Experimental determination of turbulence models applicability limits for the fluid dynamic computation in rotary divergent flow

I K Kabardin¹, N I Yavorsky¹, V G Meledin¹, M Kh Pravdina¹, M R Gordienko², D P Ezendeeva¹, S V Kakaulin¹, E V Usov², I A Klimonov², S V Kroto², A K Kabardin¹ and O Yu Sadbakov¹

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

E-mail: ivankabardin@gmail.com

Abstract. Such fundamental scientific problem arises in much industrial application. The turbulent fluid dynamic characteristics were analyzed experimentally in a rotary-divergent flow. The laser Doppler anemometry was used for turbulent mass transfer diagnostics. The verification of numerical calculations by computational hydrodynamics has been carried out according to the results of experimental studies. Verification has shown that turbulent models describe the rotational-divergent flow well only until generation of flow gaps and formation of recirculation zones.

1. Introduction
The fundamental scientific problem of this kind arises in many industrial applications. The efficiency of mass transfer is influenced by the velocity field and small-scale turbulence in the flow. Streams in real industrial plants interact under conditions of complex geometry with turning and expanding sections in which wall jets are formed, flow reconnections occur, reverse flow zones are present. Such effects usually increase the hydrodynamic resistance of the flow and disturb the turbulent mass transfer uniformity. For the numerical simulation of such phenomena, verification and modernization of turbulence models are required for an adequate description of mass transfer. The verification of numerical calculations by computational hydrodynamics has been 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 equations. The medium is considered incompressible and isothermal. For averaged equations closure, semi-empirical models of turbulence were used. For verification of numerical calculations, measurements of velocity fields in the model sections were carried out. Model sections is part of the gas flow distributor breadboard that is used to investigate purification of industrial emissions from volatile organic compounds. Measuring the two-dimensional kinematic characteristics of the flow required a modern non-contact optical method for aerodynamic flow diagnostics, laser Doppler anemometry (LDA). Meter LAD-05 based on LDA method was designed and manufactured at the Institute of Thermophysics SB RAS.
2. Experimental stand
For experimental determination of turbulence models applicability limits for the mass transfer intensification in rotary divergent flow the experimental model was made (Fig. 1.a) and the 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. Then gas flows pass through the flow meter. The temperature in the flow is measured by thermal transducer, and the overpressure $P$ is measured by pressure sensors. The hydraulic resistance of the flow is measured by differential pressure $\Delta P$ on a rotary device and a section before the second rotary device using the differential pressure transmitters. The working fluid was 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 (a) and photo (b) (1 – the 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
For verification of numerical calculations, measurements of velocity fields in the model sections were carried out. 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). Velocimeter LAD-05 based on LDA method was designed and manufactured at the Institute of Thermophysics SB RAS (Fig. 2.a). The optical velocimeter was installed on the coordinate-moving device. As a result, it allows measuring two projections of the velocity vector in the range of 0.001...400 m/s with a relative error not exceeding 0.1 %. The size of the measuring zone is 0.1x0.1x0.5 mm. The positioning device allows moving the measuring unit in the area of 250 x 250 x 250 mm with an accuracy of 0.1 mm. This method also enables measurement of local flow rate fluctuations.
3. Numerical simulation methods

The numerical simulation was carried out. The distributions of the pressure and velocity field in the model were simulated. Numerical simulation of turbulent gas flow was based on the solution of the system of continuity equations, Navier-Stokes equations and energy conservation equation [1]. The medium was considered incompressible and isothermal. For closing the averaged equations, semiempirical models of turbulence were used: Spallart-Almares model [2], k-ε turbulence model, [3], k-ω turbulence model [4], k-kl-ω turbulence model [5] and the model of Reynolds stress transfer [3].

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 exit of the computational model. The sticking condition was set on the wall.

The operating parameters of the experimental setup correspond to the developed turbulent gas flow. In an averaged turbulent flow in a pipe, two regions can be distinguished [6]. On the wall of the tube, the gas velocity is zero and gradually increases towards the center. Therefore, viscous forces predominate near the wall and the flow can be considered laminar. In the core of the flow, the flow is completely turbulent. Correct modeling of the averaged turbulent flow therefore requires the selection of a near-wall region in which the laminar flow regime prevails. The size of the region y is determined
by the condition [7]: \( y^+ < 5 \), where \( y^+ = y \cdot U^* / \nu \). Here \( \nu \) is kinematic viscosity. \( U^* = \sqrt{\tau_0 / \rho} = U \sqrt{\xi / 8} \) is pulsation velocity. \( \tau_0 \) is the shear stress at the wall; \( \rho \) is density. \( U^* \) is mean velocity. To calculate the coefficient of friction in a developed turbulent flow, you can use the estimate [8]: \( \xi = 0.316 / \text{Re}^{0.25} \).

Combining all mentioned above we can get \( y = y^+ \cdot \sqrt{U^*} < 2 \cdot 10^{-4} \text{ m} \). Thus, near the wall, the radial cell size should be less than 0.2 mm. For this reason, an uneven grid was chosen for further calculations. The grid is rectangular and thickens to a size of \( \sim 0.1 \text{ mm} \) near the wall (Figure 3.b). A uniform triangular grid is used in the flow core (Figure 3.c).

As the basic partition for the calculations, the partition of the calculation model into 1,658,000 calculation cells was chosen. To justify that the selected quantity ensures convergence, calculations were performed for different number of cells: 200,000, 300,000, 500,000, 962,000 and the base case of 1,658,000. The calculation results for the velocity profile and kinetic energy of turbulence in section 3 are presented in figures 4.1 and 4.2. To justify the convergence, a k-w-l-model was chosen, since this model, in comparison with other models, has good accuracy at relatively low time costs and therefore allows modeling within a reasonable time with varying design parameters.

As can be seen from Figures 4.1 and 4.2, the calculation results for the selection of 1,658,000 estimated volumes slightly differ from the calculations for 962,000 calculated. At the same time, with 200,000, 300,000 and 500,000, the results are very different. Thus, it can be argued that convergence is achieved with 1,658,000 calculated cells. For this reason, all subsequent calculations were carried out for a given number of calculation cells.

4. Results
Measurements of velocity, or rather its axial Vx components, were carried out in three cross section. For two cross-sections the velocity fields of axial velocity components were measured (Fig. 4 a,b).
Figure 5. a. The axial velocity distribution at flow rate $G = 250 \text{ n.m}^3/\text{h}$ in inlet cross-section before rotary section, $Z_{\text{max}} = 76 \text{ mm}$, $Y_{\text{max}} = 112 \text{ mm}$, $V_{\text{max}} = 9.7 \text{ m/s}$.

Figure 5. b. The axial velocity distribution at flow rate $G = 250 \text{ n.m}^3/\text{h}$ in cross-section after rotary section $Z_{\text{max}} = 186 \text{ mm}$, $Y_{\text{max}} = 250 \text{ mm}$, $V_{\text{max}} = 5.6 \text{ m/s}$.

For cross-sections 1, 2 the $Z$ coordinate was 38 mm, and for the cross-section 3 the $Z$ coordinate was 93 mm. The velocity field after rotary cross-section has shown that intense near wall jet was formed. Also velocity fields in Fig. 5 a,b have shown that in the stream core the velocity profile in $Z$-direction almost did not change. So the velocity profiles in the central plane were enough to compare computational results with the experiment. The results of velocity profiles measurements are shown in Fig. 6. The results of calculations for the longitudinal velocity component in cross sections along the symmetry line are shown in Fig. 7. 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 6. Velocity profiles after the rotary device at different values of mass flow rate.

Figure 7. Comparison of experiment and calculation for flow rate $G = 200 \text{ n.m}^3/\text{h}$.

The width of recirculation zone in the channel after rotary section was chosen as characteristics of applicability area for the turbulence models. The recirculation zone width in cross-section after the rotary device for different values of flow rate is shown in Fig. 8. From experimental data it can be seen that the recirculation zone width takes values $X/X_{\text{max}} = 0.24 \div 0.36$, where $X_{\text{max}}$ is 250 mm. For Reynolds-Stresses model the recirculation zone width takes the values close to the experimental one. For other turbulence models the recirculation zone widths differ from experimental widths. The model of Reynolds stress transfer adequately describes the flow in the range of flow rates from 50 to
250 n.m³/h. The k-kl-w model also was in a good agreement with experiment in the flow rate range from 150 to 250 n.m³/h.

![Figure 8. The recirculation zone width in control cross-section for different flow rate values.](image)

**Conclusion**

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 has been carried out according to the results of experimental studies. Numerical simulation of turbulent gas flow is based on solving a 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 model of Reynolds stress transfer adequately describes the flow in range of flow rates from 50 to 250 n.m³/h.

**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] Versteeg H K, Malalasekera W 1995 *An Introduction to Computational Fluid Dynamics. The Finite Volume Method* (Longman Scientific and Technical) p 257

[2] Spalart P R, Allmaras S R 1994 *La Recherche Aerospatiale* 15–21

[3] Launder B E, Spalding D B 1974 *The Numerical Computation of Turbulent Flows Computer Methods in Applied Mechanics and Engineering* 3 269–89

[4] Wilcox D C 1998 *Turbulence modeling for CFD* (DCW Industries, Inc. La Canada. California)

[5] Walters D K, Cokljat D A 2008 *Journal of Fluids Engineering* 130 121401-1–14

[6] Loytsyansky L G 2003 *Mechanics of fluid and gas* (M: Drofa) p 840

[7] Schlichting G 1974 *Theory of the boundary layer* (M: Science) p 712

[8] Kirillov P L, Bobkov V P, Zhukov A V, Yuriev Yu S 2010 *Handbook of thermal hydraulic calculations in nuclear power* (M: Publishing House) 1 p 776