

Numerical investigation using two different CFD codes of pulverized-coal combustion process characteristic in an industrial power plant boiler

Paweł Madejski^{1,*}, and Norbert Modliński²

¹AGH University of Science and Technology, Faculty of Mechanical Engineering, Department of Power Systems and Environmental Protection Facilities, Al. Mickiewicza 30, 30-059 Kraków, Poland

²Wrocław University of Technology, Faculty of Mechanical and Power Engineering, Division of Boilers, Combustion and Energy Processes, Wybrzeże Wyspiańskiego 27, 50-370 Wrocław, Poland

Abstract. Steam boilers using the coal as a basic fuel are still one of the most important techniques used to generate electricity in Power Plants. Many activities connected with optimization of steam boilers operation, investigation of combustion efficiency using different fuels, control and reducing pollutants emission are observed. Numerical modeling of large steam boilers using Computational Fluid Dynamic method can be a very way to develop and verify effects of all activities regarding combustion process optimization. Numerical modeling results of the coal combustion process in the front wall coal-fired boiler are presented in the paper. The behavior of the flow of pulverized coal through the burners was analyzed, and the temperature and velocity distribution in the combustion chamber were reproduced in the simulation. Despite the fact that the attention has been focused on boiler simulation at nominal load, it is possible to perform numerical studies concerning the analysis of coal combustion at different boiler loads (minimum load and flexible boiler operation). Analysis of different fuels and their impact on the combustion process, as well as analysis of coal mills operation, coal particles size distribution and they impact on boiler operation can be performed using developed models.

1 Introduction

Combustion and co-combustion process modeling using CFD tools (Computational Fluid Dynamics) is a known area of research and widely used by the academic and industrial community [1-6]. This technique allows conducting low-cost research on the combustion process in large steam boilers. Results obtained in this way are helpful for determining current and identify the optimal boiler operating conditions and their influences on combustion products (CO, CO₂, NO_x, UBC).

The goal of this work is simulated coal combustion process in Power Plant using developed CFD model. Presented modeling results were carried out using Open Source CFD code – Code_Saturne, which is is a dedicated tool to create numerical models of

* Corresponding author: madejski@agh.edu.pl

incompressible fluid flows on industrial applications [7,8] with the special focus on pulverized coal combustion [9] as well as heavy fuel oil or biomass co-combustion in large industrial boilers.

The main goal was to carry out a simulation of the coal combustion process in industrial boiler and to evaluate the current combustion process quality. A complex physical phenomenon which is the pulverized coal combustion and its mathematical modeling still need to pay particular attention to verification of final results. In available CFD codes dedicated to combustion process modeling, several sub-models and methods can be used in the simulation of physic-chemical processes like turbulent flow, radiative heat transfer, pyrolysis, turbulent gas combustion, and char combustion. One of the goals presented in this paper was a comparison of coal combustion results, obtained by Code_Saturne software, with the results from the most popular commercial software Ansys Fluent.

2 Boiler characterization

The boiler is characterized by the use of low-emission combustion technology consists of a staged combustion system with low oxygen combustion zone [10]. The low-NO_x combustion technology is most often implemented using low-emission burners and distribution of air supplied to the boiler. In the analyzed case, furnace installation is also equipped with a system of fuel staged by providing the coal-air mixture with better fineness in the higher regions of the furnace. The general scheme of low-emission combustion system in PC boiler (swirl burners and dropt tubes) is depicted in Figure 1. The most important properties of this kind of boiler is that boiler has front wall-fired burners, uses pulverized coal as a fuel, is equipped with the steam drum. Maximum continuous capacity is equal to 180,55 kg/s of live steam, produced at temperature 545 °C and pressure equal to 13.5 MPa. The boiler is equipped with six ring-ball mill units that supply 24 burners. Low-emission combustion system consists of two rows of main swirl burners (at each row six swirl burners are located) located at the lower part of air windbox. Additional diluted coal-air mixture is provided through 12 drop tubes located at the upper part of windbox. Air staged system is characterized by two levels of Over Fire Air nozzles (OFA) located at the front wall (6 OFA ports) and rear boiler wall (10 OFA ports). All burners are fitted in the common windbox located on the front wall of the boiler. The mill units operation configuration is different and depends mainly on load demand.

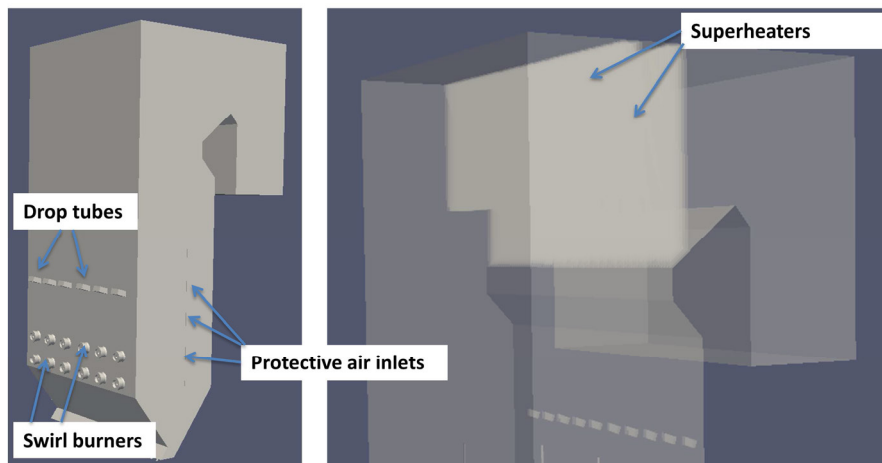


Fig. 1. Geometrical model of an industrial front-fired pulverized power plant boiler

Combustion air in the main swirl burners is separated into the core, primary, secondary, and tertiary air. The additional diluted coal-air mixture is provided through 12 drop tubes located in a single row at the upper part of windbox.

3 Industrial boiler modelling

To create the three-dimensional geometrical model and the grid, ANSYS Design Modeler and ANSYS Meshing software were used (Figure 1 and 2). Quality of mesh is one of the most important factors in the numerical simulation and the geometrical model was subdivided into fine-grid regions around the burners and coarser regions elsewhere. In order to facilitate the meshing process, the circular drop tubes have been replaced by rectangular ones and the platen superheaters have been modeled as zero-thickness horizontal planes. A final mesh cell number was to about 3.3 million and was selected after mesh independent test to obtain a compromise between solution accuracy and calculation time. Computational domain and numerical mesh close to the swirl burners region is shown in Figure 2.

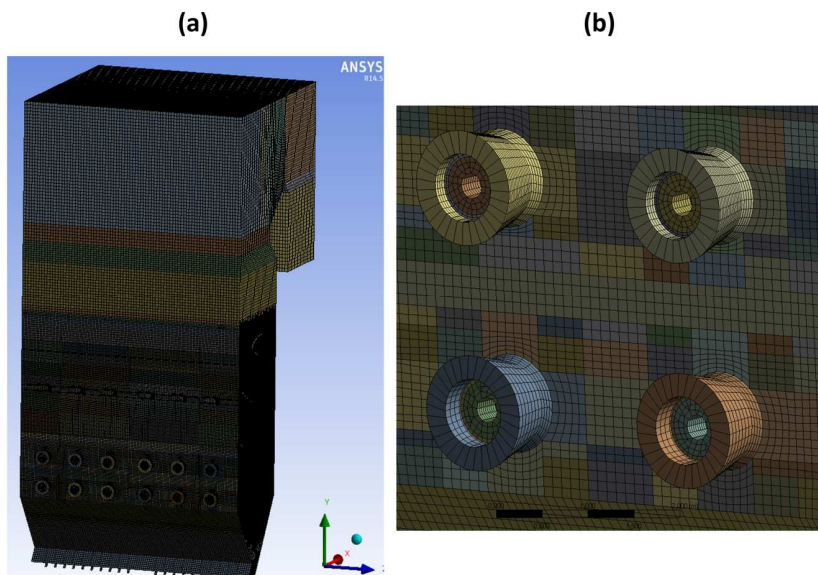


Fig. 2. The numerical mesh of boiler (a) and details of mesh close to the burner region (b)

4 CFD modelling of coal combustion using a mixture fraction approach

Pulverized coal combustion process is a one of an example of turbulent non-premixed combustion systems and can be modeled using the mixture fraction/PDF model. Mixture fraction f can be expressed as the local fuel mass fraction:

$$f = Y_F / Y_P \tag{1}$$

where Y_F and Y_P is a mass fraction of fuel and products respectively.

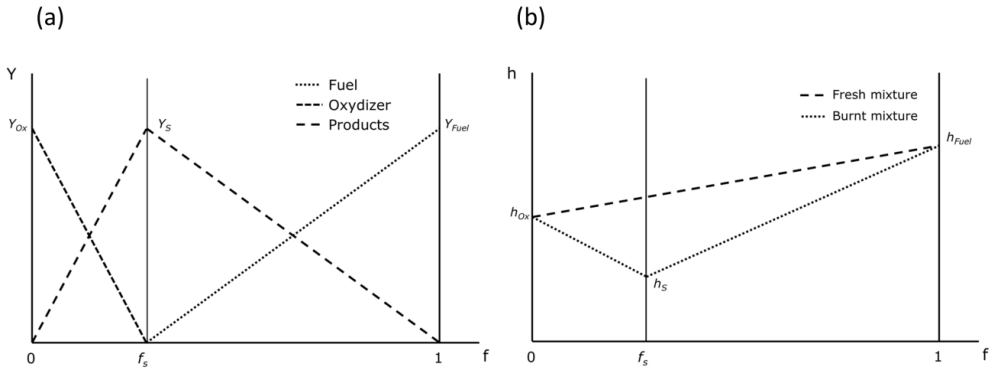


Fig. 3. Example of mixture fraction approach interpretation for a mass fraction (a) and enthalpy (b)

If $0 < f < f_s$ fuel is deficient and the mixture is called fuel lean. The mass fraction of oxidizer and products is represented in form (Figure 3a):

$$Y_i(f) = Y_{Ox} + (f/f_s) \cdot (Y_S - Y_{Ox}) \quad (2)$$

and enthalpy for fuel-lean mixture and $0 < f < f_s$ is (Figure 3b):

$$h_L = h_{Ox} + (f/f_s) \cdot (h_S - h_{Ox}) \quad (3)$$

When $f_s < f < 1$ the mixture is called fuel-rich, and the following equation can be used to calculate mass the fraction of products and fuel (Figure 3a):

$$Y_i(f) = Y_S + (f - f_s / 1 - f_s) \cdot (Y_{Fuel} - Y_S) \quad (4)$$

and for enthalpy when $f_s < f < 1$:

$$h_R = (f_s \cdot h_{Fuel} - h_S) / (f_s - 1) + (f/f_s - 1) \cdot (h_S - h_{Fuel}) \quad (5)$$

where f is the mixture fraction, f_s – stoichiometric mixture fraction, Y_S – mass fraction of products of stoichiometric reaction at $f=f_s$, Y_i – mass fraction linear functions (Y_O , Y_F , Y_P), Y_O – local mass fraction of oxidizer, Y_F – local mass fraction of fuel, Y_P – local mass fraction of products, Y_{Ox} – mass fraction of oxidizer at $f=0$, Y_{Fuel} – mass fraction of fuel at $f=1$.

The mixture fraction approach allows calculating mass fractions based on the one value. The governing transport equation of mixture fraction used to computation at each control volume is given by:

$$(\rho f) = \partial / \partial x_i (\rho D (\partial f / \partial x_i) - (\rho u_i f)) \quad (6)$$

where D is a molecular diffusion coefficient (m^2/s), u_i – velocity in direction of i (m/s), x_i – direction i in Cartesian coordinates system (m), ρ – density (kg/m^3).

Despite simplifications, the mixture fraction/PDF method allows the determination of the basic parameters of the combustion process, also in the case of coal combustion. The mixture fraction/PDF approach is commonly used in Computational Fluid Dynamics and particularly in the modeling of the turbulent reactive flows which are the most popular cases occurring in industrial practice.

5 Industrial boiler simulation

The simulation was conducted for a boiler load equal to 90%. Operating conditions have been collected from the plant on-line monitoring system by averaging two-hours measurements during steady-state boiler operation. Air/fuel distribution, temperatures, and coal mills activity are presented in Table 1.

Table 1. Boiler operating conditions

	Burner Level I	Burner Level II	Burner Level III	Sum:
Coal (kg/s)	9.17	6.55	8.46	24.18
Core Air (kg/s)	0.88	0.88	0	1.76
Primary Air (kg/s)	25.41	16.24	34.08	75.73
Secondary Air (kg/s)	6.16	6.16	3.52	15.84
Tertiary Air (kg/s)	9.02	9.02	0	18.04
OFA II (kg/s)		31.27		31.27
OFA III (kg/s)		22.99		22.99
Protective Air (kg/s)		35.81		35.81
Bottom Air (kg/s)		4.6		4.6
Total Air (kg/s)		206.05		206.05
Total Air Excess		1.11		1.11
Primary/Secondary Air Temperature (K)	386/573			

Five classes of particle diameters were used for burners in Level I and II (concentrated coal-air mixture) and for burners in Level III (thinned coal-air mixture). Table 2 present data from a laboratory test of analyzed coal.

Table 2. Coal analysis results

Proximate Analysis, wt % (as received)				
Ash	Volatile Matter		Moisture	Fixed carbon
22.34	25.77		12.75	39.14
Ultimate Analysis, wt % (daf)				
C	H	N	S	O
84.7	5.39	1.55	1.23	7.13
Density, kg/m ³		Specific heat, J/(kg·K)		Calorific value, MJ/kg (as received)
1400		1680		20.423

The boundary conditions of all inlets and walls were defined in subroutines based on the operating conditions presented in Tables 1 and 2. To prepare complete boundary condition for coal inlet the following properties and values have to be defined:

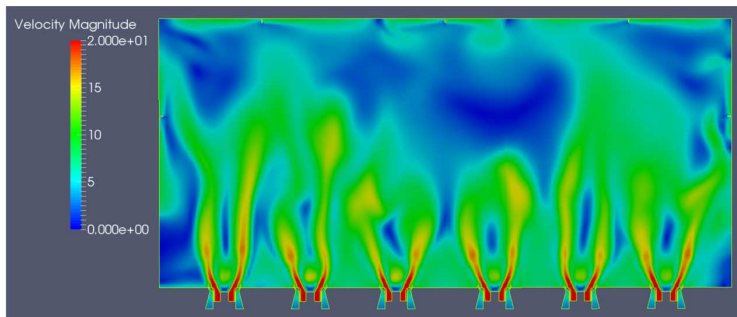
- kind of boundary condition,
- boundary zone number,
- method of inlet flux definition,
- number of oxidizers,

- the mass flow rate of oxidizer,
- the temperature of oxidizer,
- number of different coal,
- coal mass flow rate,
- number and percentage mass fraction of each coal class,
- coal and primary air temperature,
- initial velocity vector components,
- inlet hydraulic diameter,
- inlet turbulent intensity.

6 Simulation results

Velocity and temperature distribution are presented in Figures 4 and 5 and confirm proper burners operation as well as the proper value of fuel and air ratio. The shape of the flame and internal and external recirculation allows the full mixing of coal particles with secondary and tertiary air. Temperature distribution inside the furnace is uniform, even if the coal mass flow rate is varied for different burners (Figure 4). The maximum value of gas temperature is equal to 1900 K (1626.85 °C).

(a)



(b)

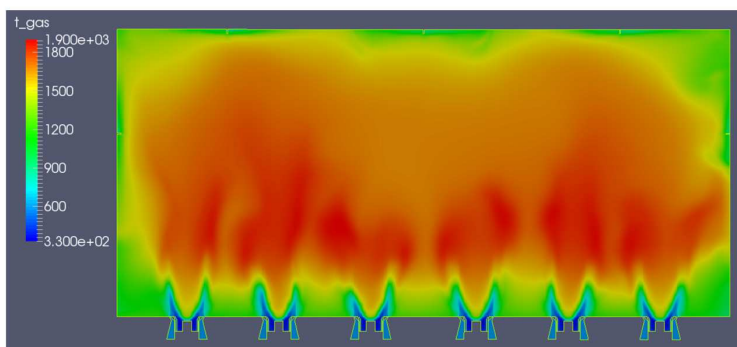


Fig. 4. Velocity (a) and temperature (b) distribution in main swirl burner plane

The complex system of over fire air allows finishing the combustion process inside the combustion chamber (Figure 6). It is very important in the boiler which is equipped with a low-NO_x furnace system and the burner belt excess air is at a very low level (equal to 0.6).

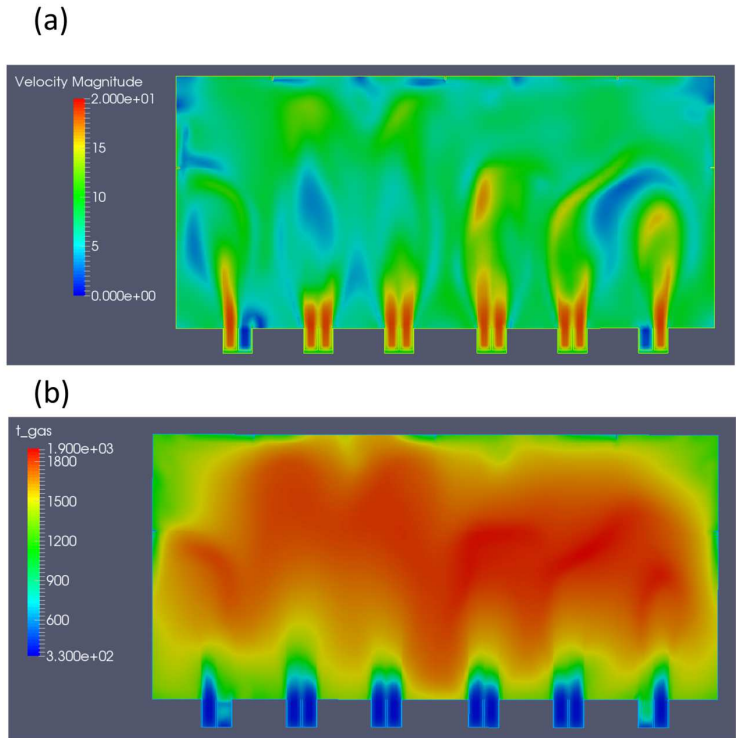


Fig. 5. Velocity (a) and temperature (b) distribution in drop tubes burner plane

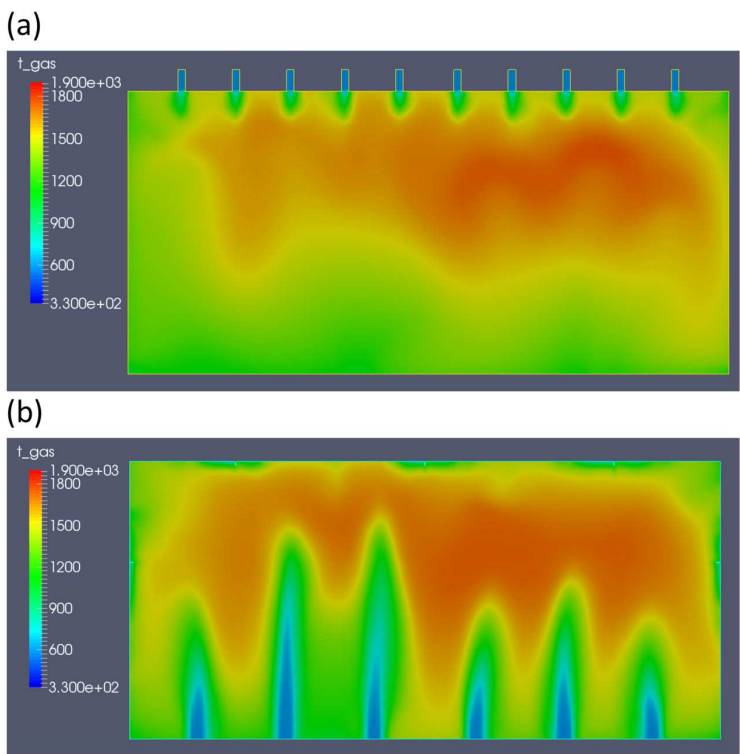


Fig. 6. Temperature distribution in the plane of OFA III (a) and OFA II (b)

In Figures 7 and 8 comparisons of simulation results obtained using Code_Saturne and Ansys Fluent are presented. Both velocity distributions, obtained using Code_Saturne and Ansys Fluent software (Figure 7), show the correct operation of burners. Swirl, which is generated by burners is reflected by velocity distribution with visible mixing zone of fuel and air. The small differences in flame length and velocity distribution came from detailed differences in turbulence models, used by the codes (Standard and Realizable $k-\epsilon$). The differences in temperature distributions are presented in Figure 8. Shapes of flames in burners located nearest the side walls are different and can indicate that the combustion process is starting at different point for these burners. The main reason for this phenomenon could be the higher fuel/air ratio at the inlet to these burners, and position close to boiler walls.

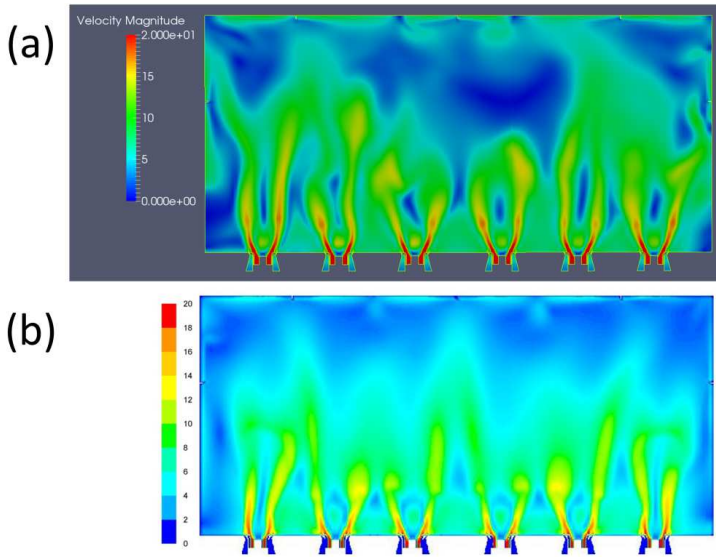


Fig. 7. Velocity distribution in main swirl burner plane obtained using Code_Saturne (a) and Ansys Fluent (b)

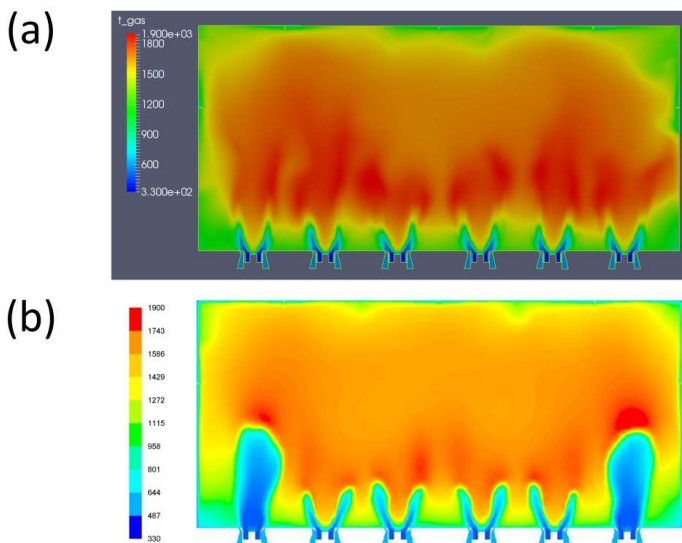


Fig. 8. Temperature distribution in main swirl burner plane obtained using Code_Saturne (a) and Ansys Fluent (b)

7 Conclusions

The complete mathematical model of the boiler was prepared in Open Source – Code_Saturne software. The main goals of this work were to create a complete model of this boiler and to carry out simulation at the load close to the nominal value using boiler operating conditions. The verification of simulation results was done by comparing the results from simulation software using Ansys Fluent software. Comparison of the results confirms the correct modeling of the combustion process in the boiler and the good quality and accuracy of results.

Despite the fact that most attention has been focused on the correct modeling of the coal combustion process and boiler operation at nominal load, it is possible to perform numerical studies concerning:

- Analysis of coal combustion in the boiler at different loads, as the minimum load and flexible boiler operation,
- Analysis of different fuels and their impact on the combustion process,
- Analysis of coal mills operation, coal particles size distribution and they impact on boiler operation.

When more accurate analysis focused on goals mentioned above is required, there is a need for thorough verification of the target results.

Presented conclusions concerning the results include general remarks and comments on the tested boiler. Results of the velocity distribution confirm the correct fuel/air ratio which are supplied swirl burners. The full vortex generated by burners was achieved, together with internal and external recirculation of fuel/air mixture inside the burner belt. In case of wrong operating conditions of burners, supplying by coal and air (fuel/air ratio, coal particles size) the mixing zone cannot exist and the combustion process in this area cannot be optimal.

Acknowledgments

This research was partially funded from GEKON program by the National Center of Research and Development and the National Fund for Environmental Protection and Water Management under research and development project No. GEKON1/O2/213655/9/2014.

References

1. S. Belosevic, M. Sijercic, S. Oka, D. Tucakovic, Three-dimensional modeling of utility boiler pulverized coal tangentially fired furnace. *Int J Heat Mass Tran*, **49**, 3371–3380 (2006)
2. J. Badur, P. Ziółkowski, D. Sławinski, S. Kornet, An approach for estimation of water wall degradation within pulverized-coal boilers, *Energy*, **92**, 142–152 (2015)
3. N. Modliński, K. Szczepanek, D. Nabagło, P. Madejski, Z. Modliński, Mathematical procedure for predicting tube metal temperature in the second stage reheater of the operating flexibly steam boiler, *Appl Therm Eng*, **146**, 854–865 (2019)
4. P. Madejski, Numerical study of a large-scale pulverized coal-fired boiler operation using CFD modeling based on the probability density function method, *Appl Therm Eng*, **145**, 352–363 (2018)
5. X. Minghou, J.L.T. Azevedo, M.G. Carvalho, Modeling of a front wall fired utility boiler for different operating conditions, *Comput Method Appl M*, **190**, 3581–3590 (2001)
6. P. Sanghyun, A.K. Jungeun, R. Changkook, C. Taeyoung, Y. Won, K. Young-Ju, P. Ho-Young, L. Hee-Chun, Combustion and heat transfer characteristics of oxy-coal combustion in a 100 MWe front-wall-fired furnace, *Fuel*, **106**, 718–729 (2013)

7. Électricité de France: Code_Saturne - general purpose Computational Fluid Dynamics (CFD) software. <http://code-saturne.org/>.
8. F. Archambeau, N. Mechtoua, M. Sakiz, Code_Saturne: A finite volume code for the computation of turbulent incompressible flows – industrial applications, *IJFV International Journal On Finite Volumes*, **1** (2004)
9. S. Dal Secco, F. Cordier, Version 3.2 tutorial - Simulating pulverized coal combustion, coal/biomass co-combustion and slagging in a furnace using the Lagrangian approach in Code Saturne. EDF Lab Chatou, France (2014)
10. P. Madejski, T. Janda, N. Modlinski, D. Nabaglo, A combustion process optimization and numerical analysis for the low emission operation of pulverized coal-fired boiler, G. Konstantinos, J. Skvaril (Eds.), Chapter in book: *Developments in Combustion Technology*, IntechOpen, (2016)