Boosting the Power Generation in Wind and Hydro Power Production

***Victor P. Bashurin*[1]*, Igor N. Budnikov*[2]*, Vitaliy Y. Khatunkin*[3]*, Vladimir A. Klevtsov*[4]*, Lev V. Ktitorov*[*5]*, Anna S. Lazareva*[6]*, Evgeny E. Meshkov*[7]*, Irina A. Novikova*[8]*, Fedor A. Pletenev*[9]*, Gennady M. Yanbaev*[10]**

[1]ITC Sistema-Sarov, SarovTechnopark, Parkovaya Street 3, Satis, Diveevo District, Nizhny Novgorod 607328, Russian Federation
e-mail: victor.bashurin@sarov-itc.ru

[2]Sarov Institute of Physics and Technology, Dukhova Street 6, Sarov, Nizhny Novgorod 607186, Russian Federation
e-mail: bin033@mail.ru

[3]Sarov Institute of Physics and Technology, Dukhova Street 6, Sarov, Nizhny Novgorod 607186, Russian Federation
e-mail: khatunkin@fakemail.po

[4]Sarov Institute of Physics and Technology, Dukhova Street 6, Sarov, Nizhny Novgorod 607186, Russian Federation
e-mail: V.A.Klevtsov@mephi.ru

[5]ITC Sistema-Sarov, SarovTechnopark, Parkovaya Street 3, Satis, Diveevo District, Nizhny Novgorod 607328, Russian Federation
e-mail: lktitorov@gmail.com

[6]ITC Sistema-Sarov, SarovTechnopark, Parkovaya Street 3, Satis, Diveevo District, Nizhny Novgorod 607328, Russian Federation
e-mail: anja.s.lazareva@gmail.com

[7]Sarov Institute of Physics and Technology, Dukhova Street 6, Sarov, Nizhny Novgorod 607186, Russian Federation
e-mail: eemeshkov@gmail.com

[8]Sarov Institute of Physics and Technology, Dukhova Street 6, Sarov, Nizhny Novgorod 607186, Russian Federation
e-mail: inovikova@fakemail.po

[9]ITC Sistema-Sarov, SarovTechnopark, Parkovaya Street 3, Satis, Diveevo District, Nizhny Novgorod 607328, Russian Federation
e-mail: fedor.pletenev@sarov-itc.ru

[10]Sarov Institute of Physics and Technology, Dukhova Street 6, Sarov, Nizhny Novgorod 607186, Russian Federation
e-mail: yanbaev@fakemail.po

Cite as: Bashurin, V. P., Budnikov, I. N., Khatunkin, V. Y., Klevtsov, V. A., Ktitorov, L. V., Lazareva, A. S., Meshkov, E. E., Novikova, I. A., Pletenev, F. A., Yanbaev, G. M., Boosting the Power Generation in Wind and Hydro Power Production, J. sustain. dev. energy water environ. syst., 4(3), pp 279-292, 2016,
DOI: http://dx.doi.org/10.13044/j.sdewes.2016.04.0022

ABSTRACT

When approaching a conventional wind turbine, the air flow is slowed down and widened. This results in a loss of turbine efficiency. In order to exploit wind or water flow power as effectively as possible, it was suggested that the turbine should be placed inside a shroud, which consists of 4 wing-shaped surfaces. Two internal air foils improve the turbine performance by speeding up the flow acting on the turbine blades, two external wings create a field of low pressure behind the turbine, thus, helping to draw more mass flow to the turbine and avoid the loss of efficiency due to flow deceleration. The system accumulates kinetic energy of the flow in a small volume where the smaller (and therefore, cheaper) turbine can be installed. A smaller system can be installed inside the bigger one, which would help to accumulate even more kinetic energy on the turbine.

[*] Corresponding author

This method implies kinetic energy summation with local flow redistribution. Both experiments and CFD simulations demonstrate a significant increase in velocity and generated mechanical power in comparison to those for a bare turbine.

KEYWORDS

Hydropower, Wind power, Ducted (shrouded) turbine, Wind turbine, Flow booster, Power augmentation, Wind turbine efficiency.

INTRODUCTION

Wind or flowing water kinetic energy can be harnessed and utilized in many types of renewable energy facilities. The efficiency of such facilities strongly depends on the kinetic energy flux in the flow. For example, a Run-Of-the-River (ROR) electric generator uses a current of a river or a naturally occurring tidal flow to create electricity. When developed with care to footprint size and location, ROR hydro projects can create sustainable energy minimizing impacts to the surrounding environment and nearby communities [1]. It may provide a renewable energy source that will have a minimal impact on the environment since it requires no dam to create a pressure head. But high-velocity water flow is still of critical importance for creating the efficiently working turbine.

Literature review and the problem context

Fluid power generation is proportional to the fluid speed cubed. Betz has shown [2] that an ideal, propeller-type wind-turbine can extract only 59.3% of the kinetic energy which flows through a stream-tube that covers the swept area of the turbine blades. In practice, the energy that can be generated with a constant wind is typically limited to a range of 7-17% of the energy contained in the wind.

Even a slight increase in fluid velocity causes a large increase in a turbine power output. A significant flow rate increase can be achieved by installing the turbine inside the specifically designed channeling device helping to concentrate the flow on the turbine. The desired flow speed increase could be obtained by creating and maintaining the low-pressure zone behind the turbine. The proper power augmentation would help to achieve the desired output by using relatively small turbine blades, therefore, would make wind power economically more competitive with other natural energy sources.

Numerous diffuser studies have been published for both wind and hydro turbines. The possibility for mass flow augmentation through the diffuser throat was confirmed and quantified in the 1950's, both theoretically [3] and experimentally [4].

In it's early stage of development, the wind turbine design usually featured a large inlet to absorb more wind, and became impractical since the duct usually turned out to be too expensive. This idea gradually evolved into the introduction of smaller and cheaper designs for wind and hydro turbines to induce more velocity rather than more airflow [5].

The sharp increase in oil prices in the 1970's reignited interest in Diffuser Augmented Wind Turbines (DAWTs). Research groups led by Gilbert and Foreman [6] at Grumman Aerospace Corporation and Igra [7] at Ben-Gurion University of the Negev published a series of studies on concentrating wind energy in a diffuser employing the method of boundary layer control that should have prevented pressure loss by flow separation and increased the mass flow inside the diffuser. Based on the same idea, a group of researchers in New Zealand [8] in early 2000's developed the Vortec 7, large-scale wind turbine prototype system with diffusers. Vortec 7 DAWT design turned out to be commercially unviable. Bet and Grassmann [9, 10] have tried to apply the boundary layer control principle to the small-scale DAWTs. A shrouded wind turbine with a wingprofiled ring structure around it is discussed in [9]. Numerical studies by Bet *et al*. [9] and Grassmann *et al*. [10] showed that aerodynamic features can boost the power output by a factor of two and five respectively. Subsequent experimental results showed an increase in the voltage output by 25%, suggesting a power increase, but the power output was not reported [10]. Gaden and Bibeau [11] tried to study the boundary layer interruption with a shaped object positioned upstream of the hydro turbine. They found a power increase of about 15%.

Ohya and Karasudanisuccessfully tested "the brimmed diffuser" [12]. For micro--scale turbine it provided a remarkable increase in the output power of approximately 4-5 times that of a conventional wind turbine. Due to a strong vortex formation behind the broad brim a low-pressure region draws more mass flow to the wind turbine inside the diffuser. Although these are just a few of the large number of studies researching the possibility of wind power augmentation published so far, only Ohya's attempt appears to have reached relatively successful commercialization [12].

Ponta and Jacovkis [13] at the University of Buenos Aires have used diffuser-shaped islands in laboratory tests to increase the power output of a hydrokinetic turbine. They found that the channeling device can significantly increase the power output and stability for low current speeds. It also provides buoyancy and serves as a pontoon. The authors of this state-of-the-art technology developed the first water current turbine design that employs the channeling of the flow as a way to improve hydro turbine performance.

Methodology

The present study focuses on determining how effectively fluid energy can be collected in the throat of the channeling device using the system of 4 wing-shaped airfoils for both water and air current. It is of particular interest to us to test the effect of jointly using several such accelerating systems of different sizes on the flow. The main aim of this study is to explore the possibility of a kinetic energy flux increase in hydrodynamic flows employing the cumulative phenomenon in the guiding surfaces system.

It was suggested using the system of specifically constructed guiding surfaces to re-direct the fluid, thus, locally modifying the flow and increasing its speed and kinetic energy flux in the throat of the channeling device. The geometry of such a system is presented below.

The present research is aimed at the development of a channeling device that would amplify the power output of a rotor of a given size, or allow to the same power output to be obtained from a smaller turbine.

Optimization of the kinetic energy flux is based on theoretical evaluations and predictions, 2D and 3D numerical simulations, and laboratory experiments.

The usable accelerating structure has been developed. It can be manufactured without difficulty using inexpensive and easily handled materials.

Design features of the flow booster

The construction under study consists of 4 airfoils optimized usinga series of numerical experiments (Figure 1). Two internal air foils (placed closer to the system symmetry plane) have a lenticular shape and the two external ones resemble more traditional Zhukovsky air foils. When the fluid passes through the system, the pressure in the narrow cross-section between the internal air foils decreases, thus providing an increase in speed according to Bernoulli's principle.

As a result, the water kinetic energy increases which can be used to develop torque in a reaction turbine. The flat geometry of the system is selected as the most suitable for the collection and utilization of the kinetic energy of the flow elongated in one direction (e.g. river, channel, etc.).

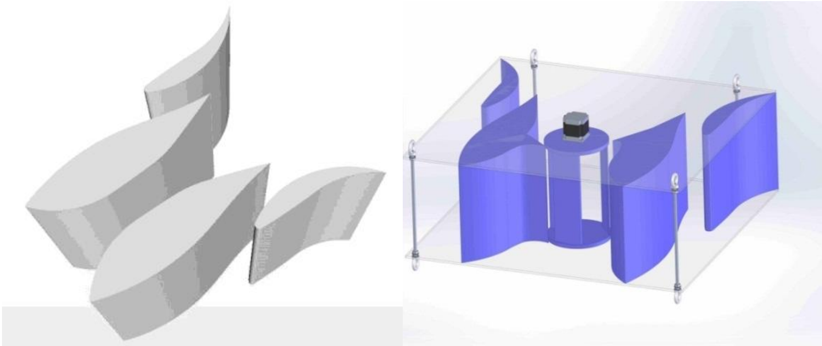

Figure 1. Accelerating system, one of the hydro turbine projects presented on the right

LAB EXPERIMENT

A laboratory tank made in the shape of a vertical rectangular channel is developed for the experimental study [14] (Figure 2). It provides a uniform laminar flow of water for up to 10 seconds. Water flows at about 15 cm/s. The flow is created once the water starts flowing through a drain hole in the bottom of the tank. Apart from the experimental system shown in Figure 1, a larger system has also been considered. Numerical simulations for the larger system have shown the possibility of 3-4 times velocity increase in a single set of guiding surfaces (one “cascade”) with the kinetic energy flux increase up to 20-25 times. The geometry of the experiment is shown below in Figure 2.

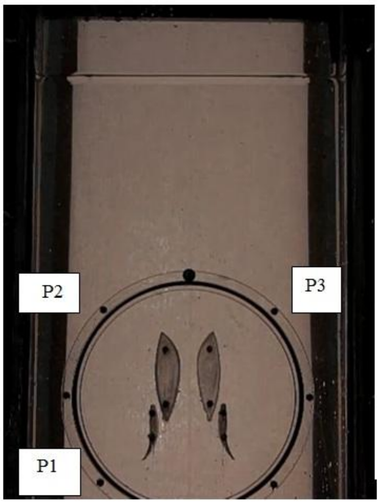

Figure 2. Laboratory tank

Points P1, P2 and P3 have the following coordinates (in millimeters): P1 = (30, 6); P2 = (34, 198); P3 = (225, 198), distance between the points P1 and P2, P2 and P3 equals 192 mm.

Experimental results

The laboratory experiment gives a velocity increase coefficient of 2.2 and an area of high velocity of about 6 cm in length. The detailed analysis of the experimental technique and the obtained results are given in [15].

Vertical coordinates of two particles of the pack and the water surface coordinate are presented below (Figure 3).

The visualization method is, to a large degree, nonintrusive. The added tracers (if they are properly chosen) generally cause negligible distortion of the fluid flow [16-17].

The method is capable of measuring an entire two-dimensional cross section (geometry) of the flow field simultaneously.

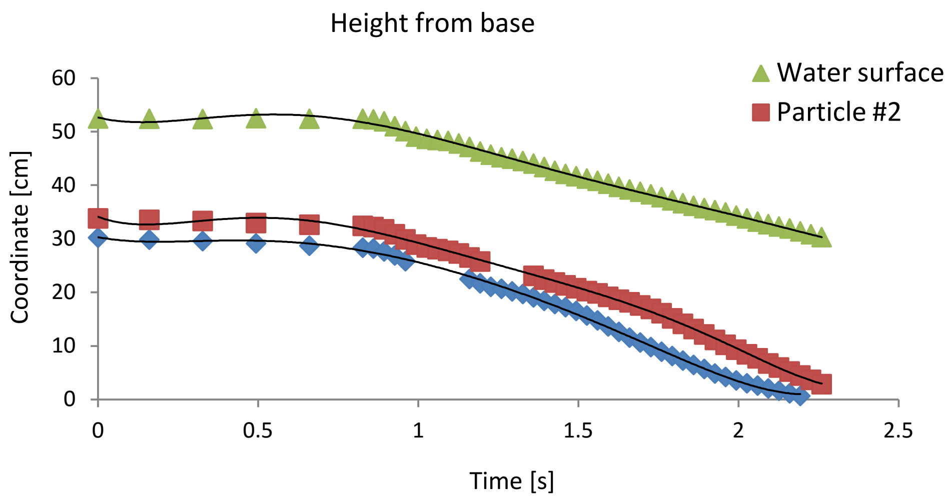

Figure 3. Vertical coordinates of the particles

The blue curve represents the measurements of the first particle's coordinate with the red curve being the coordinate of the second particle. The light green curve indicates the water level in the lab tank.

The biggest drawback of the method is that the utilized two-dimensional modification of particle image velocimetry methods is not be able in general to measure components along the z-axis (towards or away from the camera).

Numerical modeling of the lab experiment

In this part we investigate how large the water velocity increase between the internal shells of the channeling device could be in the numerical simulation of the fluid flow in the laboratory construction. Flow-speed increment and velocity peak localization are of particular interest.

The experimental results presented above were obtained for a small laboratory model (about 10 cm from the front edge to the rear end), the fluid velocity reaches 30-31 cm/s in the narrowest part of the system. This section is devoted to a discussion of the numerical modeling results for the same system. A commercially available finite volume CFD solver STAR-CCM + was used for 2D and 3D numerical simulations of the flow [18].

Steady computations of the laminar flow were performed on the grid consisting of about 0.6 million cells. The computational grid is presented in Figure 4.

The fluid outflow from the experimental tank opened to the atmosphere is calculated. Following the lowering of the liquid level, the air fills the space above the water. The accelerating system of surfaces is installed inside the tank. The computational domain is divided into two separate parts: the upper one, which is filled with water and the lower one is filled with air at the initial moment.

The atmospheric pressure is given on the surface $P \approx 10^5$ Pa. The boundaries between media are marked by the change of phase concentration "water" and "air") from 0 to1 or from 1 to 0.

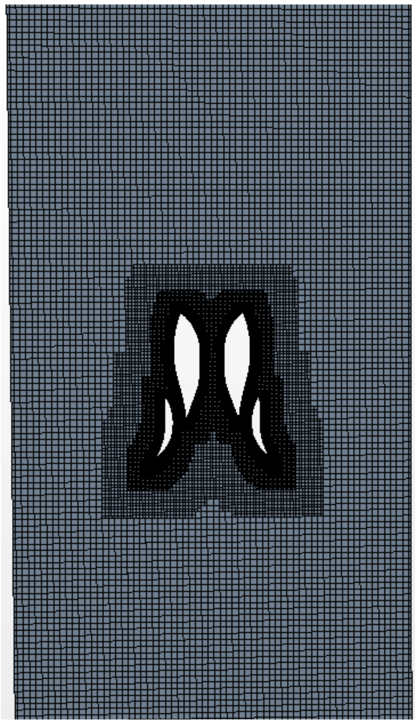

Figure 4. Example of the computational mesh used for the numerical modeling

The in-place interface providing a connection between boundaries during the meshing and analysis without physical separation is created on the border between the upper and lower areas where the drain hole is placed.

All the physical options of the CFD solver chosen for the simulation are listed in Figure 5.

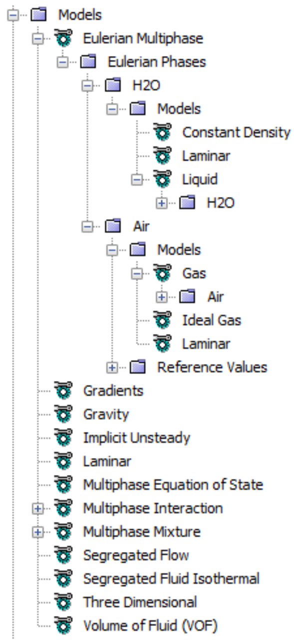

Figure 5. All the physical options of the CFD solver chosen for the simulation

The mathematical model, numerical procedures, and the computer codes are verified using experimental results.

Numerical results. In the calculations carried out for a small laboratory system of surfaces (about 10 cm from the front edge to the rear end surfaces placed inside the laboratory tank), the fluid velocity in the narrow space between the inner air foils reaches 33 cm/s while the water surface velocity equals 12 cm/s (Figure 6).

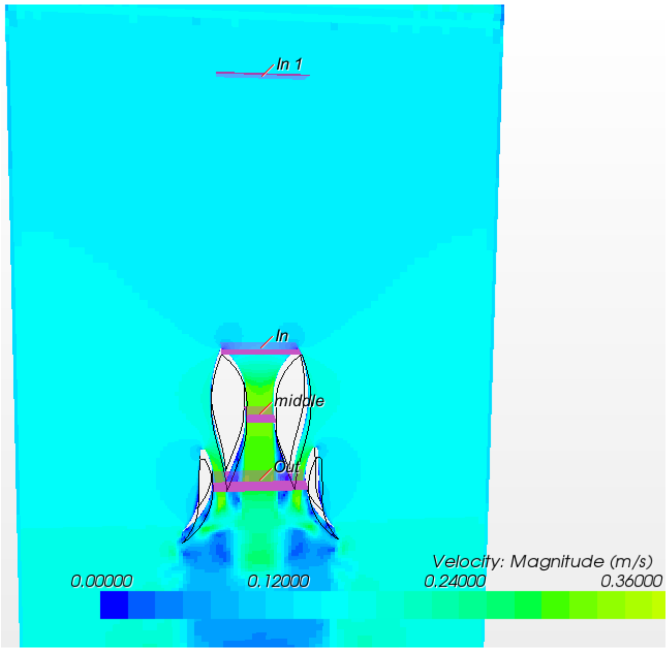

Figure 6. The water velocity in the lab tank

Figure 7 represents the kinetic energy flux through the planes shown in Figure 6 (In, In1, Middle, Out). The constant water velocity can be estimated from the mass flow through the drain hole in the bottom. It was mentioned earlier that the water velocity starts from the value of about 15 cm/s and slightly descends with time to 14.5 cm/s to the moment shown in Figure 6.

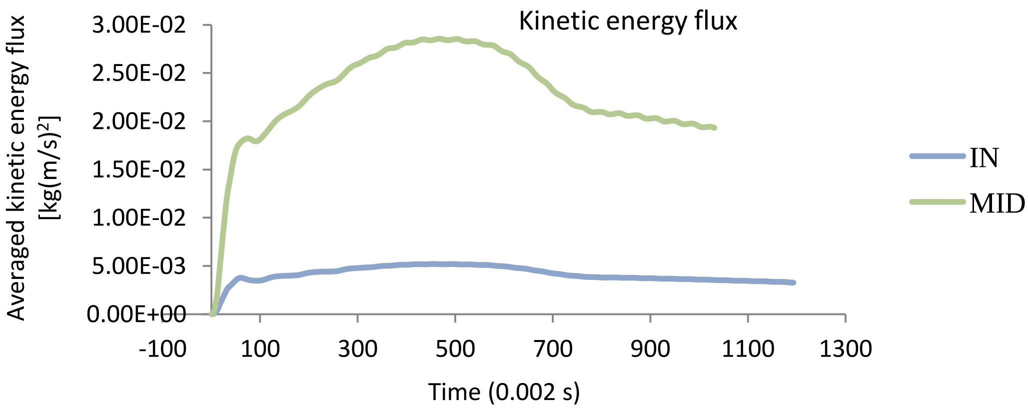

Figure 7. Kinetic energy flux through the planes IN and MID shown in Figure 5

Figure 7 compares the kinetic energy flux through the plane In1 ("far away from accelerating system") and Middle ("the highest acceleration spot inside the system"). The calculations reveal a velocity increase of up to 2.5 times, reaching values of 35 cm/s (Figure 6) while the kinetic energy flux increases 5 times (Figure 7).

Numerical results demonstrate good agreement with the experiment.

Optimization results. The results of the numerical calculations for a larger geometry are significantly different from those of the laboratory system that the numerical modeling discussed earlier. Typical geometry of such simulations is shown in Figure 8.

The flow velocity increase is from 6 to 7 times (Figure 8), and the kinetic energy flux increases 40-50 times (see Figure 9). The prototype of the channeling device that we are aiming to test in the future is 40 times larger than the lab model depicted in Figure 2. Further numerical simulations for this prototype were carried out. Our objective was to find the velocity increase dependence in the throat of the system on the relative distances between the surfaces. In all of this simulation, we assume the flow upstream from the device being laminar. The velocity of the wind equals 10 m/s.

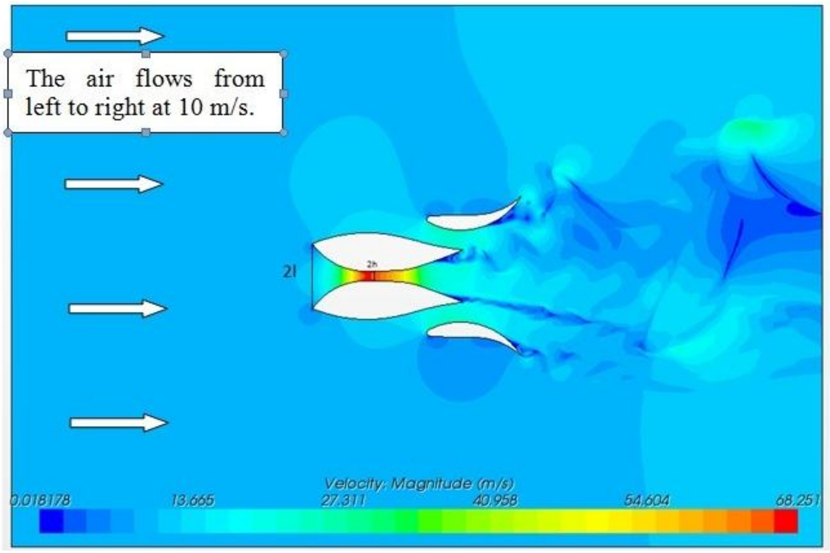

Figure 8. Typical geometry of the accelerating shroud in numerical simulations

Figure 9 shows the dependence of kinetic energy flux ratio far away from the system and in the highest velocity spot from the system "transparency coefficient" $K_{transp} = h/l$. Thus, when $h/l = 0.2$, kinetic energy flow ratio reaches it's maximum provided that the other dimensions of the system remain constant.

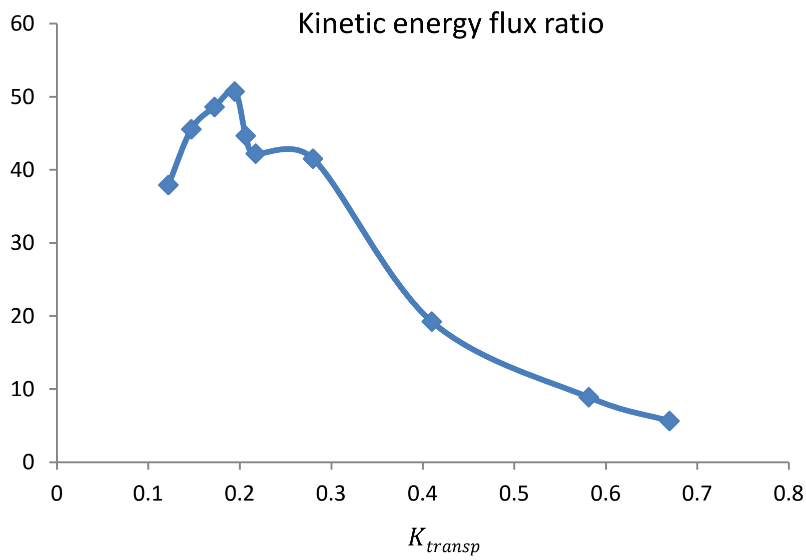

Figure 9. Kinetic energy flux ratio depending on the $K_{transp} = h/l$

Convergence. The calculations on a finer grid (consisting of 1.9 million cells instead of 0.6 million) show increase in speed for the same geometry of the problem. This can be explained by a decrease in numerical viscosity, i.e. the effective viscosity coefficient for the problem with a finer grid is closer to it's real physical value. It is important to note that the numerical viscosity in the simulations has a very little impact on the result. It means that the result of the calculation may be used as an estimation for the fluid velocity increase. The difference in the results can be explained by a change in flow configuration and regimen of the stream due to a substantial increase in the Reynolds number $Re = vL/\nu$.

FIELD TESTS AND 3D SIMULATIONS

The next phase of work involved field experiments conducted using the accelerating system prototype developed by ITC Sistema-Sarov engineers, as well as a numerical solution of corresponding two-dimensional and three-dimensional problems. The prototype under study is 5 times larger than the laboratory model: the inner air foil length from the front to the rear edge is approximately 50 cm, same as the prototype's height – about 50 cm.

In the field tests, the accelerating prototype equipped with appropriate mounting hardware is placed in the river flowing at about v_0 = 0.58 m/s (Figure 10). The rods with marks on them are used for the indirect measurement of the fluid velocity via the liquid level difference in the corresponding areas:

$$\rho g \Delta h = \rho \frac{{v_1}^2 - {v_0}^2}{2} \Longrightarrow {v_1}^2 = {v_0}^2 + 2g\Delta h \tag{1}$$

Figure 10. The general view of the accelerating system in the river

Depending on the distance between the inner air foils the velocity increase of the fluid averages from 1.5 to 2 times.

The water level locally drops due to the change of speed (and hence the pressure) in the stream. The surface in three-dimensional calculations with gravity sets the pressure profile, using which it is easy to estimate the local velocity with the help the above formula ${v_1}^2 = {v_0}^2 + 2g\Delta h$. The water level difference in this case is approximately 3.5 cm. The maximum velocity is about 1 m/s, i.e., the speed in this case increases 1.7 times (Figure 11a and 11b).

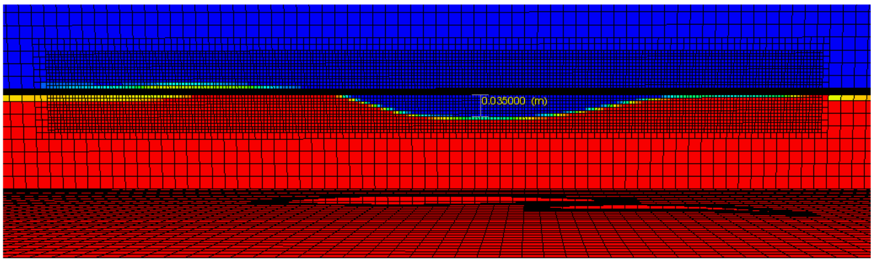

Figure 11a. 3D simulations of the water flow in the accelerating system (field test)

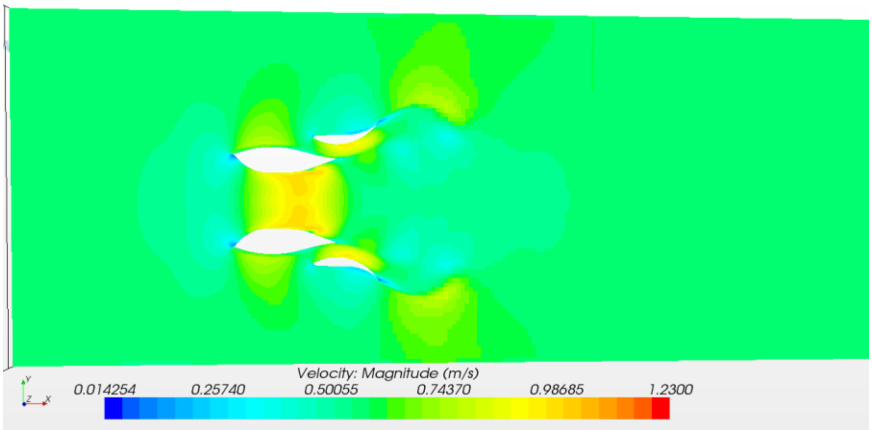

Figure 11b. 3D simulations of the water flow in the accelerating system (field test)

Further numerical testing of the channeling device

The possibility of kinetic energy local redistribution and summation was further tested in the following series of simulations. Air flows at 10 m/s. The 2D simulations with several accelerating systems – one placed inside the other – show 12.5 times speed increase compared to 6-7 times increase for a one cascade (Figure 12). It becomes possible to accelerate water even more than in a one-cascade system.

Multiple cascade systems are full of potential for practical use. This structure is capable of collecting even more energy and increase the flow speed by a factor of 10-12. The multicascade method would potentially allow using smaller, more compact and cheaper turbines with higher speed flow instead of slower and bigger ones.

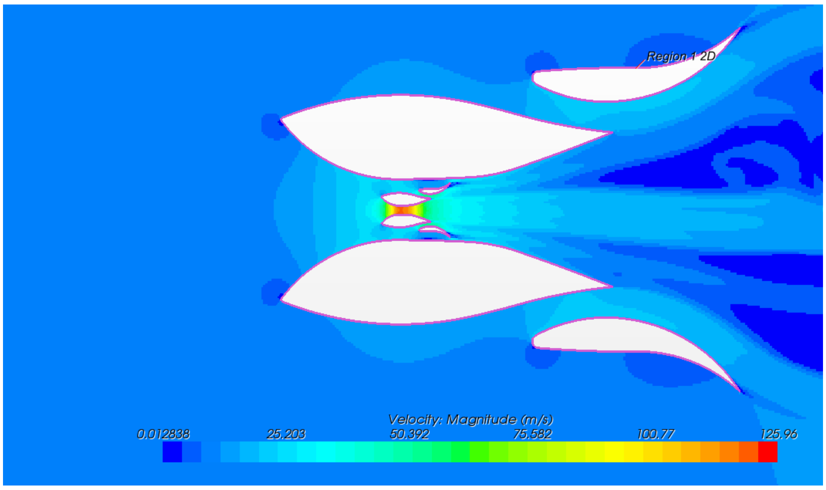

Figure 12. Multicascade flow accelerating system

2D simulations for a shrouded turbine

We conducted a series of simulations for the turbines of various shapes installed in different locations inside the accelerating system. Below are two different examples of such simulations.

Incident water flow speed is 2 m/s, the shroud is 0.5 m long and the distance between the frontal edges is 0.5 m. Rotational speed equals 45 rad/s. The Betz coefficient was chosen to be equal to 0.4, slightly higher than a practical value range, so the estimate can be considered as an upper limit for the power output.

In the first case, axisymmetric four-blade turbine (turbine radius r = 0.25 m) is installed behind the narrowest spot between the first pair of lens-shaped surfaces (Figure 13).

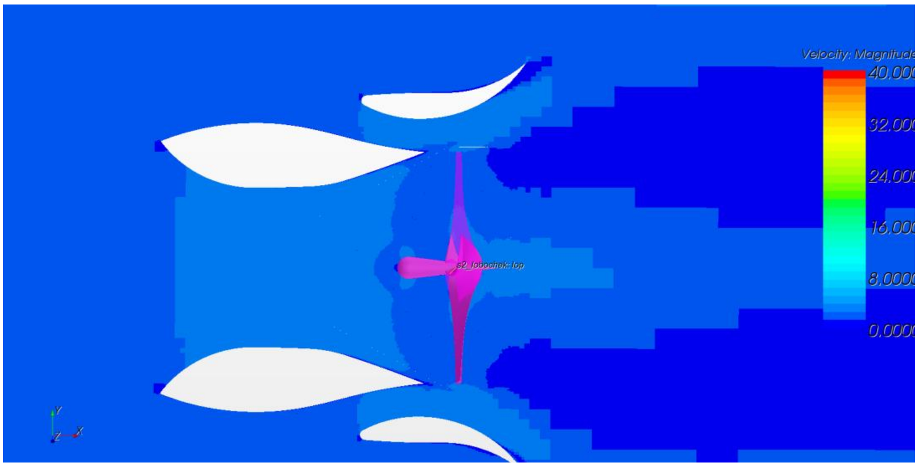

Figure 13. The turbine is installed behind the lenticular shaped wings, water flows from left to right at 2 m/s

Power output in this case equals P = 635 W (Figure 14). Theoretical power output estimate for a bare turbine:

$$P = \eta \frac{1}{2} \rho v^3 S = 0.4 \times 0.5 \times 1000 \frac{\mathrm{kg}}{\mathrm{m}^3} \times 8 \ \frac{\mathrm{m}^3}{\mathrm{s}^3} \times 3.14 \times 0.25^2 \mathrm{m}^2 \approx 314 \ \frac{\mathrm{J}}{\mathrm{s}} = 314 \ \mathrm{W} \tag{2}$$

So the power again increases to 314 W, that is a factor of 2.0, compared to the bare turbine.

Figure 14. Power output for the 1st case

In the second case, axisymmetric four-blade turbine (of slightly smaller radius r = 0.13 m) installed to the narrowest spot inside the plane-parallel shroud (Figure 15):

$$P = \eta \frac{1}{2} \rho v^3 S = 0.4 \times 0.5 \times 1000 \frac{\text{kg}}{\text{m}^3} \times 8 \ \frac{\text{m}^3}{\text{s}^3} \times 3.14 \times 0.13^2 \text{m}^2 \approx 85 \ \frac{\text{J}}{\text{s}} = 85 \text{ W} \quad (3)$$

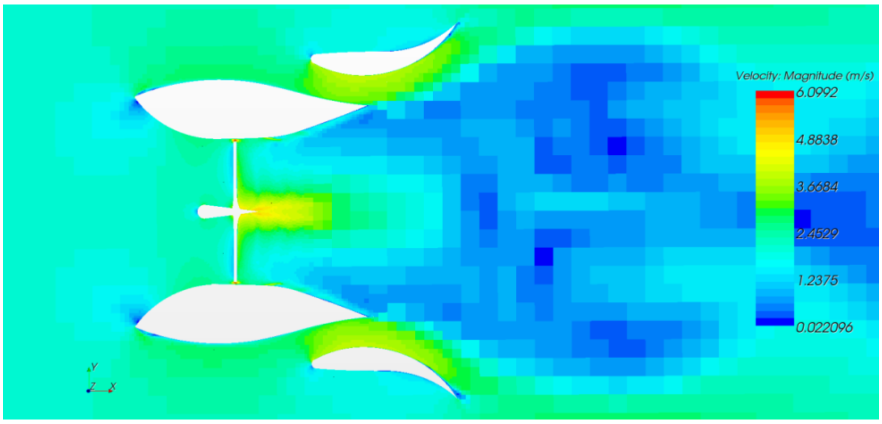

Figure 15. The turbine installed in the throat of a channeling device, water flows from left to right at 2 m/s

The numerical simulation gives the power output of P = 435 W which is more than 5 times higher than the theoretical estimate (Figure 16) for a bare turbine. The second case perfectly illustrates the positive effects of the accelerating system use. By concentrating the flow power in the channel one can reduce the size of moving parts of turbine. It could potentially simplify the design process and reduce the initial cost. For the same current speed reduction in the turbine runner size means higher rotational speed [13]. The introduction of the accelerating device makes the velocity in the neighborhood of the rotor 1.5-2 times higher than the incident current speed. This results in a remarkable increase. The introduction of channeling device makes the turbine smaller, more compact and potentially much more economically efficient in terms of price/power ratio while keeping power output the same.

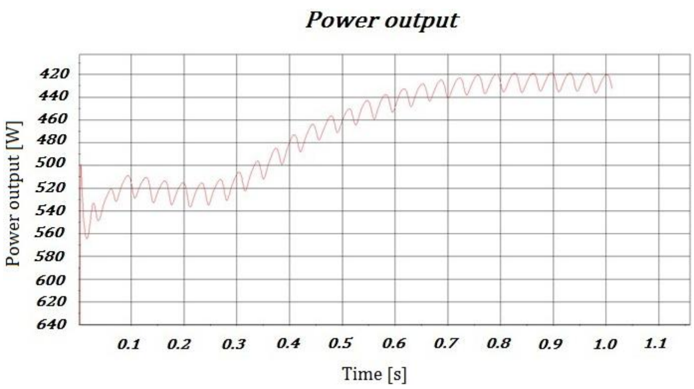

Figure 16. Power output for the 2nd case

CONCLUSIONS

The key purpose of this study was to develop a compact, autonomous and low-cost device that would make both wind and running water power production more efficient. The efficiency of air and water power facilities strongly depends on the kinetic energy flux and the velocity in the flow. A high-velocity wind or water flow plays a critical role in creating an efficiently working turbine.

The revealed numerical results are very promising and the performed experiments evidently illustrate the positive effects of the accelerating system use so far. However, more experiments are required for a systematic study of the flow acceleration inside the system, but the results in this paper demonstrate that a significant power augmentation on turbine blades could be achieved using a fairly compact booster.

Summing up the results of our study, we should mention the most significant ones:

- The experiment shows good agreement with the numerical result;
- Impressive numerical results of the water acceleration are promising so that the resulting velocity and the flow kinetic energy inside the accelerating system can be high enough to be utilized for power generation;
- The numerical results for the two cascades combined prove the possibility of additional water acceleration employing cumulative phenomena in the system of guiding surfaces. By placing one cascade inside the other one creates a multistage flow accelerator. Thus, the cumulative phenomenon is being applied to locally increase the kinetic energy flux of a free flow;
- Numerical results for the turbine installed inside the accelerating system show a significant increase in power output – up to 5-6 times. Although much work remains still to be done, the overall result is encouraging. The results of numerical simulations demonstrate that the shrouded turbine could be of substantial advantage compared to conventional turbines from an economic point of view.

To our knowledge, there has been no research work published on the use of cumulative effect in the system air foils. In our opinion, future research of multistage flow accelerating systems appears to have a particular promise for making a contribution to the field of small wind and hydro turbines.

ACKNOWLEDGEMENTS

We would like to express our most sincere gratitude to Dr. Dmitry Zhernokletov for helping us to revise the English manuscript. We are deeply appreciative of the constructive criticisms of the anonymous reviewers and the editor. We would like to express our appreciation for their valuable remarks and recommendations that helped to make the paper much more intelligible.

REFERENCES

1. Dwivedi, A. K., Raja, A. P. and Srivastava, M., Power Plant Engineering, New Delhi: New Age International, 354 p, 2006, ISBN 81-224-1831-7
2. Betz, A., *Introduction to the Theory of Flow Mechanics* (Translated by Randall, D. G.), Pergamon Press, 1966.
3. Lilley, G. M. and Rainbird, W. J., A Preliminary Report on the Design and Performance of Ducted Windmills, Cranfield College of Aeronautics Report 102, Bedford, UK, 1956.
4. Sanuki, M., Studies on Biplane Wind Planes Ventilation Tubes and Cup Anemometers, *Meteorol. Geophys. Jpn.*, Vol. 1, No. 1, pp 279-290, 1950, http://dx.doi.org/10.2467/mripapers1950.1.1_81
5. Frankovic, B. and Vrsalovic, I., New High Profitable Wind Turbines, *Ren. Ener*, Vol. 24, No. 3-4, pp 491-499, 2001, http://dx.doi.org/10.1016/S0960-1481(01)00033-7
6. Gilbert, B. L., Oman, R. A. and Foreman, K. M., Fluid Dynamics of Diffuser-augmented Wind Turbines, *J. of Energy*, Vol. 2, No. 6, pp 368-374, 1978, http://dx.doi.org/10.2514/3.47988

7. Igra, O., Research and Development for Shrouded Wind Turbines, *Energ. Conv. Manage.*, Vol. 21, No. 1, pp 13-48, 1981, http://dx.doi.org/10.1016/0196-8904(81)90005-4
8. Phillips, D. G., An Investigation of Diffuser Augmented Wind Turbine Design, *Ph.D. Thesis*, University of Auckland, Auckland, New Zealand, 2003.
9. Bet, F. and Grassmann, H., Upgrading Conventional Wind Turbines, *Renew. Energy*, Vol. 28, No. 1, pp 71-78, 2003, http://dx.doi.org/10.1016/S0960-1481(01)00187-2
10. Grassmann, H., Bet, F., Ceschia, M. and Ganis, M. L., On the Physics of Partially Static Turbines, *Renew. Energy*, Vol. 29, No. 4, pp 491-499, 2003, http://dx.doi.org/10.1016/j.renene.2003.07.008
11. Gaden, D. L. F. and Bibeau, E. L., A Numerical Investigation into Upstream Boundary Layer Interruption and its Potential Benefits for River and Ocean Kinetic Hydropower, *Renew. Energy*, Vol. 35, No. 10, pp 2270-2278, 2010, http://dx.doi.org/10.1016/j.renene.2010.03.007
12. Ohya, Y. and Karasudani, T., A Shrouded Wind Turbine Generating High Output Power with Wind-lens Technology, *Energies*, Vol. 3, No. 4, pp 634-649, 2010, http://dx.doi.org/10.3390/en3040634
13. Ponta, F. L. and Jacovkis, P. M., Marine-current Power Generation by Diffuser-augmented Floating Hydro-turbines, *Renew. Energy*, Vol. 33, No. 4, pp 665-673, 2008, http://dx.doi.org/10.1016/j.renene.2007.04.008
14. Bashurin, V. P., Budnikov, V. P., Fedorenko, Y. V., Klevtsov, V. A., Ktitorov, L. V., Lazareva, A. S., Meshkov, E. E., Novikova, I. N., Pletenev, V. A. and Yanbaev, G. M., Hydrodynamic Test Bench, *Technical Physics*, Vol. 60, No. 7, pp 1093-1095, 2015, http://dx.doi.org/10.1134/S1063784215070063
15. Bashurin, V. P., Budnikov, V. P., Fedorenko, Y. V., Klevtsov, V. A., Ktitorov, L. V., Lazareva, A. S., Meshkov, E. E., Novikova, I. N., Pletenev, V. A. and Yanbaev, G. M., Wind or Water Turbine Power Augmentation using the System of Guiding Surfaces, *Phys. Scr.*, Vol. 91, No. 4, 2016, http://dx.doi.org/10.1088/0031-8949/91/4/044002
16. Raffel, M., Willert, C. E., Werely, S. T. and Kompenhans, J., *Particle Image Velocimetry: A Practical Guide*, Springer Science & Business Media, 468 p, 2007.
17. Melling, A., Tracer Particles and Seeding for Particle Image Velocimetry, *Measurement Science and Technology*, Vol. 8, No. 12, pp 1406, 1997, http://dx.doi.org/10.1088/0957-0233/8/12/005
18. Star-CCM+ 8.02 User Guide.

Paper submitted: 22.01.2016
Paper revised: 30.04.2016
Paper accepted: 04.05.2016