Numerical simulation of two-stage swirl turbine

Luděk Koutný[1], Miloslav Haluza[2], Petr Uttendorfský[3]

[1] Brno University of Technology, Faculty of Mechanical Engineering, Energy Institute, Victor Kaplan Department of Fluid Engineering, Technická 2896/2, Brno, ykoutn04@stud.fme.vutbr.cz

[2] Brno University of Technology, Faculty of Mechanical Engineering, Energy Institute, Victor Kaplan Department of Fluid Engineering, Technická 2896/2, Brno, haluza.m@fme.vutbr.cz

[3] Brno University of Technology, Faculty of Mechanical Engineering, Energy Institute, Victor Kaplan Department of Fluid Engineering, Technická 2896/2, Brno, yutten00@stud.fme.vutbr.cz

Abstract This paper deals with numerical simulation of two-stage swirl turbine. It consists of two counter-rotating runners, the first one has three blades, the second one two blades. Between these stages there are five indifferential stator vanes. Their aim is to stabilize wakes in front of blades of the second runner. Inlet flow field of the first stage is obtained from previous simulation, its direction is mainly axial. The outgoing flow from the first stage has high tangential velocity. The blades of the second stage should make the stream flow in axial direction. Model will be manufactured to measure turbine characteristics and check design parameters.

1 Introduction

Swirl turbine has been developed by team led by Prof. Ing. František Pochylý., CSc. As mentioned in [1] this turbine covers small heads (1 to 3 meters) and large discharges. This turbine has two non-adjustable blades, governing is enabled by change of runner speed using a frequency converter. Efficiency curve is rather flat with highest efficiency over 80%. There is no need of a distributor because flow enters the runner without circulation. Due to the Euler turbine equation (1)

$$E \cdot \eta_h = -u_2 \cdot v_{u2} \tag{1}$$

(where E (J/kg) is energy of the turbine, η_h (dimensionless) is hydraulic efficiency, u (m/s) is circumferential velocity and v_{u2} (m/s) is circumferential part of absolute velocity) there is opposite direction of the runner speed and the swirl behind the runner. Rotating flow is entering a draft tube, it supports attaching of boundary layer to the draft tube. On the other hand, draft tube cannot slow down tangential velocity, so efficiency can be higher, if the flow in the draft tube is more axial.

One of ways how to improve efficiency and operating conditions of the swirl turbine is placing a second runner behind the first one. If we design its blades properly, the stream can enter the draft tube with only axial velocity. Second benefit is that the head of the turbine can be raised or – in other words – cavitation phenomena can be supressed. There is also possibility to improve governing – by deceleration of the second stage we can change the flow rate (this feature should be proved).

2 Design and numerical simulation

A model of the swirl turbine with two counter-rotating runners has been designed. This model will be arranged as a “siphon turbine” (see **Fig.1**) and tested in hydraulic laboratory of Victor Kaplan Department of Fluid Engineering. An elbow and a confuser make the inlet part of the turbine which consists of two counter-rotating runners with non-adjustable blades and five indifferential stator vanes between these two stages. The first runner has three blades, the second one two blades. Operating conditions of the second stage are very variable, because there are wakes behind blades of the first runner. Aim of the stator vanes is to stabilize wakes behind the first runner. Diffuser and another elbow make the outlet of the turbine. Draft tube continues behind the second elbow.

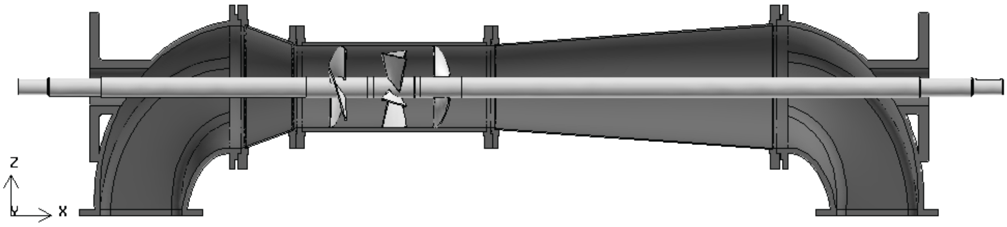

Fig.1 Model of two-stage swirl turbine

Computational domain, which starts with the end of confuser, consists of runners, chamber, diffuser and outlet pipe. The inlet armature was used for computing inlet flow field. It was not included in the computational domain. Commercial CFD software ANSYS Fluent was used for numerical computation. Setting of the software was as follows (see [2] for better understanding):

- Sliding mesh approach was used which means that the mesh is really rotating, it is an unsteady problem.
- Two-equation realizable k-ε turbulence model was selected for turbulence modelling.
- Non-equilibrium wall functions were selected for near wall treatment.
- Implicit time-stepping scheme of the 2^{nd} order was used for time-accurate calculations.
- 2^{nd} order upwind interpolation scheme was used for discretization of advective terms.

Inlet flow field was obtained by previous simulation and constant gauge pressure was prescribed at the outlet boundary condition. Angular velocity orientations of both runners are opposite, but its magnitudes can be different. In our case the magnitudes are equal.

3 Results of the numerical simulation

Three different operating points were simulated. Inlet flow field remained the same in all of these simulations and angular velocity was subject to change. Magnitude of angular velocity was the same for both runners. Simulation started with 1281 rpm, other calculated operating points were 1381 and 1181 rpm. Energy of the turbine runner can be obtained with equation (2).

$$E = E_1 - E_2 = \frac{p_1 - p_2}{\varrho} + \frac{v_1^2 - v_2^2}{2} \tag{2}$$

where subscript 1 indicates quantity on the runner inlet and subscript 2 on the runner outlet; p (Pa) is static pressure, ρ (kg/m^3) is density and v (m/s) is absolute velocity. We obtain head by dividing energy by gravitational acceleration.

$$H = \frac{E}{g} \tag{3}$$

Hydraulic efficiency of the runners is defined by equation (4). It is calculated between cross sections 1 and 2 as mentioned in equation (2), the draft tube is not included.

$$\eta_h = \frac{M\omega}{\varrho Q E} \tag{4}$$

where M (Nm) is torque, ω (rad/s) is angular velocity and Q (m^3/s) is flow rate. According to Nechleba [3] we obtain basic turbine characteristics - unit speed and unit flow rate as

$$n_{11} = \frac{nD}{\sqrt{H}};\ Q_{11} = \frac{Q}{D^2\sqrt{H}} \tag{5}$$

where n (min^{-1}) is rotational speed, D (m) is diameter of the runner and H (m) is head.

Tab.1 shows turbine characteristics of the first stage. In this table negative rotational speed and torque means it is oriented in negative way (according to right-hand rule). There is zero incidence angle of the inlet relative velocity on the first runner blades (see **Fig.2**).

Tab.1 Turbine characteristics of the first stage

n (min^{-1})	Q (m^3s^{-1})	M (Nm)	E (Jkg^{-1})	H (m)	Q_{11} (m^3s^{-1})	n_{11} (min^{-1})	P (W)	η_h (%)
-1181	0,114	-12,07	17	1,73	2,17	179,5	1493	77,22
-1281	0,114	-8,53	13,3	1,36	2,45	220,1	1144	75,65
-1381	0,114	-4,87	9,2	0,94	2,94	284,9	705	67,17

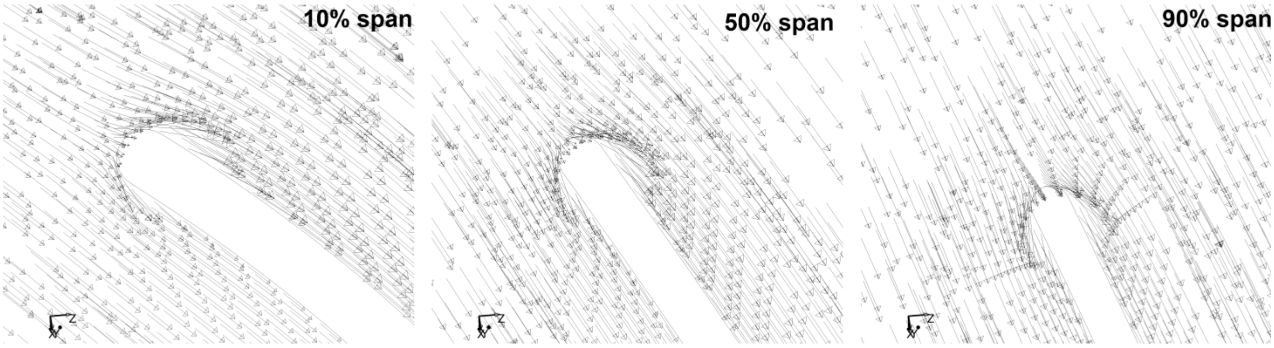

Fig.2 First stage: Inlet relative velocities at 10%, 50% and 90% span between hub and tip

Design point of the first runner is: n_{11} = 173,1 min^{-1}, Q_{11} = 1,95 m^3s^{-1} with head H = 2,192 m. In the first cascade we can see that unit speed (n_{11} = 179,5 min^{-1}) of this operating point is very close to the design point, but unit flow rate is not so close. It leads to correction of blade angles, which is not part of this paper.

In **Tab.2** we can see turbine characteristics of the second stage. It is clear that computed operating points are far from the design point, which is: n_{11} = 215 min^{-1}, Q_{11} = 2,433 m^3s^{-1} with head H = 1,408 m. The unit speed is very high and close to the runaway speed. Operating point n = 1381 min^{-1} is not turbine operation - the torque is oriented in opposite way to the angular velocity. Hydraulic efficiency of the operating point n = 1181 min^{-1} is very high (96,61%), more operating points should be simulated to obtain the efficiency curve.

Tab.2 Turbine characteristics of the second stage

n (min^{-1})	Q (m^3s^{-1})	M (Nm)	E (Jkg^{-1})	H (m)	Q_{11} (m^3s^{-1})	n_{11} (min^{-1})	P (W)	η_h (%)
1181	0,114	3,61	4,1	0,41	4,43	367	447	96,61
1281	0,114	1,46	2,34	0,24	5,83	524,1	196	73,46
1381	0,114	-0,89	0,25	0,03	17,76	1721,1	-	-

Visualization of the flow field behind the runners by means of pathlines can be seen in **Fig.3**. Absolute velocity behind the second stage has still high circumferential component (v_u from 1,45 ms^{-1} near hub to 0,95 ms^{-1} near chamber). Simulated operating points of the second stage differ from its design point. More operating point should be simulated to obtain the efficiency curve and find the design point. When we find it, we will know that the blades are well designed or not.

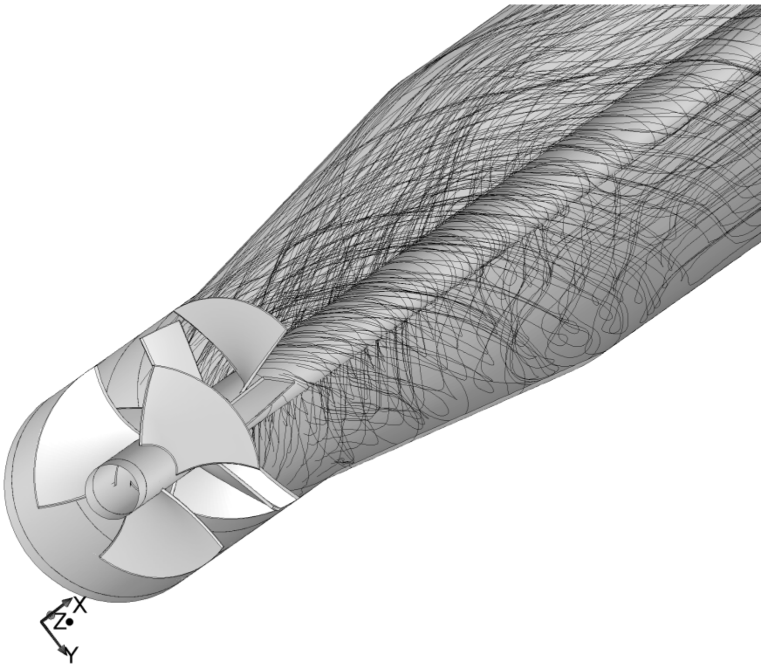

Fig.3 Flow field visualization with pathlines behind the runners

4 Conclusion

First design of counter-rotating runners of the swirl turbine has been presented in this paper. Commercial CFD software was used for numerical simulation and optimization. The first stage is ready for blade angles correction to reach designed values of unit flow rate and unit speed. Now the second stage is the point of our interest, we must find its hydraulic efficiency and unit flow rate characteristics to know if blade angles (and other) corrections will be needed.

Final design will be manufactured and its characteristics will be experimentally measured to compare with numerical results.

Acknowledgement The financial support of the Specified Research Grants no. FSI-J-12-21/1698 “Enlargement of Working Zone of Hydraulic Machines” and FSI-S-12-2 “Vortex Flow and Vortex Structures in Fluid Flow ” is gratefully acknowledged.

References

[1] POCHYLÝ, František - HALUZA, Miloslav - RUDOLF, Pavel - ŠOB, František. Design of a new low-head turbine. In *Proceedings of the XXIst IAHR Symposium, Hydraulic Machinery and Systems, Volume I.* LMH/EPFL. Baden: 2002. s. 29-34.

[2] ANSYS Fluent 12.0/12.1 Documentation. <https://www.sharcnet.ca/Software/Fluent12/index.htm>. [citated 2012-4-25]

[3] NECHLEBA, Miroslav. *Vodní turbíny, jejich konstrukce a příslušenství [Water turbines, design and accessories].* 2nd edition. Prague: SNTL. 1956.