Experimental applicability limits investigation of Spalart-Almares turbulent model and Reynolds stresses transfer model under control of rotary-divergent flow

I K Kabardin¹, N I Yavorsky¹, V G Meledin¹, M Kh Pravdina¹, M R Gordienko¹, D P Ezendeeva¹, S V Kakaulin², E V Usov², I A Klimonov², S V Krotov¹, G V Bakakin¹ and A K Kabardin¹

¹Kutateladze Institute of Thermophysics SB RAS, 630090, Lavrentyev ave. 1, Novosibirsk, Russia
²Nuclear Safety Institute RAS, 52, Bolshaya Tulskaya Street, Moscow, 115191, Russia

E-mail: ivankabardin@gmail.com

Abstract. The work continues the work of the previous year. The mass transfer characteristics were experimentally analyzed in a rotary-divergent flow with inserted blades. The laser Doppler anemometer was used for turbulent mass transfer diagnostics. The experimental investigation was used to verify the numerical calculations. As in previous year the model of Reynolds stress transfer describes the flow in range of flow rates from 50 to 250 n.m³/h more adequately than the Spallart-Almares turbulence model.

1. Introduction
This work continues the work of the previous year [1], where flow characteristics were investigated in the rotary-divergent flow. In this work one blade was inserted in the rotary section for the flow control. The verification of numerical calculations by computational hydrodynamics was carried out according to the results of experimental studies. Numerical simulation of turbulent gas flow is based on solving a system of continuity and Navier-Stokes equations. For averaged equation closure, semi-empirical models of turbulence were used: the Spalart-Almares turbulence model [2] and the Reynolds stresses transfer model [3-4]. This choice was based on the following hypotheses. Each of these models has advantages and disadvantages. The Spalart-Almares turbulence model is cost-effective model applicable for a large number of flows. But the use of an algebraic equation for the length scale is too restrictive. Transport of the length scale is not accounted for. The Reynolds stresses transfer model is applicable for complex flow, where the turbulent-viscosity models fail. The Reynolds stresses transfer model accounts for anisotropy. It has good performance for many complex flows, e.g. swirl, flow separation and planar jets. But Reynolds stresses transfer model is computationally expensive with 11 transport equations. Several terms in the transport equations must be closed. It has poor performance for some flows due to the closures introduced in the model. The aim of the work was to compare both models on the basis of verification of numerical calculations, which was made by measurements of velocity fields in the model sections. Measuring the two-dimensional kinematic characteristics of the flow required a modern non-contact optical method for aerodynamic flow diagnostics, laser Doppler anemometry (LDA) [1].
2. Experimental stand

To investigate the applicability area for turbulence models in the problems of mass transfer intensification by the control of a rotary divergent flow, the experimental model was made (Fig. 1) and an aerodynamic measuring stand for the study of flow characteristics in this model was constructed. The model consists of the following functional blocks: section before the rotary device, control section and rotary devices. Air is supplied to the model through the fan 1; then gas flows pass through the flow meter 2. The temperature in the flow is measured by thermal transducer 3, and the overpressure P is measured by pressure sensors 4. The hydraulic resistance of the flow is measured by the differential pressure ΔP on a rotary device 5 and a section before the second rotary device 6 using the differential pressure transmitters 8. The working fluid is air. The flow rate was equal to 50, 100, 150, 200 and 250 n.m$^3$/h. The maximum overpressure was 0.03 MPa. The working temperature took values in the range of 23-26°C. The Reynolds number varied in the range of $10000 - 50000$.

![Figure 1. 3D geometry of the investigated model (1 – inlet; 2 – section before the rotary device, 3 – rotary device, 4 – control section, 5 – section before the second rotary device).](image)

2.1. Velocity measuring in the experiment

To verify numerical calculations, the velocity fields in the model cross-sections were measured. Measurement of the two-dimensional kinematic characteristics of the flow required a modern non-contact optical method for aerodynamic flow diagnostics, laser Doppler anemometry (LDA)[1].

3. Flow control

In curved channels, due to the curvature of the flow, centrifugal forces, directed from the center of curvature to the outer wall of the pipe, appear. This determines the increase in pressure at the outer wall and its decrease at the inner wall when the flow passes from a straight section of the pipeline to a bent one. Therefore, the flow rate will accordingly be lower at the outer wall and higher at the inner. At this point, a diffuser effect appears near the outer wall, and a confuser effect appears near the inner wall. The transition of the flow from the curved part to the rectilinear one (after rotation) is accompanied by the inverse phenomena: the diffuser effect near the inner wall and the confuser effect near the outer [5]. Flow resistance can be reduced by installing guide vanes. The big advantage of this method is preservation of the compactness of installation.
The blades of the same shape and size are usually installed in the knees, and most often they are placed along the bend of the channel. The aerodynamic lattice in the knee, composed of guide vanes, due to the aerodynamic force developing on it, causes the flow to deflect to the inner wall. With the right choice of sizes, number and angle of installation of the blades, this deviation of the flow prevents separation of the jet from the walls and formation of a vortex region. In accordance with (Fig. 2) [5], it is proposed to use the following number of blades for the most uniform distribution of speed:

\[ n = 2.13 \cdot \frac{P}{t-1} \]  

(1)

where \( P = (S_1 + S_2)^{0.5} \), \( S_1, S_2 \) are cross-sections areas on the knee inlet and outlet. \( S_1 = a_1 \cdot b_1, S_2 = a_2 \cdot b_2 \), are tubes dimensions. The following design parameters are used: \( a_1 = 11 \text{ mm}, b_1 = 75 \text{ mm}, S_1 = 8400 \text{ mm}^2 \), \( a_2 = 50 \text{ mm}, b_2 = 167 \text{ mm}, S_2 = 223 \text{ mm}^2 \). So \( P = 223 \text{ mm} \). Given that the length of the chord of the prototype varies from 110 to 420 mm, we will evaluate it as arithmetic mean, for simplicity \( t = 265 \text{ mm} \).

The blades are evenly spaced. The distance between the chords of the blades is:

\[ A = \frac{P}{n+1} = 74 \text{ mm} \]  

(2)

In this paper, flow control with one blade is investigated. Using the calculated parameters, a design diagram was prepared and flow calculations were performed. The use of a guide vane reduces vortex formation in a bend and makes the velocity profile of the control section (section 3) more uniform (Fig. 2).

4. Numerical simulation methods

The numerical simulation was carried out like in the previous year [1]. For closing the averaged equations, two models of turbulence were used: Spallart-Almaraes model [2] and the model of Reynolds stress transfer [3-4]. Since turbulent eddies have a certain lifetime and are transported by convection, turbulence is not completely determined by the local conditions but depends also on the history of eddies. In the Spallart–Almaraes model, the turbulent viscosity relates to a transported turbulent quantity instead of relating to the mean velocity gradient. It can be mentioned that transport of the turbulent kinetic energy is taken into account in one-equation models by solving one additional particle derivative equation. A solution will determine the length scale from an additional transport equation [6].

For the Spallart-Almaraes model, the boundary conditions on a solid surface are formulated as follows. The velocity component normal to the surface is zero. For the tangential velocity component the no slip condition is used. The inlet conditions were located in cross-section 1 so that the approximations under the given conditions were not affect the results of the simulations. For a better
approximation the turbulence intensity and turbulence length scale were used. The turbulence intensity for a pipe at high Reynolds number can be estimated from [6]:

\[ I = \frac{u}{\langle U \rangle} = 0.16 \text{Re}^{-1/8} \] (3)

and the turbulence length scale is given by \( l = 0.07L \), where \( L=0.092 \text{ m} \) is the hydraulic diameter and \( U=12.2 \text{ m/s} \) is the average velocity. It is then possible to estimate \( k \) and \( \varepsilon \) from

\[ k = \frac{3}{2} \langle (U) \rangle^2 \quad \text{and} \quad \varepsilon = C_{\mu}^{3/4} \frac{k^{3/2}}{l} \] (4)

Specifying boundary conditions for the Reynolds stress transport model is more difficult, since all the stresses must be specified. The initial conditions were set in the following way. The correct initial conditions were estimated by several iterations. At the first step, the turbulence kinetic energy was equal to the experimental values. With these values, the first calculations were done. After that, the values of the velocity field and Reynolds stresses were taken from cross-section 2 and set on cross-section 1 at the inlet. Another calculation has been done. Iterations repeated until difference between the values in cross-sections 1 and 2 became less that required error value.

The nonuniform grid with \( 1.5 \times 10^6 \) cells was used in computations. The computation grid was condensed near the walls to get resolution of laminar sublayer near the wall. Near the walls the computational grid was rectangular and in the flow core the computational grid was triangular. The cell size near the walls was about 0.2 mm and in the flow core it was about 8 mm. At the inlet, the experimental measured velocity distribution was used to get better accuracy of computation. The condition on the constant pressure of 1 bar was set at the inlet of experimental setup. The comparison of the experiments and numerical simulation results are presented below.

5. Results
Measurements of velocity, or rather its axial \( V_x \) velocity components, were carried out in the three cross-sections (see Fig. 3). For cross-sections 1, 2 the \( Z \) coordinate was 38 mm, and for cross-section 3, the \( Z \) coordinate was 93 mm.

![Figure 3](image-url) Three measurement cross-sections in the model.

The results of velocity profile measurements are shown in Fig. 4. The results of calculations for the longitudinal velocity component in cross-sections along the symmetry line are shown in Fig. 5. Comparison of experimental and calculated data shows that for the flow after the rotation, there are noticeable differences for all models. The experimental data are best described by the Reynolds-Stress transfer model.
Figure 4. Experimental axial velocity surface in cross-section 3 at flow rate G = 250 m$^3$/h.

Figure 5. Comparison of experimental and calculated velocity profile at flow rate G = 250 m$^3$/h.

The difference in area under velocity profile in the channel after rotary section was chosen as a characteristic of applicability area for the turbulence models. The recirculation zone width in cross-section 3 after the rotary device for different values of flow rate is shown in Figure 6. The model of Reynolds stress transfer describes the flow in range of flow rates from 50 to 250 n.m$^3$/h more adequately than the Spallart-Almares turbulence model.

Conclusions
The turbulent mass transfer characteristics were analyzed experimentally in a controlled rotational-divergent flow. The laser Doppler anemometry was used for turbulent mass transfer diagnostics. The verification of numerical calculations by computational hydrodynamics was carried out according to the results of experimental studies. Numerical simulation of turbulent gas flow is based on solving the system of continuity, Navier-Stokes and energy conservation equations. The medium is considered incompressible and isothermal. For averaged equations closure, semi-empirical models of turbulence were used: the Spallart-Almares model of turbulence and the Reynolds stresses transfer model. The model of Reynolds stress transfer describes the flow in the range of flow rates from 50 to 250 n.m$^3$/h more adequately than the Spallart-Almares turbulence model.
Figure 6. The difference in area under velocity profile in the channel after rotary section.

Acknowledgements
The experimental study of flow kinematic properties was carried out under state contract with IT SB RAS (AAAA-A18-118051690120-2), the study of applicability area for turbulence models in the problems of mass transfer intensification by the control of a rotary divergent flow was financially supported by RFBR (Project No. 18-31-20036).

References
[1] Kabardin I K, et al. 2019 IOP Journal of physics: conf. series 1359 012096
[2] Spalart P R, Allmaras S R 1994 La Recherche Aerospatiale 15–21
[3] Wilcox D C 1998 Turbulence modeling for CFD (DCW Industries. Inc. La Canada. California)
[4] Walters D K, Cokljat D A 2008 Journal of Fluids Engineering 130 121401–14
[5] Idelchik I Ye 1992 Handbook of hydraulic resistance. 3rd ed. (M.: Mechanical Engineering) p 672
[6] Andersson B, Andersson R, Håkansson L, Mortensen M, Sudiyo L, Wachem B, Hellström L 2012 Computational fluid dynamics for engineers. 2011( University Press, Cambridge) p 212