Experimental investigation of applicability limits of K-e turbulent model and Reynolds stresses transfer model in rotary-divergent flow under control via turning blades

I K Kabardin¹, N I Yavorsky¹, V G Meledin¹, M Kh Pravdina¹, M R Gordienko¹, D P Ezendeeva¹, S V Kakaulin¹, E V Usov², V A Kolotilov², S V Krotov¹, G V Bakakin¹ and A K Kabardin¹

¹Kutateladze Institute of Thermophysics SB RAS, 630090, 1 Lavrentyev Ave., Novosibirsk, Russia
²Nuclear Safety Institute RAS, 52 Bolshaya Tulskaia 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 K-e 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 three blades was inserted in the rotary section for the flow control. This choice is based on the following hypothesis. Each of these models has advantages and disadvantages. The K-e turbulence model [2] is a complete model in the sense that velocity and length scales of turbulence are predicted with transport equations. It shows good results for many engineering applications. The K-e turbulence model is especially good for trend analysis. It is robust, economical and easy to apply. But the K-e turbulence model is limited to an eddy-viscosity assumption. The Reynolds stresses transfer model is applicable for complex flows where the turbulent-viscosity models fail. The Reynolds stresses transfer model [3] accounts for anisotropy. It has good performance for many complex flows, e.g. swirl, flow separation and planar jets. However, the Reynolds stresses transfer model is computationally expensive with 6 transport equations for stresses transfer and 1 equation for the rate of dissipation of turbulent kinetic energy. It has poor performance for some flows due to the closures introduced in the model. The aim of the work is to compare both models on the basis of verification of numerical calculations by measurements of velocity fields in the model sections. Measuring the two-dimensional kinematic characteristics of the flow requires a modern non-contact optical method for aerodynamic flow diagnostics, laser Doppler anemometry (LDA). Meter LAD-05 based on LDA method is designed and manufactured at the Institute of Thermophysics SB RAS.
2. Experimental stand
The experimental stand of previous year [1] was modernized. In the rotary section 3 three blades were inserted for the flow control. The more detailed explanation of the blades forms choice and their displacement is presented in the next section. The flow rate was 250 m³/h as in previous year experiments. Measurement of the two-dimensional kinematic was made via laser Doppler anemometry method (LDA). The detailed explanation of the method is presented in [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 one. 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 to the rectilinear part (after rotation) is accompanied by the inverse phenomena: the diffuser effect near the inner wall and the confuser one near the outer wall [4]. Flow resistance can be reduced by installing guide vanes. The great advantage of this method is the preserved compactness of the setup.

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 the separation of the jet from the walls and the formation of a vortex region. In accordance with [9], it is proposed to use the following number of blades for the most uniform velocity distribution:

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

(1)
where \( P = (S_1 + S_2)^{0.5} \), \( S_1, S_2 \) are the cross-sections of the area of the knee inlet and outlet, and \( S_1 = a_1 \cdot b_1 \), \( S_2 = a_2 \cdot b_2 \) are the 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 for simplicity as the arithmetic mean \( t = 265 \text{ mm} \). The blades are evenly spaced. The distance between the chords of the blades:

\[
A = \frac{P}{(n+1)} \approx 74 \text{ mm}.
\] (2)

In this paper, the flow control and the power of the blade is investigated. Using the calculated parameters, a design diagram is prepared and flow calculations are 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. The recirculation zone almost disappears and the near wall jet velocity maximum value is significantly decreased.

4. Numerical simulation methods

The numerical simulation was carried out similar to the previous year. The internal volume of the pipe was chosen, and 3-D geometry with a three blades was built. It was found that the radial size of the cell should be less than 0.2 mm near the wall. An irregular mesh was chosen for further calculations. Near the wall the mesh is rectangular with the cell size of 0.1 mm. A uniform triangular mesh is used in the stream core. The thickening of the mesh is present only in the vicinity of the blades (Fig. 3). The thickening of the mesh at the walls was performed using the «Inflation function» of the «Mesh module» of the computational software package. A uniform triangular mesh is used in the core, the element size is 5 mm. Each configuration has an average of 3-3.5 million cells.

For the K-\( \varepsilon \) model, the boundary conditions on a solid surface are formulated as follows [5]. All of the velocity components are equal to zero on the solid surface. Standard Wall Functions are used as well. The inlet conditions are set in cross-section 1 for the approximations under the given conditions not to affect the results of the simulations. For a better approximation the turbulence intensity and turbulence length scale are used. The turbulence intensity for a pipe at high Reynolds number can be estimated from [8]:

\[
I = \frac{u}{\langle U \rangle} = 0.16 \text{Re}^{-\frac{1}{6}}
\] (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

\[
k = \frac{3}{2} \left( \langle U \rangle \right)^2 \quad \text{and} \quad \varepsilon = C_{\mu} \frac{k^{3/2}}{l}
\] (4)
Specifying boundary conditions for the Reynolds stress transport model is more difficult, since all the stresses must be specified. The initial conditions are set as follows. The correct initial conditions are estimated by a few iterations. At the first step, the turbulence kinetic energy is equal to the experimental values. With these values first calculations are done. After that, values of velocity field and Reynolds stresses are taken from cross-section 2 and set on cross-section 1 at the inlet. Another calculation is done. Iterations are repeated until the difference between the values in cross-sections 1 and 2 becomes less than the required error value.

5. Results
Measurements of axial Vx components, were carried out in three cross section (see fig. 4) [1].

The experimental velocity surface is presented in figure 5. The calculated velocity is presented in figure 6. Comparison of experimental and calculated data shows that for the flow after rotation, there are noticeable differences for both models. The difference in the area under velocity profile (Fig. 7) in the channel after rotary section is chosen as a characteristic of applicability area for turbulence models. The model of Reynolds stress transfer more adequately describes the flow in the range of flow rates from 50 to 250 n.m³/h than The K-ε turbulence model.
Figure 5. Experimental axial velocity surface in cross-section 3 at flow rate of 250 m³/h.

Figure 6. Comparison of experimental velocity profile with calculated one in cross-section 3 for flow rate G = 250 m³/h.

Conclusions
To analyze characteristics of the turbulent flows the experimental facility and its computational model have been made. Computational model is a full digital twin of the experimental facility. The velocity profile in a rotational-divergent flow has been measured and calculated with the help of two different models of turbulence: K-e and Reynolds stress models. It is shown that Reynolds stress model is closer to the experimental data than K-e model.

Figure 7. 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 control of a rotary divergent flow was financially supported by RFBR (Project No. 18-31-20036).

References
[1] Kabardin I K, et al. IOP Journal of physics: conf. series. 2019 1382 012014
[2] Wilcox D C 1998 Turbulence modeling for CFD (DCW Industries. Inc. La Canada. California)
[3] Launder B E, Reece G J and Rodi W 1975 J. Fluid Mech. 68(3) 537–66
[4] Idechik I Ye 1992 Handbook of hydraulic resistance. 3rd ed. (M.: Mech. Engineering) p 672
[5] Andersson B, Andersson R, Häkansson L, Mortensen M, Sudiyo R, vanWachem B, Hellström L 2012 Comp. fluid dynamics for engineers (University Press, Cambridge) p 212