Aerodynamic structure of a swirling turbulent flow in a vortex furnace

Vas V Salomatov\textsuperscript{1} and VI V Salomatov \textsuperscript{1,2}

\textsuperscript{1}Kutateladze Institute of Thermophysics SB RAS, Lavrentiev ave. 1, 630090, Novosibirsk, Russia
\textsuperscript{2}Novosibirsk State University, Pirogov str. 2, 630090, Novosibirsk, Russia

Abstract. Insufficient study of the swirling flow structure, stability of high-temperature circulating zones, intensity of fuel and oxidizer mixture formation and other determining phenomena in the vortex furnaces indicates insufficiency of scientific substantiation of the methods for their calculation. Therefore, further detailed studies of aerodynamics of a turbulent swirling flow in a vortex chamber with intersection of the main flow by the inlet tangential jets aimed at the development of promising combustion devices seem to be relevant. The results of numerical simulation of the turbulent swirling flow aerodynamics with preference given to the modern models of turbulence and numerical schemes of high-order approximation are presented.

1. Introduction

The coal energy sector includes a variety of combustion schemes: layered, flare, with boiling or circulating boiling layers, in slag melt, vortex, etc. The use of a particular scheme is dictated by the quality of fuel, efficiency of ignition and burnout, environmental friendliness. The use of the vortex technology has the following advantages: an increased time of coal particles residence in the combustion space, reduction in overall dimensions of the combustion chambers, ability to hold a significant supply of fuel in volume, high heat capacity of the vortex furnace, stability of ignition and burnout processes, improvement of environmental parameters, etc. This is caused by the fact that in high-temperature and high-turbulent furnaces of the vortex type with a tangential supply of an air-fuel mixture at significant injection rates, there is a significant centrifugal effect, due to which burning particles are thrown to the periphery of the chamber, where they are immersed in the slag melt and remain in a quasi-inhibited state relative to the swirled flow of the carrier medium. As a result, on the one hand, the supply of oxidant for combustion with rapid removal of volatiles increases, and on the other hand, the conditions for thermal crushing of particles and intensive heat and mass transfer are created, and this ensures effective burnout of any low-quality fuel.

In addition to this effect, the aerodynamic structure of the vortex furnace is characterized by stable high-turbulent energy-saturated zones, which provide the highest volumetric heat stresses in such furnaces. Moreover, the mass of fuel circulating in the vortex chamber is suspended, which significantly accelerates the heat and mass transfer processes. A significant supply of fuel, rotating repeatedly in the volume of the combustion chamber, gasified and gradually burns out, which gives the required stability to
the vortex process. The advantage of vortex combustion is the ability to create a relative movement of the oxidizer and fuel. This guarantees a faster delivery of the oxidizing agent than in the case of diffusion only. Thus, the vortex transfer is the aerodynamic basis of the entire combustion process in such furnaces. The main role of the aerodynamic structure of the vortex flow is reduced to a perfect mixing of gasified fuel and oxidizer without which it is impossible to achieve neither the required burnout rate, nor the permissible characteristics of toxic emissions, as well as a high level of combustion efficiency.

Such complications of mathematical models as flow swirling, physicochemical transformations, turbulence, multicomponent structure, etc. are contain nonlinearities, which causes difficulty even when constructing a numerical algorithm. Unfortunately, the universal method, optimal with respect to resources and computation time, accuracy and efficiency of simulations has not been created yet. In the work based on the SIMPLE approach, a numerical method using Rhie-Chow interpolation on combined grids and monotone TDV-schemes of the second and third order approximation is implemented. Numerical calculations were tested using such models of turbulence as the RNG modification of the $\kappa-\varepsilon$ model, created for the swirling flows, and $\kappa-\varepsilon$ model corrected for the Richardson number. Comparison of numerical calculations with the results of our most detailed thermo-anemometric measurements carried out in a vortex furnace model using RNG type and data on near-wall swirling flows by E. P. Volchkov and V. I. Terekhov, considering the Richardson correction, proves their comparability that satisfies engineering practice.

### 2. Mathematical model and numerical algorithm

When studying aerodynamics of a vortex furnace, the factors of turbulence, complexity, and swirl complicate the mathematical model and numerical algorithm significantly. Under these conditions, the analysis of all stages of simulation is especially needed to establish the adequacy of numerically obtained results to the experimental data. It is necessary to build an algorithm that is economical with respect to resources and computation time. To date, various researchers have proposed a number of numerical methods, but, unfortunately, there is no method, optimal in terms of accuracy and efficiency of implementation. The algorithm used for the numerical solution of the Navier-Stokes equations in natural variables (the components of velocity vector, density and pressure [1]) can be divided into algorithms using the continuity equation to derive the density distribution equation, and algorithms using the pressure equation (or similar), obtained as a combination of continuity and momentum equations. The algorithms of the first group are most effective in calculating compressible flows, while the algorithms of the second group, on the contrary, are focused on calculations of subsonic velocities.

As for numerical implementation of these methods, there are the following approaches:

- Absolutely explicit finite-difference schemes. Their advantages are simplicity and clarity of implementation. The main disadvantage is the restriction of the time step of the form

\[ \tau \leq 0 \left( \frac{h}{\|u\| + a} \right) \frac{\text{Re} h^2}{\min_{s} \tau_{\text{react}}^s} , \]

where $\tau,$ $h$ are time and space steps, $u$ is local velocity vector, $a$ is sound velocity, $\text{Re}$ is Reynolds number, $\tau_{\text{react}}^s$ is characteristic time of the $s$th physical-chemical process. One of restrictions on the right side of the given inequality is often quite rigid, which significantly reduces the applicability of this approach.

- Partially implicit algorithms based on the sequential solution of equations of the original system within a single iterative procedure. With this approach for difference analogues, the terms of the equation corresponding to contributions of other variables are approximated explicitly. The main advantage of such methods is the ability to take into account easily various physicochemical
phenomena in the model without the need to revise the entire algorithm. A common disadvantage of all methods of this class is their conditional stability, and, therefore, insufficiently fast convergence.

To date, extensive experience on using these algorithms for simulation of chemically reacting and multiphase flows, which testifies to their performance on a wide class of applied problems, has been gained.

- Implicit difference schemes, the general approach to which is described in [2, 3]. These schemes are unconditionally stable, and limitation from above to the time step when solving, for instance, the stationary problems by the relaxation method is caused only by non-linearity of the original differential equations. The profitability of these algorithms follows from the possibility of using the scalar sweeps or LU factorization at the stage of solving the difference equations. It is necessary, however, to state that experience in applying these schemes to modeling the flows with complex physicochemical processes is not yet great [4], which is primarily due to the mathematical complexity of deriving the algorithmic formulation of the problem.

3. Features of numerical calculation of the swirling flows

To calculate the swirling flows of a viscous incompressible liquid numerically, most often the natural physical variables (the components of velocity vector and pressure) are used. When writing the difference analogues of convective terms in differential equations, the schemes with differences against the flow or similar ones have become common. We should note that the possibility of using the schemes of the second-order and higher-order approximation for calculating the complex flows can allow one to achieve significantly better resolution of the algorithm on large grids, which has a decisive effect on the efficiency of two-dimensional and especially three-dimensional calculations.

However, the construction of schemes of even the second order of approximation, due to their non-monotonicity, faces considerable difficulties. For example, the scheme of the second approximation order used in [5] required introduction of additional dissipative terms to suppress oscillations in a numerical solution. The need to select coefficients in expressions for artificial viscosity [5] makes it difficult to use such a numerical method for engineering calculations, since it requires additional adjustment of the numerical method when solving each specific problem. The most promising here is the use of schemes with correction of flows or TVD-schemes of a higher approximation order, which allows reduction in uniformity of the computational method and reduce oscillations significantly. We emphasize that the difference schemes that meet the TVD criterion lead to the limited numerical solutions determined by the initial and boundary conditions, and this is adequate to the physical meaning.

This study uses an approach based on the SIMPLE method. Particular attention is paid to the use of counter-flow schemes of the higher (second and third) approximation orders for the convective terms and schemes with the TVD property, and this allows high resolution of the numerical algorithm and preservation of the monotony of numerical solution in the areas of large gradients. When approximating the convective terms, the use of the CVS TVD scheme is most efficient [7]; it was preferred on the basis of the above test calculations and comparisons with experimental data. When using the combined grids, i.e., the grids, where the pressures and velocity vector components are determined at the centers of grid cells, and direct obtaining of a discrete analogue of equation for pressure is complicated by the following. The source terms of the motion equations contain pressure gradients taken at the centers of grid cells, and if these values are obtained using the central-difference approximations, then the pressure values at neighboring points are not related directly. This leads to the so-called “chess effect”, when the pressure values in the grid cells split into two alternating subsets. The same effect takes place for the irregular grids. One of the ways to solve this question is introduction of the so-called spaced grids (when the velocity components are determined on the faces of the grid cells, and the pressure and other scalar
variables are determined at the centers [1]). However, such grids lead to computational inhomogeneity of the algorithm and make it difficult to use multigrid methods of convergence acceleration. On the ordinary grids, the Rhie-Chow interpolation is an effective way to eliminate the “chess effect” when approximating the pressure gradients [8].

We should also point out that in the SIMPLE-type algorithms an equation for pressure corrections is used; in this case, a solution that satisfies the continuity equation is found through corrections to the pressure field. We used the expression for normalized residuals, where \( c_\phi \approx 10^{-3} - 10^{-5} \) as a criterion for the convergence of numerical calculation. When solving individual equations, it is often necessary to use lower relaxation to achieve convergence.

4. Modification of the \( \kappa-\varepsilon \)-model of turbulence for the swirling flow

4.1. Modification of RNG type

The latter is distinguished by the fact that it clarifies the results for the flows with fast deformations, which, in particular, include the swirling flows. The RNG modification of \( \kappa-\varepsilon \) turbulence model [7, 8] was obtained by using the apparatus of the theory of renormalized groups with respect to equations of a viscous incompressible fluid. The term “renormalization” means an approach to consideration of similar and spatial – invariant phenomena through their separation from the whole variety of observed. One of the first theories of renormalized groups to the description of the turbulent fluid flows was applied by Ma and Mazenko [9]. It is important that the values of the contact model are found analytically. It can be also noted that all RNG models are based on the assumption of Kolmogov spectrum of energy distribution of turbulent pulsations.

The expression for the efficient viscosity in the RNG modification takes form

\[
\frac{d\mu_{\phi\phi}}{dl} = A\varepsilon l^2, \tag{1}
\]

where \( A = \text{const} \), \( l \) is spatial size.

Then, the following dependence can be obtained analytically using (1)

\[
\mu_{\phi\phi} = \mu \left[ 1 + \frac{c_\mu \rho \kappa^2}{\mu \varepsilon} \right], \tag{2}
\]

where \( c_\mu = 0.0875 \), \( \mu \) is molecular viscosity of the medium. Further it can be noted that the value of empirical constant of the standard \( \kappa-\varepsilon \) model, equal to 0.09, is very close to that obtained using the RNG. Here \( \kappa \) is generation of the kinetic energy of turbulence, \( \varepsilon \) is the rate of its dissipation. Using RNG, modifications for \( \kappa \) and \( \varepsilon \) are as follows

\[
\frac{\partial}{\partial t}(\rho \kappa) + \frac{\partial}{\partial x^j}(\rho u^j \kappa) = \frac{\partial}{\partial x^j}(\alpha_{\kappa} \mu_{\phi\phi} \frac{\partial \kappa}{\partial x^j}) - \mu_s \frac{\partial \varepsilon}{\partial x^j} - \rho \varepsilon \tag{3}
\]

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x^j}(\rho u^j \varepsilon) = - \frac{\partial}{\partial x^j}(\alpha_{\varepsilon} \mu_{\phi\phi} \frac{\partial \varepsilon}{\partial x^j}) + c_1 \mu_s \kappa - c_2^\infty \rho \frac{\varepsilon^2}{\kappa} \tag{4}
\]

here \( c_2^\infty \) is given by formula

\[
c_2^\infty = c_2 + \frac{c_\mu \eta^3 \left( 1 - \frac{\eta}{\eta_h} \right)}{1 + \beta \eta^3} \tag{5}
\]
Here $\eta=s\frac{K}{\varepsilon}$ is the dimensionless parameter, characterizing the shear stress

$$s = \sqrt{2s_ys_{\mu}}, \quad s_y = \frac{1}{2}\left(\frac{\partial u'}{\partial x^i} + \frac{\partial u'}{\partial x^j}\right)$$  \hspace{1cm} (6)

The values of constants $\alpha_c$ and $\alpha_\varepsilon$ are determined using expression

$$\begin{bmatrix} \alpha - 1.3929 \\ 0.0321 \\ \alpha_\varepsilon - 1.3929 \\ - \alpha_\varepsilon + 2.3929 \end{bmatrix}^{.3679} = \mu \left[ \begin{array}{c} \mu \\ \mu_{\text{eff}} \end{array} \right] \hspace{1cm} (7)$$

where constant $\alpha_0$ is assumed to be 1. Numerical values $c_1 = 1.42; c_2 = 1.68; \eta_0 = 4.38; \beta = 0.012$ are obtained analytically. The main difference of RNG from the standard $\kappa$–$\varepsilon$-model is as follows:

• RNG–modification gives the analytical expression for the constants. This is especially significant in respect to the constants for $\varepsilon$ equation, since the methods for obtaining this equation and values of constants have been analyzed and criticized repeatedly.

• Due to the use of formulas for $\mu_{\text{eff}}$, it is possible to calculate the flows both with the areas of laminar and turbulent regimes.

4.2. Richardson corrections for the $\kappa$–$\varepsilon$–model of turbulence

Unfortunately, the standard $\kappa$–$\varepsilon$–model does not allow simulation of the flows that differ in the curvature of the current line, including the swirling flows, with a sufficient degree of accuracy [10]. This is due to the fact that the Reynolds stresses for the flows with the curved current lines differ from those observed in a free shear layer.

Various corrections of the standard $\kappa$–$\varepsilon$–model of turbulence for calculations of swirling flows were proposed in [11, 12]. These corrections are introduced either in the expression of turbulent viscosity, or as additional source terms in conservation equations. Corrections of the latter type are often introduced to the terms, describing generation and dissipation of empirical dependencies on the Richardson number

$$Ri = \frac{\kappa^2}{\varepsilon_{r^2}} \frac{\partial \nu}{\partial r}$$  \hspace{1cm} (8)

The specific form of correction terms for the equations of $\kappa$–$\varepsilon$-model was proposed by Launder [13]

$$\frac{\partial (\rho K)}{\partial t} + \frac{\partial (\rho K)}{\partial x} + \frac{1}{r} \frac{\partial (\rho v K)}{\partial r} = \frac{\partial}{\partial x} \left( \frac{\mu_s}{\sigma_x} \frac{\partial K}{\partial x} \right) +$$

$$+ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\mu_s}{\sigma_x} \frac{\partial K}{\partial r} \right) + G_{\mu_{\varepsilon}} - \rho \varepsilon$$

$$\frac{\partial (\rho r)}{\partial t} + \frac{\partial (\rho u e)}{\partial x} + \frac{1}{r} \frac{\partial (\rho r v e)}{\partial r} = \frac{\partial}{\partial x} \left( \frac{\mu_s}{\sigma_x} \frac{\partial e}{\partial x} \right) +$$

$$+ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\mu_s}{\sigma_x} \frac{\partial e}{\partial r} \right) + G_{\varepsilon} - \frac{e}{\kappa} + c_{\varepsilon 1} (1 - c_R) E_1 \frac{e^2}{\kappa}$$  \hspace{1cm} (9)

where $c_R$ is Richardson constant.
5. Testing numerical calculations of the swirling flows

5.1. Application of $\kappa$-$\varepsilon$ model corrected for Richardson number
This testing was carried out based on our thermo-anemometric data obtained on the Golovanov model of air vortex furnace, as well as the most detailed thermo-anenometric data on the near-wall swirling flows obtained by Volchov – Terekhov scientific school [14]. The calculated profiles of the longitudinal component of velocity vector $U$ are compared in Fig. 1a, and profiles of turbulence intensity $T_u$ in different cross-sections of the swirling flow in a circular channel with an open end are compared in Fig. 1b. The closest match between the calculated and experimental data was obtained using the correction for the Richardson number $c_{Ri} = 0.02$.

5.2. RNG-modification of the $\kappa$-$\varepsilon$-model
Relative to the use of RNG-modification of the $\kappa$-$\varepsilon$ turbulence model in calculations of the swirling flow with a swirl parameter of about 1 in a round channel with an open end: RNG-modifications of the $\kappa$-$\varepsilon$ models corrected for the Richardson number give the results comparable in terms of accuracy.

The aerodynamic structure of a swirling flow in a vortex furnace includes the following features:

- the average component of the azimuthal velocity component has a maximum against the output of the burner. The profile of the pulsating velocity component at the same position has a minimum, which indicates the jet character of the discharge from the burner;
- at different Reynolds numbers it follows that the flow is not self-similar;
- in addition to predominantly azimuthal motion, circulation along the axis of the combustion chamber is observed at the periphery;
- the profile of the tangential velocity component is a potential vortex;
- the profiles of radial and axial velocity components have a complex structure, which is associated with redistribution of the inlet jets in the radial direction or axial recirculation of flows;

...
Figure 1. Comparison of calculation profiles of longitudinal component of velocity vector $U$ (a) and $Tu$ (solid lines – experiment, dashed lines – calculation by $\kappa$-$\varepsilon$-model with Richardson number correction $c_{Ri} = 0.02$)

- when the flow leaves the vortex chamber, its rapid laminarization occurs;
- the relative value of velocity maximum depends both on the average flow rate through the burners and their relative width, the maximum of average velocity is accompanied by the maximum of pulsating velocity component, which indicates intense turbulent mixing at the edges of jets and less intense in the center;
- the flow in the swirling part has a “glove character”, removal occurs in the plane of symmetry between the burner nozzles;
- complex spatial movement is formed in the cooling chamber depending on the operating parameters; in most regimes, the flow is essentially uneven both in length and in height;
- on the front wall there is an area of reverse flows. The asymmetric flow pattern leads to significant nonequilibrium of the flow field; in the cooling chamber there are the areas where the absolute velocity values are several times higher than the average flow rate;
- both vertical and horizontal vortices occur over the cross-section of the cooling chamber.

The data obtained are the scientific basis for the creation of promising technological processes and technical devices with turbulent swirling flows. As it is shown in the paper, the choice of a turbulence model affects decisively the accuracy of calculations of the swirling flows.

Conclusions
Based on the SIMPLE approach, a numerical algorithm is implemented using Ray-Chow interpolation on combined grids and monotonic TVD schemes of the second and third order of approximation. Numerical calculations were tested using turbulence models such as the RNG modifications of the $\kappa$-$\varepsilon$ model, specially created for swirling flows and $\kappa$-$\varepsilon$ corrected for the Richardson number. Comparison of
numerical calculations with the most detailed anemometric measurements and data on wall swirling turbulent jets Volchkova E.P. and Terekhova V.I. taking into account the Richardson amendment, indicates their comparability, satisfying engineering practice.

References
[1] Patankar S 1980 *Numerical Heat Transfer and Fluid Flow* (Hemisphere Publishing Corporation)
[2] Launder B T, Spalding D B 1974 *Computational Meth. Appl. Mech. Engg.* 3 (1) 269–89
[3] Fletcher C 1983 *Computational Techniques for Fluid Dynamics* (New York: Springer-Verlag)
[4] Anderson D, Tannehill J, Pletcher R 1984 *Computational Fluid Mechanics and Heat transfer* (Mc-Graw Hill)
[5] Aithal S M, Subramaniam V V, Pagan J, Richardson R W 1998 *J. of Appl. Phys.* 84 3506–17
[6] Rhie C M, Chow W L 1983 *AIAA J.* 21 1525–32
[7] Yakhot V, Orszag S A 1986 *J. Sci. Comp.* 1 (1)
[8] Yakhot V, Orszag S A 1992 *PhysJFluids A.* 4 (6) 1510–5
[9] Ma S-k, Mazenko G F 1975 *Phys. Rev. B.* 11 4077–100
[10] Gupta A, Lilley D, Syred N 1984 *Swirl Flows* (Abacus Press)
[11] Sloan D G, Smith P J, Smoot L D 1986 *Prog.Energy Combust. Sci.* 12 163–250
[12] Launder B E, Priddin C H, Sharma B I 1977 *Trans. Am. Soc. Mech. Engrs, J. Fluids Engng.* 99 231–9
[13] Launder B E, Sharma B I 1974 *Lett. Heat Mass Transfer.* 1 131–8
[14] Volchkov E P, Spotar S Yu, Terekhov V I 1982 *Swirling wall jet in a cylindrical channel* (Novosibirsk: Preprint AS USSR, Sibirian Branch, Institute of Thermophysics) p 42