Numerical simulation of the processes of atomization and combustion of a coal-water slurry in different-scale installations

V A Kuznetsov*, A A Dekterev

Kutateladze Institute of Thermophysics, Novosibirsk, 630090, Russia

*victor_partner@mail.ru

Abstract. Numerical simulation of atomization of a coal-water suspension (CWS) by a pneumatic nozzle is carried out, taking into account the processes of secondary breakup of droplets. The characteristic parameters of atomization were obtained (the dispersed jet opening angle, the size and velocity of the droplets). The data obtained were used to simulate atomization and combustion of the CWS on the firing stand. The results obtained are in good agreement with experimental data. The proposed numerical technique allows research on the introduction of advanced technologies and the improvement of existing different-scale installations (from stands to pilot industrial boilers).

1. Introduction

In the world there is a large amount of waste from coal preparation plants and oil refining waste that pollute the environment. Their number is increasing every year. Coal processing waste is more than 800 million tons and oil refining waste exceeds 1 billion tons [1]. One of the possible and promising directions for disposal of the above wastes is the technology of flame-droplet combustion in the form of fuel compositions based on coal or coal waste.

An important role in the efficiency of application of the flame-droplet combustion technology is played by the process of CWF atomization into the combustion chamber using pneumatic nozzles. This process consists of interaction between three phases: solid, liquid and gas. With regard to the study of atomization and combustion of the CWS in different-scale furnace spaces, it is necessary to consider the primary breakup of the fuel jet, the secondary breakup of the formed droplets, their heating, ignition and combustion of the coal in a high-temperature environment in the presence of combustion products, etc.

Today an increase in the efficiency of fuel combustion for improving the existing and developing new technologies for the combustion of organic raw materials can be hardly imagined without mathematical modeling. Computational fluid dynamics (CFD) is a powerful, affordable and convenient tool for modeling physicochemical processes [2–4]. The development of experimental research methods and the capabilities of computer technology impose new requirements on mathematical models.

2. Problem statement and research methods

With the help of numerical modeling, the authors of the article investigated the structure of the gas flow at the outflow from a pneumatic nozzle for atomizing liquid fuel [2]. To simulate turbulence, the Reynolds stress model (RSM) and the Menter k-ω SST model in a stationary axisymmetric
formulation and URANS, DES, LES methods were used. It is shown that for complex flows the RSM, along with large eddy simulation methods, allows sufficiently correct solutions. We also tested the methodology for modeling atomization of a gas-liquid flow based on the VOF (Volume of fluid) method [5], ELSA (Eulerian-Lagrangian Spray Atomization, VOF to DPM) [6] and Lagrange [7]. The calculation results have shown acceptable agreement with the experiment by the main characteristics of atomization and allowed qualitatively correct determination of the two-phase flow structure, parameters and shapes of droplets, and the velocity and direction of their movement.

It has been determined that the VOF and ELSA (VOF to DPM) methods allow one to obtain a fairly realistic picture of the liquid fuel atomization process (simulates the primary and secondary breakup of droplets). It is worth noting that the VOF to DPM method allows you to reduce the required computing resources by reducing the number of computational cells by 30-40% by optimizing it based on the use of dynamic mesh adaptation. But these methods are still very expensive. Therefore, without huge computational resources, it becomes impossible to simulate the processes of atomization of liquid fuel for different-scale plants.

A way out of this situation is the use of a less costly method based on Lagrange to simulate atomization of liquid fuel. This method does not allow describing the process of the primary breakup of the jet. The flow of the liquid phase is set by a discrete set of droplets. But there is a wide range of models for the secondary breakup of droplets. In this work, to calculate the secondary breakup of droplets, the Reitz wave discontinuity model was chosen, which is suitable for flows with a large Weber number.

Taking into account the data obtained in the studies carried out, a complex mathematical model was proposed for describing the processes of atomization and combustion of the CWS, which includes: model of motion of a multicomponent non-isothermal gaseous medium (carrier phase) based on the RANS/URANS approach; model of radiation transfer based on the method of discrete ordinates; droplet/particle motion model based on the Lagrange approach; combustion model in the gas phase based on a hybrid model combining the mechanisms of chemical reaction and turbulent exchange; char burnout model. The coal-water fuel atomization by the nozzle supplied to the furnace is represented by a discrete set of droplets/particles with given jet opening angle, velocity, droplet size, etc. [8]. In the model, the process of ignition and combustion of a particle occurs in stages. First, the external moisture evaporates; the droplet evaporation model is used to describe this process. A model of ignition and combustion of coal particles is used to describe coal burnout.

3. Results and discussion

Numerical simulation of CWS atomization in a nozzle of the IT SB RAS with the model of the secondary breakup of drops is carried out. Schematic representation of a nozzle is in Figure 1a. The initial air pressure in the annular gas chamber of the nozzle is 1 bar. The CWS flow rate was 200 kg/h. Figure 1b shows the calculation results in the form of a gas flow velocity field and visualization of drops. It can be seen that there is a process of secondary breakup of droplets by an air flow. The graph (figure 2) shows the distribution of droplets by size and velocity at a distance of 25 cm from the nozzle.

![Figure 1. a) Schematic of the nozzle. b) Breakup of CWS droplets, the scale on the left is the velocity magnitude (m/s), the scale on the right is the droplet size (m).](image-url)
Figure 2. Distribution of droplets by size and velocity.

Figure 3 shows the calculation results in the form of the velocity field of the CWS droplets. These results demonstrate the opening angle of the dispersed jet and the characteristic droplet velocity for the selected boundary conditions. The obtained atomization parameters (opening angle, velocity and droplet size) will be used in modeling the combustion of CWS.

The fire stand of SibVTI was chosen to test the technique of numerical simulation of atomization and combustion of CWS [9]. The main elements of the firing stand include a vertical combustion chamber with a diameter of 0.4 m and a length of 6 m. In the experiments, we used a pneumatic nozzle for CWS atomization. The work [9] presents the results of an experiment for CWS prepared on the basis of Irsha-Borodinsky coal. Figure 4 shows the calculation results in the form of temperature and oxygen concentration distribution on the axis of the combustion chamber. The results obtained are in good agreement with experimental data.
4. Conclusion
An analysis of the results obtained indicates that the chosen numerical technique allows (with certain assumptions) simulation of the processes of CWS atomization and combustion in different-scale installations (from stands to pilot industrial boilers). The proposed numerical technique allows research on the introduction of advanced technologies and the improvement of existing power plants.

Acknowledgement
Computational studies were carried out within the framework of the state assignment of the IT SB RAS (reg. 121031800229-1)

References
[1] Kurgankina M, Nyashina G & Strizhak P 2019 Sci. The Tot. Envire. 671 568–77
[2] Alekseenko S, Anufriev I, Dekterev A, Kuznetsov V, Mal'tsev L, Minakov A and Sharypov O 2019 Int. J. Heat and Fluid Flow 77 288–98
[3] Alekseenko S, Kuznetsov V, Mal'tsev L, Dekterev A and Chernetskii M 2019 J. Eng. Therm. 28 (2) 177–89
[4] Butakov E, Burdukov A, Kuznetsov A, Chernova G and Kuznetsov V 2020 T and A 27 (1) 145–52
[5] Kuznetsov V, Shebelev A, Anufriev I, Shadrin E and Minakov A 2019 International Science and Technology Conference "EastConf"
[6] Kuznetsov V, Shebelev A, Minakov A 2019 J.Phys.:Conf.Ser. 1359 012080
[7] Kuznetsov V, Shebelev A, Anufriev I, Shadrin E and Minakov A 2019 J.Phys.:Conf.Ser. 1261 012019
[8] Kuznetsov V, Maltsev L and Dekterev A 2020 J.Phys.:Conf.Ser. 1677 012109
[9] Bogomolov S V, Kozlov S G, Alfimov E G and Fel’ker A A 1995 Sbornik nauchnykh trudov vserossiyskoy konferentsii “Povysheniye effektivnosti i ekologicheskoy bezopasnosti szhiganiya ugley na elektrostantsiyakh Sibiri”. Krasnoyarsk 149–55 (in Russian)