Influence of furnace cross section on the flow and velocity distribution in tangential firing furnace

A F Zainudin, H Hasini and S S A Fadhil

Centre for Fluid Dynamics, Universiti Tenaga Nasional, Jalan IKRAM-UNITEN, 43000 Kajang, Selangor, Malaysia

Abstract. This paper presents an investigation on the influence of different furnace cross sectional configurations on the flow and velocity distribution in a small-scale tangential firing furnace. The flow and combustion of natural gas in three furnaces of different cross section are simulated numerically using CFD. The results show that velocity distribution is non-uniform along the width and depth of the furnace in rectangular furnace while square cross section gives uniform distribution. The effect of furnace cross section on the tangential velocity is more obvious when the length difference of the width and depth of the furnace is larger. The axial velocity is found to be highly influenced by the furnace cross section.

1. Introduction

Tangential firing boiler is the most widely used type of boiler in the world. This is due to its advantages of uniform heat distribution in the furnace and low NOx emission which imply higher efficiency and less pollution. However, most tangential firing boilers have been experiencing steam temperature imbalance problem which has been long associated with this type of boiler. Earlier investigations on this problem suspect that the root cause is the residual swirl at the upper furnace [1-3]. The residual swirl causes flue gas velocity deviation between the left side and right side of the upper furnace which consequently results in temperature imbalance between the two sides. Excessive steam temperature on one side will shorten the usable life of the heat exchanger tubes and may eventually cause tube failure.

In the light of this issue, there have been many researchers who investigated the occurrence in the boiler utilizing the Computational Fluid Dynamics (CFD) technology [4-12]. Some of the researchers studied the influence of selected parameters on the flow and temperature imbalance occurrence in the boiler. Among the parameters which have been studied are the burner tilt (vertical) angle [7], yaw (horizontal) angle [4, 11], arrangement of the platen superheaters [1, 10], and the structure of the nose region [2]. Up to this point, no literature has been found which study on the effect of the furnace cross section on the flow and velocity distribution in the boiler. Therefore, the main objective of this study is to investigate the influence of different furnace cross section on the flow and velocity distribution in a small-scale tangential firing furnace. CFD simulation was performed on natural gas-fired tangential firing furnaces with different cross sections and the effect on the flow and velocity distribution was observed. The findings from this study may be useful for further investigation in the real full-scale boiler in order to alleviate the temperature imbalance problem.

2. Furnace Description and Case Studied

The furnace studied is a small-scale tangential firing furnace with constant cross section along its height. The typical upper part of a tangential firing boiler, starting from the nose region until the boiler exit is removed for simplicity and to reduce the computational time. In addition, for this study, only the constant-cross-section lower part of the furnace is concerned in order to exclude the renowned effect of the upper furnace structures on the flow. The furnace geometry is shown in figure 1. The geometry is constructed such that its depth, width and height are along x-, y- and z-axis, respectively. The furnace comprises two regions, namely windbox region and furnace region, as illustrated in figure 2. In the windbox region, there are a total of 28 injection ports, 7 ports being at each corner. The 7 ports consist
of 3 fuel (F), 2 primary air (PA) and 2 secondary air (SA), arranged as shown in figure 3. The fuel and air are injected tangentially into the furnace through these ports. The reaction between fuel and air produces combustion, forming counter-clockwise swirling fireball which flows upwards towards the furnace exit. As the flue gas rises, heat is transferred to the water walls.

![Image of furnace geometry with planes selected for analysis](image1)

**Figure 1.** Furnace geometry with the planes selected for analysis.

![Image of regions in the furnace](image2)

**Figure 2.** Regions in the furnace.

![Image of ports arrangement at each corner in the windbox region](image3)

**Figure 3.** Ports arrangement at each corner in the windbox region.

For the purpose of comparing the influence of furnace cross section on the flow and velocity distribution, three cases were studied. The description of the cases is summarized in Table 1. The furnace with an 8 × 8 cross section is chosen as the base case because it is expected that in a furnace with square cross section (equal width and depth), the flow will be perfectly symmetrical. The depth and height of the furnace are maintained at the length of 8 m and 30 m, respectively for all the cases. To maintain the consistency between all the cases, the injection angles of fuel and air are set such that the imaginary circle formed at the centre of the furnace has the same diameter in all the three cases. The resulting injection angles for all the cases are displayed in figure 4. Six planes at different elevations are selected for the purpose of analysing the results. Three planes are in the windbox region, located at 6 m (F1), 7.5 m (PA2), and 8.5 m (SA2) elevations, respectively. While the other three planes are in the furnace region, located at 10 m (UF1), 17 m (UF2), and 25 m (UF3) elevations, respectively. These planes are shown in figure 1.

| Case (Furnace) | Cross Section (Width × Depth) |
|----------------|------------------------------|
| A (Base)       | 8 × 8                         |
| B              | 7 × 8                         |
| C              | 6 × 8                         |

### 3. CFD Models

The numerical simulations for this study were carried out using CFD commercial code – FLUENT 14.5. The mesh, models and boundary conditions used are consistent for all the three cases. For each of the cases, the flow domain comprises a total of 1032608 hexahedral grids. The mesh scheme applied on the horizontal cross section throughout the furnace is shown in figure 5. The fuel used is natural gas, modelled as 100% CH₄ by mass. Air consists of 79% nitrogen and 21% oxygen by mole. The air-fuel ratio set for the simulation is 20.64, which corresponds to 20% of excess air. The realizable k-ε model is used to model turbulent flow in the furnace. Turbulent intensity and viscosity ratio are specified at the
inlets and outlet, with the value of 5% and 10, respectively. Detail of inlet boundary conditions imposed on the simulation is shown in table 2. The turbulence-chemistry interaction is solved using Eddy-Dissipation model. Heat transfer to the walls is calculated based on fixed wall temperature of 800K whereas radiation heat transfer is solved using P1 gray radiation model. The emissivity of the furnace wall is set to 0.7 [13] whereas the weighted-sum-of-gray-gases model (WSGGM) is used to calculate the emissivity of the flue gas.

| Parameter            | Primary Air (PA) | Secondary Air (SA) | Fuel (F) |
|----------------------|------------------|--------------------|----------|
| Mass flow rate (kg/s)| 10.32            | 5.16               | 0.5      |
| Temperature (K)      | 500              | 500                | 300      |

**4. Results and Discussions**

Generally in all the cases, the flow is the most intense in the windbox region where the mixing and reaction of fuel and air take place. Swirling fireball in counter-clockwise direction is formed at the centre of the furnace. In the windbox region and lower part of the furnace region, tangential velocity is dominant whereas at higher elevations in the furnace region, the flow is dominated by the axial velocity (z-component) as the swirl strength has weakened. This is evident from the velocity vectors shown in figure 6. As the flue gas flows upwards, most of the energy is transferred to the walls thus its velocity decreases. At higher elevations, velocity magnitude is higher near to the walls. This implies that as the height increases, the flue gas flows outward towards the walls and this enlarges the swirl diameter. From the velocity vectors, it can also be seen that the swirl remains symmetrical at the centre along the height of the furnace. Since the furnace has constant cross section along its height, the flow is not interrupted.

In case B and C, it was found that the tangential and axial velocity magnitudes along the width are not the same as those along the depth. In case A, the magnitudes are the same along both directions for tangential velocity as well as axial velocity, which can be attributed to the square (equal length of width and depth) cross section of the furnace. Based on these observations, it can be deduced that rectangular
(unequal length of width and depth) cross section of the furnace results in non-uniform distribution of
tangential and axial velocity between the width and the depth.

Figure 6. Velocity vectors on planes (a) PA2, (b) UF1, (c) UF2, and (d) UF3 for case A. Generally for case B and C, the pattern is the same, which is elaborated in the discussion.

4.1. Comparison of Tangential Velocity
The tangential velocity of the flue gas is compared between the three cases separately in two directions. The comparison is first made for the tangential velocity along the width followed by the tangential velocity along the depth of the furnace. For the first comparison, the graph in figure 7 is referred. There is not much difference between case A and case B but there is significant difference between case C and the former two cases. The magnitude of tangential velocity in case C is significantly higher than the other two cases. Based on this comparison, it can be expected that if the width of the furnace is made narrower, the tangential velocity along that direction will increase. This suggests that as the difference of length between the width and the depth is larger (narrower rectangle), the effect of furnace cross section on the tangential velocity becomes more significant.

For the tangential velocity along the depth, there is no significant difference between all the cases. The tangential velocity distributions in all the cases are almost identical, since the depth in all the cases are kept constant.

4.2. Comparison of Axial Velocity
Figure 8 shows the plot of axial velocity along the width at selected planes for all the cases. Based on the plot, it can be seen that the axial velocities for case C are obviously higher than those for the other two cases. The axial velocities for case B are also significantly higher than those for case A. Similarly for the axial velocity along the depth, the highest axial velocity is found in case C, followed by case B, and case A, as shown in figure 9. This trend clearly indicates that the smaller cross section produces higher axial velocity magnitude. The reduction in cross-sectional area forces a relatively large portion of the flue gas to flow upwards in the furnace, making the axial velocity component larger. Since the fuel and combustion air are injected horizontally (no tilting), the flue gas will tend to flow mostly in the horizontal plane first before flowing upwards due to the effect of buoyancy. But when the area becomes smaller, some portion of the flue gas will be forced to take the vertical path.
Figure 7. Tangential velocity plot along furnace width for all cases.

Figure 8. Axial velocity plot along furnace width for all cases.

Figure 9. Axial velocity plot along furnace depth for all cases.

5. Conclusion

CFD simulations of flow and velocity distribution in boiler furnace of different cross-sections were carried out. The influence of different furnace cross sections on the flow characteristics in tangential firing furnace were numerically investigated. In general, the flow in the furnace is intense in the windbox region due to swirl which is caused by the tangential firing system. The intensity deteriorates as the combustion gas flows upwards towards the furnace exit. At higher elevations, axial velocity is dominant while the tangential component of velocity weakens. Square furnace cross section produces uniform velocity distribution along the depth and width of the furnace, whereas rectangular cross section causes the tangential and axial velocity components to be non-uniform. The effect of furnace cross section on the tangential velocity is more significant in narrower rectangle, with higher magnitude along the narrower direction. The axial velocity component is highly influenced by the furnace cross section. The smaller the cross-sectional area, the higher the magnitude of axial velocity in the furnace. This can be attributed to the nozzle effect that forces the velocity to increase in smaller cross-sectional flow area.

Acknowledgement
The authors would like to acknowledge the financial support received from the Ministry of Education Malaysia through the Science Fund (06-02-03-SF0273).
References

[1] Yin C, Rosendahl L and Condra T J 2003 Further study of the gas temperature deviation in large-scale tangentially coal-fired boilers Fuel 82 1127–37

[2] 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 Int. J. 44 378–387

[3] Xu M, Yuan J, Ding S and Cao H 1998 Simulation of the gas temperature deviation in large-scale tangential coal fired utility boilers Comput. Methods Appl. Mech. Eng. 155 369–380

[4] Park H Y, Baek S H, Kim H H, Kim Y J, Kim T H, Lim H S and Kang D S 2016 Reduction of main steam temperature deviation in a tangentially coal-fired, two pass boiler Fuel 166 509–516

[5] Tang G, Wu B, Johnson K, Kirk A, Fu D and Zhou C Q 2016 Numerical study of a tangentially fired boiler for reducing steam tube overheating Appl. Therm. Eng. 102 261–271

[6] Ahmad A, Hasan H, Kannan M, Noor N A W M and Lim M T 2015 Numerical study on the imbalanced flue gas temperature at boiler rear pass area of a tangential-fired coal power plant J. Mech. Eng. Autom. 5 12–18

[7] Tian D, Zhong L, Tan P, Ma L, Fang Q, Zhang C, Zhang D and Chen G 2015 Influence of vertical burner tilt angle on the gas temperature deviation in a 700 MW low NOx tangentially fired pulverised-coal boiler Fuel Process. Technol. 138 616-628

[8] Yang M, Shen Y Y, Xu H T, Zhao M, Shen S W and Huang K 2015 Numerical investigation of the nonlinear flow characteristics in an ultra-supercritical utility boiler furnace Appl. Therm. Eng. 88 237–247

[9] Constenla I, Ferrín J L and Saavedra L 2013 Numerical study of a 350 MWe tangentially fired pulverized coal furnace of the As Pontes Power Plant Fuel Process. Technol. 116 189–200

[10] Zhou Y, Xu T, Hui S and Zhang M 2009 Experimental and numerical study on the flow fields in upper furnace for large scale tangentially fired boilers Appl. Therm. Eng. 29 732–739

[11] Zhou Y, Zhang M, Xu T and Hui S 2009 Effect of opposing tangential primary air jets on the flue gas velocity deviation for large-scale tangentially fired boilers Energy and Fuels 23 5375–82

[12] Belosevic S, Sijercic M, Tucakovic D and Crnomarkovic N 2008 A numerical study of a utility boiler tangentially-fired furnace under different operating conditions Fuel 87 3331–38

[13] Yin C 2013 Refined weighted sum of gray gases model for air-fuel combustion and its impacts Energy and Fuels 27 6287-94