Numerical study of the methane-air combustion in the direct-flow burner of the boiler 300 MW TGMP-314 boiler

N E Fomenko¹,², Vojislav Jovicic², V B Prokhorov¹, M V Fomenko¹

¹ Moscow Power Engineering Institute, National Research University (NRU MPEI), Krasnokazarmennaia str., 14, Moscow, 111250, Russian Federation
² Friedrich-Alexander University (FAU) of Erlangen-Nuremberg, Institute of Fluid Mechanics (LSTM), Cauerstrasse 4, Erlangen, Germany

Email: fomenko.n.e@yandex.ru

Abstract. This paper presents the results of the numerical simulation of the methane-air combustion in the direct-flow burner. The construction of the burner was proposed by the Department of Thermal power plants MPEI for organization the staged combustion. The burners are designed for installation on boiler 300 MW TGMP-314 at thermal power plant in Moscow. The simulation of five modes of operation of the burner depending on the boiler load was carried out using modern packages of computational fluid dynamics ANSYS. The goal of this work is to investigate effects of replacement of existing vortex burners in previously described facilities with the direct-flow burners with organization of the new combustion process. As results after the simulation, the parameters of the numerical values of the turbulence parameters and the mass fractions of various substances for the gas-air flow in the boiler inlet plane was obtained. On the basis of the obtained results analysis of the proposed design of the burner was made. Numerical values of the parameters of the gas-air flow in the plane of the inlet to the boiler will be used later as input data for the numerical simulation of the boiler furnace.

1. Introduction and Motivation

Significant changes in the environmental legislation of the Russian Federation [1] lead to a significant increase of the fees for exceeding the specific emissions of pollutants into the atmosphere. As a result, many of the older facilities are working on the different optimization methods to improve their system performances and reduce the pollution emissions.

This research investigates the possibility for such a pollutant emission reduction in the gas-oil fired boilers TGMP-314 installed in the thermal power plant in Moscow. These systems currently work with specific emissions of nitrogen oxides (NOx) of 200-225 mg/m³. Since the standard value of specific emissions should stay below 125 mg/m³ [2], certain optimization of the components could be of advantage.

The goal of this work is to investigate effects of replacement of existing vortex burners in previously described facilities with the direct-flow burners with organization of the new combustion process. The working hypothesis is that use of the direct-flow burners will result in better system performances at the low costs for the reconstruction of the boiler.
2. Used method
In order to achieve previously described goals, computational fluid dynamic (CFD) simulations were used in the first step of the research described in this paper, in order to investigate aerodynamics and flow field within the newly proposed direct-flow burners. Defining in details boundary conditions (BC) in the plane of fuel and oxidizer into the furnace is essentially important for the appropriate modeling of the conditions within the boiler. Commonly, these BC are set as averaged values of temperatures, velocity profiles, turbulence parameters and distribution of chemical elements [3, 4] at the boiler inlet planes. However, this approach leads to the loss of information about the flow parameters and distorts the simulation results, when the mixing of air and fuel occurs before entering the boiler. It also disables numerical calculation of the pollutant emissions since they are strongly depending on the conditions within the burner (mixing, ignition, flow field, etc.). On the other hand, modeling and numerical simulation the aerodynamics and heat and mass transfer within the boiler furnace together with the burners, requires significant computing resources and creates severe problems due to the several order of magnitude difference in size of the for example fuel inlet and cross-section area of the boiler. Therefore, in this work, authors proposed to divide the simulation process into two stages for carrying out the numerical simulation of the furnace with the more precise results and optimized use of the resources. In first step, described in this manuscript, numerical simulation of processes in the burner were observed (velocity, temperature and air/fuel concentrations were calculated). In the next step, using the results of this research, numerical simulation of the boiler furnace could be conducted with lower computational costs if the results of the simulations from the first step are used as the input BC by extrapolating the results of modeling burners. The proposed two-stage process of simulation the furnace is more precise and has optimized time and computational power consumption. The simulation results obtained in this way are also commonly closer to the real values [5].

3. The design of direct-flow burner
The sketch of an investigated direct-flow burner with a cut in the ZX plane is presented in the Fig. 1. The burner consists of a central primary air channel (fuel-air channel) with an internal diameter of 706 mm, inside which six fuel distribution pipes with a diameter of 60 mm each are mounted [9]. The length of the channel from the supply duct of the primary air to the inlet to the boiler is 1100 mm. The distance between the gas outlet point from the gas pipes and the plane of the entrance to the furnace is 150 mm. For distribution of natural gas fuel-distribution nozzle is made with one central hole diameter 38 mm and eight side holes diameter 10 mm. The length of the gas pipe from the supply duct of the primary air to the end of the gas nozzle is 950 mm. The length of the cylindrical part of the gas pipe after the study of the grid on the results of variable calculations is 227.5 mm.

4. Numerical simulation
Two commercial software packages were used to perform the simulation: SolidWorks was used to build a 3D model of a burner system and ANSYS to generate a mesh, select solver models, perform calculations and analysis of the results. The work was conducted through the following steps:

1) Creating a 3D model of the computational domain (Solid Works). The geometrically computational domain consists of six repeating segments, therefore only 1/6 of the burner was modeled using the Periodic boundary condition and then the Rotation function was used in the CFD Post processor. On the Fig. 1, the area of the burner, which was created as 3D model, was shown in color. This area contains a V-shaped volume of primary air (highlighted in green) and one natural gas distribution pipe with a fuel-distribution nozzle (highlighted in yellow). Total length used in this model, from the plane of the jet-entrance into the furnace (highlighted in blue) was selected to be 15 m. This was necessary in order to obtain the real velocity profile of the jet directed into the boiler furnace and to eliminate the influence of the model boundary for the calculation. The directions of X, Y, Z-axes and the point of origin of the coordinate system, which is located inside the gas pipeline, are also marked in the Fig. 1. The Z-axis is located in the center of the gas pipe and its positive direction coincides with the direction of flow.
2) Generation of meshes and the selection of the BCs for the simulated model in ANSYS ICEM. The constructed 3D model is made of two types of mesh: hexa- and tetra-meshes. The model geometry was therefore divided into 6 parts, 5 of which are meshed with hexa-mesh and one with tetra-mesh. The resulting mesh obtained by connecting six meshes in ANSYS ICEM and consist of structured (hexahedra) and unstructured elements (tetrahedra + pyramids), where 84% of the mesh is occupied by structured elements. The total number of mesh elements was 4,926,719.

![Diagram](image-url)

**Figure 1.** Direct-flow burner; 
a) – view from the furnace to the burner embrasure; b) cut in the ZX plane.

In ANSYS ICEM BC types were also selected. For the planes on which the "cutting" of the V-shaped volume from the original burner design was made, the "Periodic" BC was used. For gas and air inlet planes the type of "Mass Flow Inlet" BC was selected. For the outlet plane of the gas-air
mixture from the 3D model the “Pressure Outlet” was selected. For surfaces simulating walls - “Wall” BC was set.

3) Setting the numerical values of BC in ANSYS Fluent. For two of the above types of BC (“Mass Flow Inlet” and “Pressure Outlet”) numerical values were set in ANSYS Fluent. These values were calculated for each of the five modes of operation of the boiler burners. Parameters of steam, gas and air are taken from the summary table of results of testing of the TGMP-314 boiler at its work on gas and are given in Table 1. Reg.1 mode corresponds to the mode at maximum steam capacity and Reg.5 to the mode at minimum steam capacity. A static pressure of 0 Pa was set for the “Pressure Outlet” boundary condition. Natural gas was used as fuel. The mass fraction of methane in this natural gas is 93.4%.

### Table 1. Parameters of air and gas at different modes.

| Parameter | Reg. 1 | Reg. 2 | Reg. 3 | Reg. 4 | Reg. 5 |
|-----------|--------|--------|--------|--------|--------|
| Load of the boiler, % | 93.7 | 87.0 | 73.4 | 69.8 | 64.8 |
| Mass flow of the gas to the boiler, kg/s | 21.50 | 19.72 | 17.39 | 16.53 | 15.56 |
| Mass flow of the gas per burner gas pipe, kg/s | 0.217 | 0.194 | 0.159 | 0.151 | 0.138 |
| Air mass flow through 1/6 fuel-air channel, kg/s | 1.587 | 1.456 | 1.284 | 1.221 | 1.149 |
| Gas temperature, °C | -3.1 | -3.3 | -1.9 | -1.9 | -1.5 |
| The temperature of the primary air, °C | 285 | 277 | 275 | 270 | 265 |

4) Setting of solver models in ANSYS Fluent. The k-ω SST model was chosen as a turbulence model, which proved itself for the use of similar problems and showed better convergence with the experiment than the RNG k-ε model [6]. The requirements for the dimensionless parameter Y+ have been met.

Since in this case gas and air are not premixed when entering the burner, i.e. the process of diffusion combustion is in progress a “Non-preamixed combustion” model has been chosen [7, 8]. The criteria for the end of the iterative solution of the modeling problems considered in this work was to achieve the accuracy of the residuals for calculating the value of 0.001 and the unbalance of the mass flow at the inlet and outlet to the model less than 1 %

5. Results Analysis in ANSYS CFD-Post

The results of calculations of the five modes of operation are the fields of velocity, temperature, concentrations of substances in the horizontal ZX plane and in the plane of the inlet to the boiler parallel to XY, averaged parameters of the gas flow in the plane of the inlet to the boiler.

Fig. 2 shows the field of methane distribution in the horizontal plane ZX. As a result of the analysis of the figure, it can be noticed that the chemical reaction between the fuel and air occurs before the mixture enters the boiler. That can be seen in the Fig. 2 where the areas with the mass fraction of methane different from the initial 0.934 is shown. From the central hole of the fuel-distributing nozzle with a diameter of 38 mm, the gas comes out at a high velocity (Fig. 4) of up to 200 m/s.

Due to this high inlet velocity, fuel has not enough time to mix with the air before entering into the boiler furnace. Therefore, before entering to the boiler, the primary air reacts only with relatively limited amount of fuel, supplied through the eight small streams coming out of the side holes with a diameter of 10 mm. The velocity of these streams is much lower than the velocity of the main gas jet (Fig. 4). In general, the distribution of mass fractions of methane in the plane of the entrance to the boiler does not depend on the mode of operation, because the distribution patterns are very similar. This should be done by an independent process of ignition of the fuel from the operation of the boiler.

Fig. 3 provides the overview of the change in the average temperature of the jet at the inlet plane into the boiler combustion chamber. From Fig. 2 it can also be seen that natural gas begins to react with air before reaching the inlet into the boiler. This is noticeable in an increase of the temperature at the inlet to the boiler. The drop in a gas and the air flow rates is also noticeable, in the case of a decrease steam production. This further leads to a decrease in the average velocity of the fuel-air flow in the inlet plane to the boiler. The worst mixing occurs for the case with a lowest steam production.
In this case, an increase in the average mass fraction of O$_2$ at the inlet to the boiler is related with a decrease in steam amount, leading to a directly proportional dependence of the average temperature at the inlet to the boiler on the amount of the steam.

**Figure 2.** Distribution of CH$_4$ mass fractions in the horizontal ZX plane.

**Figure 3.** Change in the average velocity at the boiler inlet.

**Figure 4.** Velocity fields in the horizontal ZX plane for the mode with maximum (Reg.1) and minimum (Reg.5) steam capacity.
The design of the fuel-distribution nozzle influences the output velocity of the fuel from the pipe. Fig. 5 shows an increase in velocity in the Z-direction, from -0.03 to 0.02 m on the gas pipe axis. This corresponds to the position of the narrowing of the flow section (Fig.1), which affects, (like de Laval nozzle), the velocity profile of the fuel flow in the center of the pipe (Fig. 5). The higher the initial velocity of the fuel, the higher the effect of this design element of the burner, which is expressed in an increase in the difference between the fuel velocity before and after this narrowing. The presence of this section in the fuel pipe is a disadvantage of the fuel-distributing nozzle design because the fuel flow after the fuel pipe has high local velocities of the order of 130 - 190 m/s. The average velocity of the fuel-air flow in this plane is 70 – 40 m/s.

![Figure 5](image-url)

**Figure 5.** Values of gas velocities on the Central axis (co-directional with the Z axis) for the mode with maximum (Reg.1) and minimum (Reg.5) steam capacity.

6. **Conclusions**

The numerical simulation of five modes of operation of the burner were performed. It has been found that:

- as a result of mixing natural gas and air before entering the boiler, their partial reaction occurs. This leads to an increase in the temperature of the fuel-air mixture and the creation of combustion products in the flow;
- the flow of the air and fuel mixture has a local maximum velocity of about 130 - 190 m/s in the plane of the inlet to the boiler. It is consequence of the presence of a narrowing section in the fuel-distributing nozzle of the fuel pipe, which can be called a disadvantage of this design and requires further refinement;
- the lower of load of the boiler, the lower of energy of the interaction of fuel with air, and as a consequence, lower is the proportion of the substance reacted. This leads to a decrease in the temperature of the fuel-air mixture at the inlet to the boiler;
- the distribution of substances in the plane at the inlet to the boiler does not depends on the mode of burner operation, the distribution pattern of the mass fractions of methane and oxygen on the plane at the inlet to the boiler furnace similar that allows to make a conclusion about the independence of the process of ignition of the fuel from the operation of the boiler.
The obtained numerical values of the turbulence parameters and mass fractions of various substances in the boiler inlet plane are the main results of the work and will be used as initial data in the numerical simulation of the boiler furnace TGMP-314 in the next step of research.

References
[1] Federal law of 21.07.2014 N 219-FZ (ed. of 03.07.2016) "On amendments to the Federal law "On environmental protection" and certain legislative acts of the Russian Federation"
[2] GOST R 50831-95. The installation of the boiler. Mechanical equipment. General technical requirements 1996 (Moscow: Publishing standards)
[3] Audai Hussein Al-Abbas, Jamal Naser and David Dodds 2011 CFD modelling of air-fired and oxy-fuel combustion of lignite in a 100 KW furnace Fuel 90 1778–95
[4] Jin Yan, Xiaofeng Lu, Quanhai Wanga, Yinhu Kang, Jianbo Li, Jinliang Zhou, Yi Zhang, Zhuo Lv and Sicong Sun 2017 Experimental and numerical study on air flow uniformity in the isobaric windbox of a 600 MW supercritical CFB boiler Applied Thermal Engineering 122 311–21
[5] Adnan Dugum and Kemal Hanjali 2019 Numerical simulation of coal-air mixture flow in a real double-swirl burner and implications on combustion anomalies in a utility boiler Energy 170 942-53
[6] Kirichkov V S Development, study and optimization of burning of fuel in a direct-flow – vortex flame in a steam boiler - the dissertation on competition of a scientific degree of candidate of technical sciences. 2014 (Moscow).
[7] Pranit Gaikwad, Hrishikesh Kulkarni and S Sreedhara 2017 Simplified numerical modelling of oxy-fuel combustion of pulverized coal in a swirl burner Applied Thermal Engineering 124 734-45
[8] Poinso T and Veynante D 2005 Theoretical and numerical combustion (RT Edwards, Inc.)
[9] Warnatz J, U Maas and R Dibble 2006 Combustion (Germany: Springer)

Acknowledgments
This work was supported by a Russian Scientific Foundation grant (project no. 16-19-10463П dated May 20th, 2019) which is here gratefully acknowledged. The authors also gratefully acknowledge the financial support of Professor Klaus Riedle-Foundation for enabling conduction of this work at the Institute of Fluid Mechanics (LSTM) at Friedrich-Alexander University (FAU) in Erlangen, Germany.