Development of a simplified methodology for furnace aerodynamics with vortex combustion of organic fuel modeling

V B Prokhorov¹, N E Fomenko¹, M V Fomenko¹
¹National Research University "Moscow Power Engineering Institute", Krasnokazarmennaya str., 14, Moscow, 111250, Russia

Email: fomenko.n.e@yandex.ru

Abstract. This paper describes the process of developing a simplified methodology for furnace aerodynamics during the development or modernization of combustion schemes with direct-flow burners. This technique is based on the use of numerical modeling of air movement and turbulence phenomena in the furnace volume and allows for a relatively short period of time to analyze a large number of options for the burners and nozzles location. This is its advantage in comparison with the use of experimental modeling or numerical simulation with combustion when analyzing a variety of schemes. The model was developed on the basis of validated results of combustion processes numerical simulation in the K-50 boiler furnace. The paper presents the results of calculations performed for several variants of the simplified methodology. For further use, the option that best corresponds to full-scale studies taking into account the fuel combustion process has been selected. The main states of the methodology are formulated.

1. Introduction

Aerodynamics and fuel combustion processes modeling in the combustion furnaces of power boilers is necessary at the design stage and during the boiler units modernization and is especially relevant for combustion schemes with direct-flow burners because they are collective work burners. Furnace aerodynamics simulation allows us to qualitatively assess the jets interaction in the furnace volume due to its influence on the fuel ignition process and mixing of fuel with an oxidizer. Modeling of the processes occurring in the furnaces is currently performed in the following ways: experimental [1-4], using numerical simulation [5] and a combined method [6].

Experimental isothermal modeling has proven itself well as a tool for a comprehensive aerodynamics assessment. It allows to qualitatively study the furnace aerodynamics excluding the heat exchange processes and fuel combustion. The error in the results is introduced by the isothermal approach to modeling, the "flow markers" inertia, an increasing of burners and nozzles size in relation to the furnace dimensions on the test installation to ensure dynamic pressures equality in the experimental model and the real furnace.

Numerical modeling of furnace processes is performed using modern computational fluid dynamics (CFD) packages, such as ANSYS, Open Foam, Star CCM+. Such programs contain a large range of validated mathematical models that allow to simulate the flow trajectories, turbulent processes, convective heat exchange, particle trajectories in the flow, radiation, chemical reactions occurring...
during fuel combustion. The method disadvantages are the need for a powerful computer or a
calculation server and a large amount of calculation time which depending on the power of the
calculation computer.

The combined method modeling merges experimental physical modeling and numerical
modeling elements. The jets aerodynamics in the isothermal test installation is numerically
modeled for various variants of the nozzles and burners location on the physical model. This
allows to save money on the creation of physical model several variants and limit researchers only
to the experimental installation implementation for the initial combustion scheme (for numerical
model validation of the experimental installation) and the final scheme obtained as a variable
calculation results in the CFD program.

It is necessary to consider many options for the burners and air nozzles arrangement when
designing a fuel combustion scheme with direct-flow burners. The experimental study requires
money costs for a large test installation number and furnace numerical modeling with combustion
processes of each model will be performed for a long time. In this regard, it became necessary to
create a simplified methodology for furnace aerodynamics modeling for analyzing a variety of
variant schemes.

2. The process of developing a simplified methodology for furnace aerodynamics modeling

2.1. The research object and the tasks set when developing a simplified methodology for furnace aerodynamics modeling

The simplified method involves performing numerical simulation. The ANSYS program was used for
this. The study object is the K-50 boiler furnace since there are validated [5] results of the K-50 furnace numerical simulation with parameters taken from a really working boiler. The fuel for the boiler is coal and a step-by-step combustion is organized in the furnace. The calculation 3D boiler furnace K-50 model is shown in Figure 1.

![Figure 1. The calculation 3D boiler furnace K-50 model.](image)

The development was carried out by performing a calculations series with a sequential numerical
model complication. The results of calculation and K-50 furnace modeling were compared. The
selected calculations results are considered here: Calculation A, Calculation B, Calculation C and Calculation D.

The main tasks that were set for the numerical model for simplified methodology:

- furnace simulation with the actual overall furnace dimensions, burners and nozzles;
- furnace simulation with the actual overall furnace dimensions, burners and nozzles;
- lack of combustion processes modeling to simplify the calculation and reduce the calculation time;
- taking into account changes in the air thermodynamic properties as a function of temperature in the numerical model;
- the possibility of obtaining quantitative and qualitative results on mixing of primary, secondary and tertiary air as a modeling result;
- the maximum possible similarity of the current lines coming out of the nozzles and burners, behavior of the velocities change, the turbulent kinetic energy, the densities inside the furnace volume obtained during combustion modeling with the results of calculation using a simplified method.

2.2. General approach description to the calculation

The air movement was described by the continuity (mass conservation), Navier-Stokes (conservation of momentum). The turbulent phenomenon was described by the k-ε Realizable model [7]. The convective heat transfer was described by the energy conservation equation. These models were used for all calculations. Air was chosen as the simulated substance, however, in some calculations its thermodynamic properties were set differently. The boundary conditions at the entrance to the burners, nozzles and planes that simulate air suction into the furnace were the "mass-flow-inlet" and air temperature. These parameters were taken from the K-50 boiler test results [5], however, for some calculations, the primary air temperature was different from the real one. The boundary condition at the furnace outlet from the model was set by the "pressure outlet" type. The solution was considered complete when the residuals were reached for all equations the order $10^{-4}$ and there was no unbalance in mass flow rates, there were no back flows at the model outlet.

The boundary conditions numerical values for the A - D calculations and the features of each calculation are given in Table 1. Individual features of the calculations are highlighted.

The calculations results are vector velocity fields, streamlines, temperature fields, density fields, mass fraction fields of primary, secondary and tertiary air, charts of the temperatures, velocities, densities distribution along the line lying in the plane passing through the burners, secondary and tertiary air nozzles.

2.3. Calculation A

The boundary conditions are given in Table 1. The simulated substance is air with constant thermodynamic properties, the values of which were automatically set by the calculation program: density $1.225 \text{ kg/m}^3$, viscosity $1.7894 \times 10^{-5} \text{ kg/m} \cdot \text{s}$, heat capacity $1006.43 \text{ J/kg} \cdot \text{K}$, thermal conductivity $0.0242 \text{ W/m} \cdot \text{K}$. Figures 2 and 3 show a streamlines comparison obtained in the calculation A and in the calculation with combustion process from [5]. In the pictures “PA” is primary air, “SA” is secondary air and “TA” is tertiary air.

The results showed that the flow trajectories of the air flow inside the furnace are different. In the calculation A the primary air jet suppresses the secondary air jet in the furnace center and leads to the vortex motion formation at the side furnace wall, thus, the current lines in the calculation A and the calculation with combustion do not coincide. In calculation A the air density in the entire furnace volume is constant and the primary air mass flow rate is more than 2 times greater than that of secondary air. This leads to the fact that the primary air jet kinetic energy is greater than that of the secondary air. It provides a deeper penetration into the furnace. The picture turns out to be different when simulation with combustion, since the density of the primary air jet decreases as it penetrates into the furnace (Figure 4). Accordingly, the jet kinetic energy gradually decreases, since the resulting
Combustion processes lead to heat release in the furnace. This leads to an increase in the temperature in the furnace and, accordingly, to a drop in the air density. This leads to a decrease in the jet range, so there is no direct interaction of the primary and secondary air jets.

### Table 1. Boundary conditions and features for the calculations A-D.

| Calculation                  | A       | B       | C       | D       |
|------------------------------|---------|---------|---------|---------|
| **Inlet temperatures, °C**   |         |         |         |         |
| Primary air                  | 70      | 70      | 2000    | 2000    |
| Secondary air                | 392.65  | 392.65  | 392.65  | 392.65  |
| Tertiary air                 | 392.65  | 392.65  | 392.65  | 392.65  |
| Planes that simulate air     | 30      | 30      | 30      | 30      |
| suction                     |         |         |         |         |
| **Mass flow inlet, kg/s**   |         |         |         |         |
| Primary air                  | 0.752 (per 1 burner) | 0.752 (per 1 burner) | 0.752 (per 1 burner) | 0.752 (per 1 burner) |
| Secondary air                | 0.29 (per 1 nozzle) | 0.29 (per 1 nozzle) | 0.29 (per 1 nozzle) | 0.29 (per 1 nozzle) |
| Tertiary air                 | 1.195 (per 1 nozzle) | 1.195 (per 1 nozzle) | 1.195 (per 1 nozzle) | 1.195 (per 1 nozzle) |
| Planes that simulate air     | 3.627 (per 1 nozzle) | 3.627 (per 1 nozzle) | 3.627 (per 1 nozzle) | 3.627 (per 1 nozzle) |
| suction                     |         |         |         |         |
| **Mass fraction of primary/secondary/tertiary air at the model inlet** |         |         |         |         |
| Primary air                  | -       | -       | 1/0/0   | 1/0/0   |
| Secondary air                | -       | -       | 0/1/0   | 0/1/0   |
| Tertiary air                 | -       | -       | 0/0/1   | 0/0/1   |
| Planes that simulate air     | -       | -       | 0/1/0   | 0/1/0   |
| suction                     |         |         |         |         |
| **The simulated substance**  |         |         |         |         |
| Air                          | +       | +       | -       | -       |
| Mixture (primary, secondary, tertiary air) | -       | -       | +       | +       |
| **Relation $\rho(T)$**       |         |         |         |         |
| Primary air                  | const   | real polynomial | real polynomial | inverse polynomial |
| Secondary air                | const   | real polynomial | real polynomial | real polynomial |
| Tertiary air                 | const   | real polynomial | real polynomial | real polynomial |
| Planes that simulate air     | const   | real polynomial | real polynomial | real polynomial |
| suction                     |         |         |         |         |
The results showed that the flow trajectories of the air flow inside the furnace are different. In the calculation A the primary air jet suppresses the secondary air jet in the furnace center and leads to the vortex motion formation at the side furnace wall, thus, the current lines in the calculation A and the calculation with combustion do not coincide. In calculation A the air density in the entire furnace volume is constant and the primary air mass flow rate is more than 2 times greater than that of secondary air. This leads to the fact that the primary air jet kinetic energy is greater than that of the secondary air. It provides a deeper penetration into the furnace. The picture turns out to be different when simulation with combustion, since the density of the primary air jet decreases as it penetrates into the furnace (Figure 4). Accordingly, the jet kinetic energy gradually decreases, since the resulting combustion processes lead to heat release in the furnace. This leads to an increase in the temperature in the furnace and, accordingly, to a drop in the air density. This leads to a decrease in the jet range, so there is no direct interaction of the primary and secondary air jets.

**Figure 2.** Streamlines, calculation A.  
**Figure 3.** Streamlines, calculation with combustion.  
**Figure 4.** Density field, calculation with combustion.  
**Figure 5.** Charts of the densities distribution along the line lying in the plane passing through the burners, secondary and tertiary air nozzles.
Figure 6. Charts of the velocities distribution along the line lying in the plane passing through the burners, secondary and tertiary air nozzles.

The charts of the velocities and densities distribution along the line lying in the plane passing through the burners, secondary and tertiary air nozzles (Figures 5 and 6) show that these variables and the nature of their changes in the combustion furnace height are not comparable with the obtained dependencies when modeling a furnace with combustion processes. Also the use of constant thermodynamic properties of air leads to distortion of the results.

The results of the calculation A do not allow us to apply the numerical model used to simulate the furnace aerodynamics in the first approximation, since the qualitative and quantitative results of the simulation differ from those obtained in the numerical calculation of the combustion in furnace.

2.4. Calculation B

The numerical model for the calculation B is generally similar to the one used in the calculation A. This calculation is necessary for the gradual complication of the numerical model. A distinctive calculation feature is the use of polynomial dependences of density, heat capacity, dynamic viscosity and thermal conductivity on temperature. The relations \( \rho(T) = \text{polynomial} \) are obtained from [8-9] for standard dry air and represent fifth degree polynomials.

A streamlines comparison based on the calculation B results and calculation with combustion showed that the splitting of the secondary air jet by the primary air stream, as in calculation A, is present, however, to a lesser extent. Since there is a slight decrease in density in the propagating primary air stream (Figure 7) due to its heating from 70°C to 120...180°C, which leads to a decrease in the primary air jet range.

Despite the fact that it is impossible to note the similarity of the stream lines in calculation B with the combustion calculations the model has become more accurate in describing the nature of the density change in vertical planes compared to calculation A. In Figures 5 and 6 the curves peaks are clearly visible, located at approximately the same level as in the simulation with combustion.

There are no coincidences in the density change at the primary and tertiary air supply level due to the low temperature level in these zones compared to the combustion calculation. At the secondary air supply level the average temperature and air density level (Figure 5) have become closer to the average value obtained when combustion calculation, since the density is calculated by a polynomial relation on temperature. However, the nature of the density change does not coincide. In the places where the density chart has peaks after combustion calculation, the line obtained when calculating B has dips marked in Figure 8 in color. This is due to the fact that a jet of primary air with a temperature
about 100°C lower enters the secondary air supply zone. The flows mix and cool down, which leads to a certain increase in density.

Figure 7. Density field, calculation B.

Figure 8. Charts of the density distribution along the horizontal line lying in the secondary air supply plane.

The mathematical model used in the calculation B does not give a correct idea of the streamlines of primary, secondary and tertiary air in the furnace volume. However, the model is more accurate than the model for calculating A, since it takes into account the influence of temperature on thermodynamic parameters. This allows to estimate the nature of changes in the velocity, temperature and density of the flow inside the furnace in different sections as an initial approximation.

2.5. Calculation C

A significant difference between the qualitative and quantitative calculation B results is due to the low temperature level inside the furnace as a whole, since there is no heat generation resulting from the combustion process. In order to increase the temperature in the proposed combustion zone it is proposed to supply primary air with a temperature of 2000°C to the burners. The 2000°C temperature is taken as a value for which thermodynamic parameters can be determined by the dependencies [8-9].

According to the boundary conditions the calculation C coincides with the calculation B with the exception of the primary air inlet temperature, which here became equal to 2000°C (Table 1).

For quantify the mixing of primary, secondary and tertiary in this numerical model instead of a homogeneous substance (air), a Mixture was given. The mixture consists of three components: three air type with the same thermodynamic properties, but different names. Primary air - Air 2, secondary air and modeling air suction - Air 3, tertiary air - Air 4. The component transfer process simulation was modeled using the Species Transport model, which works by solving the conservation equations of the mixture chemical components. To use this model, the mass fractions of each substance were set at each burner inlet, nozzle inlet and plane modeling air suction inlet. Their values are given in Table 1. The thermodynamic properties of each component are set in accordance with the relations [8-9].

The obtained streamlines do not resemble the combustion simulation. This is due to the fact that in calculation C the primary air velocities turned out to be several times higher than in the combustion simulation. Since the inlet temperature is set to 2000 °C in accordance with the relation \( p(T) = \text{polynomial} \). [8-9] the density has become equal to 0.151 kg/m³, whereas at a 70°C temperature it is equal to 1.02 kg/m³. According to the continuity equation, when the density decreases and the mass flow rate and cross-section remain constant the air velocity increases. In this case, it became equal to 168 m/s. Further, as the primary air jets penetrated and cooled their density increased. The
combination of these phenomena led to inconsistent results. In this regard, the results of this calculation are not presented in Figures 5 and 6.

However, this calculation allowed us to obtain a higher temperature in the furnace volume and at the furnace outlet the temperature is in the range of 900-1000°C. In the zone of secondary air, a dry bottom hopper, the air temperature became close to that in the simulation with combustion. The tertiary air also began to enter into the zone with a higher temperature, compared to calculations A and B. All this contributes to the fact that the obtained thermodynamic parameters, calculated from the relations [8-9], allow us to obtain results that are closer to the calculation with combustion.

This calculation made it possible to evaluate the possibility of the approach to modeling air in the form of an air mixture. As results it was possible to obtain the mass fractions distribution of primary, secondary and tertiary air on any planes in the furnace volume.

The calculation of C did not allow us to obtain an aerodynamics consistent with that obtained during the combustion simulation, however, the supply of primary air with a temperature of 2000°C allowed us to increase the overall temperature level in the simulated furnace volume, which should make the results more accurate. Also on this model, a solution for modeling the air mixing in the furnace was tested.

2.6. Calculation D
In this calculation, the advantages of the C calculation were used and its disadvantages were corrected. As noted, the primary air jets have a velocity level that does not correspond to the real physical flow distribution due to the low air density at the burners inlet. In this regard, changes have been made to the mathematical model for the primary air component (Air 2). The viscosity, heat capacity and thermal conductivity are set according to the relation [8-9]. The air density is given as an inverse relation  \( \rho(T) = \text{inverse polynomial} \), which is shown in Figure 9.

![Figure 9. Inverse polynomial relation \( \rho(T) = \text{inverse polynomial} \).](image)

Thus, the 2000°C temperature will correspond to the air density that would be at a 70°C temperature and vice versa. This approach should make it possible to approximate the change in the primary air jet density to the type obtained during combustion: at the burner inlet the jet density will correspond to a 70°C temperature and as it penetrates into the boiler it will decrease. This decrease in density during combustion simulation is due to the flow heating when it enters the heat release zone. In model D the density will decrease due to the application of the polynomial inverse relation \( \rho(T) = \text{inverse polynomial} \).

The obtained streamlines (Figures 10 and 11) are consistent with the combustion simulation results. The primary air jet penetrates to the furnace middle. Under the primary air flow there is a flue gases recirculation to the torch root due to the swirling of the flow by secondary air. The tertiary air jet
moves vertically and does not come into direct contact with the primary air flow. Similar consistent results were obtained for the temperatures, densities, and velocities fields.

Figure 10. Streamlines, calculation D.

Figure 11. Streamlines, calculation with combustion.

An important parameter for evaluating the furnace aerodynamics is the turbulent kinetic energy $k$. It characterizes the intensity of velocity pulsations and is defined as half the sum of the squares of the velocity components standard deviations. Charts and fields of the turbulent kinetic energy allow us to identify zones of intense turbulent motion, which play an important role in mixing air and fuel jets. The results obtained for the turbulent kinetic energy in the calculation D and in the calculation with combustion are compared (one of the comparison options is shown in Figure 12).

Figure 12. Charts of the turbulent kinetic energy distribution along the line lying in the plane passing through the burners, secondary and tertiary air nozzles.

The charts result show that the obtained $k$ values do not numerically coincide with those obtained in the combustion calculations. However, it can be noted that at some points these values are quite close and there is a similarity in the appearance of these dependencies. These results allow us to assert that the model D allows to evaluate the presence of turbulent processes occurring in the furnace volume and their intensity.
The use of a air components mixture allowed to obtain the distribution of primary, secondary and tertiary air mass fractions in the furnace volume (Figure 13-15).

![Figure 13. Fields of primary air mass fraction.](image1)

![Figure 14. Fields of primary air mass fraction.](image2)

![Figure 15. Fields of primary air mass fraction.](image3)

The results give an idea of the global components distribution from 0 to 1. It is also possible to estimate the local components distribution, for example, at the furnace outlet. These visualizations show that the primary air partially penetrates into the secondary air supply zone and mixes with it. Tertiary air is absent below the level of its supply and moves participating in the fuel afterburning.

Based on the results obtained, the numerical model used in the calculation D can be taken as a numerical model for a simplified methodology for furnace aerodynamics modeling.

3. Formulation of the main states of the simplified methodology for furnace aerodynamics modeling

Basic general principles of the simplified methodology:

- using the CFD software package for numerical simulation;
- the furnace 3D model must be made in real dimensions;
- burners and nozzles are modeled by planes;
- using instead of air a mixture consisting of the i-th number of components, which corresponds to the n-th number of flows types involved in combustion. The researcher can also select additional flows into separate components, if it is necessary to evaluate their mixing in the volume;
- using the polynomial relations [8-9] of density, viscosity, thermal conductivity and heat capacity in the calculation for all inlet flow type, except for the fuel-air mixture flow (primary air);
- using real flow and temperature boundary conditions for all flow types included in the model, except for the fuel-air mixture flow (primary air);
- to calculate the viscosity, thermal conductivity and heat capacity of the fuel-air mixture flow (primary air) use the relations [8-9]. To calculate the density use the inverse relation on the temperature $\rho(T) = inverse polynomial$, whose polynomials for the fuel-air mixture temperatures of 70-250°C are shown in Table 2 and the graphic image is shown in Figure 9.
- the inlet boundary conditions for the fuel-air mixture (primary air) are set as follows: the mass flow inlet is set as real and the temperature is 2000°C;
- modeling of air movement, turbulence phenomena and convective heat exchange is performed.
Table 2. Polynomials coefficients $\rho(T) = \text{inverse polynomial}$.

| The fuel-air mixture temperatures, °C | Polynomials coefficients $\rho(T) = \text{inverse polynomial}$ $\rho(T) = a_0 + a_1 \cdot T + a_2 \cdot T^2 + a_3 \cdot T^3 + a_4 \cdot T^4 + a_5 \cdot T^5$ |
|--------------------------------------|--------------------------------------------------------------------------------------------------|
| 70                                  | $a_0$ $a_1$ $a_2$ $a_3$ $a_4$ $a_5$ |
| 100                                 | $-4.767e-02$ $1.186e-03$ $-2.528e-06$ 2631e-09 $-1.259e-12$ 2.331e-16 |
| 150                                 | $-6.006e-02$ $1.199e-03$ $-2.473e-06$ 2.505e-09 $-1.173e-12$ 2.135e-16 |
| 200                                 | $-8.556e-02$ $1.250e-03$ $-2.450e-06$ 2.376e-09 $-1.075e-12$ 1.899e-16 |
| 250                                 | $-8.061e-02$ $1.076e-03$ $-1.929e-06$ 1.753e-09 $-7.527e-13$ 1.282e-16 |

4. Conclusions

The existing methods of modeling the power boilers furnace aerodynamics are not always suitable for working out a large number of options for the burners and nozzles location in the furnaces when developing or modernizing the fuel combustion scheme. If it is necessary to analyze a variety of fuel combustion schemes with direct-flow burners to evaluate the furnace aerodynamics, a simplified methodology was developed using numerical modeling in the ANSYS software package.

The development of the numerical model was made out by carrying out a variety of calculations with their gradual complication. The K-50 boiler furnace was chosen as the study object, since there are results of numerical simulation with combustion for it validated with the parameters obtained on a really working boiler. The results of the variant calculations were compared with the results of modeling a furnace with combustion. The article presents the results for the most significant A-D calculations.

The calculations differed from each other in the boundary conditions, the method of calculating the air thermodynamic parameters and numerical models. Model D is chosen as a numerical model for the implementation of a simplified methodology for furnace aerodynamics modeling. The simulated substance was a mixture: the program identified individual components with air properties that characterized the primary, secondary and tertiary air. The primary air temperature is set to 2000°C. The viscosity, heat capacity and thermal conductivity of all components of the mixture are calculated using polynomial relations on temperature. The density for secondary and tertiary air is also calculated. For primary air, an inverse density relation on temperature is given. Thanks to the use of the inverse density relations on temperature for the primary air and the use of a 2000°C temperature, it was possible to achieve a similar furnace aerodynamics as during combustion. The obtained charts of changes in the density, velocity, kinetic energy of turbulence, and temperatures are close to the dependences of the combustion results by the nature of the curves.

The main states of the simplified methodology, which is planned to be used for further research, are formulated.

References

[1] Prokhorov V B et al Physical modeling of the M-shaped boiler furnace aerodynamics 2020 J. Phys.: Conf. Ser. 1683 042072
[2] Paramonov A P, Kadyrov M R, Trinchenko A A Research on Influence of the Furnace Chamber Aerodynamics on Ecological Indicators of Boiler Plants (Part 1: Model of a Low-temperature Swirl Furnace) 2017 Procedia Engineering 206 pp 546-51.
[3] Anufriev I et al Modeling of aerodynamics in vortex furnace 2011 International Symposium on Coal Combustion (Berlin) pp 1037-47
[4] Song M et al. Secondary air distribution in a 600 MWe multi-injection multi-staging down-fired boiler: A comprehensive study 2020 *Journal of the Energy Institute* 93 3 pp 1250-60

[5] Prokhorov V B et al Solid fuel combustion processes modelling in the furnace in terms of the boiler K-50-14-250 2020 *J. Phys.: Conf. Ser.* 1683 042050

[6] Prokhorov V B et al Numerical variational study of the M-shaped boiler invert furnace 2020 *J. Phys.: Conf. Ser.* 1683 042033

[7] Shih T H, Liou W W, Shabbir A, Zhu J A New k-e Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation 1995 *Computers Fluids* 24 3 pp 227-38

[8] Eric W Lemmon, Richard T Jacobsen, Steven G Penoncello, Daniel G Friend Thermodynamic Properties of Air and Mixtures of Nitrogen, Argon, and Oxygen from 60 to 2000 K at Pressures to 2000 MPa 2000 *J. Phys. Chem. Ref. Data* 29 3 pp 331–85

[9] Lemmon E W and Jacobsen R T Viscosity and Thermal Conductivity Equations for Nitrogen, Oxygen, Argon, and Air 2004 *Int. J. Thermophys* 25 1 pp 21–69

**Acknowledgments**

The reported study was funded by RFBR, project number 20-38-90170/20.