

3D CFD simulation of aerodynamics of a 406 MW_t CFB boiler

Oleg M. Koksharev^{1,*}, Andrey V. Gil¹, and Lebedev V.M.²

¹National Research Tomsk Polytechnic University, 634050 Tomsk, Russia

²Omsk State Transport University, 644046, Omsk, Russia

Abstract. Modeling of various processes in CFB boilers (combustion, aerodynamics, hydrodynamics, etc.) is connected with the solution of systems of differential equations with a large number of unknowns that leads to their simplification and, accordingly, affect the quality of the calculation. The work consists of several parts and aims to assess the applicability of some or other mathematical algorithms to obtain a quantitative simulation of processes in power boilers with circulating fluidized bed. In part, presented in this article, the simulation of CFB boiler aerodynamics using ANSYS FLUENT is considered.

1 Introduction

The modeling methods for CFB furnaces can be classified as fundamentals-oriented and practice-oriented models. In the fundamentals-oriented models, the modeling of the fluid dynamics is attempted with the currently available fundamental theories, such as the Eulerian–Eulerian multiphase models applying the kinetic theory of granular flow. In the practice-oriented models (or engineering models), the theories are simplified and adjusted by empirical correlations to improve the calculation speed and fit the model results with the measured data [1, 2]. In this paper, the 3-D model simulation of the aerodynamic structure of CFB boiler has been resolved by applying ANSYS FLUENT 12.1.

2 Problem statement

It is worth noting that a proper understanding of heat and mass transfer in furnaces with circulating fluidized bed is important in order to make the best design with efficient and reliable options for this type of system. During the individual review of the physical phenomena such as gas flow with the particles, turbulence, chemical reactions and heat transfer, are quite difficult to forecast and study. This study becomes more complicated when these physical processes are interconnected in a large-scale system, such as boilers with CFB.

The results of studies that have been carried out on the basis of the geometrical model presented in [3, 4] (Fig. 1), showed the ambiguity in terms of layout decisions of secondary

* Corresponding author: omk2@tpu.ru

air nozzles, burners and return leg for particles flow from a cyclone arrangements that were accepted based on empirical dependences (Fig. 1).

Therefore it was decided to create a 3-D model, where the calculation takes into account not only the volume of combustion chamber as well as the processes in the cyclone and standpipe with loop seal (Fig. 2).



Fig. 1. The initial object of study: a - 3-D model; b – mesh.

The object of study is a CFB boiler with a capacity of 406 MW_t that is equipped with two cyclones for particulate control. The height of the combustion chamber is 24.5 meters. The furnace is equipped with 5 burners on the rear wall in 1 row and 10 nozzles of secondary air at the front wall in 2 rows. At the exit of the furnace the particles are separated into 2 streams – one is separated and through the standpipe and loop seal returns to the furnace bed layer, another flow leaves the cyclone and directed to the electrostatic separator. It should be noted that the boiler before the simulation was originally calculated on thermal design basis for boilers with CFB in accordance with [5]. 3-D model and computational grid (mesh) of the boiler are shown in Fig. 2.

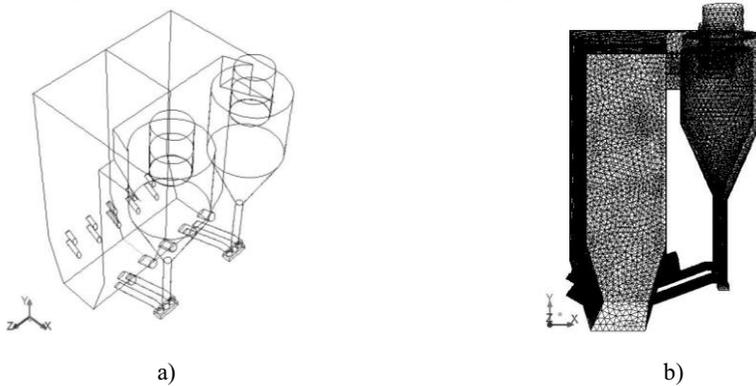


Fig. 2. The research object: a - 3-D model; b – mesh.

3 Results and discussion

Fig. 3–4 show the contours of static pressure in horizontal and vertical sections of the boiler. The obtained results are in the agreement with the results obtained by other authors [6]. Accordingly, the highest values are observed in the lower and upper part of the combustion chamber and comprise about 17000 Pa , the average value through the volume of the furnace is 15000 Pa . The vacuum increases as it approaches the exit of the cyclone.

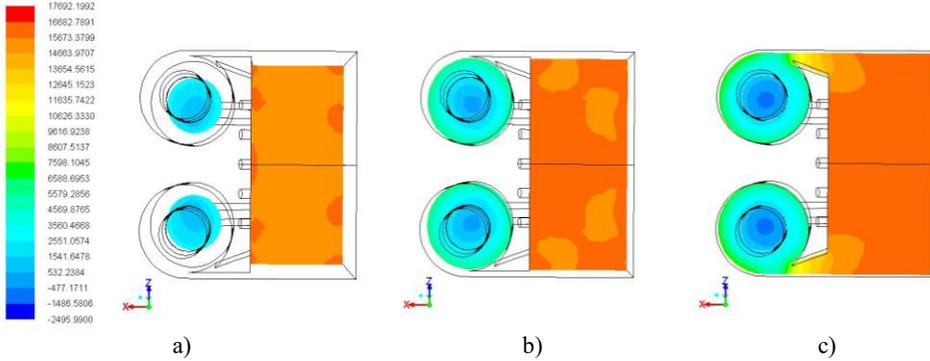


Fig. 3. Contours of static pressure in horizontal sections of the CFB boiler (Pa): a – at 15 meters height; b – at 20 meters height; c – at 23 meters height.

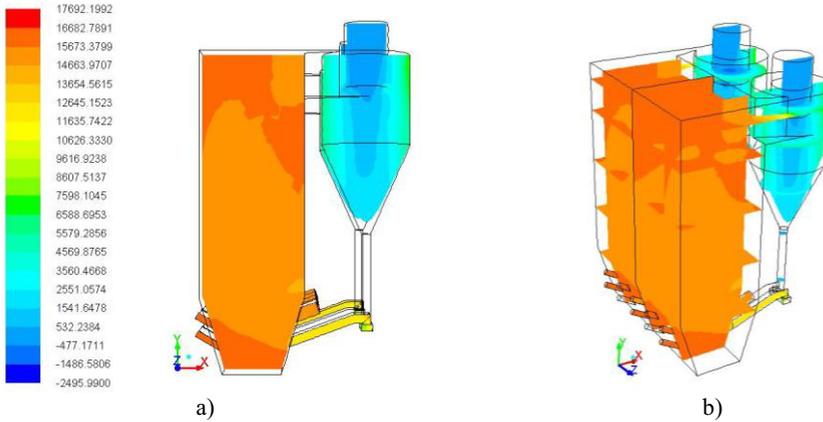


Fig. 4. Contours of static pressure in vertical sections of the CFB boiler (Pa): a – vertical section at a distance of 2.75 meters from the side wall; b – isometric projection with horizontal and vertical sections.

According to the results, the blower fan should be selected in such a way as to create the necessary pressure for the air supply. For the loop seal under the pressure of 7000 Pa and 16500 Pa for the primary air.

Fig. 5 shows graphs of static pressure distribution over the furnace chamber height. In the case of Fig. 5a the pressure drops to 8000 Pa in the area of the recirculation pipelines inlet from the cyclone and the burners embrasure at a height of 2.5 meters in the furnace. Further, the pressure is equalized in the volume of the furnace. In the case of Fig. 5b, the pressure varies from 15200 Pa to 16400 Pa.

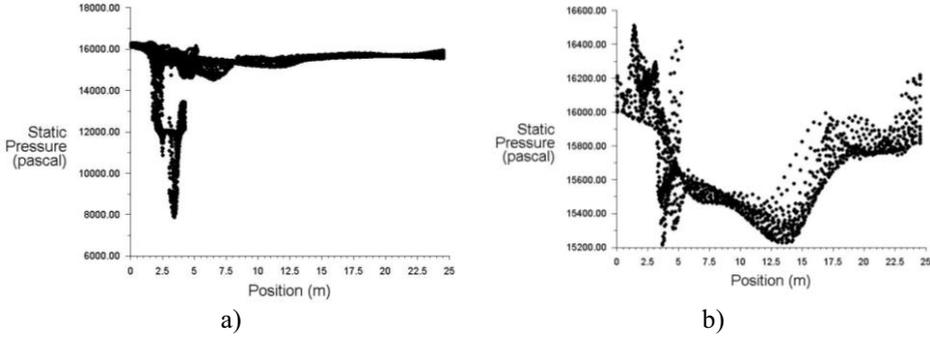


Fig. 5. Graphs of static pressure distribution over the furnace chamber height: a – at a distance of 2.75 meters from the side wall; b – at the center of the furnace.

Fig. 6 presents the results of velocities numerical simulation in the volume of the combustion chamber and cyclone. High-speed flow in the furnace volume fairly uniform and is 6-7 m/s. According to Fig. 6c – f it may be noted a more rapid upward flow in the central zone of the combustion chamber caused by the arrangement of the burners on the back wall of the furnace at the bottom. Then, along the furnace height speed becomes higher in the peripheral parts, where the cyclones are located. Thus in this area can be more intensive abrasive damage of the heating surfaces.

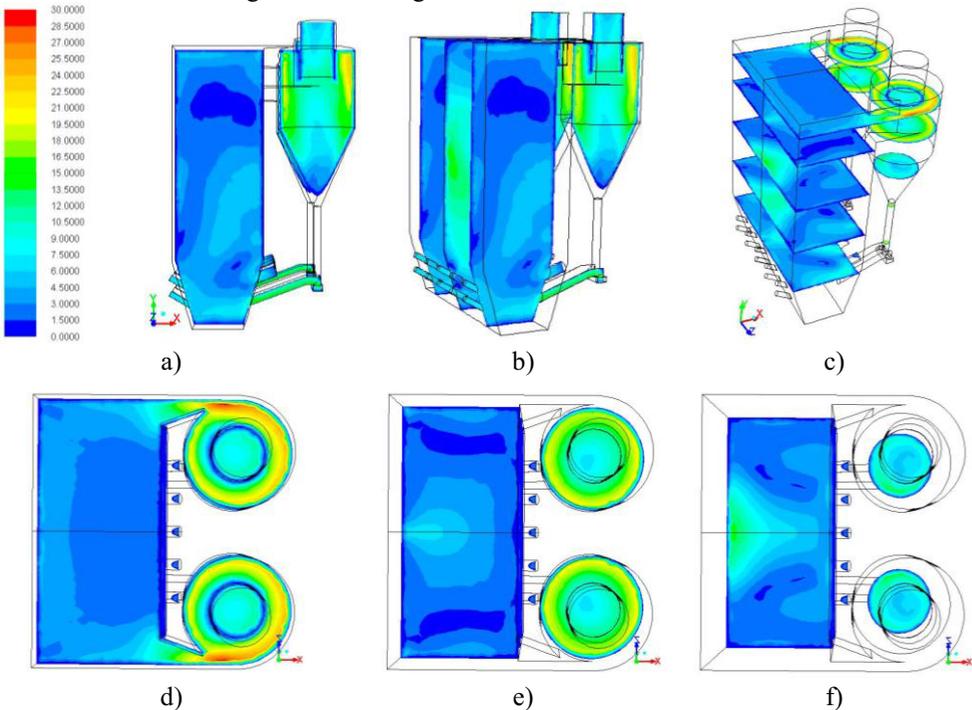


Fig. 6. Contours of velocity distribution in the volume of the boiler with CFB (m/s): a, b – distribution in vertical planes at a distance of 2.75 meters from the side wall; c – distribution in the horizontal planes (5, 10, 15, 20 and 23 m); d, e, f are the velocity distribution at the level of 23, 20 and 15 meters, respectively.

A further increase in speed (Fig. 6, d) corresponds to the desired value for the effective separation of solid particles in cyclones. At the entrance to the cyclone, speed reaches values of 27 m/s.

The speed of fluidization at 850 °C is 4.6 m/s, but the maximum speed in the furnace according to the calculations accepted with a 50% reserve, equal to 7 m/s. As can be seen from Fig. 6, the average velocity over the cross section of the furnace is 7 m/s. At the inlet to the cyclone, the velocity of the particles is 20 m/s. In both cases, the calculated values by the method of mathematical simulation coincide with empirical data.

Consider the trajectory of the particles inside the volume of the boiler. Fig. 7 shows a graph of the particle trajectory colored by the particles ID. For clarity, a part of the particles was not included in graph.

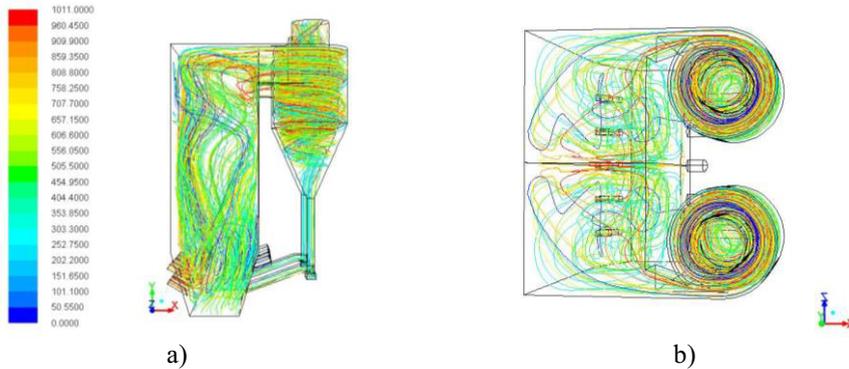


Fig. 7. Pathlines colored by particle ID: a – side view; b – top view.

4 Conclusions

In this work the analysis of the numerical simulation results of the aerodynamic structure of furnace volume, the cyclone and the elements for particles return in CFB boiler with the help of ANSYS FLUENT was made. Presented plots of the velocity distribution and static pressure are correlated with the empirical data. It is worth noting that the adopted design of the boiler, the arrangement of the burners and secondary air nozzles provide a reliable operation and maintenance of the boiler with a stable supply of the fuel and the organization of the external circulation.

References

1. M.H. Bordbar, K. Myöhänen, *Appl. Therm. Eng.* **76**, 344 (2015)
2. A.V. Gil, A.Y. Gil, *EPJ Web Conf.* **82**, 01044 (2015)
3. A.V. Gil, D.A. Baturin, *MATEC Web Conf.* **72**, 01009 (2016)
4. A.V. Gil, D.A. Baturin, *MATEC Web Conf.* **23**, 01009 (2015)
5. Pr. Basu, *Circulating Fluidized Bed Boilers: Design, Operation and Maintenance* (Springer, Switzerland, 2005)
6. N. Zhang, B. Lu, W. Wang, J. Li, *Chemical Engineering Journal*, **162**, 821 (2010)