TRANSACTIONS OF THE INSTITUTE OF FLUID-FLOW MACHINERY

No. 123, 2011, 71–84

MACIEJ KANIECKI[1*], ZBIGNIEW KRZEMIANOWSKI[1] and MARZENA BANASZEK[2]

Computational fluid dynamics simulations of small capacity Kaplan turbines

1 The Szewalski Institute of Fluid-Flow Machinery of the Polish Academy of Sciences, Hydraulic Machinery Department, Fiszera 14, 80-231 Gdańsk, Poland

2 Gdansk University of Technology Faculty of Mechanical Engineering Narutowicza 11/12 80-233 Gdańsk, Poland

Abstract

In the presented paper authors focused on numerical analysis of the flow through selected constructions of Kaplan and semi-Kaplan machines. The paper contains three computation examples; each of them concentrates on different aspects of computational fluid dynamics (CFD) utilization. The numerical tests of various turbulent models for Kaplan turbine CFD applications, numerical determination of the cam curve characteristic and numerical analysis of operating parameters of the semi-Kaplan machine have been presented. The methodology of the implemented numerical computations and final results of calculations have been discussed in each of the presented study cases.

Keywords: Hydraulic Kaplan turbine; CFD simulations; Determination of operating parameters; Cam curve characteristic; Hill diagram

1 Introduction

In the recent decade computational fluid dynamics (CFD) technique achieved high level of reliability and occurred a practical tool for operating behaviour determination of different types of fluid-flow machinery. With properly defined calculations

*Corresponding author. E-mail address: kaniecki@imp.gda.pl

domain (i.e. precise representation of geometry, fine grid constructing, appropriate computation model and correct definition of boundary conditions) the CFD simulations seem to be very reasonable both with their qualitative and quantitative results. In respect to hydraulic turbines, the computational methods enable supporting the design processes of the flow systems and determining the operating conditions with large and small scale flow phenomena. The Institute of Fluid-Flow Machinery of the Polish Academy of Sciences has some experiences in performing CFD simulations of the flow through flow systems of hydraulic turbines.

In the presented paper authors focused on numerical analysis of the flow through selected constructions of Kaplan and semi-Kaplan machines. As it was mentioned above, the paper contains three computation examples of different aspects of CFD utilization. Started with the first example, authors compared numerical calculations with the test results of the model Kaplan turbine. The tests were carried out on the model test rig located at the Gdansk University of Technology. For the optimal operating point, the numerical analysis was conducted for a series of different turbulence models. Authors observed that the final results of determination of global operating parameters were significantly sensitive to the applied turbulence model. In the second example, the methodology of the Kaplan cam curve determination by means of CFD has been presented with the great focus to the local phenomena of the flow and global operating parameters of the turbine. The last example is dedicated to calculations of the operating parameters of the model semi-Kaplan turbine, which has been developed from first principle in the Institute. Final results of the CFD simulations for newly designed semi-Kaplan turbine have been presented.

2 Numerical modelling of the flow through the model Kaplan turbine

2.1 Motivation and brief characteristic of the computational model

The primary aim of present investigation was the numerical analysis of an existing model Kaplan turbine with adjustable position of the blade system (Fig. 1). The turbine under consideration is mounted on a test stand at the Fluid Mechanics and Turbomachinery Center of Gdansk University of Technology Faculty of Mechanical Engineering, and has been thoroughly experimentally investigated, resulting in the identification of optimum setting combination of the guide vanes and the runner blades. This experimentally determined efficiency-optimal work point has been taken as the basis for numerical modeling. Computational fluid dynamics is well established in the analysis of turbomachinery flows for over 20 years. The literature, however, enumerates computational results using commercial solvers

Figure 1. Schematic view of the flow system of investigated model turbine.

with arbitrarily chosen turbulence models, typically selected on the basis of user experience. It should be noted that there is no single, universally applicable turbulence model; its appropriate selection for a specific computational case rests on the expert judgment. As the authors had at their disposal accurate and comprehensive experimental results, it was decided to deliberately carry out computation employing a broad range of turbulence models with variants. The following models were employed [5]: one-dimensional Spallart-Almaras, two dimensional k-ε and k-ω models. For the k-ε model the analysis has been carried out for assorted variants of the model, distinguished by various methods of boundary layer modeling (standard, nonequilibrium, enhanced) and by different viscosity models (algebraic or differential). Altogether twelve numerical variants were used, and the varying results were checked against the experimental standard. Numerical analysis was carried out using the ANSYS/Fluent [7] commercial solver, a popular choice for numerous technical applications. The fundamental geometrical and operating parameters of the tested model turbine are presented in Tab. 1.

The computational mesh for the given geometry (Fig. 1) was prepared in two stages. At the first stage, the mesh for the rotor and for the stators was prepared using the Numeca AutoGrid [9] tool. After transferring the fragmentary mesh to Gambit [8], the mesh was completed by meshing the draft tube and the offset inlet. The complete mesh consists of over 7 million elements. The mesh consists entirely of hexahedral elements, and was structured to achieve values of the nondimensionless wall distance (y^+) in the range 1 to 3 [5]. The computation has been carried out with the aid of the Fluent solver included in the ANSYS 12.1 [7] package, with the additional assumption of stationary flow and using a second order discretization scheme. Carried out in double precision, the computation

Table 1. Fundamental geometrical and operating parameters of the model turbine.

Nett head H_{net} [m]	2.725
Discharge Q [m^3/s]	0.074
Rotational speed n [rpm]	650
Power output P [kW]	1.7
Runner diameter D [m]	0.25
Number of runner blades	6
Number of guide vanes	12
Number of stator vanes	6

required the order of $2{\times}10^4$ iterations. As noted in the introduction, the computations were carried out using several turbulence models for comparison. Three fundamental turbulence models were used: namely the one-dimensional Spallart-Almaras, two dimensional k-ε and two-dimensional k-ω model.

The results of computation are presented in Tab. 2 and Figs. 2 and 3. Table 2 summarizes the key quantities calculated for several computational variants and measured in the experiment, specifically the torque, power, mass flow rate and hydraulic efficiency. In Figs. 2 and 3 the CFD results of static pressure and velocity fields are shown.

Table 2. Numerical results for various turbulence models and experiment measurement of model Kaplan turbine.

No.	Experiment and turbulence models	Torque [Nm] (Power) [W]	Mass flow rate [kg/s]	Hydraulic efficiency [%]
	Experiment	25.9 (1763)	74.3	88.7
1.	Spalart-Allmaras	25.7 (1750)	74.1	88.5
2.	k-ε standard enhanced	24.3 (1654)	73.3	84.6
3.	k-ε RNG standard	24.5 (1664)	74.1	84.3
4.	k-ε RNG standard – differential viscosity model	25.3 (1725)	73.8	87.9
5.	k-ε RNG non-equilibrium	23.4 (1594)	74.2	80.7
6.	k-ε RNG non-equilibrium – differential viscosity model	25.6 (1741)	74.8	87.4
7.	k-ε RNG enhanced	25.2 (1712)	73.8	87.2
8.	k-ε RNG enhanced – differential viscosity model	25.1 (1708)	73.7	86.9
9.	k-ε realizable standard	24.9 (1700)	74.2	85.9
10.	k-ε realizable non-equilibrium	23.9 (1632)	74.1	82.6
11.	k-ε realizable enhanced	25.8 (1755)	74.2	88.7
12.	k-ω SST	25.8 (1754)	74.3	88.5

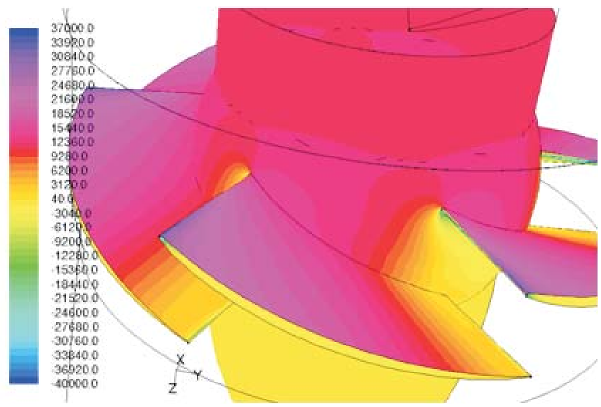

Figure 2. Distribution of the static pressure on the runner surfaces.

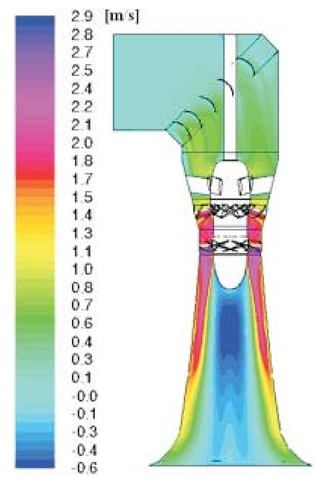

Figure 3. Distribution of the axial velocity in the turbine flow system.

2.2 Conclusions

The computation has brought to light significant and expected discrepancies between the obtained values for flow efficiency yielded by the disparate methods of turbulence modeling. The highest degree of consistency, and hence the closest to experimental validation, has been returned by the k-ε realizable enhanced model (88.7%), closely followed by the results accomplished using the k-ω SST model (88.5%) and final being the Spalart-Allmaras model (88.5%), the experimental reference value being 88.7%. The worst performer in this regard has been the k-ε RNG non-equilibrium model (80.7%). Analyzing the differences in individual quantities impacting efficiency leads to the conclusion that the largest disparity is due to differences in computed and measured torque on the turbine shaft. All the computed variants yielded values of the torque significantly lower than the value actually measured. The discrepancies between predicted and measured values of mass flow rate resulted in a similar, but smaller effect.

3 Determination of the cam curve characteristic by CFD applications

3.1 Methodology of CFD cam curve determination

Crucial advantage of the double regulation Kaplan turbine is the ability of operating with relatively high efficiency in the broad range of loading. To achieve a high efficiency Kaplan turbine performance the optimization research of relative position of the runner blades and guide vanes are necessary [2,4]. As the final result of the research the cam curve characteristic for the specific Kaplan turbine is determined. Practically the optimisation research should be carried out for every new installation at the hydropower plant. Unfortunately the costs of such activities are really high, especially for small hydro installations. Due to limited financial abilities, small hydro power companies and manufactures are interested in cost reduction, which is realized among others by virtual calculations of the cam curve characteristics for low capacity hydraulic turbines. The concerned company, which decides to apply the CFD for optimisation process of the Kaplan turbine operation, should be aware of the limited accuracy of such approach. The crucial discrepancies between CFD analyses and real test results can be generated both by simplifications of real conditions and numerical errors of utilized model. The Szewalski Institute of Fluid-Flow Machinery has some experience in such kind of CFD applications speciallyt developed for small hydropower companies. The presented example shows the application of CFD using the finite volume method for determination of the operating conditions and cam curve characteristics of low head bulb turbine. The main parameters and the draft of the flow system of the tested turbine are presented in Tab. 3 and Fig. 4.

Table 3. Main geometrical and operating parameters of tested turbine.

Nett head H_{net} [m]	3.0
Discharge Q [m^3/s]	3.54
Rotational speed n [rpm]	305
Power output P [kW]	93
Rotor diameter D [m]	1.09
Number of runner blades	3
Number of guide vanes	16

The methodology of the numerical cam curve determination is practically similar to the real conditions research. For a constant value of rotational speed and the nett head the specific runner blade angle β_1 is determined. For the case of a single runner blade position considered were five to six guide vanes

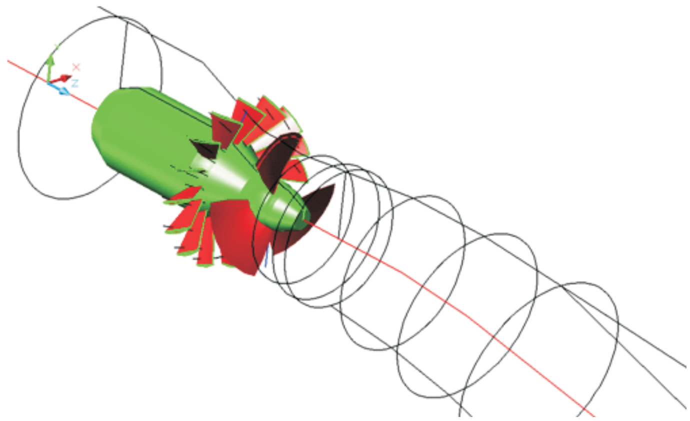

Figure 4. Draft of the flow system of the tested bulb turbine.

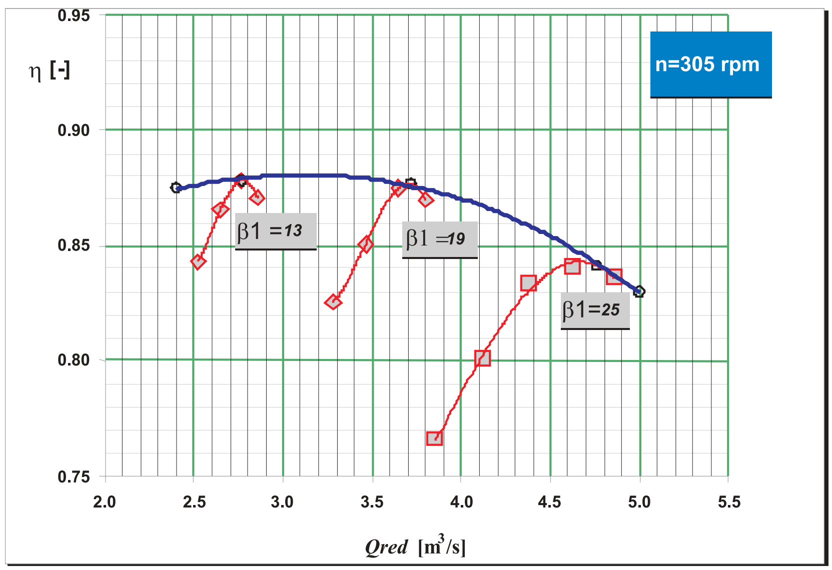

Figure 5. The envelope curve (–) and the propeller curves (□) for determined rotational speed $n = 305$ [rpm] of bulb turbine.

opening β_2. At every relative positions of blade system of the turbine the CFD calculations are carried out to determine operating parameters of the turbine. In Fig. 5 the exemplary characteristics of the hydraulic efficiency via discharge factor Q_{red} (reduced for $H_{net} = 3$ [m]) is presented. The characteristic was prepared for a given rotational speed of the runner and the net head. The successive operating points are the results of the conducted CFD computations. The envelope through the best efficiency points at every propeller curve has been drawn. Basing on that envelope curve the cam curve has been finally determined which is the relation between the runner blade angle and guide vanes angle for the best efficiency points (Fig. 6).

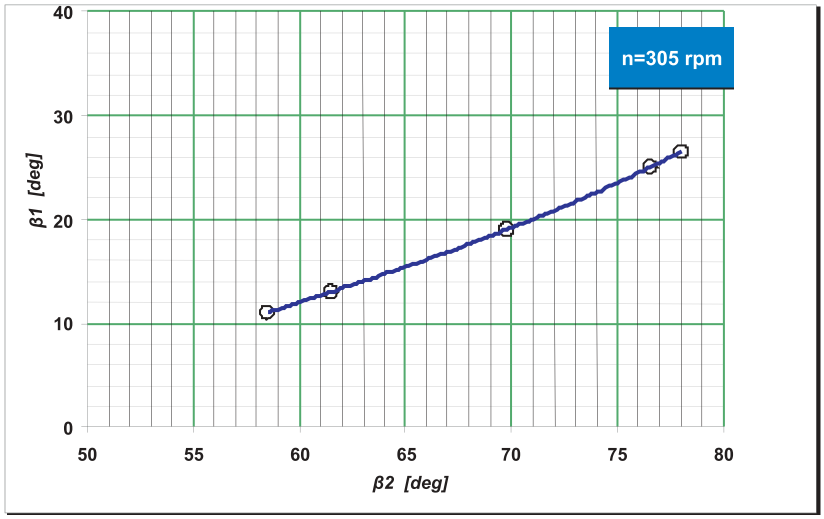

Figure 6. The cam curve characteristic for a given rotational speed of bulb turbine.

3.2 Computational model

The numerical calculation of liquid flow throughout the flow system of the presented bulb turbine has been carried out using the Numeca/Fine [9] and ANSYS [7] commercial software. The main purpose of numerical calculations was to determine the operating parameters for a wide range of performance. Every single calculation process was realized to specify the relative position of turbine blade system. The calculated global values of operating parameters were marked on the specified propeller curve. The entire geometry has been divided into three blocks — inlet pipe, blade system and the draft tube — connected with the interfaces. Finally, the structured grid was applied in the whole area of calculation domain. In the blade system block, the structure of the grid was the O-shape with the gradation of the elements thickness near the walls. The value of dimensionless y^+ was approximately defined using the empirical formula (the calculated value of y^+ was close to 2). The programme solves the governing integral equations for the conservation of mass and momentum for the steady state flow. The turbulence flow has been taken into account by application of the turbulence k-ε realizable enhanced model. In Figs. 7 and 8 a few screen shots of the grid structure and obtained results have been presented.

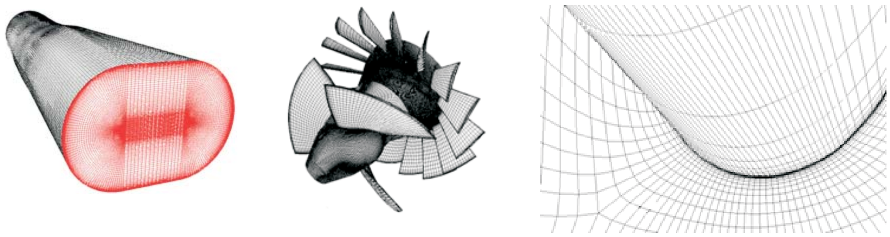

Figure 7. Grid structure on the chosen elements of the turbine flow system.

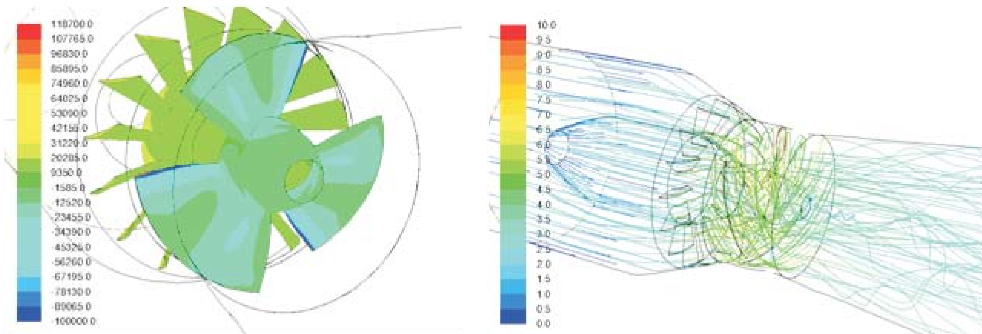

Figure 8. Distribution of the static pressure and streamlines for optimal operating point W19K67, $n = 305$ rpm.

3.3 Conclusions

Basing on the CFD calculations the optimal positions of the Kaplan blades system can be obtained with relatively fine accuracy for a wide range of applications. As it was notified above the precise representation of the flow system geometry and properly constructed CFD model (fine grid with appropriate y^+ value, appropriate turbulent model, second order discretization scheme with at least 10^4 iterations per process) have a significant effect on numerical results. For small hydro applications the of CFD based determination of the cam curve is reasonable and economically profitable.

4 Calculations of the operating characteristic of the model semi-Kaplan turbine

During the last thirty years the Szewalski Institute of Fluid-Flow Machinery conducted some experimental and theoretical activities on the development of hydraulic turbines designed for small hydro power applications. Currently, the semi-Kaplan low head model turbine has been realized under the granted government project (the so-called 'TNS' series of types). The turbine is equipped with the banded draft tube and steady state guide vanes palisade. The fundamental geometrical and operating parameters of the turbine are presented in Tab 4. The schematic view of the flow system is shown in Fig. 9.

Table 4. Fundamental geometrical and operating parameters of the model turbine.

Nett Head H_{net} [m]	2
Discharge Q [m^3/s]	0.21
Rotational speed n [rpm]	850
Power output P [kW]	2.7
Rotor diameter D [m]	0.3
Number of runner blades	3
Number of guide vanes	10

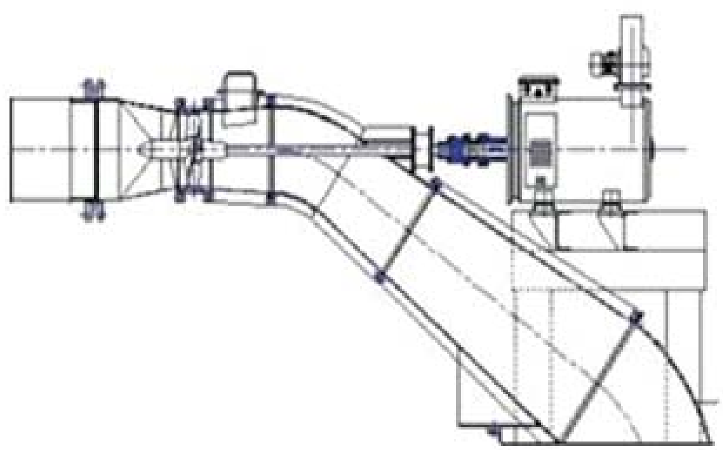

Figure 9. Schematic view of the flow system of TNS model turbine.

One of the main purposes of the project is the turning of applied CFD model by comparison with the experimental results. Up to now the CFD analysis is completed with the final results of the computations in the form of the hill diagram. The experimental test will be carried out in a period of few months. The numerical analysis possessed the attributes of the real conditions laboratory tests. For chosen speed factors $n11$ the operating parameters of discharge factors $Q11$, power output factors $P11$ and the hydraulic efficiency η_h have been calculated with a wide range of runner blade opening positions. The hydraulic efficiency characteristic via discharge factor has been determined for selected values of the speed factor (Fig. 10) and finally, as a consequence of former steps, the hill diagram has been drafted (Fig. 11). The following figures present single efficiency characteristic and the hill diagram of the model TNS turbine.

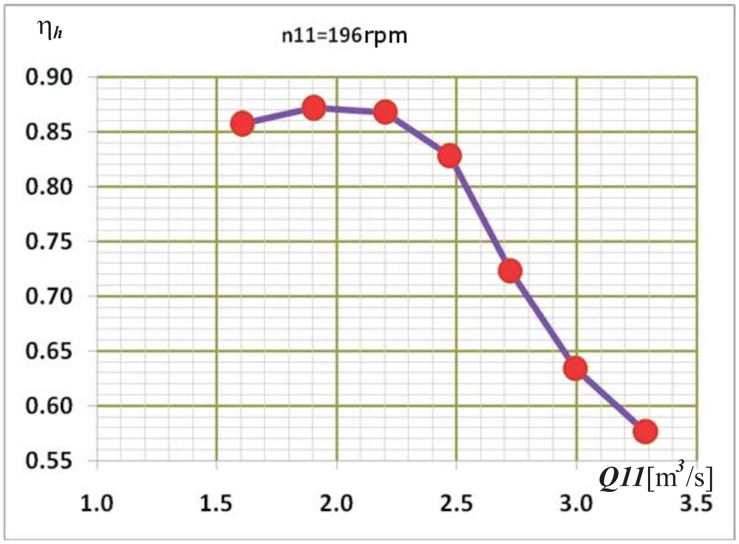

Figure 10. The single hydraulic efficiency characteristic via discharge factor obtained by the numerical calculations for TNS model turbine..

4.1 Computational model

The grid preparation and numerical calculation of the TNS model turbine have been carried out, as above, using the Numeca/Fine [9] and the ANSYS [7] commercial software. Determinations of the fundamental operating conditions as well as the investigations of the local phenomena of the flow were the crucial purposes of the TNS numerical researches. The calculations were realized for six runner blade positions, which required generation of the same number of structural grids domains. The quality of the grids were satisfactory. In all cases between 1 to 3 nodes were placed in the near-wall region $y^+ \leq 3$. As it was imposed for previous examples, the applied model realized the steady state flow. The turbulence flow has been taken into account by application the turbulent SST k-ω model. In the pictures (Figs. 12 and 13) the grid structure and obtained results of pressure field and the streamlines distributions have been presented.

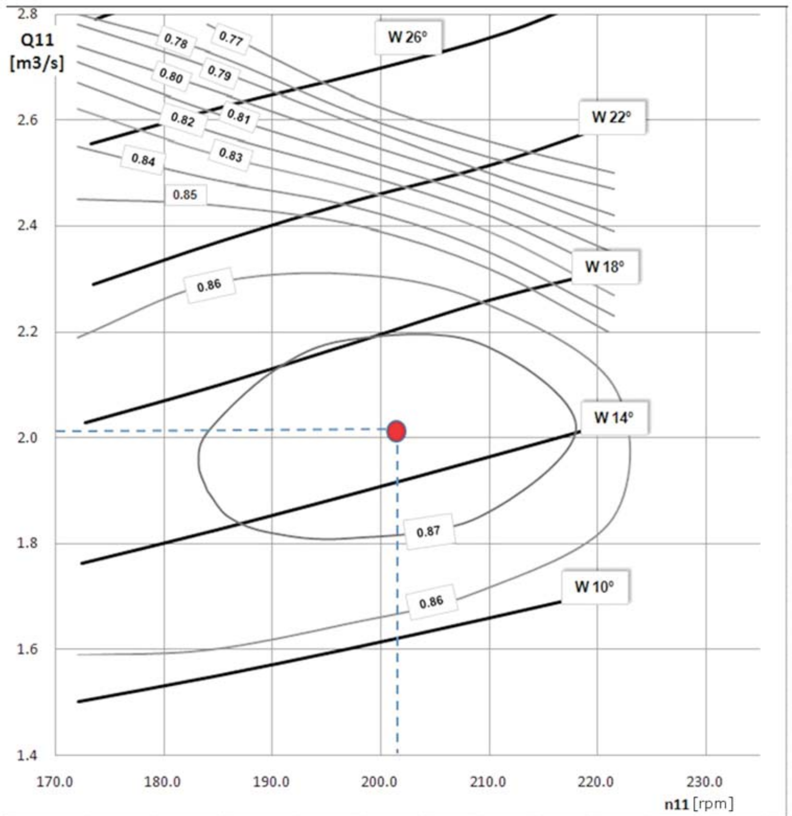

Figure 11. Hill diagram obtained by the numerical calculations for TNS model turbine. Discharge factor $Q11$ (reduced for $H_{net} = 1$ [m] and $D = 1$ [m]) via speed factor $n11$ (reduced for $H_{net} = 1$ [m] and $D = 1$ [m]).

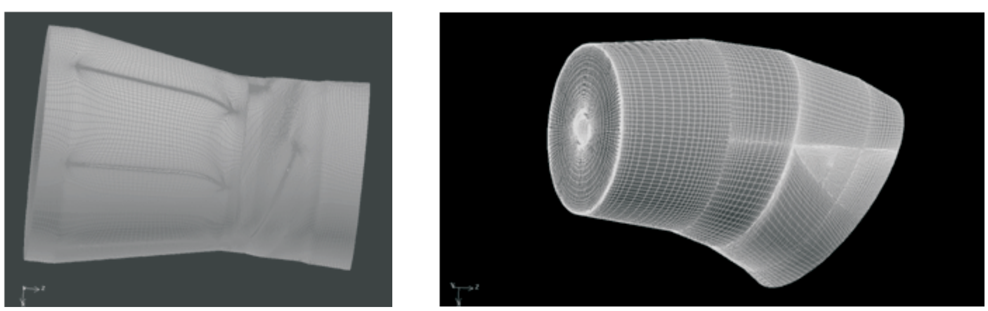

Figure 12. Structure of the grids in the blade system and the draft tube of the TNS turbine.

4.2 Conclusions

The final verification of conducted calculations will be realized after the series of laboratory tests of model semi-Kaplan TNS turbine. However obtained CFD

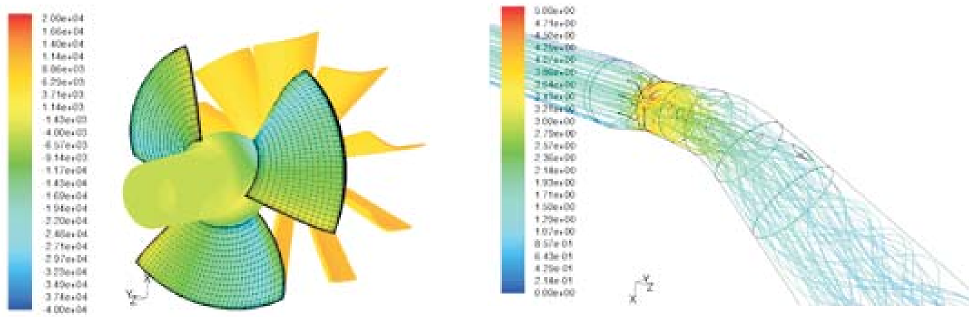

Figure 13. CFD results of static pressure and streamlines distributions in the TNS turbine flow system.

results seem to be qualitatively similar to the literature data [1,3] and authors own experience. Obviously the quality of the CFD projection depends on the several aspects which were taken under the consideration in the previous study cases — the detailed representation of real geometry, fine grid with appropriate y^+ value, appropriate turbulent model, second order discretization scheme with at least 10^4 iterates per process.

Acknowledgments Preparation of the paper has been funded from an the Szewalski Institute of Fluid-Flow Machinery Pol. Ac. Sci. statutory activity.

Received 25 October 2011

References

[1] Anton I.: *Hydraulic Turbine*, Facla, Timisoara 1979.

[2] Gordon J.L.: *Turbine selection for small low-head hydro developments*. In: Proc. Waterpower XII, Buffalo 2003, USA.

[3] Gorla R., Khan A.: *Turbomachinery — Design and Theory*. Marcel Dekker Inc., New York 2003.

[4] Nechleba M.: *Hydraulic Turbines — Design and Equipment*. Artia, Prague 1957.

[5] Perić M., Ferziger J.M.: *Computational Methods for Fluid Dynamics*. Springer-Verlag, Berlin 1996.

[6] Kaniecki M., Krzemianowski Z., Banaszek M.: *Determination of the operating parameters for Kaplan turbines utilizing the CFD calculations*. In: Proc. Int. Conf. Hydro 2011, Prague 17–19 Oct. 2011.

[7] *ANSYS Fluent 12.1 Theory Guide*. Ansys Inc. April 2009.

[8] *Gambit Tutorial Guide*. Fluent Inc. May 2000.

[9] *Numeca. Numeca's Flow Integrated Environment. User Manual — Fine Version 6*, September 2006.