ANALELE UNIVERSITĂȚII
"EFTIMIE MURGU" REȘIȚA
ANUL XX, NR. 1, 2013, ISSN 1453 - 7397

Florin Pomoja

Computer Simulation of Water Flowing inside Turbine HYE 10a, a Modern Method for Checking of the Structural Improvements

This paper presents theoretical results of flow simulations, performed on cross-flow type turbine of 8.25 kW, and also the results obtained on the improved turbine, with modified shapes for the rotor, case, and guiding palette.

***Keywords**: Computational Fluid Dynamics, cross-flow turbine, 8.25 kW*

1. Introduction

Computational **F**luid **D**ynamics (on short CFD) is a process of flow simulation for a fluid (gas or liquid), made with specialized applications using numerical methods of calculation.

The programs used in fluid mechanics are based on certain algorithms to analyze and then solve user requirements in terms of flow simulation, or more precisely the interactions between fluid (gas or liquid) and surrounding bodies. These bodies (be it for pallets, pipes, casings etc.) are configured in CFD by surfaces defined in their turn by *boundary conditions*.

Flow analyses presented in this paper were performed using the TascFLOW CFX, ANSYS software package v.12. There have been two simulations: one for the original version and other for the proposed, improved version of the turbine.

2. The two versions of the turbine

In the case of the original turbine, the angle between the axis of the injector and the axis of suction pipe is 90°, and the suction pipe axis is vertical. For proposed case the angle between the two axes was increased to 108°, and the axis of the suction pipe been deviated from vertical with an 18° angle - Figure 1. In the figure the damper is shown in the fully closed position.

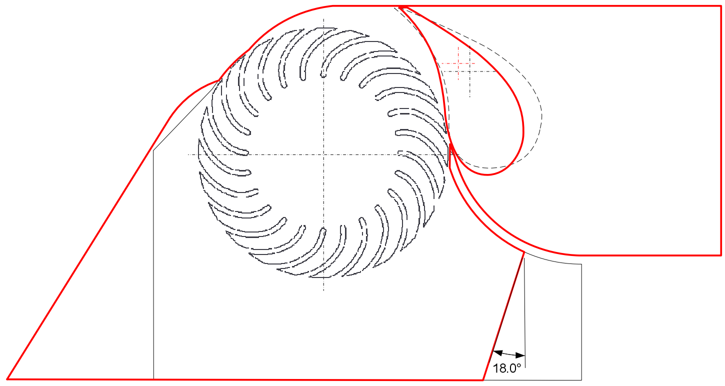


Figure 1. Shapes of the initial turbine (black) and the shapes of the proposed turbine (red), with new damper in fully closed position.

We gave up at the parallelism between the two walls of the turbine case, and we proposed an angle of +14° between them – as shown in Figure 2. In that figure the damper is shown in the fully open position.

Value of 14° for the angle between the walls turbine is considered to be optimal in terms of water discharge.

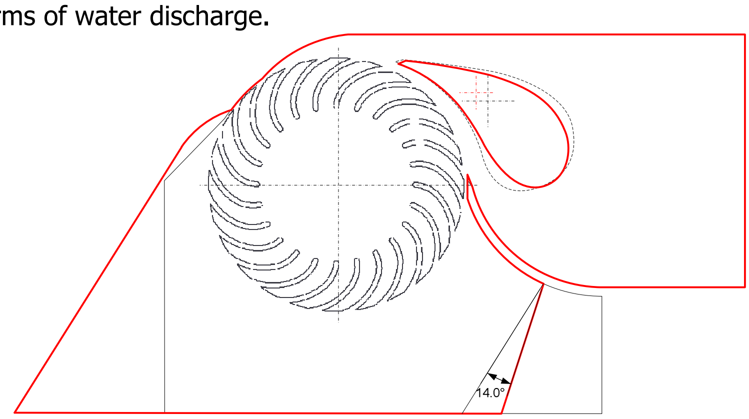


Figure 2. Shapes of the initial turbine (black) and the shapes of the proposed turbine (red), with new damper in fully open position.

In the Figures 1 and 2, the shape of the initial turbine case is drawn with black, and the new shape is drawn with red. The same is the situation for the damper shape.

Also, as shown in the two figures, the volume of the chamber in front of the case fell through constructive geometry changes instead increased in the opposite side, with about 18% for better drainage. We do so and because the simulations of flow presented below shown us that this area is characterized by an intense water recirculation phenomenon.

New form of case has been modified because we want to reduce the friction between running water and the case walls. In this way it is intended and possible to increase the rotational speed of the rotor (it is known that cross-flow turbines increased rotor angular speed leads to efficiency grow).

Another modification of the constructive shapes was the removal of the mounting shaft between the two discs. This shift was replaced with two exterior half-shafts, attached to two flanges fixed on the outer surface of discs.

The cross-section of the damper also was decreased - by decreasing her thickness from 76 mm to 59 mm - with 22.36% (see Figure 3), to achieve a better shape in terms of hydrodynamic profile. When amending the shape of the damper was taken into account and the changes of its axis of rotation, so that it fully closes the inlet inside the turbine case.

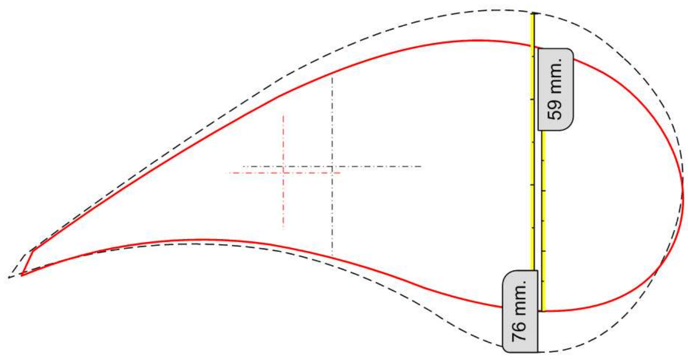


Figure 3. The shape adopted for the new damper.

The new damper shape is closer to shape of a drop of rain, which - as is known - is the form with the lowest aerodynamics coefficient. In this way we intended to reduce the friction between the water jet and the damper, and the secondary effect was the increase of amount of water that enters through the bottom of the damper (control valve).

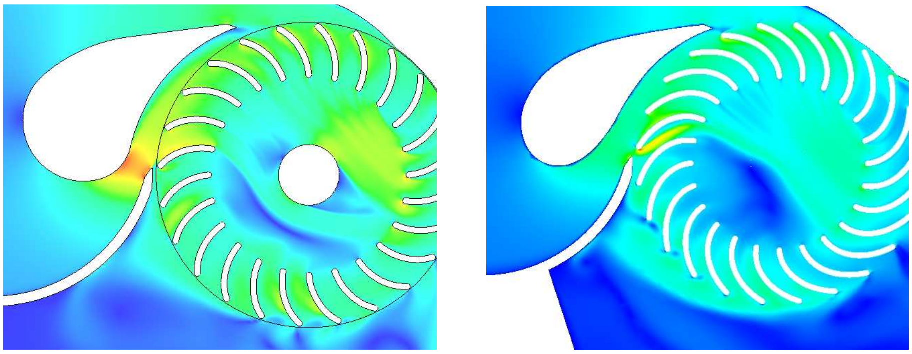
3. The results of simulations - comparative examples

After examining the results of computer simulations of water flow through the two turbines under review, the following conclusions could be drawn:

1) The relative velocity of the water is more uniform when enter into the intra-palletizing area, as a result of improved shape of the damper (Figure 4).

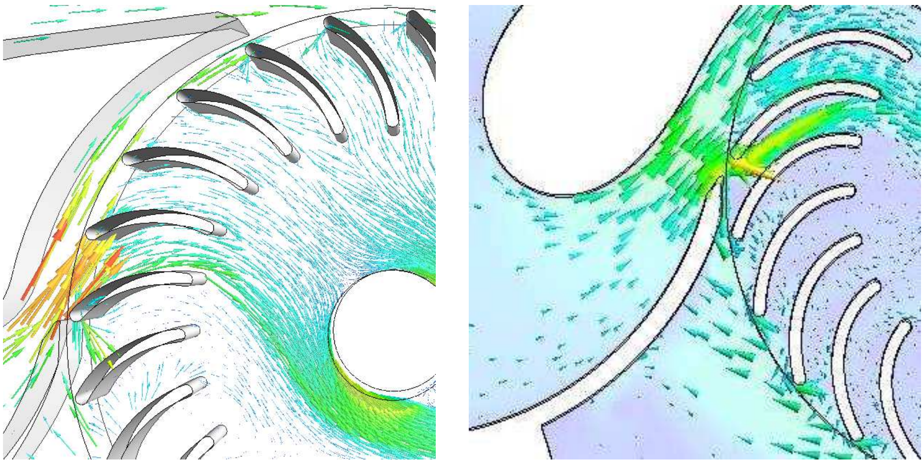
Due to change of the case shape and remove of the shaft between the two discs, inside the case appeared an improvement in water flow.

2) For the proposed rotor we see that there is a uniform field of the velocity of water in the core zone of the rotor, as shown in Figure 5.



a) b)

Figure 4. The relative velocity of the water - compared:
a) initial rotor; b) proposed rotor.

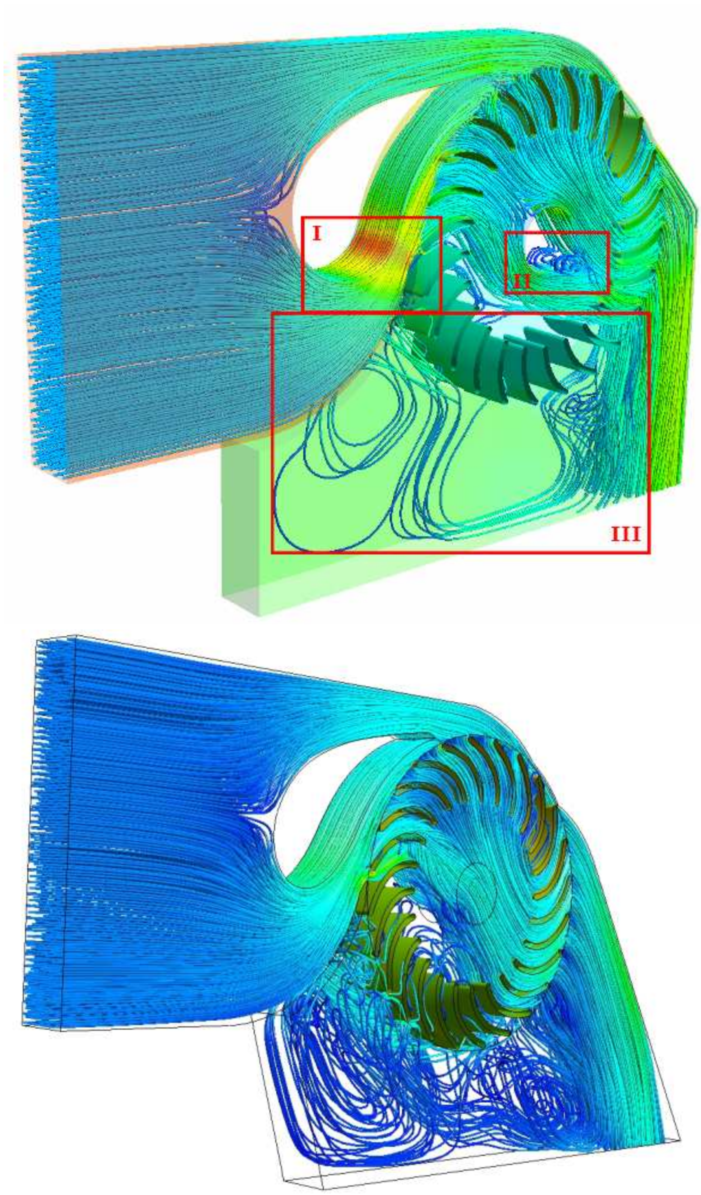


a) b)

Figure 5. Vector representation of the water velocity – compared:
a) initial rotor; b) proposed rotor.

3. Distribution for water flow lines in the existing turbine is shown in Figure 6, which revealed three "sensitive" areas in terms of flow lines:

- in zone I, where is a large increase of velocity;
- in zone II are seen turbulences concentrated in a very small space;
- in zone III (leaving water phase) are two areas of stagnation.



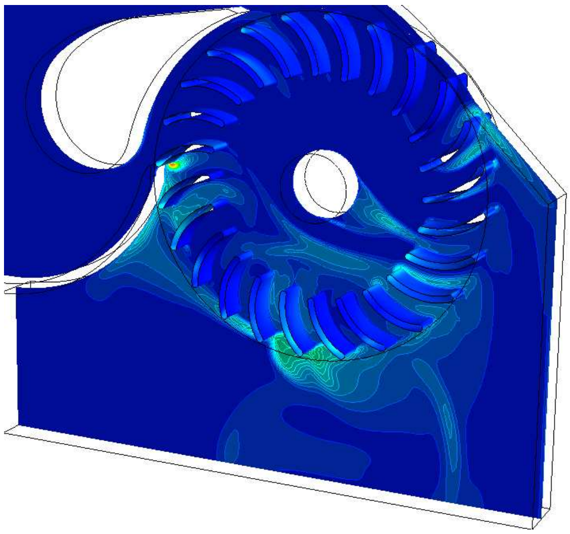
a)

b)

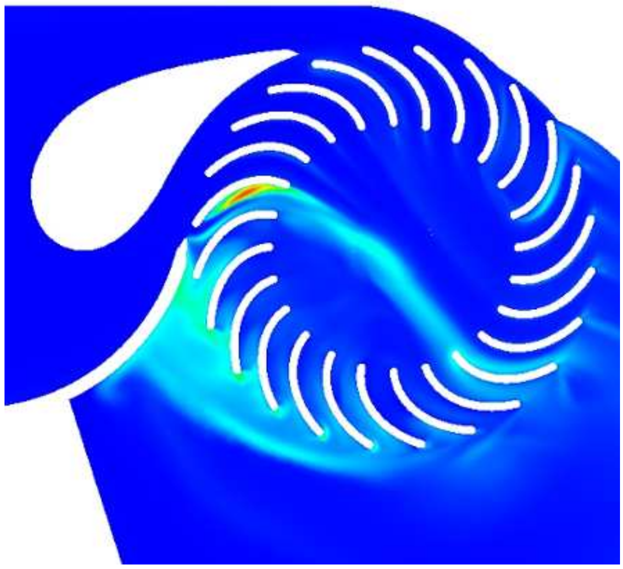
Figure 6. Distribution of the flow lines - compared:
a) initial rotor, b) proposed rotor.

In Figure 6 b) can be observed an enhanced water flow, for turbine proposed versus the original version of turbine, for all the three areas.

4. An improved situation for proposed version of turbine can be seen in Figure 7 b), in terms of the distribution of turbulent kinetic energy of water in the turbine casing.



a)



b)

Figure 7. Distribution of turbulent kinetic energy of water in the turbine casing, – compared: a) initial rotor; b) proposed rotor

Losses of the turbulent kinetic energy at the bottom of the existing turbine rotor were transferred only on two blades and in the water jet, as seen in Figure 7 b). So in the case of the proposed turbine rotor, the turbulent kinetic energy losses were reduced.

4. Conclusions

Although eliminating the internal shaft itself is not a new idea, the proposal of the author proved to also be a contributing factor to improve water flow through the rotor proposed, taking into account the particularities of turbine HYE 10. Simulations were performed for values of flow and water head identical to those where actual measurements were made, at the same rotational speed.

Table 1 presents the results obtained in terms of turbine efficiency, calculated by TaskFLOW CFX from ANSYS application, for flow simulation on the two variants of the turbine: initially and proposed, both with the same variant of blade on rotors.

Table 1.

No.	Flow Q	Head H	Rotational speed n	Efficiency η	
				Initial rotor	Proposed rotor
	[m^3/s]	[m]	[rot/min]	–	
1	0.015	32		0.797749	0.812063
2	0.016	35		0.801336	0.821605
3	0.017	40		0.824986	0.843838
4	0.018	45		0.827256	0.847331
5	0.020	50	1.885	0.822548	0.842507
6	0.021	55		0.816159	0.836047
7	0.023	65		0.796569	0.815017
8	0.025	75		0.773581	0.789351
9	0.027	88		0.745555	0.763942

In the table have been highlighted the highest values of simulations for the efficiency, and it is relevant that the modified version of the turbine has higher values in all cases. The highest values for efficiency – namely 82.72% for initial version of the rotor, and 84.73% for proposed variant of the rotor - were obtained at a rate of 0.018 cube meter of flow and a head of 45 m, and this is the optimal point of turbine operation.

With Table 1 data, we draw diagrams shown in Figure 8, in which they appear turbine efficiency according to water flow, for the two types of rotor: initial and proposed.

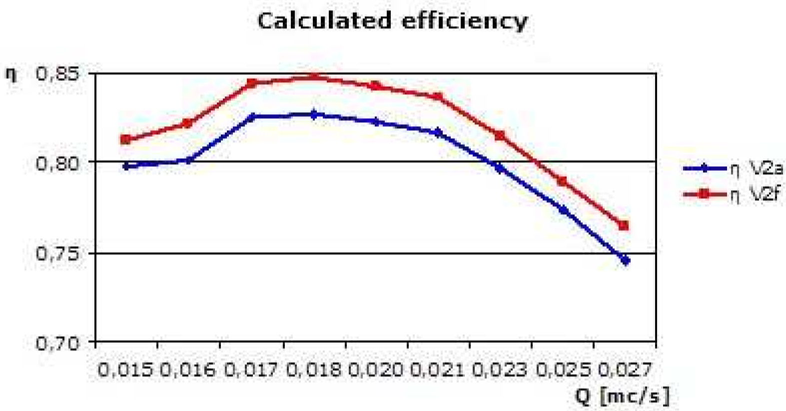


Figure 8. Calculated efficiency, according to flow

We consider that the occurrence of maximum efficiency at 0.018 m^3/s flow and 44 m water head was expected, because the turbine was designed for a maximum flow of 20 l/s, and a head of 50 m.

As seen from the diagram, the two efficiency curves have a similar allure for the all nine operating points. The turbine efficiency is improved with a maximum of 2.01 percent.

While this amount may seem insignificant, for the cross-flow turbines the value itself is still a significant increase.

References

[1] Pomoja F., Mănescu T. Şt., Hopotă A., *Improving Efficiency of an Micro-hydro Aggregate using a Banky 10 kW Micro-turbine, by Blade Design Optimization,* Proceedings of 7th International Symposium Machine and Industrial Design in Mechanical Engineering – KOD 2012, 24-26 May 2012, Balatonfüred.

[2] White F. M., *Fluid Mechanics,* McGraw-Hill, 2010.

[3] Kiyoshi K. et all, *Internal Flow Analisys on a Micro Cross-flow Type Hydro Turbine at Very Low Speed Range,* The 11st Asian International Conference on Fluid Machinery, November 21-23, 2011, ITT Madrad, India.

[4] www.ansys.com

Address:

- Dipl. eng. Florin Pomoja, "Eftimie Murgu" University of Reşiţa, Piata Traian Vuia , No. 1 - 4, 320085, Reşiţa, f.pomoja@uem.ro