Influence degree of turbulence on heat-and-mass transfer in the combustion chamber

Askarova¹, A. Bekmuhamet¹,*, S. Bolegenova², V. Maximov³, M. Beketayeva⁴, Sh. Ospanova⁵

1, 2, 3: Department of technical physics and thermal physics, Al-Farabi Kazakh National University Almaty, Kazakhstan
4, 5: Physico-Technical Faculty, Al-Farabi Kazakh National University Almaty, Kazakhstan
*Correspondent author: kazsat2006@gmail.com

Abstract The urgency of the given problem and growing attention it relate to the work of existing power plants, the creation of new combustion chambers, with increase in quantity of the polluting substances entering in atmosphere. Investigated object in the given work the combustion chamber of copper RK 39. The block 300 MBT, steam capacity 475 T/hour is chosen. The copper is established on Aqsw power stations (Kazakhstan). The calculation area for carrying out computational experiments and creation of a database for modelling with useful program in complex PREPROZ. In created files geometrical data of investigated process, initial and boundary conditions for process modelling heat-and-mass transfer in reacting streams contain.[1]

1. Introduction

At the present time the main subject of engineers devoted mainly towards the utilization of the energy sources in more efficient and economical way. The efficient combustion of solid fuel in combustion chambers and the efficient heat transfer to water and steam in steam generators are essential for the economical operation of power plants. Heat transfer problems pertaining to the combustion in industrial furnaces are of great importance to the engineering designer of boilers and steam generators. In most industrial flame applications, the achievement of high heat transfer rates is a main target and is desirable.

When creating a geometric model, each wall of the combustion chamber is described separately in the form of numerical codes. First introduced to the walls of their angular points. Input and output are defined primarily as a type of hole, and then set the coordinates of the spatial position of the burners on certain walls. Hole Burner (input) is described as concentric circles (for round burners), the output - in the form of the rectangle with the corresponding coordinates in space. Reason of this paper for the limitations when creating a geometric model, circular holes were replaced with equal rectangular area, which also affected the accuracy of the calculations. In accordance with a given geometry of the lattice is created for numerical simulation (Figure 1).

Influence of initial level of turbulence on the basic characteristics of burning process which shown has been investigated, that change of turbulence degree of dust gas a stream essentially affects distribution of the basic characteristics of burning process in top internal space. We according to investigating that, comparing the obtained data for concentration of CO, CO₂, CH₄ for two degrees of turbulence Tu=10 and Tu=5. It is possible to draw a conclusion, that increase in degree of turbulence there is maximum hashing of mix and the minimum emission of harmful substances in environment. So for example, on exit concentration CO at degree of turbulence Tu=10 decrease on 52 % in comparison with exit of the same substance at Tu=5 (Value CO on an exit for Tu=10 - 0.00933 kg/kg, For Tu=5 - 0.00611 kg/kg).[2]

In this paper, numerical study of the effect of turbulence degree on the complex physical and chemical processes occurring during combustion of pulverized fuel in the combustion chambers is
carried out.

2. The basic equations of mathematical models

For conclusion of balancing ratios of the stationary control element of volume or control element of weight (Figure 2). It is assumed that the center of gravity of the selected element moves with the velocity of flow. This corresponds to a stationary control volume sound approach for the Euler’s flow. Change the value of the transport is described in a single fluid element. Value of transport size is defined in each point of considered area.

![Figure 2. Control volume for the generalized transport equation](image)

When converting from a finite limit to the infinitesimal volume element is obtained by controlling the differential equation describing the conservation of the transport variable $\phi$:

$$\frac{\partial(\rho \phi)}{\partial t} = - \frac{\partial(\rho u \phi)}{\partial x_i} + \frac{\partial(\rho u_i \phi)}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \Gamma \phi \frac{\partial \phi}{\partial x_i} \right] + \frac{\partial}{\partial x_i} \left[ \Gamma \frac{\partial \phi}{\partial x_i} \right] + S \phi$$

(1)

Replacing in equation (1) the convective and diffusive transport of flux density, cross-border control volume, we obtain:

- Density of the convective flow: $\hat{O}(K), j = \rho u_j \phi$
- Density of diffusion flux: $\hat{O}(D), j = \Gamma \frac{\partial \phi}{\partial x_j}$

Then, based on the data relationships (1) can be written as:

$$\frac{\partial(\rho \phi)}{\partial t} = - \frac{\partial\hat{O}(K), j}{\partial x_j} + \frac{\partial\hat{O}(D), j}{\partial x_j} + S \phi$$

(2)

We write equation (2) in vector form:

$$\frac{\partial(\rho \phi)}{\partial t} = div\left(- \rho \bar{u} \phi + \Gamma \phi \nabla \phi \right) + S \phi$$

(3)
And in tensor form, equation (2) becomes:

\[
\frac{\partial (\rho \phi)}{\partial t} = - \frac{\partial \left( \rho u_j \phi \right)}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \Gamma \phi \frac{\partial \phi}{\partial x_j} \right] + S_\phi
\]  

(4)

Here the index \( j \) characterizes the spatial direction component of the velocity \( u_j \) in the chosen Cartesian coordinate system with coordinates \( x_j \), where \( j = 1, 2, 3 \).

Then, we can use:

\[
u = \rho \frac{\partial \phi}{\partial t}
\]

(5)

Value of \( \phi \) change by time:

\[
\frac{\partial (\rho \phi)}{\partial t}
\]

\( \phi \) change due to convective transport:

\[
\frac{\partial \left( \rho u_j \phi \right)}{\partial x_j}
\]

\( \phi \) change due to molecular exchange phenomena:

\[
\frac{\partial}{\partial x_j} \left[ \Gamma \phi \frac{\partial \phi}{\partial x_j} \right]
\]

(Diffusive exchange).

The source (sink) term for the quantity of \( \phi \):

\[
S_\phi
\]

2.1 simulation of turbulence

The ‘k-\( \varepsilon \) model’ is an eddy viscosity turbulence model based on the Boussinesq hypothesis of relating the Reynolds stresses to the mean velocity gradient, where the turbulent viscosity is expressed in terms of turbulent kinetic energy (k) and its dissipation rate (\( \varepsilon \)). For modeling of turbulent viscosity and the closure of the system we used the standard k-\( \varepsilon \) turbulence model:[5]

\[
\frac{\partial (\rho k)}{\partial t} = - \frac{\partial }{\partial x_i} \left( \rho u_i k \right) + \frac{\partial}{\partial x_i} \left[ \frac{\mu_{\text{eff}}}{\sigma_k} \frac{\partial k}{\partial x_i} \right] + \frac{\partial}{\partial x_i} \left( \rho \varepsilon \right)
\]  

(6)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} = - \frac{\partial }{\partial x_i} \left( \rho u_i \varepsilon \right) + \frac{\partial}{\partial x_i} \left[ \frac{\mu_{\text{eff}}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right] + c_{\varepsilon 1} \frac{\varepsilon}{k} \Pi - c_{\varepsilon 2} \rho \frac{\varepsilon^2}{k}
\]  

(7)

Where, \( \Pi = \left[ \mu_i \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{i,j} \right] \frac{\partial u_i}{\partial x_j} \) - production of turbulent kinetic energy.

Convert the kinetic energy of pulsational motion into internal energy (dissipation). \( \sigma_k, \sigma_\varepsilon \) -- corresponding to the Prandtl number.

The system of equations is not closed, since the turbulent flow regime the effective transport coefficients \( \mu_{\text{eff}}, a_{\text{eff}}, D_{\text{eff}} \), are unknown functions. According to the Reynolds analogy, we can use the following relations:

\[
\rho a_{\text{eff}} = \frac{\mu_{\text{eff}}}{Pr_{\text{eff}}}, \quad \rho D_{\text{eff}} = \frac{\mu_{\text{eff}}}{Sc_{\text{eff}}}
\]

Where \( Pr_{\text{eff}}, Sc_{\text{eff}} \) -- effective of Prandtl number and Schmidt, respectively, whose values are set...
to be constant.

The effective viscosity is determined by modeling the relationship of the Kolmogorov-Prandtl:

$$\mu_{eff} = \mu + C_\mu \rho \frac{k^2}{\varepsilon} \quad (8)$$

Where $\mu$ - physical viscosity, $C_\mu$ - an experimental constant.

3. Results of numerical researches

Numerical simulation of turbulent flows with chemical reactions, including thermo-dynamic, kinetic and three-dimensional computer modeling of combustion chambers, with the lowest cost will investigate in detail the turbulent combustion of powdered coal in a real combustion chambers, and give practical advice on the use of new technology of low-grade solid fuel burning.

The mathematical model describing reacting flow in the combustion chamber, includes the nonlinear differential equations: the equation of continuity, motion of a viscous medium, heat distribution and diffusion components of the reaction mixture and the reaction products, the equation of state and the equation of chemical kinetics. The solution of these equations was carried out on the basis of a program complex for three-dimensional modeling of FLOREAN.

In figures 3-14 results of numerical modelling for two values of turbulence degrees at $T_u=0.05$ and $T_u=0.1$, also their analysis is carried out.

Figure 3. Temperature distribution by height of the combustion chamber At $T_u=0.05$
* - experiment [2]

Figure 4. Temperature distribution by height of the combustion chamber At $T_u=0.1$

Analysis for the graphs in Fig. 3-5, which shows the temperature distribution shows that the output of the combustion chamber have a lower gas temperature. That is explained by the conditions of high ignition of Ekibastuz coal is reduced, and heat transfer in the furnace, related pollution screens layer of
flying ashes. Ash settling on the heating surface, affects the heat transfer and increases resistance to flue pipes and causes big harm to the equipment. At the same time, it may be noted that the conditions for ignition at $\text{Tu} = 10\%$ better, and the output, we have very small differences in temperatures.

![Figure 5](image5.png)

*Figure 5. Comparison of temperature distributions along the combustion chamber for two turbulence degrees at $\text{Tu}=0,05$ and $\text{Tu}=0,1$."

![Figure 6](image6.png)

*Figure 6. Concentration profile of CO by height of the combustion chamber at $\text{Tu}=0,05$"

![Figure 7](image7.png)

*Figure 7. Concentration profile of CO by height of the combustion chamber at $\text{Tu}=0,1$"

It is known that at higher turbulence is a more complete combustion and fewer emissions of harmful gases, which has a positive impact on the environment. We can show it in the following figures. Distribution maximum, minimum and average on section of values of concentration of CO (carbon
monoxide), and comparison of their mean values for the two turbulence degrees in the combustion chamber height are shown in Figures 6-8. And the figures 9-11 are the same distribution for the carbon dioxide CO2.

Figure 8. Comparison of concentration profile of CO by height of combustion chamber for two turbulence degrees $T_u=0.05$ and $T_u=0.1$.

Figure 9. Concentration profile of CO2 by height of the combustion chamber at $T_u=0.05$

Figure 10. Concentration profile of CO2 by height of the combustion chamber at $T_u=0.1$
From the graphs Figures 6-11 shows that the concentration profile of the height of the combustion chamber, depending on the gas (CO, CO2) for different values of turbulence (Tu = 0.05; Tu = 0.1) are not qualitatively different. At the same time, there are quantitative changes, which can be explained by the fact that the more turbulent flow, the better the mixing of gases within the chamber. This in turn leads to more complete combustion and reduce emissions from coal-fired.

Figure 12. Concentration profile of CH4 by height of the combustion chamber at Tu=0.05

Figure 13. Concentration profile of CH4 by height of the combustion chamber at Tu=0.1
The importance and convenience of numerical simulation of complex phenomena that occur in the combustion chamber are quite obvious. As a result of computer simulation for the boiler PK-39 Ekihsztuz power plant in the present study was obtained by an extensive bank characteristics of the fuel process, the temperature field, concentration of combustion products, the energy released by chemical reactions for the two turbulence degrees $Tu = 0.05$ and $Tu = 0.1$. It is shown that the chosen mathematical model allows to calculate the parameters of a satisfactory flow and thermal characteristics of devices.

We obtain detailed characteristics of temperature, concentration of coal combustion products (including harmful CO, CH4, CO2) around the combustion chamber for different values of the turbulence degree: $Tu = 0.05$ (5%) and $Tu = 0.1$ (10%).

The comparison showed that the nature of the temperature and concentration curves reasonably well modeled, and coincides with the experimental data. This shows the correctness of the applied in this paper a mathematical model of a turbulent jet and its coal-dust distribution in the combustion chamber operating power plant. The greatest differences in the calculated and experimental values can only be seen in the area of inflammation and decay.

It is shown that modeling the formation of harmful substances (CO, CO2, etc.) through use in the model and software package is quite possible. The simulation results allow to optimize the combustion process of high-pulverized coal to reduce emissions and allow power plants to create a "pure" and the spectacular use of coal.

**Reference:**


