Numerical study of swirling flow in a steam generating element model with double-sided heating

A V Mamaev, S M Dmitriev, R R Ryazapov, A E Sobornov, A V Kotin
NNSTU n.a. R.E. Alekseev, Minin Street, 24, Nizhny Novgorod, 603155, Russia

e-mail: aleks_may@mail.ru

Abstract. The paper is devoted to numerical modelling of the swirling flow in an annular channel with an inner twisted pipe. The computational model is designed. The technique of swirling flow calculation is tested for CFD packages LOGOS and ANSYS CFX. The velocity and pressure calculated fields are obtained. Experimental and calculated velocity profiles over the channel cross section are presented. The loss coefficient values is obtained. Experimental and calculated comparison of results is made.

1. Introduction
The study of the hydrodynamic processes features is an important stage, both in the heat exchangers design, and at the stage of substantiating the operation parameters and modes of the nuclear power plant. The problem of hydrodynamic flow calculating in channels is most often arisen in engineering practice, it consisted from flow velocity and pressure fields. The internal swirling flows is paid particular attention, which are widely used in modern heat and power equipment due to their specific features. This flow type is characterized by comparable components values of a velocity field. The necessary intensification of the heat and mass transfer processes in the power plants channels is achieved due to swirl the flow [1, 2]. The using of pipe-in-pipe channels with an inner spiral twist pipe is one of the most effective ways to heat transfer intensify. It provides a heat transfer processes intensification of the heating and heated flows in high-stress heat-exchange equipment, in particular in the steam generator unit of low-power nuclear power plants.

It makes possible to significantly reduce the facility weight and dimensions and to increase the technical and economic indicators in general. [3]. The complex structure of this flows, high capital cost of experiments, and the empirical dependencies limitation cause difficulties in choosing the most rational approach for hydrodynamic calculations.

The rapid increase of the multiprocessor computing systems performance makes it possible to use computational fluid dynamics (CFD) methods for complex flows modeling. CFD makes it possible to obtain hydrodynamic characteristics with an acceptable accuracy for practical purposes and to avoid costly experiments. Numerical study methods of aerodynamics and heat and mass transfer processes are based on mathematical models of turbulence structure, which require verification to each separate class of research problems [4, 5].

2. Experimental branch and technique
The results of an experimental study of the hydrodynamic characteristics of a single-phase air flow in an annular channel consist of a "pipe in a pipe" channel with an inner twisted pipe were used as a database for computational model verifying.
The study was carried out at the experimental branch, which is part of the research laboratory in the NNSTU (Nizhny Novgorod state technical university) and it is an aerodynamically open circuit with air pump.

The experimental model (Figure 1) is a reduced physical model of the steam generating element with double-sided heating. The model consists of a casing pipe (1) with an inner diameter of 57 mm, equipped with a coordinate device consist of: a hot-wire anemometer (5) for measuring flow velocity and angle, static pressure sampling (3), (4) and a special protractor. The hot-wire anemometer is located at a distance of 1000 mm from the model inlet. The studies were carried out on a model with an internal twisted displacer (2) with an outer diameter of 40 mm and a twisted step of 200 mm.

![Figure 1. Experimental model](image1.png)

The channel pressure drop and the distribution of the axial and tangential velocity components of a single-phase swirling flow in the range of Re numbers $4 \times 10^3 \div 10^5$ were obtained. Figure 2 shows the channel cross-section diagram with azimuthal angles, at which the flow velocity was measured. The channel radius ($r$) was counted from the inner tube wall.

![Figure 2. The channel cross-section diagram](image2.png)

The aim of the study is to evaluate the CFD methods to simulate a swirling air flow in an annular channel with an inner spiral wound pipe. The following tasks can be distinguished:

- design of a computational model;
- numerical research;
- analysis of the results.

The software ANSYS CFX and LOGOS were used for the computational study. LOGOS is developed at the All-Russian Research Institute of Experimental Physics of the Russian Federal Nuclear Center.

3. Mathematical model

The solution of the Reynolds-averaged Navier-Stokes (RANS) equations is main approach of the swirling flow numerical study. As a result of the averaging operation, the flow characteristics are represented in the sum of mean and pulsation components, and the 6 new unknown components in the
equation of motion, forming the Reynolds turbulent stress tensor, are appeared. The resulting system of equations is inconsistent, which requires the additional equations for solution. One of the methods is using the semi-empirical models based on the turbulent viscosity concept [6].

The Spalart-Allmases (SA) model has positively proven for the analysis of many aerodynamic flows. The SA model, like all models of turbulent viscosity, assumes the isotropic structure of turbulence. It is requiring some special corrections for flow swirling calculation. A modification of the SA model with a correction for streamline curvature and rotation, called SARC. It has a wider range of using to the problem of analyzing swirling flows. This model was implemented in the LOGOS code and was used for calculation in this work.

The BSL RSM (Reynolds Stress Model BaseLine) model was chosen to close the Reynolds equations in ANSYS CFX. It uses nonlinear equations for the stress tensor components. It allows to examine the complex flow structure, which is not explored by turbulent viscosity models. This model assumes solution of equations corresponding to the Menter k-ω BSL model and additional differential transport equations for Reynolds stresses [7]. The BSL RSM model requires solving a larger number of equations in comparison with the SARC model. It negatively affects the calculation convergence.

4. Computational study

The three-dimensional channel geometry was prepared for further building of computational grids. The computational grid building on the original geometry was complicated by a large number of poor-quality elements in the area of contact between the displacer and the casing tube. Contact of two surfaces was simplified to achieve the normal quality of the mesh model. An area along the line of surfaces tangency was cut off.

It is preferable to use hexahedral block grids for swirling flows simulation, because streamlines and grid lines are oriented in the same way. The using of unstructured tetrahedral grids is causes the solution accuracy loss. It is need to additional shredding the computational domain to achieve acceptable results, which leads to convergence time of the solution increasing. The selected turbulence models do not use the wall function, which required a mesh thickening at the outer and inner surfaces of the channel. The first element thickness is corresponding to the recommendations of CFX manual: $Y^+ \sim 1$.

A mesh model consists of 7.2 million elements and 6.9 million calculated nodes with a thickening factor equal to 1.2 was generated. The hexahedral mesh is shown in Figure 3. So significant element number is dictated by mesh local shredding due to the decreasing width of the channel towards the contact line. The mesh was generated on the geometry of one twisted step, next, it is copying and gluing to ensure the required length of the computational model.

![Figure 3. The hexahedral mesh](image)

The calculations were made using the ideal gas model (Air Ideal Gas). The physical properties of the flow is corresponding to the experiment. The boundary conditions were set on from the operation parameters of the experimental study. The normal component of the velocity was set at the inlet of computational domain. The relative pressure is equal to 0 Pa was set at the exit of computational domain.
5. Results

The velocities distribution in the channel cross-section is a determining factor of heat transfer and pressure drop in steam-generating channels with a twisted pipe. The distributions of the relative axial and tangential velocity were plotted with experimental result comparison (Figures 4-5).

**Figure 4.** Profiles of the axial component for the average velocity $\bar{v}=17.4$, m/s

**Figure 5.** Profiles of the tangential component for the average velocity $\bar{v}=17.4$, m/s

The best congruence between the calculated and experimental velocity profiles was obtained for the axial component, both using the SARC model and the BSL RSM model. It is important to note both models have a similar velocity distribution. Qualitative differences of calculated profiles are noted at
angles $135^\circ$ and $225^\circ$, but values from the BSL RSM model are closer to the experimental data. Figures 6-8 show the calculated and experimental distributions of relative velocity axial and tangential components, where $R$ is the relative radius.

**Figure 6.** Profiles for the azimuth angle $180^\circ$ and average velocity $\bar{v} = 11.8$, m/s a) axial velocity component $v_z$ and b) tangential velocity component $v_r$; 1- experiment; 2-BSL RSM; 3- SARC

**Figure 7.** Profiles for the azimuth angle $180^\circ$ and average velocity $\bar{v} = 17.4$, m/s a) axial velocity component $v_z$ and b) tangential velocity component $v_r$; 1- experiment; 2-BSL RSM; 3- SARC

**Figure 8.** Profiles for the azimuth angle $180^\circ$ and average velocity $\bar{v} = 22.3$, m/s a) axial velocity component $v_z$ and b) tangential velocity component $v_r$; 1- experiment; 2-BSL RSM; 3- SARC
The calculated distributions from both turbulence models are in qualitative agreement with the experimental data. At the near-wall region the SARC model gives some different values, in comparison with BSL RSM, which is most clearly viewed with an increase in the average velocity.

Much attention is paid to the value of the loss coefficient (ξ) of the pipe-in-pipe channels with inner twisted displacer. The static pressure distribution along the length of the model were measured to determine ξ. Figure 9 shows the calculated and experimental dependences ξ on the Re number.

![Figure 9. Dependences of the loss coefficient ξ on the Re number: 1- experiment; 2-BSL RSM; 3- SARC](image)

Different values of the coefficient β* (BetaStar) to achieve the required calculation accuracy on the BSL RMS model was calculated. It is included in the pressure-strain rate correlation in the equation for the Reynolds turbulent stress tensor and is responsible for the energy redistribution between the tensor components. The selection of β* from the range: 0.05-0.09 was made for the operation mode with highest average flow rate. The dependence of the ξ coefficient on β* is shown in Figure 10. The calculated values of ξ obtained on the BSL RMS model differ from the experimental ones by no more than 3%. The SARC model also gave acceptable results: the deviation from the experiment did not exceed 8%.

6. Conclusion

The flow in a channel with an inner twisted tube is studied with the LOGOS and ANSYS CFX software packages. The computational model made on the base of the generated tetrahedral mesh. The measured velocity fields are profiles of the axial and tangential components over the channel section in the direction of different azimuthal angles. Velocity distributions obtained from the RSM BSL and SARC turbulence models are similar. A qualitative difference is observed at angles 130° and 225°. Both models showed the good match for the axial velocity component. Dependencies for the loss coefficient were determined. The β* parameter of the BSL RMS model was determined to be 0.07 to achieve the calculation accuracy. The deviation of loss coefficient calculated values from the experiment values was for the BSL RMS model - no more than 3%, for the SARC model - no more than 8%.

The good congruence of the numerical results with the experimental data allows us to conclude that, computational model is applicable for describing the flow in a channel with an internal twisted tube. The BSL RMS model match better, but the SARC model also showed satisfactory results. In addition, the SARC model being less demanding on computational resources, it is much more profitable for performing calculations on fine grids.

Acknowledgments

Study is made with the financial support from the Ministry of Education and Science of the Russian Federation within the national project "Science and Universities" initiated for the youth research labs creation (scientific topic: "Hydrodynamic and heat and mass transfer processes in nuclear power plants equipment").
References

[1] Shchukin V K and Khalatov A A 1982 *Heat transfer, mass transfer and hydrodynamics of swirling flows in axisymmetric channels* (Moscow: Mechanical Engineering) p 200

[2] Mitrofanova O V 2010 *Hydrodynamics and heat transfer of swirling flows in channels of nuclear power plants* (Moscow: FIZMATLIT) p 88

[3] Budov V M and Dmitriev SM 1989 *Forced heat exchangers of NPP* (Moscow: Energoatomizdat) p 176

[4] Kharlamov S N 2008 *Algorithms for modeling hydrodynamic processes* (Tomsk: TPU) p 80

[5] Kozelkov A S 2014 *Mathematical models and algorithms for simulation of hydrodynamics and aerodynamics problems* (Nizhny Novgorod: NSTU) p 164

[6] Snegirev A Yu 2009 *High performance computing in technical physics. Numerical Modeling of Turbulent Flows* (St. Petersburg: Publishing House of Polytechnic University) p 143

[7] ANSYS CFX Solver Modeling Guide 2013 (ANSYS Inc. - ANSYS CFX Release 15)