Investigation of an axial hydrokinetic turbine with the CFD program Flow Simulation

K Tuzharov\textsuperscript{1}, S Iliev\textsuperscript{1,*}, V Vilag\textsuperscript{2} and J Vilag\textsuperscript{2}

\textsuperscript{1}University of Ruse, Faculty of Industrial Agriculture, Faculty of Transport, 7017 Ruse, Bulgaria,
\textsuperscript{2}COMOTI Romanian Research and Development Institute for Gas Turbines, Department "Aviation and industrial turbines. Gas turbine assembly", D Iuliu Maniu Bd., sector 6, cod 061126, OP 76, CP174

\*E-mail:spi@uni-ruse.bg

Abstract. In the work, the characteristics of a hydrokinetic axial turbine are obtained by numerical CFD modelling. Using the new research tool, Fluid Computational Dynamics, arises the question as to the accuracy of the solution. That’s why for solving the problem, has been used new numerical method “Sliding mesh”, which is included in the “Flow Simulation” soon. Besides, the numerical results for frequency characteristics are compared with the characteristics obtained by modified analytical model. It was found that for both solutions - for the output powers of the turbine, the differ is (5...15)\% and for the torques of the working shaft the differ is (4...18)\%. Because it’s accepted that the analytical solution is more accurate as it is done by a modified classical method, the compliance of the shape and energy performance characteristics is considered to be a numerical model verification obtained by simulation with the Flow Simulation CFD.

1. Introduction

The Interest of hydrokinetic turbines began in 1973 due to the worsening energy and ecological crises on the planet. Recently, the new research tool, Fluid Computational Mechanics (CFD), has been used for their research. In this approach, the flow area is calculated by means of different forms of the Navier-Stokes' private differential equations, including closing turbulent equations known as turbulent models, with which the differentiate system is reduced into a solvable state. The fluid area (computing area) is divided into multiple volumes (cells) for each of which the flow parameters are calculated. This method does not make too many simplifying prerequisites. The question arises as to the accuracy of the solution as compared to the analytical methods of research.

2. Exposition

The object of the study is a model of the hydrokinetic turbine (figure 1), consisting of a grid 1, a rotor 2 which acts as a casing of the electric generator, the flow channel in the housing 3, the impeller 4 consisting of four blades, a hub, the diffuser 5 and the carrier 6 for connecting the hydro unit to the turbine housing. The impeller is with an outside diameter \(d_2=0.935\) m , internal diameter \(d_1=0.180\) m, the aerodynamic profile of the blades is Espero with relative thickness\(\delta=0.2\), with chords at external and internal diameters, respectively \(c_2=0.160\) m, and \(c_1=0.135\) m. The flow of the working fluid...
through the hydrokinetic turbine actuates the impeller 4, achieving the equality between torque created by the turbine and the resistor torque.

**Figure 1.** Hydrokinetics turbine. 1-protecting grid; 2- electric generator housing; 3-turbine housing; 4-wheel drive (blades, hub); 5-diffuser; 6-struts for connecting the hydro unit to the turbine housing.

The purpose of the report is to obtain the frequency characteristics of the hydrokinetic turbine which are dependent on the rotational force and shaft power from the impeller speed at constant velocity $v=2\text{m/s}$ of the flow. To accomplish the goal, the following tasks were performed: a physical model of the Solid Works turbine and a numerical model (figure 2a) was created with the CFD Flow Simulation program using the Wizard option (built-in project management program). The type of analysis is internal, three-dimensional, with the computing area before and after the turbine being extended (not shown in figure 2a) in order to achieve convergence and higher accuracy of the solution.

Prerequisites for the study are that the working fluid is pure water, the flow is turbulent with a laminar sublayer on the walls, the walls are thermoinsulation (the flow is adiabatic). Taking into account that there are sealing cavities in the geometric model to be excluded from the pattern, the **Exclude cavities without flow conditions** options are activated.

The general project settings are shown in table 1. The table shows that the default conditions are left by default by activating the Pressure potential option. In Flow Simulation, turbulent fluid flows are modeled with the Navier-Stokes equations [1, 2] which describe the laws of storage of matter, impulse and energy. Flow parameters are averaged over time, according to Reynolds. As a result, the equations
have additional members - Reynolds tensions. To close the system of equations in Flow Simulation, the equations for the transfer of turbulent kinetic energy and its dissipation within the turbulent model $\varepsilon - k$ [1, 2] are used.

**Table 1. The general project settings.**

| Physical features | Rotation |
|-------------------|----------|
| Wall Conditions   | Default smooth walls |
| Initial and Ambient Conditions | Thermodynamic parameters: Temperature=293 K; Pressure: 101325 Pa; Turbulence parameters: turbulence intensity 5%, turbulence length 0.01 m; turbulence energy $k=1$J/kg and dissipation $\varepsilon = 1$ kW/kg. |

| Result and Geometry | Level of Result resolution equal to 5; Minimum gap size = 0.01 m, other options according to default. |

An auxiliary body was created to simulate the rotation, which covered the turbine with a reserve. The body diameter is 1050 mm, the width is 250 mm. These dimensions are determined by the condition that they exceed the outside diameter and the width of the impeller. For the rotating area, the current values are set to the stepped angular velocity $\omega = 0.20 \text{ s}^{-1}$ with step $\Delta \omega = 5\text{s}^{-1}$ and the Disable attribute is assigned, which turns the body into a transparent for the flowing environment.

The boundary conditions are set by closing the edges of the extended computing area with lids. The velocity of the flow ($v=2$ m/s) and the outlet pressure ($p=101325$ Pa) and ambient temperature ($T=273$K) are set on the surface of the inlet. The roughness of the walls is neglected.

To set up the home network, the Automatic settings mode is turned off, and the number of cells of the base network by coordinates, the criterion for partitioning the dividing network with the liquid medium, narrowing the narrow channels to 3 cells is determined, provided the system can a refining level is achieved 6.

![Figure 3](image1.png)

**Figure 3. Computing meshes about planes x-y and z-y.**

To optimize the network around the impeller, a functionalized *Local Initial Mesh* is used. A body has been created in advance that covers the area. The *Tolerance Refinement Criterion* in the Solid / Fluid Interface option has been increased to 6 by setting the crushing level of the cells. Split to level 2 all the cells that fall in the Refining cells option and increase to level 4 the desired number of cells across the narrow channels in the Narrow channels option.
Fragments of the resultant mesh in isometry are shown in figure 2b, and in transverse x-y and longitudinal z-y sections in figure 3. It yields 142,745 liquid and 41,587 partial cells. As far as the result is concerned, the mesh is sufficiently rational - along the blades there are 30..40 cells and in the width of 12..20 cells, the mesh thickens at the blades and the hub walls, as well as in the gap between the rotor and the flow tube.

A way to compare the theoretical currents obtained by different methods is by comparing the results for control points, control lines or control loops. Figure 4a shows two control loops (a circle with a radius of 350 mm and a rectangle with sides 700x1350) and control lines in the direction of the blades with a length of 350 mm.

For the purposes of the project, the determination of the pressure and flow distribution in the flow, the average values of pressure and velocity in the inlet and outlet sections, the reactive torque in relation to the z axis, which the flow creates on the rotor, the inlet power of the flow, the output mechanical power over turbine shaft, torque and power ratios, and turbine speed.

The pressure and flow velocities in the flow are obtained as a result of the solution of the Navier-Stokes private differential equations. The torque $M_z$ and the force $F_z$ values for different rotation angles at a certain pressure distribution are modeled using the built-in universal functions:

$$M_z = \int_{r_1}^{r_2} dM_z(r)$$
$$F_z = \int_{r_1}^{r_2} dF_z(r)$$  \hspace{1cm} (1)

where $r_1, r_2$ are the inner and outer radii of the impeller; $r$ - the current radius along the blades of the impeller; $dM_z$ and $dF_z$ - the current values of the elementary torque generated by the pressure distribution under and above the blades and the elementary axle force generated by the difference between the pressures before and after the impeller.

![Figure 4](image.png)

**Figure 4.** Control loops around the rotor and blade, and velocity and pressure distribution along a circle with a radius of 350 mm.
The average torque values and the front force are calculated on the basis of the data for the instantaneous values with the dependencies: 

\[ M = \frac{\sum N M_z}{N} \quad \text{and} \quad F = \frac{\sum N F_x}{N}, \]

where \( N \) is the number of steps for one complete rotation of the rotor.

The powers and the dimensionless turbine coefficients are calculated by inserting dependencies for the input power with the Insert Equation Goals option:

\[ P_1 = \frac{\pi}{8} \rho D^2 v^3 \quad (2) \]

- The output power

\[ P_2 = M \omega \quad (3) \]

- The speed of rotation

\[ n = \frac{30}{\pi} \omega \quad (4) \]

- The speed ratio

\[ \lambda = \frac{\pi \, R}{30 \, v} \quad n \quad (5) \]

- The torque ratio

\[ C_M = \frac{8M}{\rho \pi D^2 v^3} \quad (6) \]

- The power factor

\[ C_p = C_M \lambda \quad (7) \]

3. Results and analysis

As a whole, in the course of the solution, the task exhibits stability and rapid convergence especially for the power indicators - torque and axle force, as well as for all project objectives – the power in the input and output of the rotor, the torque and power coefficients. Figure 5 shows that the task reaches at the 150\textsuperscript{th} iteration.

![Figure 5](image-url)
As stated in the introduction with the Reynolds equation, the distribution of velocities and pressure through the turbine flow channel are determined.

The results for the velocity field in the longitudinal and cross section of the flow channel at rotor speeds $n=76.5 \text{ min}^{-1}$ are shown in figure 6, and in figure 7 for the static pressure field in longitudinal section. It can be seen that the flow before the rotor is steady, uniform with an average constant velocity 2 m/s (figure 6a). The carriers 6 (figure 1) negligibly disturb the flow, but the impeller exchanges energy with the water flow, creating a strong redistribution of speed. Its highest values are on the front edge and back of the blades, and the lowest on the low part of the blades caused by the circulating velocity around them. Because of the contraction, the velocity through the rotor and beyond it has higher values 2.7 m/s - on $n= 76.5 \text{ min}^{-1}$ and 3.3 m/s at $n=214 \text{ min}^{-1}$. Separation of the flow after the impeller creates a highly swirling zone with low speeds 0 ..0.5 m/s which, as the rotational speed of the rotor increases, moves away from the plane in which it is located.

The velocity field forms the pressure field, which has three clearly defined zones - one before the rotor, and two after the rotor. With average pressure values, respectively, $p_1=104600 \text{ Pa}$, $p_2=97500 \text{ Pa}$, and $p_3=93600 \text{ Pa}$.

The pressure field forms the force interaction of the flow with the rotor, respectively the power indicators - the torque and the force from the front pressure.
Another characteristic phenomenon that occurs after the rotor is the rotation of the flow. It complicates the flow and is illustrated with the current lines shown in Figure 8. In front of the rotor they are parallel and then rotate in the direction of rotation of the blades with an angular velocity depending on the rotor angular velocity. Current lines are combined with the isotachs (lines at constant speed). The absolute speed after the impeller increases to 3.5 m/s due to the contraction of the flow path and the flow rotation.

The distribution of velocity and pressure on closed control loops (figure 4b, c) around the impeller, for example, on a 700 mm diameter circle, as mentioned above, can serve as a benchmark for different test methods.

**Figure 8.** Trajectories and isotachs at $n=214 \text{ min}^{-1}$.

**Figure 9.** Frequency characteristics obtained by CFD simulation.

**Figure 10.** Aerodynamic characteristics $Cm = f(\lambda)$ and $Cp = f(\lambda)$ obtained by CFD simulation.
The results of the calculations in the form of torque $M$ frequency characteristics and shaft power $P_2$ are shown in figure 9. The value of the starting torque at zero rotor speed is 100 Nm, at speed $n=50\,\text{min}^{-1}$ decreases to value of 75 Nm at the nominal speed $n=125\,\text{min}^{-1}$, rises to a nominal value $M_{\text{nom}}=110\,\text{Nm}$, then decreases smoothly and at zero synchronous rotation speed $n=190\,\text{min}^{-1}$ reaches zero. The power curve is parabola with the maximum at the rated rotation speed, which value is $P_{\text{max}}= 1400\,\text{W}$.

After undimensioning of the frequency characteristics the aerodynamic characteristics, which are dependencies between the torque coefficients $C_m$ and power coefficient from the speed ratio $\lambda$, are obtained (figure 10). In figure 11 are given the frequency characteristics of the shaft power and torque.
of the hydrokinetic turbine, obtained by the analytical method at different water velocities [4]. The comparison of the results is based on characteristics at water velocity \( v=2 \text{ m/s} \) (lines with triangular marks). The match between the characteristics is very good at rotations of 100...150 min\(^{-1}\), that is, in the field of nominal operating modes.

4. Conclusions

From the frequency characteristics for the torque and power (figure 9) and the hydrodynamic characteristics between the torque coefficients and the power versus speed ratio (figure 10) it is apparent that they correspond qualitatively and quantitatively to the experimentally obtained characteristics presented in the literature.

In the comparison of the frequency characteristics of figure 9 with the analytically obtained frequency characteristics [4] shown in figure 11, it was found that for both solutions at the rotational speeds \( n = 100..150 \text{ min}^{-1} \), the powers differ by (5 ... 15)\% and the torques with (4..18)\%. It is accepted that the analytical solution is more accurate as it is done by a modified classical method [3, 4].

Compliance of the shape and energy performance characteristics is considered to be a numerical model verification obtained by simulation with the Flow Simulation CFD.

Acknowledgments

The present paper has been written with the Project No 2020-RU-03’s financial assistance.

References

[1] Terziev A 2014 Specifics in numerical modeling of flow past a square-cylinder *Science Conference of Ruse University* **53** 143

[2] Tuzharov K, Popov G, Zheleva I, Klimentov K, Nikolaev I, Kostov B and Ahmedov A 2015 Theoretical study of tachometric flowmeter with CFD product Flow Simulation. *Science Conference of Ruse University* **46**

[3] Fateev F M 1958 *Wind turbines and wind installations* (Agrycultureedition, Moskva)

[4] Tuzharov K, Popov K G, Klimentov K, Nikolaev I, Kostov B 2017 Characteristics of Modeling Hydrokinetic Turbine *56th Science Conference of Ruse University* Ruse