V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

OPTIMIZATION OF DIFFUSER WITH CFX TECHNOLOGY

Ferenc Szlivka*[1], Gergely Éder*[2],

[1] Szent István University, Faculty of Mechanical Engineering, Institute of Environmental Systems, Department of Environmental and Building Engineering
Gödöllő, H-2103, Hungary Tel: +36-28-410-975 ext 1475/ Fax: +36-28-420-997
e-mail: szlivka.ferenc@gek.szie.hu

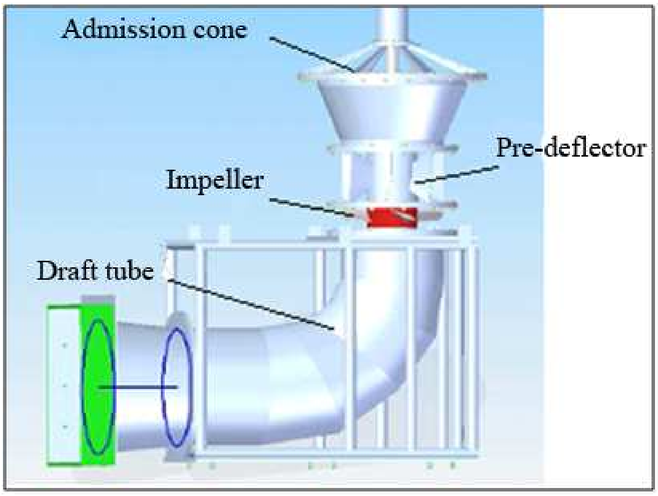
[2] Szent István University, Faculty of Mechanical Engineering, Institute of Mechanics and Machinery
Gödöllő, H-2103, Hungary Tel: +36-28-410-975 ext 1475/ Fax: +36-28-420-997
e-mail: eder.gergely@gek.szie.hu

Key words: Fluid Dynamics, Water turbine, Diffuser

Abstract. *We have set as an aim to optimize the parameters of a conduit-turbine's diffuser, what was planted into an eel-catcher shaft on the Sió canal, in Hungary. The aim was to increase the efficiency of the turbine. The length and the bend of the draft tube were also given. In the course of optimization the size of the diffuser's outlet diameter was changed. The vortex of water-spout which entering the diffuser was optimized with given draft tube geometry [1] by F. Szlivka and P. Kajtár. The main goal of our work was by changing the diameter of diffuser (draft tube) reach the best coefficient of efficiency. The simulation was accomplished with ANSYS CFX software.*

INTRODUCTION

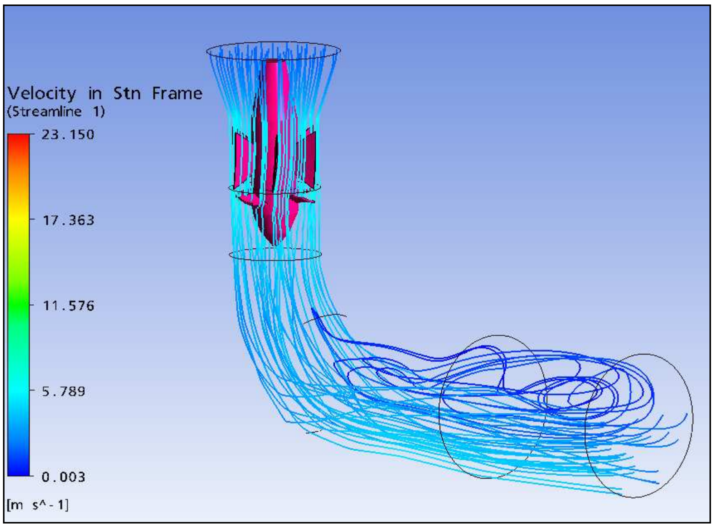
We have set as an aim to optimize the parameters of a conduit-turbine's diffuser, what was planted into an eel-catcher shaft on the Sió canal, in Hungary. The aim was to increase the efficiency of the turbine. In the course of our work we proceeded a previous researching where were examined the needed parameters of planting of the turbine and the controlling of the stream parameters with CFD simulation. During the designing of the turbine's main parts the following problems were realized: there were not enough places in the shaft to place an adequate diffuser. The shaft was too small for a sufficient diffuser what possess draft tube with small angle. For this reason was required the reducing of the diffuser's length and the increasing of the aperture angle. It was the cause of the efficiencies deterioration, however the loss was reducible. Our aim was the further reduce of the loss with the optimization of the diffuser's dimensions. In the course of optimization the entering size of the draft tube was given (900 mm) which accords with the diameter of turbine. The length and the bend of the draft tube were also given. In the course of optimization the size of the diffuser's outlet diameter was changed. The vortex of water-spout which entering the diffuser was optimized with given draft tube geometry [1] by F. Szlivka and P. Kajtár. The main goal of our work was by changing the diameter of diffuser (draft tube) reach the best coefficient of efficiency. The simulation was accomplished with ANSYS CFX software.
Similar optimization was accomplished by J.G.I. Hellström 1, B.D. Marjavaara, T.S. Lundström [2] with using CFX-5.7.1 software. Flow optimization with using diffuser was accomplished by L.F. Gaden*, Eric L. Bibeau [3].



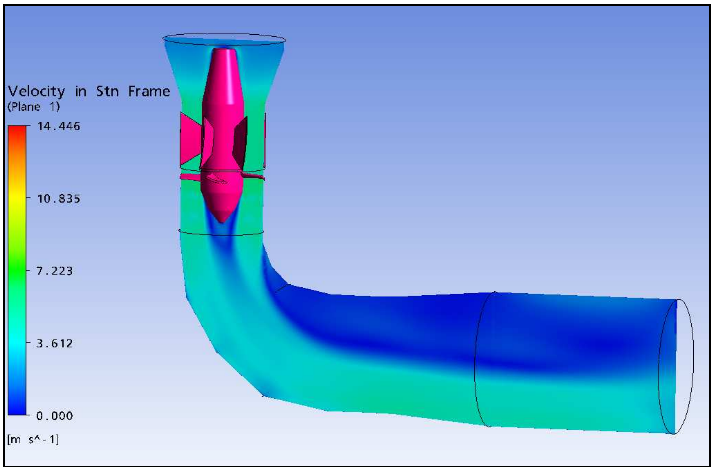
Picture 1: Micro turbine

CFD SIMULATION

In the previous work [1] the first step of the CFD simulation was the modeling of the stream area. The modeling was easily executable in the ANSYS Design Modeler module with the previously made 3D CAD model. The three main parts of the model were: influent part, the turbine and the diffuser. The next critical part was the preparing of the CFD mesh. In the meshing it is necessary to choose the good element size and the thickness of the boundary for the actual problem. With using too small element size the computation would consume too much time, however with big element size the impropriety is going to grow. The whole model was four meter high thus the hydrostatic pressure was also an important parameter and therefore it was necessary to define the gravitational acceleration. It can be seen from the results of the earlier examination that the evolved flow patter in the draft tube was not suitable. The flow has stagnated in the upper part moreover backflow was detectable in some section. The main cause of it was the not suitable place for create the elbow and the diffuser. After short section the flow has come off from the wall in the expanded elbow. The flow in the diffuser was improved by giving vortex for the stream at the inlet point. This vortex was made by the impeller. The optimal parameters of the vortex were determined with a distinct calculation.



Picture 2: Stream lines in micro turbine [1]



Picture 3: Velocity field in micro turbine [1]

The first step of the optimization process was to determine the magnitude of the vortex (eg. the circumferential velocity) in the outgoing flow from the impeller.

THE OPTIMIZATION OF THE VORTEX

From the calculation it could be seen that the next circumferential velocities were the optimal along the determined axial velocities. The diameter of the draft tube was 1490 mm.

Axial velocity [m/s] Va	Circumferential velocity[m/s] v_k	Inlet pressure [Pa]	Outlet pressur [Pa]	Pressure difference [Pa]	Efficiency of the diffuser
3	1,05	-3860,46	1,03467	3861,49467	0,908

	1,08	-3864,23	1,20387	3865,43387	0,904
5	1,7	-10730,6	2,75478	10733,35478	0,913
	1,8	-10751	3,41737	10754,41737	0,906
7	2,3	-20996	4,72023	21000,72023	0,916

Table 1: The optimum circumferential velocity at different axial velocities [1]

OPTIMIZATION OF THE DIFFUSER

As we mentioned it earlier the flow has stagnated in the upper part, and in some section backflow was detectable as well as the enlarging of the length is not possible. Therefore we would like to improve the flowing features and the investment costs with the optimization of the draft tube's diameter, to determine the outgoing section diameter of the diffuser. At the measurement the diameter of the draft tube were changed 1050 mm, 1150 mm, 1250 mm, 1350 mm, and 1450 mm were used instead of the original 1490 mm, which was possible maximum.

EFFICIENCY OF THE DIFFUSER

We have calculated the average axial and circumferential velocities on the outgoing section of the diffuser. The results are summarized in the Table 2.

Inlet Axial velocity	Outlet Axial velocities				
900 mm	1050 mm	1150 mm	1250 mm	1350 mm	1450 mm
3 m/s	2,203	1,836	1.555	1.333	1.156
5 m/s	3,672	3,061	2.592	2.222	1.926
7 m/s	5,141	4,285	3.628	3.111	2.697
Inlet Circum. velocity	Outlet Circum. velocities				
900 mm	1050 mm	1150 mm	1250 mm	1350 mm	1450 mm
0,985	0,3775	0,4135	0,709	0,657	0,612
1,6425	0,629	0,6894	1,1826	1,095	1,019
2,2995	0,8813	0,9652	1,655	1,533	1,427

Table 2: Axial and circumferential velocities at different diameters

Pressure enlargement	Diameters				
Axial v.	1050 mm	1150 mm	1250 mm	1350 mm	1450 mm
3 m/s	1951,67	2737,623	3231	3550,75	3762,17
5 m/s	5508,81	7682,945	9047,53	9933,2	10517,07
7 m/s	10874,6	15125,13	17792,4	19524,3	20663,48

Table 3: Pressure enlargement at draft tube

The efficiency of the diffuser was calculated with usage of the next formula:

$$\eta_{diff} = \frac{pressure\ growth}{\frac{\rho}{2} \cdot \left[\left(v_{ain}^2 + \bar{v}_{cin}^2\right) - \left(\bar{v}_{aout}^2 + \bar{v}_{cout}^2\right)\right]}$$

$\bar{v}_k$ Average circumferential velocity

$\bar{v}_{aki}$ Average axial velocity in outlet cross-section

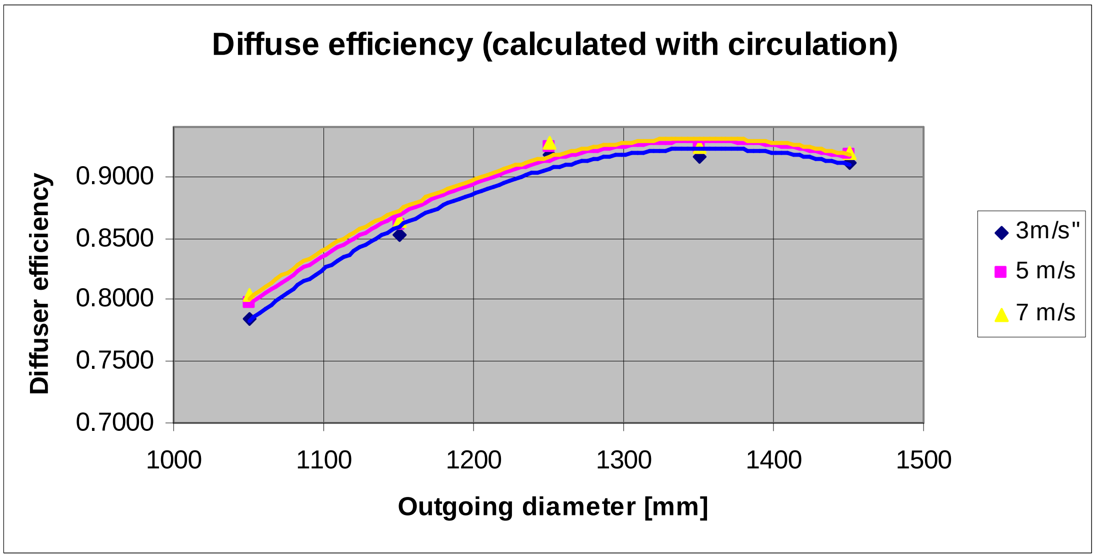
$\bar{v}_{kki}$ Average circumferential velocity in outlet cross-section

Efficiency	Diameters				
Axial v.	1050 mm	1150 mm	1250 mm	1350 mm	1450 mm
3 m/s	0,7847	0,8517	0,9167	0,9149	0,9110
5 m/s	0,7973	0,8607	0,9241	0,9214	0,9165
7 m/s	0,8031	0,8644	0,9270	0,9240	0,9188

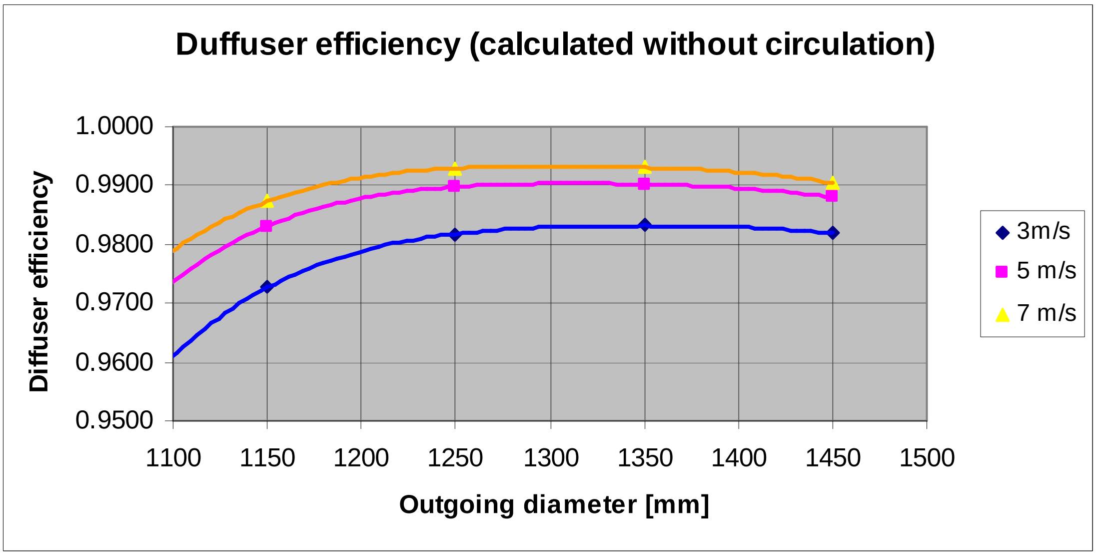
Efficiency	Diameters				
Axial v.	1050 mm	1150 mm	1250 mm	1350 mm	1450 mm
3 m/s	0,9413	0,9727	0,9818	0,9832	0,9818
5 m/s	0,9567	0,9831	0,9898	0,9902	0,9880
7 m/s	0,9636	0,9873	0,9929	0,9931	0,9904

Table 4: Efficiencies at different diameters (calculated with- and without circulation)

As it can be seen from the results the efficiency was almost the same with the different diameters or even it was a bit better with smaller size. It follows from this we can reach the same or a bit better efficiency with lower investment cost.



Graph 1: Diffuse efficiency (calculated with circulation)



Graph 2: Diffuse efficiency (calculated without circulation)

The results were represented in graphs and maximum point was detectable on both approximately at 1300 mm outgoing diameter of the diffuser. This optimum diameter is a little bit less than the possible maximum (1490 mm). But it is only the fluid mechanical optimum. Naturally the economic optimum would be the building in of the draft tube with smallest diameter. Therefore our work is economically advantageous whereas it is not necessary to build in the possible maximum magnitude.

Further possibility to optimize the system to optimize the magnitude of the vortex (circumferential velocity) after the impeller (inlet of the diffuser) with using the calculated outlet diameter (1300 mm). Whereas the optimization is multivariate and prospectively non-linear thus the optimization of the different variables are not performable independently from each other. We are going to accomplish further calculation heranent. This micro-turbine is built meanwhile and it works adequately.

REFERENCES

[1] Szlivka Ferenc, Kajtár Péter: Magyarországi törpe vízturbina fejlesztése: *Publication of Dunaújváros College XXIX*, Dunaújváros (2009)

[2] J.G.I. Hellström, B.D. Marjavaara , T.S. Lundström: Parallel CFD simulations of an original and redesigned hydraulic turbine draft tube; *Division of Fluid Mechanics, Lulea˚ University of Technology,* SE-97187 Lulea, Sweden; Received 12 September 2005; accepted 3 August 2006; Available online 1 November (2006)

[3] David L.F. Gaden*, Eric L. Bibeau: A numerical investigation into the effect of diffusers on the performance of hydro kinetic turbines using a validated momentum source turbine model; *Renewable Energy* **Vol 35** (2010) p 1152–1158