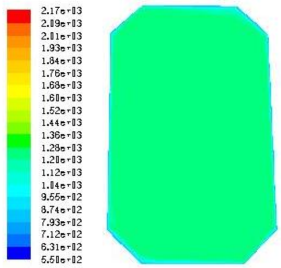
Figure 5: Temperature contour at F1 plane

(a) Full load RNG k-ε turbulence (b) Reduced load RNG k-ε turbulence

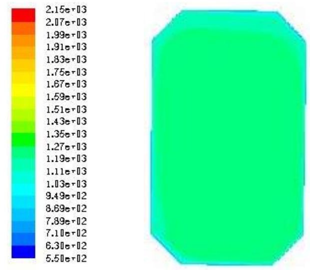
Figure 6: Temperature contour at SA plane

(a) Full load RNG k-ε turbulence (b) Reduced load RNG k-ε turbulence

Figure 7: Temperature contour at Slant_b plane



(a) Full load RNG k-ε turbulence

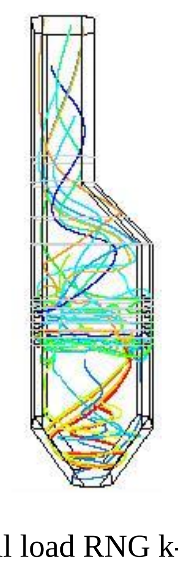


(b) Reduced load RNG k-ε turbulence

Figure 8: Temperature contour at Reheater plane

3.2 Flow pattern inside the furnace

Fig 9 illustrates the general flow pattern for combustion with heat transfer case for full load condition and reduced load condition. In general, intense and highly swirling flow is found at the windbox region where the fuel and air are injected into the furnace. Once entering the furnace, the gas tends to concentrate at the center of the furnace then flow towards the upper region. It can be seen some flow pattern to travels downward to hopper region especially when using RNG k-ε turbulence model where the observed intensive at hopper region. The intensity of the combustion gas reduced slightly as the gas flows upwards 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 slanted region, the gas moves towards the right and front wall, where the flow deviation is corrected.



(a) Full load RNG k-ε turbulence

(b) Reduced load RNG k-ε turbulence

Figure 9: General flow pattern inside the furnace

4 CONCLUSIONS

CFD simulation of flow characteristics and combustion inside oil fired full scale boiler. Two cases have been simulated with the effect of heat transfer to the furnace wall for full load and reduced load condition. In general, the results of the simulations show that flow inside the furnace is highly swirling with intense combustion and comparing between full load and reduced load conditions, the temperature magnitude is not much different. Highest exhaust gas temperature is predicted at the combustion region in the center of the furnace and height increases, the temperature decreases due to heat loss radiation as well as heat transfer to the furnace wall.

5. Acknowledgements

The authors would like to express their University Tenaga Nasional support in carrying this work. The funding received in presenting this paper from the management of University Tenaga Nasional is also acknowledged.

References

Coelho, P.J.and Carvalho ,M.G.(1996) Evaluation of a Three-Dimensional Mathematical Model of a Power station Boiler. Journal of engineering for Gas Turbine and Power , 118, 887-895.

Fan, J., Qian, L., Ma, Y., Sun, P., & Cen, K. (2001). Computational modeling of pulverized coal combustion processes in tangentially fired furnaces. Chemical Engineering Journal, 81(1), 261-269.

Hasini and Maruf .(2006) CFD analysis of flow combustion process in an oil-fired boiler at design condition. proceeding of the Eleventh Asian Congress of Fluid Mechanics.

Lee,Y.P., Xu,J.Y.,Zhang, Q.,Gu,F.and Xu,Y.Q.(2001) Investigation on Residual Swirl and Flue Gas Imbalance in Large Tangentially Fired Boiler .JSME international journal ,44,378-386.

Yusoff, M. Z., Hasini, H., Majid, K. A.,Ramli, M. R., Hassan, H., Hussein, I., Boosroh, M. H., 2002, “Modeling of Combustion in a 120 MW Gas Fired Full Scale Industrial Boiler: Preliminary Simulations,” Proceedings, 6th Asia-Pacific International Symposium On Combustion and Energy Utilization, Kuala Lumpur, Malaysia, pp. 155-160.