Burner tilting angle effect on velocity profile in 700 MW Utility Boiler

K M Munisamy 1, M Z Yusoff 1, S K Thangaraju 1, H Hassan 2, A Ahmad 2
1Universiti Tenaga Nasional, Jalan IKRAM-UNITEN, 43009, Kajang
2Tenaga Nasional Research, Jalan Ayer Hitam, 43000 Kajang.

E-mail: kannan@uniten.edu.my

Abstract. 700 MW of utility boiler is investigated with manipulation of inlet burner angle. Manipulation of burner titling angle is an operational methodology in controlling rear pass temperature in utility boilers. The rear pass temperature unbalance between right and left side is a problem caused by fouling and slagging of the ash from the coal fired boilers. This paper presents the CFD investigation on the 0° and -30° of the burner angle of the utility boiler. The results focusing on the velocity profile. The design condition of 0° burner firing angle is compared with the off-design burner angle -30° which would be the burner angle to reduce the rear pass temperature un-balance by boiler operators. It can be concluded that the -30° burner angle reduce the turbulence is fire ball mixing inside the furnace. It also shift the fire ball position in the furnace to reduce the rear pass temperature.

Introduction
A large portion of the electrical energy generated for global usages are produced from the combustion of fossil fuel resources in thermal power plants. Tangentially fired coal fired boilers are the mostly used type of boiler in the power generation. The coal combustion process in thermal power plant consists of phenomena such as turbulence, heat transfer by convection and radiation, and chemical reactions. Many factors affect the phenomenon stated above such as furnace scale thickness, air velocity, mass flow rate and temperature of the heated air and many more. By varying these factors we can actually vary the temperature distribution In the coal fired boiler furnace which has direct impact on the efficiency of the boiler. The figure below shows the parts of a coal fired boiler. A re-heater is a component of a coal fired power plant boiler which is responsible to reheat the steam known as cold reheat which is exiting the high pressure turbine. After it is reheated it is send back to the second steam turbine. This re-heater has boiler tubes placed in panels. While this process is going on, it was identified that there are un-even temperature distribution in the re-heater. This un-even temperature distribution can cause heat to be concentrated only on a certain region at the boiler tubes causing tube to tube temperature differences. When the temperatures are different along the tubes it will cause irregular expansion processes which will lead to cracking in the boiler tubes of the re-heater. Crack in boiler tubes will reduce the efficiency of the boiler system.

The purpose of this project is to find a solution to reduce the un-even temperature distributions in the left and right-hand side of the re-heater of a 700 MW boiler. The fluid that is being studied in this project is the flue gas. Flue gas or also known as combustion exhaust gas which is exiting to the
atmosphere via a flue, which is a pipe or channel for conveying exhaust gases from a furnace, boiler or steam generator. Its composition usually consists of mostly nitrogen derived from the combustion of air, carbon dioxide, and water vapour as well as excess oxygen. It further contains a small percentage of a number of pollutants, such as particulate matter (like soot), carbon monoxide, nitrogen oxides, and sulphur oxides. The first CFD simulation is done for the existing plant operation in terms of the fuel and air flow configurations by using 3-dimensional CFD simulation. Once the simulation for the existing plant operation is done, then it will be followed by the designing of the improved configuration of the plant. Once the new design of the plant operation is obtained, then CFD simulation will be performed for the new design in terms of the fuel and air flow configurations. Data from both the simulations will recorded and compared. The results gained in this investigation will be useful to solve the un-even temperature distributions in the re-heater which will save cost of repair of the boiler tubes and further increases the efficiency of a coal fired boiler system.

2. Summary of CFD simulation in Coal and Gas fired Boiler Applications

A model of a waste heat recovery boiler was set up to study the effect of temperature of char and dust within the boiler on the deposition of these particles on the boiler walls by using Computational Fluid Dynamics (CFD) software[1]. According to Manickam et al.[1], the deposition of particles onto the walls of the boiler can be reduced if the temperature at which the particle becomes sticky is higher than the particle temperature when it hits the wall of the boiler. The deposition of these particles on the walls causes a non-uniform temperature distribution on the surface of the wall[1].

R. Vuthaluru and H.B. Vuthaluru [2], proposed a study on the effects of different operating conditions in a wall fired furnace using Computational Fluid Dynamics (CFD) software, FLUENT. R. Vuthaluru and H.B. Vuthaluru [2], used a 3-D combustor model to determine the temperature and heat flux profiles for a 500MW utility boiler using volatile coal as their firing medium. Different operating conditions such as effect of excess air on the furnace temperature pattern and temperatures achieved in different parts of the boiler during the combustion process have been analysed. Figure...shows the specifications of the 500MW furnace geometry. Figure 2 shows the operating conditions of the boiler.

There are 4 operating conditions in this study. First is the main boiler operating condition which is the Based on the FLUENT software simulation results, R. Vuthaluru and H.B. Vuthaluru [2], concluded that the full and partial loads with different burners in operation produces a significant impact on the temperature distribution and furnace exit temperatures.

R. Ben-Mansour et al. [3] proposed that the temperature distribution in the combustion chamber and in the exhaust gas are effected by various operating conditions such as excess air factor with varying the air mass flow rate, combustion air temperature and the number of tripped burners. This study is done by using CFD FLUENT software.

According to R. Ben-Mansour et al. [3], as the combustion air temperature increases, furnace temperature increases. The furnace average temperature decrease as the excess air to fuel ratio factor increases for a given air mass flow rate [3]. According to R. Ben-Mansour et al. [3], tripping one or two burners either adjacent or opposite or tripping four results in regions of high temperature gases close to the walls and the temperature is distorted when any of the burners are tripped.

Indrusiak et al. [4], studied the effect of air leakage in the bottom of the boiler which was simulated using CFD code CFX and the result obtained was compared with the situation where the leakage does not exist. According to Indrusiak et al. [4], three situations were taken into account which is the study of flow field, temperature field and the heat flux generated in the furnace. The tree situations are Cases A,B and C. Case A are the value of bottom air leakage evaluated by the power plant staff [4]. Case B is
the value of total amount of air which is assigned to the bottom air leakage that is computed by stoichiometric balance \[ 4 \]. Case C is the value assigned to the air leakage in case A which is now added to the secondary air \[ 4 \].

S. Mohammad et al. \[ 5 \] presented an investigation on the flow and combustion process in an oil fired furnace by using RNG turbulence model based commercial CFD-code FULENT. Two stages of simulation were performed, one with the effect of heat transfer to the furnace wall for full load and the other is for reduced load condition \[ 5 \].

Based on the simulation results, it can be observed that the temperature at the centre of the furnace is higher for the full load case compared to the reduced load case \[ 5 \]. For both the cases the temperature distribution are high and non-uniform at the windbox region where combustion process takes place \[ 5 \]. The temperature distribution is found to be more uniform at lower temperature at location beyond the windbox region due to heat loss radiation and heat transfer to the furnace wall \[ 5 \]. Higher swirling and intense combustion can be observed for the full load case compared to reduced load conditions \[ 5 \].

3. Model Development for 700mW Utility Boiler

The model of the 700 MW boiler is developed in 3-D CAD software then it is imported to CFD software for mesh generation. The inlet of the primary and secondary air is modelled. The inlet condition is exploited with 0° and -30° inlet condition and the velocity profile is presented. The inlet condition is determined from the actual data measured from the utility plant as figure 1.

**Figure 1.** Primary and secondary air flow for Unit 2 – 700MW Boiler.
4. Result and Discussion

The results of the velocity profile from Plane 1 to Plane 6. The figure 4 shows the 0° results show the fire ball from plane 1 to 6. In comparison, figure 5 for -30° burner tilting angle shows the fire tracing is not as clear as the design condition. The fire ball is shifted below than the plane 1.
5. Conclusions
It can be concluded that the -30° burner angle reduce the turbulence is fire ball mixing inside the furnace. It also shifted the fire ball position in the furnace to reduce the rear pass temperature. However, the efficiency of the furnace will be penalised by reducing the burner tilt angle.

References

[1] M. S. a. J. M. Manickam, "CFD modelling of waste heat recovery boiler," *Applied Mathematical Modelling*, vol. 22, p. 823 to 840, 1998.

[2] H. V. R. Vuthaluru, "Modelling of wall fired furnace for different operating conditions using FLUENT," *Fuel Processing Technology*, vol. 87, pp. 633-639, 2006.

[3] M. H. a. H. I. A. R. Ben-Mansour, "Influence of Inlet Conditions and Burner Tripping on NO and Temperature Characteristics in a Tangentially Fired Furnace," *International Journal of Environmental Application & Science*, vol. 4, no. 1, pp. 1-21, 2009.

[4] C. L. a. H.-C. Zhou, "Deduction of the two-dimensional distribution of temperature in a cross section of a boiler furnace from images of flame reduction," *Combustion and Flame*, vol. 143, pp. 97-45, 2005.

[5] E. H. a. Z. S. Mohammad, "Simulation of 120MW Industrial Boiler at Design Condition," *Journal of Advanced Science and Engineering Research*, vol. 3, pp. 286-293, 2013.