ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

NUMERICAL INVESTIGATION ON THE FLOW FIELD, TEMPERATURE DISTRIBUTION AND SWIRL IN SMALL-SCALE TANGENTIAL FIRING FURNACE

Abdullah Farhan Zainudin, Hasril Hasini and Siti Sarah Ain Fadhil
Fluid Dynamics and Risk Reduction Research Group, Institute of Sustainable Engineering, Universiti Tenaga Nasional, Jalan Ikram-Uniten, Kajang, Selangor, Malaysia

ABSTRACT

This paper presents a numerical investigation of the flow and combustion behaviour in a small-scale tangential firing furnace to study the occurrence of flow and temperature deviation without the influence of upper furnace structures. Particular emphasis is given to the flue gas flow field, velocity, temperature distribution, and swirl intensity at different furnace elevations. The CFD simulation result shows that the swirl and velocity distribution are perfectly symmetrical along the furnace. The swirl diameter becomes larger as furnace height increases. Flow and temperature deviation, which normally occur in full-scale furnace were absent in this case. The swirl intensity was found to be highly influenced by the tangential velocity. These initial findings are useful in understanding the flow behaviour in the furnace and thus, find the root cause of flow and temperature deviation in the full-scale furnace.

Keywords: temperature deviation, velocity distribution, swirl.

INTRODUCTION

Among the most widely used boiler for thermal power plant application is of the tangential firing type. Boiler with tangential firing system offers higher efficiency by the uniform heat distribution in the furnace and low NO_x emission. The fuel and air are injected from the corners of the boiler which produces better mixing and ensures higher chance of complete combustion. However, there is an inherent steam temperature deviation problem associated with the tangential firing boiler. In the upper furnace, the steam temperature on one side is higher than that on the other side. The higher steam temperature on one side may cause the superheater or reheater tubes to deform over time and shorten its usable life.

In parallel with the development of Computational Fluid Dynamics (CFD), many researchers have studied this issue using numerical method [1-6, 9]. The influence of burner injection angles [1-3], nose structure [4-5], and platen heat exchangers [5, 9] on the temperature deviation has been studied and significant effects were found. Generally, it is believed that the main cause of the temperature deviation is the residual swirl in the upper furnace. Therefore, the intensity of the swirl becomes one of the aspects of interest in some literatures, in which the variation of swirl intensity along the boiler height was also studied [1, 5-6, 9].

Based on the literatures, it is known that the flow and temperature deviation occur starting from the nose region due to its structure [4-5]. However, limited works were done on the flow distribution in the constant-cross-section lower part of the boiler without the effect of upper furnace structures in determining whether the deviation has initiated before entering the nose region. Therefore, the purpose of this study is to investigate numerically the flow field, velocity distribution, temperature distribution and swirl intensity in a small-scale tangential firing furnace without the effect of the upper part. In this study, the typical upper part of the boiler, starting from the nose region until the boiler exit, is excluded from the furnace model. This is to investigate whether the flow and temperature deviation could have initiated even before the nose region.

FURNACE MODEL

The small-scale furnace has an 8 × 8 × 30 m (width × depth × height) dimension. It has constant cross section until the exit, which means the nose region and upper furnace are not considered in the calculation. The model of the furnace is shown in Figure-1. The furnace can be divided into two main regions, namely burner region and furnace region which are critical to the investigation. These regions are illustrated in Figure-2. The fuel and air are injected into the furnace from each of the four corners through the inlet ports in the burner region. There are a total of 28 injection ports for the furnace, 7 ports being at each corner. At each corner, there are 3 fuel (F), 2 primary air (PA), and 2 secondary air (SA) ports, arranged as in Figure-3. Fuel and air mixture reacts in the burner region to form anti-clockwise swirling flow at the center. The firing angles of fuel and air are set to follow the firing angles in one of the power plant boiler in operation in Malaysia, as shown in Figure-4. For the purpose of analysis, six planes at different elevations were created. Three of the planes are in the burner region, at 6 m (F1), 7.5 m (PA2), and 8.5 m (SA2) elevations, respectively. While the other three planes are at 10 m (slightly above the burner region, UF1), 17 m (middle of furnace region, UF2), and 25 m (near the exit, UF3) elevations, respectively. The planes are as shown in Figure-1.

VOL. 12, NO. 14, JULY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

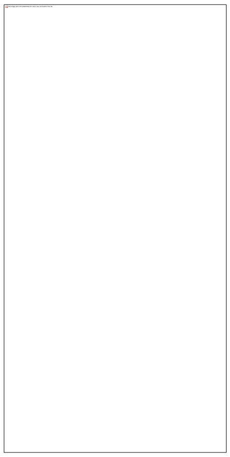


Figure-1. Furnace geometry with planes selected for analysis.

Figure-2. Regions in the furnace.

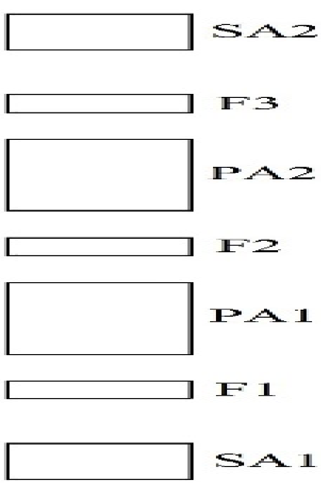


Figure-3. Arrangement of ports in the burner region at each corner.

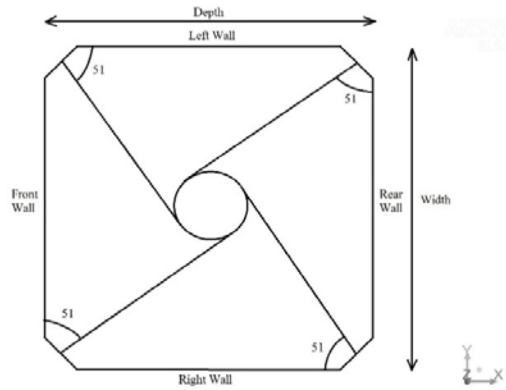


Figure-4. Injection angles of fuel and air.

CFD MODELS

The numerical simulation was performed using CFD commercial code - FLUENT 14.5. The flow domain comprises a total of 1032608 hexahedral grids. This number of grid suffices to produce a reliable and grid-independent solution. As shown in Figure-5, when the total number of grid is increased to 1319552, the deviation of overall solution is found to remain less than 5%. When the total number of grid is 780800, the solution deviates more than 5%, especially in the burner region. The mesh scheme applied on the cross section throughout the furnace domain is illustrated in Figure-6.

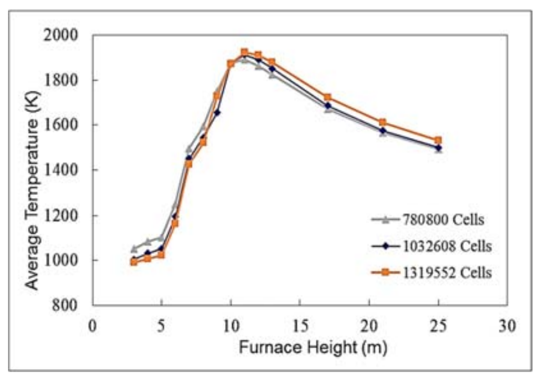


Figure-5. Grid independence test.

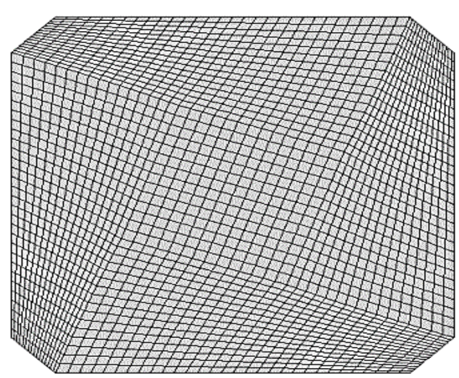


Figure-6. Mesh of the horizontal cross section throughout the furnace.

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Natural gas is used as the fuel, modelled as 100% CH_4 by mass. The air-fuel ratio set for the simulation is 20.64, which corresponds to 20% of excess air. Generally, the solver solves equations for mass and momentum conservation, energy, turbulence, radiation, and interaction between turbulence and chemistry reaction. Turbulent flow is solved using the realizable k-ε model, which has been proven to be better in solving complex flows than the standard and RNG k-ε model [7]. 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-1. The turbulence-chemistry interaction is solved using Eddy-Dissipation model. Heat transfer to the walls is calculated based on fixed wall temperature of 800 K. In a flow involving combustion, radiation is the most important mode of heat transfer. For this study, radiation heat transfer is calculated using P1 gray radiation model. Emissivity of the wall is set to be 0.7 [8] while the weighted-sum-of-gray-gases model (WSGGM) is used to calculate the emissivity of the flue gas.

Table-1. Inlet boundary conditions at each port.

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

RESULTS AND DISCUSSIONS

Flow field and velocity distribution

Velocity vectors are captured on planes PA2, UF1, UF2, and UF3 and these are shown in Figure-7(a) – 7(d). The flow is the most intense in the burner region where fuel and air are injected and combustion takes place. High-velocity swirling flow is observed at the center of the furnace in anti-clockwise direction. As the flue gas rises, it loses energy, mainly in the form of heat which is transferred to the water walls and the velocity of the flue gas decreases especially near the center. As the elevation increases, the flue gas velocity near the walls is found to increase slightly. From this observation, it may be understood that the flue gas flows outward towards the wall, making the swirl diameter larger as it rises. It can also be observed that the swirl formed in the burner region remains symmetrical at the center as it rises until the exit. Due to the constant furnace cross-sectional area, the flow is not interrupted by any structure of the furnace.

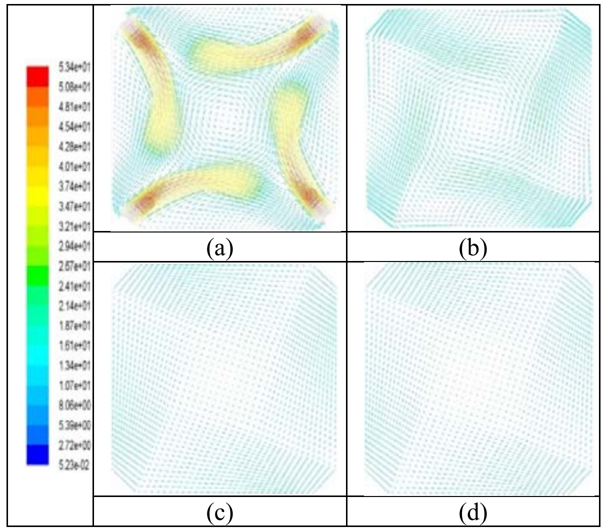


Figure-7. Velocity vectors on (a) PA2, (b) UF1, (c) UF2, (d) UF3.

Figure-8(a) and 8(b) show the velocity vectors on planes near the left side wall and right side wall of the furnace, respectively. Near the left wall in the furnace region, the flue gas flows towards the front wall whereas near the right wall it flows towards the rear wall due to the anti-clockwise direction of the swirling flow. It was previously reported that such flow near the side walls results in velocity deviation between the two sides hence flue gas temperature deviation [1, 9]. Since the upper part of the furnace is excluded, such finding cannot be observed in this study.

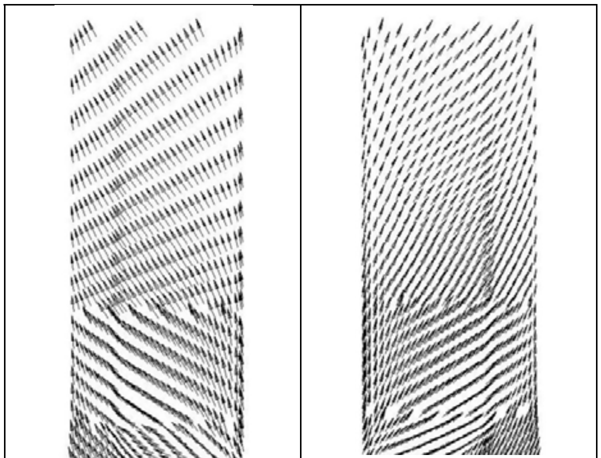
Tangential velocity along the width of the furnace on planes PA2, UF1, UF2, and UF3 are plotted as displayed in Figure-9. It can be observed that the tangential velocity distribution along the width is perfectly symmetrical. The magnitudes are equal between the left side and the right side of the furnace, only differ in direction. The symmetrical velocity distribution remains until the exit of the furnace. As the height increases, the velocity magnitude decreases and becomes higher near to the walls. This implies that the swirl strength reduces but the swirl diameter is enlarged as the furnace height increases. These observations from the tangential velocity plot match well with the observations from the velocity vectors discussed previously. Figure-10 shows the axial velocity plot along the width on planes PA2, UF1, UF2, and UF3. From the plot, it is also evident that at higher elevations, the axial velocity of the flue gas is higher near the walls. This shows that the swirl diameter is enlarged as it goes upwards.

Temperature distribution

In order to investigate the possibility of flue gas temperature imbalance occurring in the furnace, analysis of temperature distribution is carried out. The plot of average temperature along the furnace height is shown in Figure-11. From the burner region, the temperature increases rapidly as the height increases until the lower

ARPN Journal of Engineering and Applied Sciences

part of the furnace region (11 m). At this elevation, the average temperature reaches maximum at 1913 K. Downstream of this elevation, the temperature begins to drop until the exit. In the furnace region, most of the heat energy is being absorbed by the wall which explains the declining trend of flue gas temperature.



(a) (b)

Figure-8. Velocity vectors (a) near the left wall, (b) near the right wall.

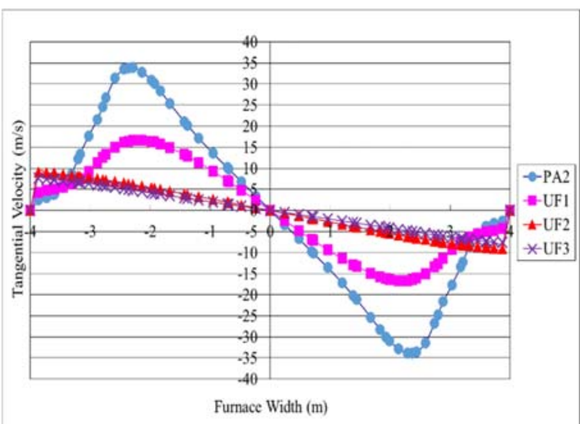


Figure-9. Tangential velocity plot along furnace width.

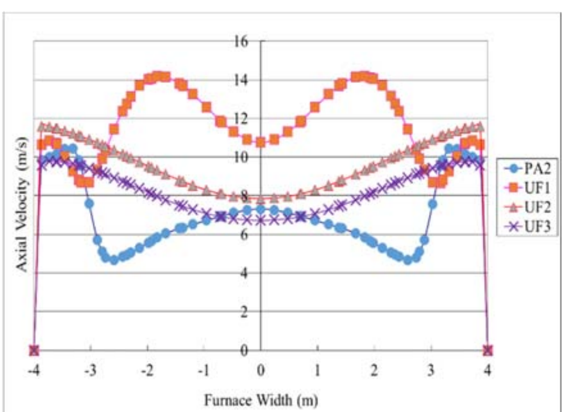


Figure-10. Axial velocity plot along furnace width.

The temperature contours on planes F1, PA2, SA2, UF1, UF2, and UF3 are shown in Figure-12. At F1 plane, relatively low temperature distribution is observed as most of the fuel is still not burnt completely. When air is injected at PA2, combustion occurs intensely and higher temperature is seen along the lines of injections. As the elevation increases in the burner region, more air is injected thus more fuel burnt completely and the temperature becomes higher, as shown in Figure-11(c). After the burner region, the temperature distribution becomes more uniform as all the fuel has burnt completely. From the contours, it can also be seen that the swirl remains symmetrical along the furnace height until the exit.

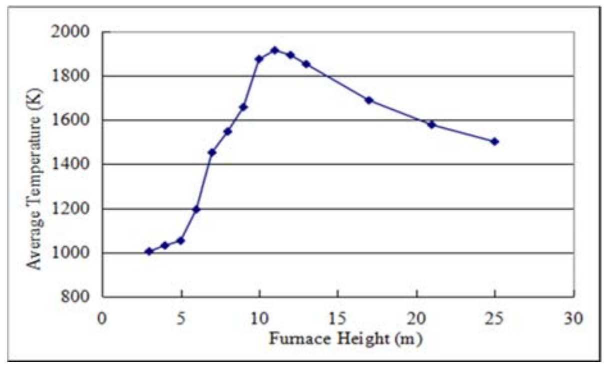


Figure-11. Average temperature plot along furnace height.

Swirl intensity

Steam temperature deviation problem has been long associated with the residual swirl in the upper furnace which causes velocity deviation hence flue gas temperature imbalance. Therefore, it is important to study the swirl intensity when investigating on the temperature deviation of tangential firing furnace. Swirl intensity is represented by a parameter called swirl number (SN), which is shown in Equation. (1).

$$SN = \frac{\int_0^R \rho W U r^2 \, dr}{R \int_0^R \rho W^2 r \, dr} \tag{1}$$

where ρ, W, U, R, and r are flue gas density, axial velocity, tangential velocity, hydraulic radius, and distance from the center of the cross section to the measuring point, respectively.

The swirl number is calculated at several elevations along the furnace height and its variation with the furnace height is plotted, and is shown in Figure-13. The highest swirl number occurs at 6 m elevation which is in the lower part of the burner region. After the 6 m elevation, the swirl number drops quite rapidly until the end of burner region. Since the swirl number is a function of velocity, which is decreasing with increasing height, it continues to decrease gradually until the exit. At the exit, there is still a small intensity of swirl exists which is about 10% of its highest intensity in the furnace. From Figure-9,

VOL. 12, NO. 14, JULY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

the tangential velocity distribution shows the same trend that is, weakening rapidly in the burner region and decreases slowly in the furnace region. Based on this, it can be related that the tangential velocity distribution governs the swirl number distribution.

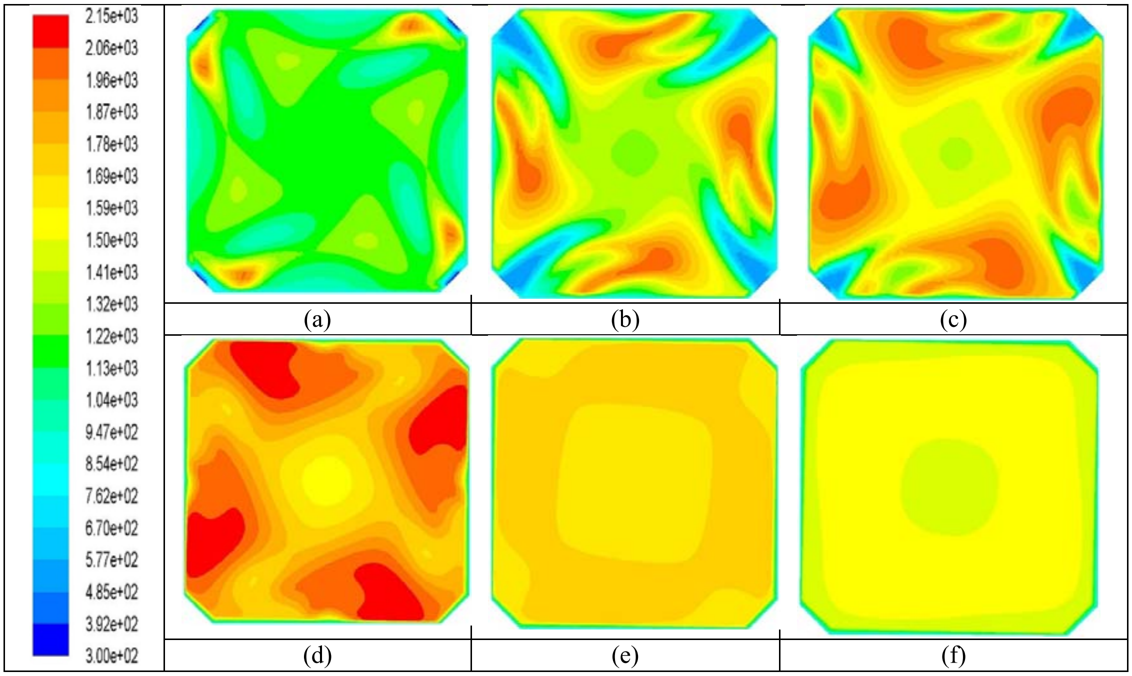


Figure-12. Temperature contours on (a) F1, (b) PA2, (c) SA2, (d) UF1, (e) UF2, (f) UF3.

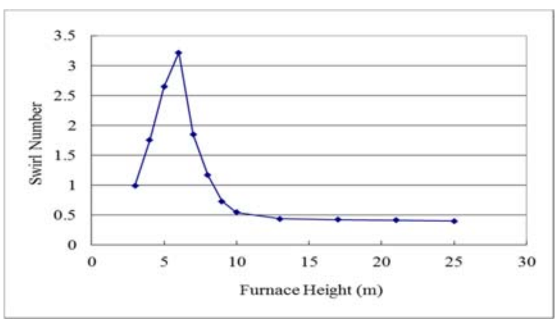


Figure-13. Swirl number plot along furnace height.

CONCLUSIONS

The flow field, velocity distribution, temperature distribution, and swirl intensity in a small-scale constant cross-sectional area furnace are numerically investigated. Based on the simulation results, the following conclusions could be withdrawn:

i. The velocity distributions are perfectly symmetrical at all furnace elevations. Swirl remains at the center along the furnace height. There is no velocity deviation observed along the furnace height until the exit.

ii. As the height increases, velocity magnitude decreases and higher velocity region is found near the walls. This implies that the swirl is enlarged as it flows upwards.

iii. The temperature is the highest in the burner region and the distribution on any horizontal plane becomes uniform after the burner region when all the fuel has completely burnt. In the furnace region, the temperature decreases with increasing height as most of the heat is absorbed by the water walls. Temperature deviation does not occur along the furnace.

iv. The swirl is the strongest in the lower part of burner region. After that, its intensity declines rapidly in the burner region, and decreases slowly in the furnace region. This trend is in good accordance with that of the tangential velocity. Therefore, it can be said that the tangential velocity distribution governs the distribution of swirl number.

ACKNOWLEDGEMENT

The authors would like to thank the Ministry of Education of Malaysia under the Fundamental Research Grant Scheme 2015 (20150211FRGS) for the financial support.

REFERENCES

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

[2] H.Y. Park, S.H. Baek, Y.J. Kim, T.H. Kim, D.S. Kang, D.W. Kim, Numerical and experimental investigations on the gas temperature deviation in a large scale, advanced low NO_x, tangentially fired pulverized coal boiler, Fuel. 104(2013) 641-646.

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

[4] H. Hasini, M.Z. Yusoff, N.H. Shuaib, M.H. Boosroh, Analysis of flow and temperature distribution in a full scale utility boiler using CFD, in Proc. 3rd Int. Conf. Energy and Environment, Malacca, Malaysia. (2009) 208-214.

[5] Y.-P. Lee, J.-Y. Xu, Q. Zhang, F. Gu, Y.-Q. Xu, Investigation on residual swirl and flue gas imbalance in large tangentially fired boiler, JSME Int. J. 44 (2001) 378-387.

[6] Y. Zhou, M. Zhang, T. Xu, S. Hui, Effect of opposing tangential primary air jets on the flue gas velocity deviation for large-scale tangentially fired boilers, Energy and Fuels. 23(2009) 5375-5382.

[7] ANSYS Fluent Theory Guide, Release 14.0, ANSYS, Inc., Canonsburg, PA, 2011.

[8] C. Yin, Refined weighted sum of gray gases model for air-fuel combustion and its impacts, Energy and Fuels. 27(2013) 6287-6294.

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

[10] Y.A. Cengel, M.A. Boles, Thermodynamics: An Engineering Approach, 8th ed., New York: McGraw-Hill, 2015.