https://doi.org/10.33271/nvngu/2024-2/052

D. L. Bondzyk[1],
orcid.org/0000-0003-3123-1971,
O. V. Baranyuk[1,2],
orcid.org/0000-0001-6008-6465,
M. V. Vorobyov[*1,2],
orcid.org/0000-0001-9621-7658,
M. V. Chernyavskyy[1],
orcid.org/0000-0003-4225-4984,
O. V. Kosyachkov[1],
orcid.org/0000-0002-9445-8738

1 – Thermal Energy Technology Institute of the National Academy of Sciences of Ukraine, Kyiv, Ukraine
2 – National Technical University of Ukraine “Igor Sikorsky Kyiv Polytechnic Institute”, Kyiv, Ukraine
[*] Corresponding author e-mail: vorobiov.nikv@gmail.com

CFD-MODELING OF CRITICAL DEVIATIONS OF COMBUSTION PROCESSES IN PULVERIZED COAL BOILERS. PART 1. CONSTRUCTION OF THE TPP-210A BOILER CALCULATION MODEL

Purpose. Development of a mathematical model for predicting critical deviations of furnace processes during the operation of anthracite boilers converted to burning sub-bituminous coal, including in non-design modes of operation. Verification of the calculation model of the TPP-210A boiler of the 300 MW power unit on the design fuel – anthracite for further analysis of its operation on sub-bituminous coal.

Methodology. Modelling of solid fuel combustion was performed with the help of finite-element CFD models of the boiler unit in the ANSYS-Fluent software complex with the determination of the relevant characteristics of the boiler.

Findings. Finite-element CFD models were developed according to the working drawings of the TPP-210A boiler and its burner device, reconstructed for burning sub-bituminous coal. Verification of the results of CFD modeling was carried out for the case of burning the design fuel – anthracite in the design operating modes. It is shown that the discrepancy between the results and the experiment does not exceed 6.5 %, which allows the use of the developed computer model for simulating the burning of sub-bituminous coal, including in non-design operating modes. The obtained results are the basis for further calculations of the operation of the TPP-210A boiler of the 300 MW power unit on sub-bituminous coal with the determination and minimization of the factors of critical deviations of furnace processes that lead to thermal damage of wall screens.

Originality. For the first time, CFD modelling has taken into account all the design features of the burner devices of the TPP-210A boiler, reconstructed for burning sub-bituminous coal with the introduction of a steam ejector of a high-concentration dust pipe under rarefaction to the central air channel. For the first time, the distribution of thermal resistances has been applied as boundary conditions on the walls of the fuel tank for the case of burning sub-bituminous coal. The validity of these approaches was confirmed by the verification of the results.

Practical value. The verified computer model of the TPP-210A boiler of the 300 MW power unit converted to burning sub-bituminous coal will allow determining and minimizing the factors of critical deviations of furnace processes which lead to thermal damage of wall screens. This will contribute to increasing the reliability and improving the technical and economic indicators of the boilers of the Trypilska TPP, where 3 such boilers are operating, and other power plants (Zmiyivska, Kryvorizka, Prydniprovska), where similar anthracite boilers are planned to be converted to sub-bituminous coal.

Keywords: *pulverized coal boiler, anthracite, sub-bituminous coal, CFD modelling, radiant and convective heat exchange*

Introduction. The initiatives of developed countries to urgently switch to “carbon-free” energy, followed by Ukraine [1], face objective difficulties. On a global scale, this is the unbearable volume of necessary investments into renewable generation for many countries and a significant rise in the price of natural gas, which was considered as a fuel for the transition period, taking into account the sanctions on Russian energy resources; in view of this, the reduction of energy use of coal in the world by 2050 is projected to be no more than 54 % [2]. For Ukraine, during wartime and the first postwar years, these difficulties are even greater. At the same time, the temporary occupation and/or destruction of a number of mines in the Donbas over the past two years has resulted in the loss of less than 25 % of commercial reserves and only 15 % of steam coal production [3]. Therefore, it can be expected that in Ukraine, as in the rest of the world, the development of renewable sources will go on for at least 10–20 years in parallel with energy production from fossil fuels, including coal [1].

Ukraine’s coal-fired power industry faces a specific problem in that, due to the significant deposits of Donetsk anthracite, about half of the power units of thermal power plants had boiler units designed specifically for burning this low-reactive fuel, and since 2017, they have been facing a cessation of supplies of the design fuel [1, 3]. Meanwhile, the remaining solid fuel resources in the Pavlohrad region and in the Lviv-Volyn basin are almost exclusively represented by hard coal – the so-called “gas group” (according to the national classification DSTU 3472:2015) or subbituminous coal (according to ASTM D388-23) [4]. Subbituminous coal differs from anthracite and semi-anthracite in its high volatile matter yield for a dry ash-free state (35–45 % vs. less than 18 %) and high reactivity of the coke residue, which determine its greater propensity to spontaneous ignition and explosiveness in dust systems and a higher burning rate in a pulverized coal flame. Therefore, the use of subbituminous coal in anthracite boilers required their certain reconstruction, which primarily covered the dust system (use of flue gases with an oxygen concentration of less than 16 % instead of hot air for dust drying and transport) and burners [5].

In 2017–2021, 4 anthracite boiler units of TP-90 type (150 MW) at Prydniprovska TPP, 3 boiler units of TP-100 type (200 MW) at Zmiivska TPP, 3 boiler units of TPP-210A type (300 MW) at Trypilska TPP, and 1 unit of P-50 type at Kryvyi Rih TPP were converted to subbituminous coal combustion [3]. 2 of the 4 TP-90 boilers had tangential direct-flow burners, which were replaced with swirl burners when they were converted to subbituminous coal, while the rest of the boilers had swirl burners from the very beginning. All boilers have wet bottom ash removal, but differ significantly in the shape of the furnace chamber: TP-90 and TP-100 boilers have a prismatic,

---

so-called open type, while TPP-210A and P-50 boilers have a so-called semi-open type furnace chamber with a constriction that separates the combustion chamber with fully shotcreted screen surfaces in the lower part and the afterburner and cooling chamber with open screen surfaces in the middle and upper parts.

For a long time, all boilers operated without incident. At the end of 2021, cases of thermal damage to the screen surfaces on the front and rear sides of the furnace in the area above the constriction started in boilers of the TPP-210A type. In [1], based on studies and tests performed by TETI of NAS of Ukraine in difficult wartime conditions, it was substantiated that the cause of the damage was not the conversion of boilers to burn "gas" coal in itself, but a combination of factors associated with changes in the operation of boilers, features of furnace, thermal-hydraulic processes, and properties of non-designed fuel. The recommendations provided to minimize the impact of these factors significantly reduced the probability of damage, but did not eliminate it completely.

The difficulty of preventing critical deviations in furnace processes lies, in particular, in the fact that a number of their parameters, such as fields of concentrations of oxidant, unburned carbon, temperatures and heat flows along the height and cross-section of the furnace, cannot be directly measured, so that tests can only identify their irreversible consequences, not the factors of deviations themselves. The solution in this case is mathematical modelling, which has long been effectively used to optimize furnace processes. In addition, mathematical modelling is irreplaceable in determining the peculiarities of operation in a wide range of boiler load control, which is especially important with the strengthening of the regulatory function of thermal energy [1, 3].

Purpose. This work is concerned with the development of a mathematical model for predicting critical deviations of furnace processes during the operation of anthracite boilers converted to subbituminous coal combustion, including in non-designed operating conditions. In this part of the work, finite-element CFD models of the TPP-210A boiler and its burner, reconstructed for the combustion of subbituminous coal, were developed, and the results of CFD modelling were verified for the case of combustion of the design fuel – anthracite in the design operating conditions.

Literature review. World scientific practice shows that all new research is carried out using both experimental research and numerical calculation methods [6], in particular CFD (Computational Fluid Dynamics) modelling, which makes it possible to significantly speed up scientific research and reduce the material costs for conducting it.

Modern studies of heat transfer processes are largely based on the use of such well-known software packages as ANSYS-Fluent and OpenFoam. The use of such complexes allows for a more complete study of heat transfer processes in a wide range of boundary/initial conditions, taking into account the maximum possible factors that affect these processes. Among them, the geometry of the fuel chamber, the design of the burners, the distribution of fuel and oxidant supply, and the aerodynamics of flows through the burner channels are crucial. As mentioned above, the TPP-210A boiler is a wet bottom ash removal boiler, its fuel chamber contains a constriction zone, and the lower part (pre-furnace) is equipped with opposing swirl burners.

It is noted in [5] that this design of the fuel chamber was widespread mainly for boilers using low-reactive coal, which requires high combustion temperatures and the organisation of "early ignition" of the pulverised coal flame due to the ejection of high-temperature combustion products from the flame core to the burner mouth under the influence of low pressure created by swirling flows of primary and secondary air. However, for such a fuel and such a design, taking into account the high flame temperatures and the required excess of oxidant, high generation of thermal nitrogen oxides is typical. Given the gradual tightening of environmental restrictions, in recent years, the world has been developing technologies for combustion of subbituminous coal and lignite with reduced generation of nitrogen oxides, possibly with the addition of biomass fuel to reduce sulphur dioxide emissions and specific CO_2 emissions. Mathematical modelling of furnace processes using computational fluid dynamics (CFD) methods and the ANSYS FLUENT software package was also developed in these directions [7].

The first direction is to create a collective swirl flame through the interaction of jets from direct-flow tangential burners located in several levels in height, with a corresponding decrease in the temperature in the core and a more complete filling of the volume. For example, in [8], the aerodynamics of a collective distributed flame was studied using CFD modelling and the optimal ratio between the reduction of nitrogen oxides generation and the degree of burnout of Australian sub-bituminous coal was sought. Researchers [9] studied the combustion characteristics of coal from the Chinese provinces of Zhundong and Beitashan, which is close in composition to subbituminous coal, for a 600 MW tangentially combusting boiler using numerical modelling. The correlation between the intensity of high-temperature corrosion and NO_x formation at different oxidant flow rates in the flame core was optimised.

The study [10] investigated the combustion of Malaysian subbituminous coal and lignite, also in a fuel boiler with multi-tiered tangential direct-flow burners. The standard Navier-Stokes-Reynolds (RANS) averaged equation was used to solve the continuity, momentum, and energy equations. The implemented k-ε model was used as a closure for the turbulent model. The combustion equation was adopted assuming a model without pre-mixing. Coal particle tracking was performed using a discrete phase model, and the distribution of coal particles was assumed to follow the Rosin-Rammler distribution. A somewhat unexpected result was the shift of the maximum temperature zone to the upper part of the furnace when using lower-calorific lignite; however, this correlates well with the greater heat transfer from the flame core due to the higher specific (relative to the heat of combustion) flow rate of combustion products of lignite with a higher moisture and oxygen content in the organic part compared to subbituminous coal.

In [11, 12], for the same geometry of the fuel chamber and burners, the co-combustion of fuels with different calorific value and reactivity was studied – low-reactive and subbituminous coal as well as subbituminous coal and sludge from waste processing plants, respectively. From a methodological point of view, these studies are valuable because they extend the standard ANSYS FLUENT package, which is designed for a single fuel and is usually applicable for mixtures only with an average of their characteristics, to the case of feeding two fundamentally different types of fuels to different burner levels.

In the second direction, swirl burners with a high-temperature flame core are retained, the reduction of nitrogen oxides generation is achieved by the lack of oxidant in the core, and the combustion of the generated CO occurs in the tertiary air supplied by the level above (OFA technology). This method is suitable for both subbituminous [13] and low-reactive coals [14]. However, in CFD modelling studies of such multi-stage combustion, such as [13, 14], detailed simulations of the processes in the swirl burner are usually sacrificed for the sake of a detailed description of the interaction of the flame with the secondary and tertiary oxidant streams.

The smallest number of works is devoted to modelling a "classical" furnace with counter swirl burners. In [15], the combustion of high-ash Kazakhstani bituminous coal in a boiler with a steam capacity of 75 t/h is modelled using ANSYS FLUENT, and in [16] subbituminous coal is modelled in a boiler of a 620 MW power unit. In both cases, the geometry of the furnace is prismatic, open (of constant cross-section in height). The equipment of the boilers with diagnostic tools allowed us to verify the models in detail, achieving a high degree of coincidence with the test data.

The works by national researchers are mainly devoted to CFD modelling of the promising for Ukraine co-combustion of coal with peat and various types of solid biofuels. In [17], the results of a numerical study of the processes of co-combustion of peat and lignite in a 2.5 MW swirl furnace are presented. It was established that at an ash content of $A^r = 30–35$ % and a moisture content of $W^r = 30–35$ %, the combustion process is unstable due to the problem of removing moisture and ash from the furnace volume. The temperature of flue gases at the furnace outlet during the combustion of peat and lignite is 1,711 and 1,888 °C, respectively, which, under the condition of excess air $\alpha = 2.0–2.3$, provides a high degree of particle burnout even with a limited time in the furnace.

The paper on mathematical modelling of the operation of a low-power boiler using pellets from wood and agricultural waste [18] gained experience in calculating the combustion of non-design fuels without changing the geometry of the furnace and verifying the calculation model.

CFD modelling of the processes in the TPP-210A boiler was performed in [19, 20] for the co-combustion of anthracite with fuel biomass, and in [21] for the co-combustion of anthracite with subbituminous coal. As in [11, 12], in these studies, the ANSYS FLUENT standard package was adapted to separate the supply of two fundamentally different types of fuels to different burners or burner channels. In all cases, earlier ignition and deeper burnout of anthracite in the presence of an impurity of 10–30 % by heat of the more highly reactive fuel was observed.

The work by M. Nekhamin [22], which was performed to model the operation of the TPP-210A anthracite boiler using AN-SYS FLUENT, is most closely related to the direction of this study. Despite the fact that the geometric model [22] had certain constructive simplifications of the furnace and burners, the valuable feature of this work is the proving that the most correct boundary condition on the boiler walls is not just the temperature of the wall surfaces, but the thermal resistance. The ANSYS FLUENT software [7] makes it possible to use such a boundary condition without the need to adjust the mesh model (i.e., selecting walls of a certain thickness and creating a separate mesh for them). This possibility is realised by specifying the wall thickness, thermal conductivity of the material, and temperature on the outside of the wall, which is taken into account in the program using an additional analytical solution of the one-dimensional equation along the normal to the wall. In addition, the use of the fuel temperature of the medium from the boiler's passport thermal calculation for the height distribution boundary conditions allowed us to obtain results that correlate with the known data from anthracite combustion tests in this boiler (Shagalova, 1976), which indicates the correctness of the CFD model development. In particular, an increase in the flame temperature and heat fluxes in the zone above the constriction was found, which, although insufficient for thermal damage to the screens, is completely consistent with the risk zone when burning anthracite, but completely coincides with a risk zone when subbituminous coal is burned.

Unresolved aspects of the problem. To achieve the purpose set in this work, it is necessary to fully take into account the design features of the semi-open fuel chamber and burner devices of the TPP-210A boiler, reconstructed for the combustion of subbituminous coal, which has not been done before. After verification of the developed model on anthracite, it will be necessary to perform for the first time computational studies of the distribution of temperatures and heat fluxes over the height and cross-section of the fuel chamber during the combustion of subbituminous coal during design and non-design modes of operation of the TPP-210A boiler, in particular, with a lack of oxidant, excessive air intake, uneven distribution of dust and oxidant over the burners, etc.

Methodology. The supercritical pressure direct-flow boiler TPP-210A with a steam capacity of 950 t/h and the technical solutions used to convert it to subbituminous coal combustion are described in detail in [23]. The boiler has two identical independent U-shaped shells. The fuel chambers of the shells are about 34.5 m high, 10.9 m wide at the front, and 7.8 m deep. The depth clamp, which reduces the cross-section of the fuel chamber by approximately half, is located 8.5 m above the bottom. Each shell is equipped with six or seven pulverised coal and gas burners, which are located in a single level on the front and rear walls of the pre-furnace at a height of about 4 m from the bottom. The swirl burners are arranged in a one level opposite each other, 3 on the front and rear walls, the middle ones are directed in such a way as to create a swirling flow in the clockwise direction, the rest swirl in the opposite direction. The burner has a maximum capacity of 12 tonnes per hour for coal dust. When working on anthracite, dust from the intermediate bin was transported to the burners through high concentration dust pipelines under pressure, which were cut into the primary air pipelines 2–3 m before the burners, so that an air mixture of dust and hot primary air entered the primary air ducts. The specific feature of this type of swirl burners is the presence of two secondary air channels, which makes it possible to maintain the air velocity in the inner channel at partial loads by partially or completely covering the peripheral channel.

When the TPP-210A boilers were converted to sub-bituminous coal combustion [23], the dust system was mainly reconstructed. The combination of "ball mill – separator – cyclone – mill fan" with the dust collected in the cyclone being dumped into a common intermediate bin remained unchanged, but a mixture of hot air and flue gases taken from the convection shaft with an oxygen content of no more than 16 % was supplied to the mill as a drying agent. The spent drying agent was directed to the primary air ducts of the existing burners instead of the discharge nozzles, which were plugged. Due to the explosion and fire safety requirements, the high concentration dust transport under pressure created by a high-pressure fan was replaced by the high concentration dust transport under vacuum created by steam ejectors with Laval nozzles. The outlet pipes of the steam ejectors with conical outlet dividers are introduced along the axis of the central air channel (Fig. 1), so that the dust, which is dispersed evenly in all directions by the dividers, mixes with air at the outlet of the burner's central air channel. The rest of the design and geometry of the burners remained unchanged.

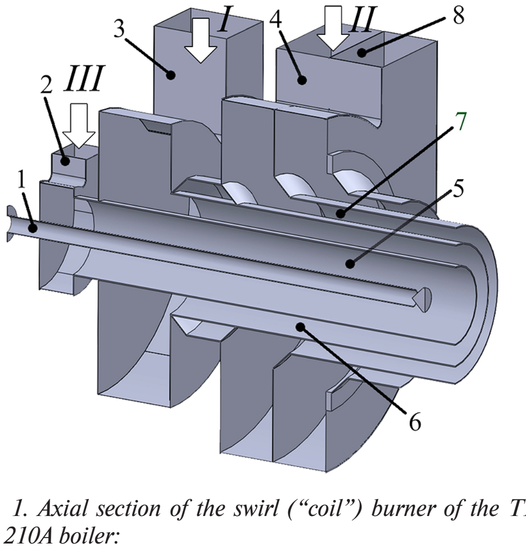

Fig. 1. Axial section of the swirl ("coil") burner of the TPP-210A boiler:

I–III – supply of primary, secondary and cooling air, respectively; 1 – steam ejector; 2 – central air swirler; 3 – primary air swirler; 4 – secondary air swirler; 5 – central air duct; 6 – primary air duct; 7 – secondary air ducts; 8 – secondary air flow control valve between the internal and peripheral channels

The most difficult tasks are to model a semi-open fuel chamber and a swirl burner of complex geometry with dust supply by a steam ejector with a splitter. To reproduce the operation of the steam boiler, a geometric model was created, on the basis of which a finite element model was built (Fig. 2) with the corresponding boundary conditions and a mathematical description of the calculation process. The ANSYS-Fluent software was used to implement the model. In this study, a hybrid mesh was used, which is a combination of a structured and unstructured finite element mesh. The distance between the mesh nodes was regulated by the Revelance Centre parameter with a value of "Fine", which determines the mesh density within the specified limits. The size of the grid elements was set in the Element Size field. The MultiZone method was used to combine the structured and unstructured calculated finite element grids.

The CFD model was used to determine the flow of the continuous gas phase and its interaction with the discrete phase of coal particles. When coal particles pass through the gas, they release gaseous combustibles that serve as a source for the combustion reaction. The well-known species transport model or the non-premixed combustion model can be used to model the reactions.

Fig. 3 shows a finite-element mesh view of a swirler designed to swirl the secondary air supplied to the combustor. The surfaces on which the boundary layer should form are indicated near the solid wall using the Inflation function. This is done for correct modelling of the flow in the swirler. To ensure a correct flow in the boundary layer, 10 layers are used when setting up the Inflation function.

The view of the finite element swirler mesh of the primary air supply is shown in Fig. 4. The continuum inside the swirler was approximated by a set of tetrahedral finite elements, since the design of the steam ejector with a conical splitter at the end did not allow the use of other types of finite elements.

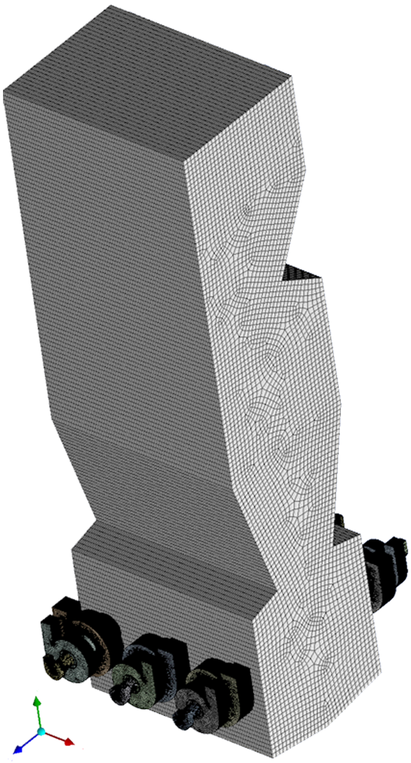

Fig. 2. The appearance of the finite element mesh of the fuel chamber in three-dimensional space

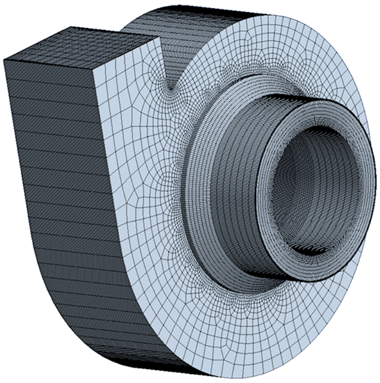

Fig. 3. Exterior view of the finite element mesh of the secondary air swirler

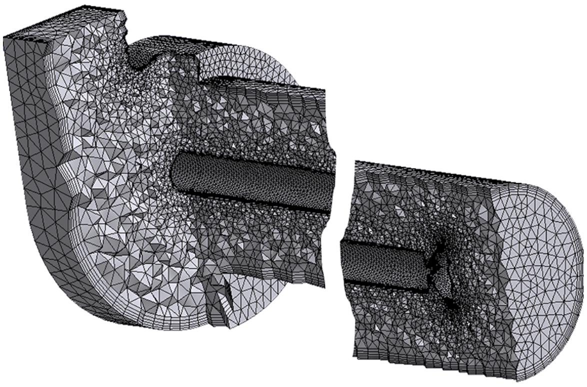

Fig. 4. Visualization of the finite element mesh image inside the central air supply swirler model

Fig. 4 shows a thickening of the finite element mesh near the solid walls of the model. This is the result of using the Inflation function to model the development of the boundary layer. As in the case of the swirler simulation for the secondary air supply, 10 layers are used in this model when setting up the Inflation function to ensure correct flow in the boundary layer.

As a result of the adjustment experiments, the value of the density of the computational mesh was determined, which does not affect the solution. The evaluated finite element mesh quality parameters to be used in further studies included a Skewness of no more than 0.8 and an AspectRatio of no more than 40. These parameters help to ensure a high-quality and accurate approximation of the geometry and flows in the model, which is important for reliable calculation results.

The computer model was developed using the ANSYS-Fluent software package and contains the continuity equations, Reynolds-averaged Navier-Stokes equations, energy equations, and the transport equation for the i^{th} component of the mixture. To close the Reynolds-averaged Navier-Stokes equations, the Realizable k-ε turbulence model was used, since this model is designed to calculate flows containing streams and simulate areas with highly streamlined curvature (in this problem, the surface of swirlers that twist the flow). The governing equations used are the Navier-Stokes system of equations for the flow of a viscous compressible fluid with variable properties [7, 10]

$$\frac{\partial \rho u_i}{\partial \tau}+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial p'}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)\right]+S_m,$$

where S_m is equalizer of mass forces; μ_{eff} is effective viscosity, which is the sum of molecular and turbulent viscosity; p' is modified pressure, which is defined as follows [24]

$$p'=p+\frac{2}{3}\rho k+\frac{2}{3}\mu_{eff}\frac{\partial u_k}{\partial x_k}.$$

The additions on the left side of the system of equations determine the change in flow properties over time and the amount of motion of the molecules in the medium. The additions on the right side of the equations determine the effect of mass forces, pressure forces, and viscosity forces. Thus, the equations are considered as a balance of inertial forces (left side), mass forces, pressure forces, and viscosity forces (right side) acting on a particle of the medium.

Reynolds-averaged energy equations

$$\frac{\partial \rho h_{tot}}{\partial \tau}-\frac{\partial p}{\partial \tau}+\frac{\partial}{\partial x_i}(\rho u_j h_{tot})=$$
$$=\frac{\partial}{\partial x_j}\left(\lambda\frac{\partial T}{\partial x_j}+\frac{\mu_t}{Pr_t}\frac{\partial h}{\partial x_j}\right)+\frac{\partial}{\partial x_j}\left[u_i(\tau_{ij}-\rho\overline{u_i'u_j'})\right].$$

Although the transformation of the molecular diffusion term may be inaccurate if the enthalpy h_{tot} depends on parameters other than temperature, the latter equation, written in terms of turbulent diffusion, is correct, provided that the eddy-diffusivity hypothesis is satisfied. Moreover, since turbulent diffusion is usually much larger than molecular diffusion, small errors in the latter can be ignored.

Together with the continuity equation, the Navier-Stokes and energy equations form a closed system

$$\frac{\partial \rho}{\partial \tau}+\frac{\partial(\rho u_j)}{\partial x_j}=0.$$

The dependence for calculating the turbulent viscosity is

$$\mu_t=C_\mu\rho\frac{k^2}{\varepsilon},$$

where C_μ is a constant; k and ε are the kinetic energy of turbulent pulsations and their dissipation rate, respectively.

To simulate turbulent fluid motion, the Navier-Stokes equations in the Reynolds form are closed using the k-ε turbulence model

$$\underbrace{\rho u_i\frac{\partial k}{\partial x_i}}_{Convection}=\underbrace{\mu_t\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)\frac{\partial u_j}{\partial x_i}}_{Generation}+\underbrace{\frac{\partial}{\partial x_i}\left\{(\mu_t/\sigma_t)\frac{\partial k}{\partial x_i}\right\}}_{Diffusion}-\underbrace{\rho\varepsilon}_{Destruction};$$

$$\underbrace{\rho u_i\frac{\partial k}{\partial x_i}}_{Convection}=\underbrace{C_{\varepsilon 1}\left(\frac{\varepsilon}{k}\right)\mu_t\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)\frac{\partial u_j}{\partial x_i}}_{Generation}+$$
$$+\underbrace{\frac{\partial}{\partial x_i}\left\{\left(\mu_t/\sigma_t\right)\frac{\partial k}{\partial x_i}\right\}}_{Diffusion}-\underbrace{\rho C_{\varepsilon 2}\left(\frac{\varepsilon^2}{k}\right)}_{Destruction},$$

where $k=\frac{1}{2}\overline{u_i u_j}$ is kinetic energy of turbulent pulsations; $\varepsilon=\nu\overline{\frac{\partial u_j}{\partial x_i}\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)}$ is the rate of dissipation of turbulent kinetic energy.

Non-equilibrium wall functions are used to model the flow in the boundary layer. Their advantages are that they are designated to take into account the sensitivity of the logarithmic law, which postulates the distribution of the flow velocity in the boundary layer, to the pressure gradient. Thus, they are better at predicting the occurrence of negative pressure gradients and the appearance of backflows than standard near-wall functions.

Weakened assumptions of local equilibrium for CHP in neighbouring cells

$$\frac{\tilde{U}C_\mu^{\frac{1}{4}}k^{1/2}}{\tau_w/\rho}=\frac{1}{\kappa}\ln\left(E\frac{\rho C_\mu^{\frac{1}{4}}k^{1/2}y}{\mu}\right),$$

where

$$\tilde{U}=U-\frac{1}{2}\frac{dp}{dx}\left[\frac{y_v}{\rho\kappa^* k^{1/2}}\ln\left(\frac{y}{y_v}\right)+\frac{y-y_v}{\rho\kappa^* k^{1/2}}+\frac{y_v^2}{\mu}\right],$$

where y_v is the distance from the impenetrable solid wall to the upper boundary of the viscous sublayer (usually is $y+\approx 5$); y_v is the distance from the impenetrable solid wall to the upper boundary of the turbulent core.

The temperature in the boundary layer is calculated as follows

$$T^*=\frac{(T_w-T)\rho C_\mu^{1/4}k^{1/2}}{\dot{q}''},$$

where C_μ according to the Realizable k-ε turbulence model used is

$$C_\mu=\frac{1}{\left(A_0+A_s\frac{U^*k}{\varepsilon}\right)}.$$

After setting the hydrodynamic characteristics of the flow, a non-premixed combustion model was used. The starting point for its use is the creation of a PDF table (PDF – probability distribution function), which contains information on the dependence of the content of components and temperatures on the fractional composition of the mixture. ANSYS-Fluent uses this table to obtain the values of these parameter values during the calculation. When using the combustion model without pre-mixing, all thermodynamic parameters are extracted from the prePDF chemical database and entered into ANSYS-Fluent as a pdf-mixture material.

The Equilibrium Chemistry equations, which are much more accurate, were used to determine the PDF table. With this model, it is possible to include the effects of intermediate reactions and dissociation reactions, creating more realistic flame temperature predictions than the conventional Eddy-Dissipation model. To correctly predict NO_x formation, the Laminar Flameless option is used, which enables to include aerodynamic deformation to account for non-equilibrium effects such as super-equilibrium radical concentration and sub-equilibrium temperatures. The need to simulate the combustion of both anthracite (for model verification) and subbituminous coal of different composition determines the purpose of setting the values of the chemical components of the reaction in the computer model, which are part of both the dry-ash-free basis and the "received" fuel content (ultimate analysis).

In establishing the boundary conditions on the walls of the boiler unit, the approach of M. Nekhamin [22] was used and the values of effective thermal resistances determined by him on the basis of the boiler passport and its hydraulic scheme. The temperature of the outer surface of the wall was taken as the temperature of the heat carrier (steam or water) T_{STEAM} flowing in the screens. Thus, the value of the effective thermal resistance of the walls takes into account the resistance of the boundary layer of the coolant. It is substantiated that since the operation of the boiler and turbine itself should not change when changing fuel, the same values should be preserved for the case of burning subbituminous coal. The parametric boundary conditions are shown in Fig. 5.

During the modelling, it is assumed that coal is fed into the boiler furnace at a rate of 60 tons per hour. Atmospheric pressure is set at the outlet of the fuel furnace.

To determine the absorption coefficient, the wsggm-domain-based model is used, where the dependence of the absorption coefficient on the composition is applied using the weighted-sum-of-grey-gases model.

The flow of solid fuel particles is modelled using a discrete phase model. This model assumes the motion of individual particles and includes the exchange of momentum, heat and mass between the gas and coal particles. Calculations of particle trajectories alternate with calculations of the continuous

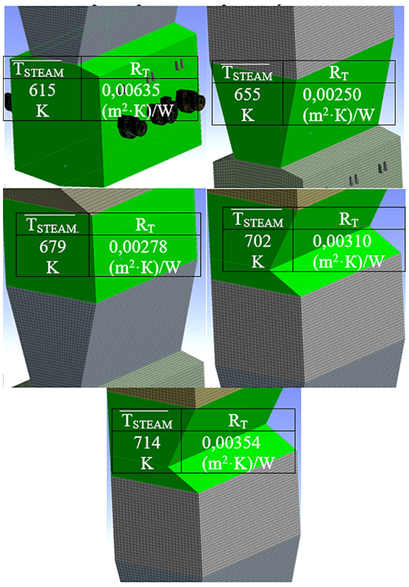

Fig. 5. Parameters of thermal boundary conditions on the walls of the TPP-210A boiler

gas phase. The coal particles have a uniform diameter distribution in the range from 70 to 200 micrometres. The size distribution corresponds to the Rosin-Rammler equation with an average size of 134 micrometres and a spread parameter of 4.52. The chemical composition of the fuel used in this study was selected on the basis of the research of the coal chemical laboratory of the Separate Enterprise "UKRNDIVUGLEZBAHACHENNYA" of the State Enterprise "VUGLEINOVATSIA". The data on the chemical composition of the fuel are shown in Fig. 6.

Fuel lower calorific value in terms DAF dry-ash-free, kJ/kg, is calculated using the well-known Mendeleev formula

$$Q_l^P = 339C^P + 1{,}030H^P + 109(O^P - S^P) - 25W^P.$$

Fuel high calorific value, kJ/kg

$$Q_h^P = Q_l^P + 25(W^P + 9H^P).$$

Verification results. The computational model was verified using the data from M. Nekhamin's computational experiment with the TPP-210A anthracite boiler [22] and the data from tests of anthracite combustion in this boiler (Shagalova, 1976), which M. Nekhamin also used to verify his model.

The temperature distribution along the central axis of the fuel chamber (Fig. 7) shows that the flue gas temperatures determined by the computational model developed by the authors are in good agreement with the modelling results [22] and test data (Shagalova, 1976) at a height of 10 to 35 m. As expected, the distribution of flue gas temperatures over the height of the TPP-210A boiler fuel chamber above the level of the burners is monotonic, with the temperature decreasing from a maximum of 1,600 °C above the burners to 1,010 °C at the outlet of the fuel chamber due to heat exchange with the screen walls. The modelling results [22] show a temperature 100–180 °C higher than the test results, as if the calculated flame is "late" compared to the real one. In our opinion, this is due to the not quite perfect modelling of the burner itself, which leads to a delay in the ignition of the model flame. In the proposed model, the calculation differs from the test data by only 30–100 °C with a tendency to converge temperatures along the course of the combustion products. Hence, it can be concluded that the proposed model is more consistent with the real process in the fuel and satisfactory verification based on the test results.

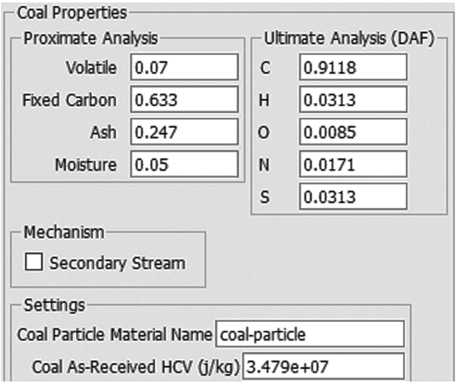
Coal Properties
Proximate Analysis
Volatile 0.07
Fixed Carbon 0.633
Ash 0.247
Moisture 0.05
Ultimate Analysis (DAF)
C 0.9118
H 0.0313
O 0.0085
N 0.0171
S 0.0313
Mechanism
☐ Secondary Stream
Settings
Coal Particle Material Name coal-particle
Coal As-Received HCV (j/kg) 3.479e+07

Fig. 6. Chemical composition of the fuel, designed according to the requirements of ANSYS-FLUENT to generate a PDF table for subbituminous coal

The maximum deviation of the authors' simulation results from the experimental data does not exceed 6.5 %. This indicates that the ANSYS-Fluent program simulates the flow of heated gases in channels bounded by heated walls quite well.

Figs. 8, 9 show the temperature distribution in the longitudinal and transverse sections of the boiler and the trajectory of the flow particles. The longitudinal section was chosen so that it intersects the burner, which is closest to the observer. It can be seen that the area occupied by the flame and the shape of the flame torch are classical for the pulverized coal combustion.

As can be seen from Figs. 8, 9, streams from opposite burners collide, which leads to the formation of zones with circulation flow "above" and "below" the gas stream. These circulation zones occupy space in the corners of the furnace and lead to acceleration of the fuel flow in the area of the narrowest cross-section. The flue gas temperature in this area corresponds to the flame temperature. As a result, the highest values of heat fluxes and heat transfer coefficients from flue gases to the walls of the furnace screens should be expected in this area.

The calculated distribution of heat fluxes along the height and plane of the furnace (Fig. 10) correlates well with the results of [22] and indicates an increase in heat flux to the walls in the zone above the constriction. It can be expected that modelling the operation of a boiler using subbituminous coal, the degree of burnout of which in the pre-furnace will increase due to its greater reactivity, will show an additional increase in the heat flux to the walls in this zone, which fully corresponds to the cases of thermal damage to the screen surfaces on the front and rear sides of the furnace in the zone

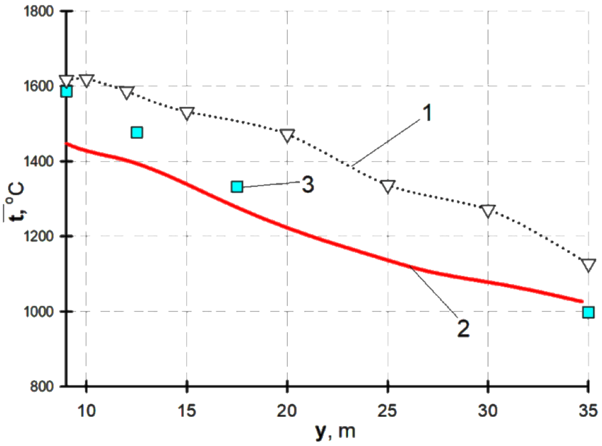

Fig. 7. Verification of the computational model of the TPP-210A boiler in the case of anthracite combustion:

1 – data by M. Nekhamin [22] *(average mass temperatures); 2 – author's original modelling results; 3 – test data* (*Shagalova, 1976*)

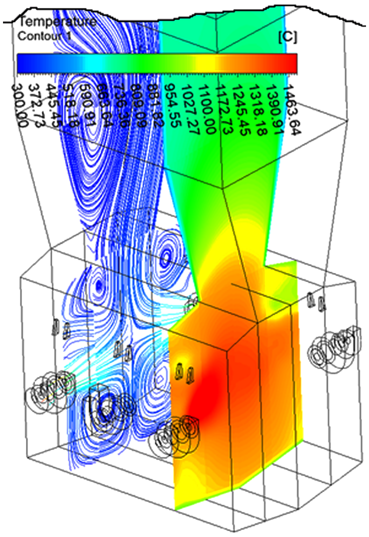

Fig. 8. Temperature distribution in the longitudinal (across the burners) section of the computational model of the TPP-210A boiler

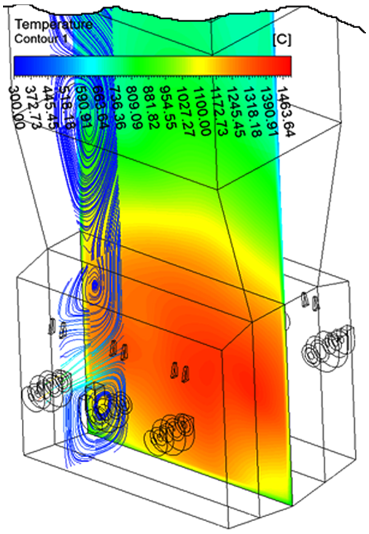

Fig. 9. Temperature distribution in the cross-section of the computer model of the TPP-210A boiler

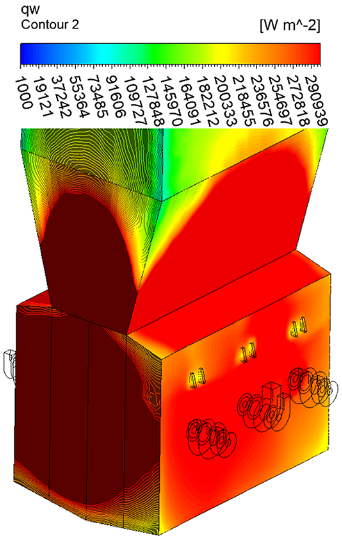

Fig. 10. Distribution of heat flows on the walls of the computer model of the TPP-210A boiler

above the overpressure described in [1]. The second part of the paper will be devoted to these computational studies, including non-designed operating conditions (with excessive air intake into the furnace and insufficient excess oxidizer in the burners, uneven distribution of dust and oxidizer over the burners, etc.

Conclusions and prospects for further research development. In this work, finite-element CFD models were developed according to the working drawings of the TPP-210A boiler and its burner device, reconstructed for the combustion of subbituminous coal. For the first time, CFD modelling has taken into account all the design features of the burner devices of the TPP-210A boiler, reconstructed for the combustion of subbituminous coal with the introduction of a steam ejector of a high concentration dust duct under a vacuum into the central air channel. For the first time, the distribution of thermal resistances was used as boundary conditions on the walls of the furnace for the case of combustion of subbituminous coal. The validity of these approaches is confirmed by verification of the results for the case of combustion of the design fuel – anthracite in the design operating conditions. It is shown that the deviation of the results from the experiment does not exceed 6.5 %, which makes it possible to use the developed computational model to simulate the combustion of subbituminous coal, including in non-design operation modes. The obtained results are the basis for further calculations of the operation of the TPP-210A boiler of the 300 MW power unit using subbituminous coal with the identification and minimization of the factors of critical deviations of furnace processes that lead to thermal damage to wall screens, which will be described in the second part of the article.

The results of the work will help to increase the reliability and improve the technical and economic performance of boilers at Trypillia TPP, where 3 TPP-210A boilers were converted to subbituminous coal, and other power plants (Zmiivska, Kryvorizka, Prydniprovska), where similar anthracite boilers are planned for conversion to subbituminous coal.

Grant support. The work was carried out with the grant support of the National Research Foundation of Ukraine within the framework of the project of the Thermal Energy Technology Institute of the National Academy of Sciences of Ukraine 2022.01/0058 "Modernization of coal-fired boiler units of TPPs and CHPs taking into account the diversification of their fuel base to ensure stable heat and energy supply and power system load regulation".

References.

1. Dunayevska, N. I. (2023). Problems and technologies of thermal processing of fuels in energy installations of thermal power plants. *Visnyk of the National Academy of Sciences of Ukraine*, (4), 72-84. https://doi.org/10.15407/visn2023.04.072.

2. World Energy Outlook (2023). *Chapter 3: Pathways for the energy mix*, (pp. 101-153). International Energy Agency. Retrieved from https://iea.blob.core.windows.net/assets/42b23c45-78bc-4482-b0f9-eb826ae2da3d/WorldEnergyOutlook2023.pdf.

3. Chernyavskyy, M., Miroshnychenko, Y., Provalov, O., & Kosyachkov, O. (2023). Conversion of anthracite boiler units of chp plants for combustion of sub-bituminous coal in the war conditions. *Energy Technologies & Resource Saving, 76*(3), 3-20. https://doi.org/10.33070/etars.3.2023.01.

4. Makarov, V., Kaplin, M., Bilan, T., & Perov, M. (2023). Forecasting of coal production in Ukraine. *System Research in Energy, 1*(72), 35-45. https://doi.org/10.15407/srenergy2023.01.035.

5. Dipak, K. Sarkar (2015). *Thermal power plant: Design and Operation*. Elsevier Incorporated. https://doi.org/10.1016/B978-0-12-801575-9.00001-9.

6. Seheda, M. S., Beshta, O. S., Gogolyuk, P. F., Blyznak, Yu. V., Dychkovskyi, R. D., & Smoliński, A. (2024). Mathematical model for the management of the wave processes in three-winding transformers with consideration of the main magnetic flux in mining industry. *Journal of Sustainable Mining, 23*(1), 20-39. https://doi.org/10.46873/2300-3960.1402.

7. ANSYS FLUENT 14.5 Theory Guide (2012). ANSYS Inc. ANSYS Help. Retrieved from https://ansyshelp.ansys.com.

8. Hart, J., Bhuiyan, A. A., & Naser, J. (2018). Aerodynamics of burner jet in a tangentially-fired boiler: A CFD modelling and experiment. *International Journal of Thermal Sciences, 129*, 238-253. https://doi.org/10.1016/j.ijthermalsci.2018.02.030.

9. Liu, H., Guo, X., Chen, L., Zhang, X., & Qing Wang, Q. (2021). Numerical simulation of high-temperature corrosion and NOx generation characteristics of a boiler. Energy Sources, *Part A: Recovery, Utilization, and Environmental Effects,* 1-22. https://doi.org/10.1080/15567036.2021.1931566.

10. Noor Akma Watie Mohd Noor, Hasril Hasini, Muhamad Shazarizul Haziq Mohd Samsuri & Meor Mohd Faisal Meor Zulkifli (2020). CFD Analysis on the Effects of Different Coal on Combustion Characteristics in Coal-fired Boiler. *CFD Letters, 12*(10), 128-138. https://doi.org/10.37934/cfdl.12.10.128138.

11. Liu, X., Zhang, J., Tan, H., Mo, Q., Wang, X., & Wan, Y. (2021). Numerical and experimental study on co-firing of low volatile coal in a 330 MW tangentially fired boiler. *Journal of the Energy Institute, 96*, 242-250. https://doi.org/10.1016/j.joei.2021.03.015.

12. Wei, D., An, D., Wang, T., Zhang, H., Guo, Yo., & Sun, B. (2023). Influence of fuel distribution on co-combustion of sludge and coal in a 660 MW tangentially fired boiler. *Applied Thermal Engineering, 227.* https://doi.org/10.1016/j.applthermaleng.2023.120344.

13. Madejski, P. (2018). Numerical study of a large-scale pulverized coal-fired boiler operation using CFD modeling based on the probability density function method. *Applied Thermal Engineering, 145*, 352-363. https://doi.org/10.1016/j.applthermaleng.2018.09.004.

14. Deng, L., Zhang, Y., Ma, S., Zhu, Z., Liu, H., & Belošević, S. (2021). Numerical study on combustion characteristics and heat flux distributions of 660-MW ultra-supercritical double-reheat tower-type boiler. *Asia-Pacific Journal of Chemical Engineering, 16*(3). https://doi.org/10.1002/apj.2631.

15. Askarova, A., Georgiev, A., Bolegenova, S., Beketayeva, M., Maximov, V., & Bolegenova, S. (2022). Computational modeling of pollutants in furnaces of pulverized coal boilers of the republic of Kazakhstan. *Energy, 258*(1), 124826. https://doi.org/10.1016/j.energy.2022.124826.

16. Laubscher, R., & Rousseau, P. (2020). Coupled simulation and validation of a utility-scale pulverized coal-fired boiler radiant final-stage superheater. *Thermal Science and Engineering Progress, 18*, 100512. https://doi.org/10.1016/j.tsep.2020.100512.

17. Norchak, V. (2022). Combustion of composite peat-coal mixtures in a vortex fire-place. *Collected scientific works of Ukrainian State University of Railway Transport, 199*, 23-31. https://doi.org/10.18664/1994-7852.199.2022.258647.

18. Vorobyov, N., Baranyuk, A., & Dunayevska, N. (2023). CFD Modeling of Water Heating Boiler Operation with Different Types of Fuel Pellets. *International Journal of Energy for a Clean Environment, 24*(1), 115-127. https://doi.org/10.1615/InterJEnerCleanEnv.2022043888.

19. Dunayevska, N. I., Zasyadko, Y. I., Zasyadko, P. Y., & Shchudlo, T. S. (2017). Mathematical model of the processes of co-combustion of biomass with coal in the boiler of a thermal power plant. *Electronic Modeling, 39*(3), 89-104.

20. Dunaevska, N. I., Bondzyk, D. L., Nehamin, M. M., Miroshnichenko, Ye. S., Beztsenny, I. V., Yevtukhov, V. Ya., & Shudlo, T. S. (2020). Technology of Anthracite and Solid Biofuels Co-Firing in Pulverized Coal Boilers of TPP and CHP. *Science and Innovation, 16*(5), 86-96. https://doi.org/10.15407/scin16.05.086.

21. Nekhamin, M., Beztsennyi, I., Dunayevska, N., & Vyfatnuik, V. (2020). On using the ANSYS FLUENT software for calculating the process of burning a mixture of particles from different types of solid fuels. *Eastern-European Journal of Enterprise Technologies, 4*(8(106)), 48-53. https://doi.org/10.15587/1729-4061.2020.209762.

22. Nekhamin, M. (2020). Design by program ANSYS FLUENT processes in heating of caldron of tpp-210a. *Energy Technologies & Resource Saving,* (3), 35-40. https://doi.org/10.33070/etars.3.2020.03.

23. Chernyavskii, N. V., Miroshnichenko, E. S., & Provalov, A. Y. (2021). Experience in Converting TPP-210A Boilers with 300 MW Power Units to Burning Gas Coal at the Tripillya Thermal Power Plant. *Power Technology and Engineering, 54*(5), 699-706. https://doi.org/10.1007/s10749-020-01273-0.

24. Baraniuk, O. V., Vorobyov, M. V., & Rachynskyi, A. Y. (2023). *CFD-modeling of heat transfer and hydrodynamics processes by means of a software package: monograph.* Kyiv: Igor Sikorsky Kyiv Polytechnic Institute, Polytechnic Publishing House. ISBN 978-966-990-062-3.

CFD-моделювання критичних відхилень топкових процесів у пиловугільних котлах. Частина 1. Побудова розрахункової моделі котла ТПП-210А

Д. Л. Бондзик[1], *О. В. Баранюк*[2,1], *М. В. Воробйов*[*2,1], *Н. В. Чернявський*[1], *О. В. Косячков*[1]

1 – Інститут теплоенергетичних технологій Національної академії наук України, м. Київ, Україна
2 – Національний технічний університет України «Київський політехнічний інститут імені Ігоря Сікорського», м. Київ, Україна
[*] Автор-кореспондент e-mail: vorobiov.nikv@gmail.com

Мета. Розробка математичної моделі прогнозування критичних відхилень топкових процесів при роботі антрацитових котлів, переведених на спалювання суббітумінозного вугілля, у тому числі в непроектних режимах експлуатації. Верифікація розрахункової моделі котла ТПП-210А енергоблоку 300 МВт на проєктному паливі – антраციті для подальшого аналізу його роботи на суббітумінозному вугіллі.

Методика. Моделювання спалювання твердого палива виконувалось за допомогою скінченно-елементних CFD-моделей котельного агрегату в середовищі програмного комплексу ANSYS-Fluent з визначенням відповідних характеристик роботи котла.

Результати. Розроблені скінченно-елементні CFD-моделі відповідно до робочих креслень котла ТПП-210А та його пальникового пристрою, реконструйованого для спалювання суббітумінозного вугілля. Проведена верифікація результатів CFD-моделювання для випадку спалювання проєктного палива – антрациту в проєктних режимах експлуатації. Показано, що розбіжність результатів з експериментом не перевищує 6,5 %, що дозволяє використовувати розроблену комп'ютерну модель для моделювання спалювання суббітумінозного вугілля, у тому числі в непроєктних режимах експлуатації. Отримані результати є основою для подальших розрахунків роботи котла ТПП-210А енергоблоку 300 МВт на суббітумінозному вугіллі з визначенням та мінімізацією чинників критичних відхилень топкових процесів, що призводять до термічних пошкоджень стінових екранів.

Наукова новизна. Уперше при CFD-моделюванні враховані всі конструктивні особливості пальникових пристроїв котла ТПП-210А, реконструйованих для спалювання суббітумінозного вугілля з введенням парового ежектора пилопроводу високої концентрації під розріdженням до каналу центрального повітря. Уперше застосовано розподіл теплових опорів в якості граничних умов на стінках паливні для випадку спалювання суббітумінозного вугілля. Правомірність цих підходів підтверджена верифікацією результатів.

Практична значимість. Верифікована комп'ютерна модель переведеного на спалювання суббітумінозного вугілля котла ТПП-210А енергоблоку 300 МВт дозволить визначити й мінімізувати чинники критичних відхилень топкових процесів, що призводять до термічних пошкоджень стінових екранів. Це сприятиме збільшенню надійності й покращенню техніко-економічних показників котлів Трипільської ТЕС, де працює 3 таких котли, та інших електростанцій (Зміївської, Криворізької, Придніпровської), де подібні антрацитові котли плануються для переведення на суббітумінозне вугілля.

Ключові слова: *пиловугільний котел, антрацит, суббітумінозне вугілля, CFD-моделювання, променевий і конвективний теплообмін*

The manuscript was submitted 05.10.23.