Modeling of the conjugation of vortex flows with downstream in ANSYS

L E Schesnyak
Peoples' Friendship University of Russia (RUDN University), Moscow, Russia
Email: shchesnyak-le@rudn.ru

Abstract. Using the new technology of longitudinal circulation currents which allows dissipating the excess kinetic energy, thus preventing riverbed erosion and increasing the reliability and environmental safety of the structure. In modern practice of the design of water discharge structures of high-pressure dams, those special properties of flows are widely used. This allows to solve a number of important engineering problems arising in the practice of operating the structures by using the new innovative approaches to. In the paper, the author offers an interesting solution to the engineering problem – A new mathematical modelling has been created. For this study, the standard software package Fluent in ANSYS R 19.0 is used and the required outcome is compared with experimental results. Consequently, the necessary parameters of the swirling flow which are required when evaluating the conditions for the coupling of the water discharged with the downstream of the dam structure are depend on the attained results.

Keywords: vortex flow, downstream, water discharge, spillway, drifting flow, modelling, ANSYS.

1. Introduction
The conjugation of vortex flows with downstream is the component where the flow existing the runner is decelerated, thereby converting to the excess of kinetic energy in to the static pressure, as a consequence increase the global power of the turbine. In practice of hydraulic engineering, recently, the new technologies for dissipating the excess kinetic energy of the stream are introduced, particularly in spillway structures of hydroelectric power stations and rotary-conjugating structures on canals. The principle of these technologies lies in the use of special properties of longitudinal circulation currents that allow solving a number of important engineering problems arising in the actual operation of structures, namely:

- Dissipating the excess kinetic energy of the flow within the culvert;
- Prevention of dangerous riverbed erosion and the collapse of coastal slopes in the downstream;
- Protection of streamlined surfaces from cavitation damages allowing to increase the reliability and environmental safety of hydraulic engineering objects.

As an example of using these technologies, where the features of longitudinal circulation currents are implemented, can be mentioned the vortex shaft spillway of the Tehri Dam in India [1]. In recent years, extensive scientific research has been carried out on longitudinal circulation currents and methods of their hydraulic calculation [2-8]. However, high estimation of the work of the vortex spillways of the Tehri Dam is given by different authors, who are engaged in the design of spillway structures. It provides a basis for the continuation and more detailed study of longitudinal circulating currents, especially in the propagation of swirling turbulent jets which are discharged from the spillways into the downstream. Those scientific and engineering problem requires new innovative approaches for its solution. To solve these problems, it is reasonable to use inexpensive and affordable models describing the conjugation of vortex flow with downstream. The same physical phenomenon admits a variety of different models, which are divided into physical and mathematical [9-11]. Physical models are extremely reliable and
convenient for solving engineering problems, but they are very expensive, particularly when modeling the fundamentally non-linear once its scale cannot be changed. The creation of physical models of real hydroelectric power stations in the laboratory presents difficulties.

On the contrary, mathematical models offer an extremely economical solution to engineering tasks but always should be remembered that every mathematical model takes into account only some phenomena with approximate accuracy. According to Samarsky et al., mathematical modeling includes three stages [10]:

- Designing on the basis of general laws of nature, and one or another empirical material of a correct mathematical task, called the mathematical model of the phenomenon;
- Creation of an algorithm for solving this task
- Verification of the model by comparison with full-scale experiment and with other simpler models.

In hydrodynamic tasks the boundary-value problems for partial differential equations are used as a model. Such problems are rarely solved analytically, therefore, they usually apply numerical and grid methods for solving differential equations. Theoretically, the flow of water during discharge is most accurately described by the Navier-Stokes equation [12–17].

The application of grid method implies that the scale order of the stack step of the solution can be well approximated by linear functions but in turbulent problems this assumption is not fulfilled. However the Navier-Stokes equation appears to describe well acceptable structure of the flows therefore, it is poorly computable by the finite difference method. So, models based on the direct solution of the Navier-Stokes equations are not used in engineering calculations. From the mathematical point of view, the Navier-Stokes equations are extremely complicated in addition at this time the existence and smoothness of the solution for Navier-Stokes equations in R3 have not been proved [18]. At first, the Reynolds decomposition between mean and fluctuating (e.g., turbulent) components is introduced and applied to the Navier-Stokes equations, resulting in the so-called Reynolds-Averaged Navier-Stokes equations (RANS, in the Favre formulation valid for compressible flows) [20–24]. An unclosed system of differential equations is obtained. Because of the nonlinearity behavior of the equations. Its closure system is performed in various ways, which leads to different models of turbulent flow. Known closure methods are based on certain assumptions where the substantiation of which presents significant difficulties and contains parameters that are determined empirically. At the same time, these parameters were selected for several decades on flows whose parameters varied in very large ranges. Therefore, it can be expected that these models will create an adequate model of conjugation of vortex flow with downstream for the hydroelectric power stations on the right scale.

In the present paper, the $k–\varepsilon$ model is used to create a complete mathematical model of a hydroelectric power station with a vortex discharge, and this is verified by comparing with a reduced physical model of the hydroelectric power station. For calculations, the standard tool package Fluent in ANSYS R 19.0 is used for working with empirical models of turbulent flows. This system provides an access to many packages which focused on modeling of phenomena from various areas of physics but combined with the general idea – discretization of partial differential equations by the method of finite elements analysis (FEA), and by the modern classification refers to FEA software. The choice of this system is primarily due to the fact that the ANSYS Inc. was established in 1970 specifically for the development of FEA software and now has accumulated a lot of experience in this matter. Currently, Fluent ANSYS is the most famous and universal software for modeling the turbulent flows, new software packages are always tested to coincide with the results of Fluent [24]. It also described in detailed documentation and furthermore included with ANSYS, incl. the User Guide and the training literature.

It should be noted that we have at our disposal a mathematical model of the vortex flow, which does not take viscosity into account and cannot describe the conjugation with stream, but it is solved analytically. In this study, a comparison will be made with this simple mathematical model, specifically, as suggested by Samarsky et al. [9], a hierarchy of vortex flow models will be designed. In Summary, the aim of the present study can be formulated through creation and testing of a model for the
conjugation of vortex flow with downstream along with its verification by carrying out a complete experiment on a small scale.

The objectives of this study is to develop a mathematical model of conjugation of vortex spillways with the downstream by ANSYS modelling and proving the outcome with the experimental results.

2. Methods
2.1. Mathematical Model
The vortex flow is described as a pipe of constant circular cross-section and in the physical model its diameter is 50 mm. An impeller was placed at the entrance of the flow to create swirl (Figure 1). The downstream for simplicity is described as a rectangular parallelepiped and its dimensions are taken smaller than the physical model in order to save the calculating resources.

![Figure 1. Discharge with the 4-channel impeller included in the downstream.](image)

The modeling is based on the continuity equation and the averaged Navier-Stokes equation (Reynolds-averaged Navier-Stokes equation, RANS)

\[
\frac{\partial U_i}{\partial x_j} = 0
\]

\[
\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\delta}{\partial x_j} (2\mu S_{ij} + \rho \tau_{ij})
\]

where \( U \) and \( P \) – averaged velocity and pressure, respectively, and

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)
\]

For the density and viscosity of water, the standard values are taken. In order to close this system, it is also necessary to determine the components of the Reynolds stress tensor \( T_{ij} \). We will use the standard model proposed by Launder and Sharma in the early 1970s [9]. This is the most popular model for describing turbulent flows [25]. In the Fluent module, other popular models of turbulent flow are implemented, computational experiments have shown that the \( k-\varepsilon \) and \( k-\omega \) models give approximately the same results, so we are considering the \( k \) and \( \varepsilon \) model [11, 25]. To describe this model, it is necessary to introduce the kinetic energy of turbulent pulsations

\[
K = \frac{1}{2} u_i^* u_i^*
\]

And the rate of dissipation

\[
\varepsilon = \nu \frac{\delta u_i^*}{\delta x_k} \frac{\delta u_j^*}{\delta x_k}
\]

Within the framework of the model, it is assumed that the Reynolds stress tensor is expressed through the functions as follows:

\[
\tau_{ij} = 2\nu T S_{ij} - \frac{2}{3} k \delta_{ij}
\]
where the kinematic viscosity $\nu_T$ is determined from considerations of dimensionality as $\nu_T = C_{\mu} k^2/\varepsilon v_T$.

The coefficient $C_\mu$ is assumed to be constant, and was determined by the empirical data as $C_\mu = 0.009$. The dynamics of these values, characterizing turbulence, are described by two equations which are similar to the Navier-Stokes equation. The dynamics of the energy $k$ is described by the equation below:

$$\frac{\delta k}{\delta t} + U_j \frac{\delta k}{\delta x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\delta}{\delta x_j} \left[ \left( v + \nu_T \right) \frac{\delta k}{\delta x_j} \right]$$

$$\frac{\delta \varepsilon}{\delta t} + U_j \frac{\delta \varepsilon}{\delta x_j} = C_{\varepsilon 1} \varepsilon \frac{\tau_{ij} \frac{\partial U_i}{\partial x_j}}{k} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} - \sigma_\varepsilon + \frac{\delta}{\delta x_j} \left[ \left( v + \nu_T \right) \frac{\delta \varepsilon}{\delta x_j} \right]$$

where $\delta_k = 1.0$. The dynamics of the rate of dissipation is described by equation and the coefficients are taken empirically as given below:

$$C_{\varepsilon 1} = 1.44, C_{\varepsilon 2} = 1.92, \sigma_\varepsilon = 1.3$$

Adding these two equations to RANS gives 6 equations with 6 unknowns, four of which $P, U_1, U_2$ and $U_3$ are characterized as the flow on average, and the remaining two unknowns considered with the turbulence of the flow. We consider only the steady flow, i.e. substitute $\delta t = 0$. We can add boundary conditions of the impermeability of the walls of discharge and stream for these six equations then set the rate at the discharge entrance. In all calculated experiments, a velocity of 10 cm/sec was taken at the inlet of the discharge, which ensured a noticeable swirling of the flow in the discharge. The conditions at the ends of the stream were selected in such a way that to provide a drifting flow of 4 cm/sec.

Within the framework of the described geometry and model, the ANSYS R 19.0 system was used for calculations. For the numerical solution of the system of partial differential equations with the resulting system of partial differential equations, the discretization by the finite element method was performed. A more detailed grid was used for the discharge than for the stream. The system of algebraic equations obtained after such a discretization is solved by means of an iterative process. The convergence is estimated from the difference of speeds, $k$ and $\varepsilon$, found at neighboring steps. In all experiments, for 100 iterations, we were able to achieve accuracies for velocities about $10^{-3}$ m/sec (Figure 2). This is more than enough for engineering calculations.

![Figure 2](image.png)

*Figure 2. Deductions for estimating the convergence of the iterative method, the velocity values are within $10^{-3}$ m/sec.*

In this Method, conjugation devices were used for the water outlet below the downstream level. Different angles of the drifting flow (0 - 90°) were used for the study, which are 30°, 45° and 90°. As a
downstream, a hydraulic chute was used, which allows to create a drifting flow and also a dimensional grid was applied in the place of the outlet of the swirling flow.

The control of velocity in the downstream (behind the swirling device) was carried out using FlowTrecker2 (FT2). FT2 uses a reliable Acoustic Doppler Velocimetry (ADV) technology, tested by experts from around the world for decades in hydraulic laboratories and in a wide range of operating conditions. The modified and improved acoustic ADV sensor for the FT2 provides unprecedented accuracy, especially with minimal water flow and in shallow water. 2D data in the horizontal plane (2D and 3D options available) allows to perform the most comprehensive quality control and get an idea of the flow parameters.

Determination of the actual parameters of the swirling jet in a fixed space filled with the same fluid, as well as in a drift flow for different modes with different residual swirl including the initial section (L ≤ 10d). Choose the modes with the most suitable for economic efficiency and safety of operation of hydraulic structures. Confirm the relationship between model data and actual values.

3. Results and Discussion

A series of computational experiments was performed in which the swirling flow entered the hydraulic chute at various angles, and the obtained results were compared with real observations. The following three cases were studied:

1. The vortex flow is perpendicular to the flow in the downstream (Figure 3);
2. The vortex flow enters the downstream at an angle of 30° (Figure 4);
3. the vortex flow enters the downstream at an angle of 60° (Figure 5).

![Figure 3](image1.png)

**Figure 3.** Velocity module for the 1st case, horizontal section (a) and vertical section (b); vertical section was made along the plane, marked on the horizontal section by the black line.

![Figure 4](image2.png)

**Figure 4.** Velocity module for the 2nd case, horizontal section (a) and vertical section (b); vertical section was made along the plane, marked on the horizontal section by the black line.
A velocity of 10 cm/sec on the course to the discharge and 4-channel impeller provides a noticeable vortex flow in the discharge. As predicted by the analytical model of Zhivotovsky [5], the maximum velocities were reached near the walls, and a barely noticeable reverse flow appeared on the axis. Since our model, in contrast to the analytical model, takes viscosity into account, the vortex flow is straightened after a certain distance from the swirler. In the stream near the entrance of the vortex flow, a pocket emerges at very high velocities. These velocities are directed against the drifting flow, but not in the direction of the discharge axis, i.e. the velocity is extinguished in the direction of flow. The smaller the angle between the discharge and the drifting flow, the more noticeable this phenomenon.

4. Conclusion
On the basis of the experimental study, the following results were achieved:
1. A mathematical model of conjugation of vortex spillways with the downstream has been created;
2. A model was implemented in the ANSYS Fluid system and a series of numerical experiments was performed;
3. A physical model of conjugation of vortex spillways with the downstream has been created;
4. A series of experiments was performed using modern measuring equipment FlowTrecker2;
5. A mathematical model is verified by comparison with the real experiment.
The obtained results allow to determine the necessary parameters of the swirling flow required in evaluating the conjugation conditions of the spillway structures with the downstream.

References
[1] Bhosekar V V C, Deolalikar P B and Sridevi M I 2012 Hydraulic design aspects for swirling flow at vertical drop shaft spillways ISH Journal of Hydraulic Engineering 13(3) pp 78–89
[2] Zhivotovsky B A 1995 Spillway and interfacing structures with swirling flow (Moscow: RUDN University Press)
[3] Galant M A, Zhivotovsky B A, Novikova I S, Rodionov V B and Rozanova N N 1995 Features of vortex tunnel spillways and hydraulic conditions of their operation Hydrotechnical Construction 9 pp 16–22
[4] Volshanik V V, Zuykov A L and Mordasov A P 1990 Swirling flows in hydraulic structures (Moscow: Publishing House Energoatomizdat) p 280
[5] Volshanik V V, Orekhov G V, Zuykov A L and Karelin V Y 2000 Engineering hydraulics of swirling fluid flows Hydrotechnical Construction 11 pp 23–26
[6] Mitrofanova O V, Podzorov G D and Pozdeeva I G 2000 Vortex structure of swirl flows International Journal of Heat and Mass Transfer 65 pp 225–234
[7] Gesheva E S, Shtork S I and Alekseenko S V 2017 Investigation a single-spiral vortex in a swirl flow MATEC Web of Conferences 115 02025
[8] Akhmetov D G and Akhmetov T D 2015 Swirl flow in vortex chamber Physical and mathematical Sciences 4(6) pp 109-120
[9] Samarsky A A and Mikhailov A P 2001 Primery Mathematical modeling: Ideas, Methods (Moscow: Publishing House Fizmatlit) p 320
[10] Samarskii A A and Mikhailov A P 2018 Principles of Mathematical Modeling: Ideas, Methods, Examples (Frorida: CRC Press) p 360
[11] Kartashev A L and Krivonogov A A 2016 Mathematical Modelling of Vortex Generation Process in the Flowing Part of the Vortex Flowmeter and Selection of an Optimal Turbulence Model Bull. South Ural State Univ. Ser. Mathematical Model and Program 9(4) pp 105–116
[12] Sangwha Y 2019 Exact Solution of Navier-Stokes Equations International Journal of Advanced Research in Physical Science 6 pp 39–43
[13] Vellando P, Puertas J, Bonillo J and Fe J 2000 Finite element solution of the Navier-Stokes equations using a SUPG formulation Advances in Computational Engineering and Sciences pp 856–861
[14] Alam M and Saha S 2017 Normal stress differences and beyond-Navier-Stokes hydrodynamics EPJ Web of Conf 140 11014
[15] Ramm A G 2019 Solution of the Navier–Stokes problem Appl. Math. Lett. 8 pp 160–164
[16] Ramm A G 2019 On the Navier-Stokes problem J. Adv. Math. 16 pp 8262–8266
[17] Willis A P 2017 The Open pipe flow Navier–Stokes solver Software X 6 pp 124–127
[18] Launder B 2003 Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc Int. Commun. Heat Mass Transf. 1(2) pp 131–137
[19] Sun S, Jenkins E W, Zhangxing C and Geiser J 2011 Mathematical and numerical modeling of flow and transport Journal of Applied Mathematics Article ID 901380.
[20] Parra T, Castro F, Rodriguez M A, Szasz R Z, Robert Z, Gutkowski A and Perez R 2015 Numerical Simulation of Swirling Flows - Heat Transfer Enhancement Journal of Fluid Flow, Heat and Mass Transfer 2 DOI:10.11159/jffhmt.2015.001
[21] Westerweel J 2009 Advanced Experimental Methods For Turbulent Shear Flows Proc. 6th Int. Symp. on Turbulence and Shear Flow Phenomena pp 779–784
[22] Teresa P, Perez J R, Szasz R Z, Rodriguez M A and Castro F 2015 Numerical Modelling of flow pattern for high swirling flow EPJ Web of Conferences 92 02059
[23] Belov I A and Isaev S A Modeling of turbulent flows (St. Petersburg: Publishing House)
[24] ANSYS Fluent Theory Guide. Release 15 2013 (Canonsburg: ANSYS Inc.)
[25] Georgiev D J, Roberts A J and Strunin D V 2007 Nonlinear Dynamics on Centre Manifolds Describing Turbulent Floods: K-W Model Discrete and Continuous Dynamical Systems 2007 pp 1-10