NUMERICAL PERFORMANCE EVALUATION OF DESIGN MODIFICATIONS ON A CENTRIFUGAL PUMP IMPELLER RUNNING IN REVERSE MODE

Ioannis Kassanos[a], Marios Chrysovergis[a], John Anagnostopoulos[a], Dimitris Papantonis[a] and George Charalampopoulos[b]

[a] *National Technical University of Athens, Department of Mechanical engineering, Laboratory of Hydraylic Machines, Heroon Polytechneiou 9, Zografou, 15780, Athens, Greece*
[b]*Drakos-Polemis Pumps, Kolokotroni 16, Kryoneri, Greece*

Abstract. In this paper the effect of impeller design variations on the performance of a centrifugal pump running as turbine is presented. Numerical simulations were performed after introducing various modifications in the design for various operating conditions. Specifically, the effects of the inlet edge shape, the meridional channel width, the number of blades and the addition of splitter blades on impeller performance was investigated. The results showed that, an increase in efficiency can be achieved by increasing the number of blades and by introducing splitter blades.

Keywords: Hydroelectric Power. Pump as Turbine, Impeller Design. Numerical Modelling
PACS: 88.60.K-

INTRODUCTION

The use of centrifugal pumps as turbines has been a popular alternative over the years, especially for mini hydroelectric plants. They offer the advantage of decentralized power generation, significant reductions in capital costs due to their simple construction and the ability to use standard of the shelve pumps and components with minimal modifications [1, 2]. The above have further strengthened their popularity in mini hydro power plants. However, an important disadvantage is the reduced efficiency, which may be up to -8.5% of the pump efficiency [2], the limitation of the operating envelope in a narrow range of conditions and the inability to control the operating point. Another issue has been the difficulty in predicting the operational characteristics of the pump in turbine mode. Several efforts have been made to this end using empirical correlations [1], CFD or experimental measurements [2, 3, 4, 5]. Furthermore, it is not a priori known what effect small geometrical changes in the impeller will have. This paper studies the effect of various modifications in the impeller/runner, on the turbine mode characteristics for various operating conditions using commercial computational fluid dynamics (CFD) software. An off-the-shelve low specific speed centrifugal pump was obtained and modeled in reverse mode and several design modifications were analyzed, aiming in the increase of efficiency.

MODEL DESCRIPTION

The numerical model used in this paper is based on a low specific speed centrifugal pump with a design head of 38mWc, flow rate 18m^3/h and a rotational speed of 2900 rpm, while the blade count of the impeller was 5. In the following table, the main geometric parameters of the pump are presented.

TABLE1. Geometric characteristics of centrifugal pump under investigation

Parameter	Value
Pump Type	LDP 32-160
Rotational Speed [rpm]	2900
Impeller width [mm]	5
Impeller diameter [mm]	177
Suction diameter[mm]	53.1
Discharge diameter [mm]	41.1

Motor power [kW]	5.5

IMPELLER DESIGN VARIATIONS

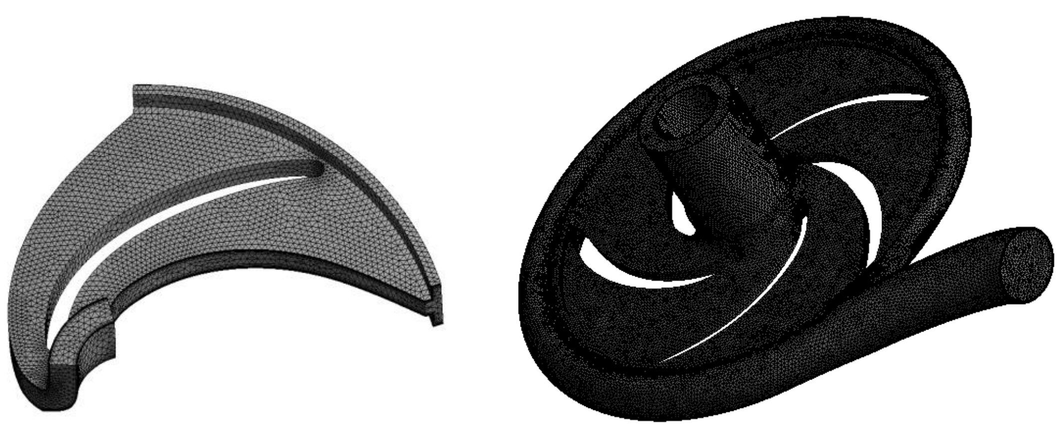
Apart from the initial impeller geometry, five other design variations, as listed in table 1, were investigated. The effects of the inlet edge blade profile rounding, the increase of blade number, the increase of the meridional flow channel width and the use of splitter blades on turbine performance were investigated.

TABLE2. Design variations under consideration.

Case#	Design variation
0	Initial Geometry
1	Inlet edge blade rounding
2	20% width increase
3	6 blades
4	7 blades
5	Splitter blades

NUMERICAL METHOD

The computational domain under consideration consisted of a) a single blade-to-blade passage and b) the full centrifugal pump model including the impeller and the spiral casing. For both cases, an unstructured tetrahedral mesh was used, while for the flow analysis the steady state RANS equations using a 2nd order discretization scheme and the realizable k-e turbulence model were solved. The boundary conditions were derived in a) from a spiral casing flow simulation where the absolute and tangential velocity components were obtained and in b) from empirical correlations as presented in [1] that connect the design flow in turbine mode to the operational characteristics in pump mode. The boundary conditions were: inlet flow rate of $29m^3/h$, zero outlet static pressure, and rotational speed of 2900 rpm. All simulations were performed used Ansys Fluent.

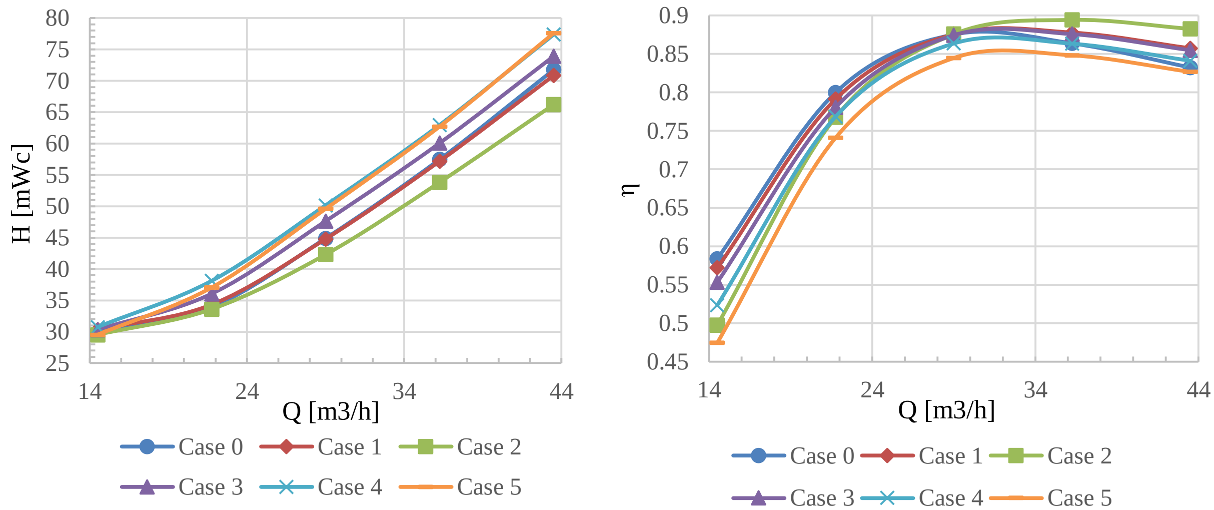


(a) (b)

FIGURE 1. (a) Numerical mesh of blade-to-blade domain; (b) Numerical mesh of full model

RESULTS

In the following figures the performance curves for the various cases considered are shown. From these figures it can be seen that qualitatively the curves follow the expected trend. An interesting observation is that the theoretical design point that was calculated through empirical correlations, well agrees with the numerical value of $29m^3/h$ shown below. From the H-Q curves it can be observed that cases 4 and 5 show similar performances while for all other cases the same flow is achieved at lower heads. Comparing cases 0 and 1 it can be seen how the rounding of the inlet edge improves the overall turbine performance, especially at larger flow rates, though a small effect in the head characteristic appears. At higher flows, the flow velocities increase and the potential mismatch in the flow and velocity angles leads to increased losses. On the other hand, the increase in impeller width reduces the head across the flow range studied. In Fig.2b, an increase in blade number leads to a reduction in efficiency as a result of the increased friction losses. Furthermore, it is evident that an increase in impeller width leads to a shift of the maximum efficiency point to higher flow rates.



(a) **(b)**

FIGURE 2. (a) Head-Flow characteristic of design variations; (b) Efficiency-Flow characteristic of design variations

To further check the results shown above, cases 1, 2, 4 and 5 were studied using also the full model, including the spiral casing for the estimated nominal flow rate in turbine mode.

TABLE3. CFD Performance results of different design variations.

Case #	Head[m]	Power[W]	Efficiency[%]
1	65.89	3689	70.9
2	61.257	3323	68.57
4	68.83	4053	74.63
5	66.96	3927	74.33

In table 3 the operational characteristics are compared. The results show that in cases 4 and 5 the hydraulic efficiency is increased by 3.4 % and 3.6 %, respectively, compared to case1. The use of more blades leads also to a reduction of the specific speed of the pump/turbine due to the increase of the operating head, while the opposite is observed when increasing the impeller width. The differences in performance from the blade-to-blade analysis and the full model simulations are attributed to the circumferential irregularity of the velocity distribution at the runner inlet. This was observed by plotting the span wise velocity distribution at the inlet of the runner, and the corresponding distribution at the spiral casing exit obtained from an isolated casing simulation. The increased losses, compared to the blade-to-blade calculation, can also be seen in Fig.3, where the velocity vectors at the mid span of the casing and the stream lines are plotted. Large recirculation regions are predicted for the initial geometry, which are reduced by increasing the blade number. These can be further reduced by adjusting the blade angles, while further performance improvements can be achieved by reducing the blade thickness and thus minimizing friction losses within the flow passage. By doing so, the cavitation characteristics of the impeller can also be improved.

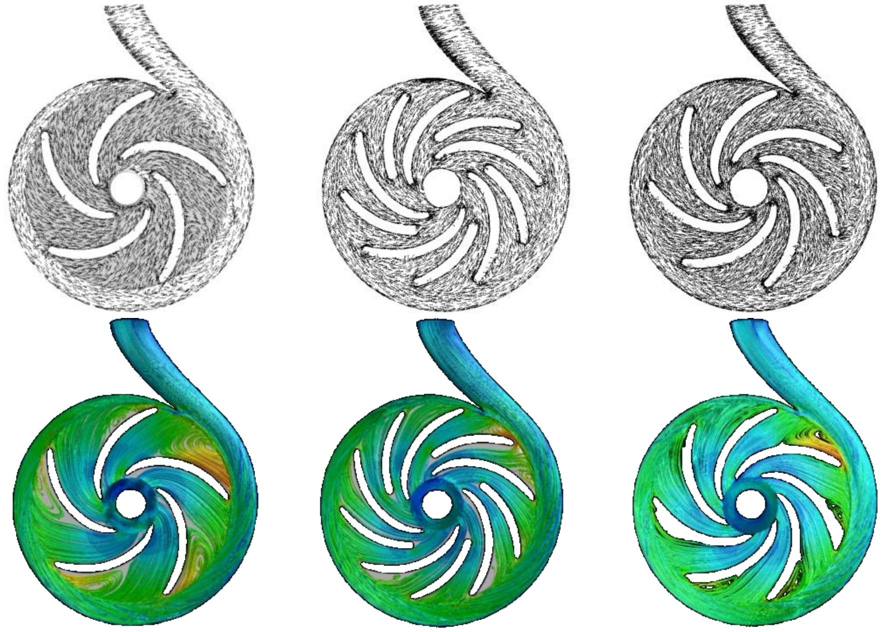

(a) (b) (c)

FIGURE 3. Relative velocity vectors and flow streamlines for case1 (a); case5 (b); and case4 (c)

CONCLUSIONS

In this paper the effect of various design modification on turbine mode operation of a centrifugal pump were shown. From the results presented here, it was revealed that the small number of impeller blades in conjunction with the change of width at the inlet causes a severe recirculation in the flow passage and increased losses. These losses could not be predicted by ignoring the interaction of the spiral casing and the impeller. For the proper performance analysis in turbine operation this interaction needs to be taken under consideration either by solving the full model or by adjusting the inlet velocity profile to account for this effect. Furthermore, the use of a larger number of blades leads to a shift of the performance curve to larger heads as well as higher efficiencies for the same flow rate. From the analysis of the flow the following design improvements are also proposed; modification of the blade outlet angles to reduce flow recirculation there and reduction of blade thickness in order to reduce the outlet edge velocity magnitude and to improve the cavitation characteristics of the impeller in turbine mode. The results presented here will be used as guidelines in the design optimization of the same runner, which will also be manufactured and laboratory tested both in turbine, as well as in pump operation. In this way, the numerical results presented here will be validated and the effect of the proposed design modifications in pump operation will also be quantified.

ACKNOWLEDGMENTS

Co-funded by Greece-National Strategic Reference Framework 2007-13 and the EU-European Regional development fund

REFERENCES

1. A. Williams, "The turbine performance of centrifugal pumps: a comparison of prediction methods", *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 59-66 (1994).
2. S. Jain, A. Swankar, K. Motwani, R. Patel, "Effects of impeller diameter and rotational speed on performance of pump running in turbine mode", *Energy Conversion and Management* **89**, 808-824 (2015).
3. S. Yang, S. Derakhshan, F. Kong, "Theoretical, numerical and experimental prediction of pump as turbine performance", *Renewable energy* **48**, 507-513 (2012).
4. S. Yang, F. Kong, W. Jiang, X. Qu, "Effects of impeller trimming influencing pump as turbine", *Computers & Fluids* **67**, 72-78 (2012).
5. D. Giosio, A. Henderson, J. Walker, P. Brandner, J. Sargison, P. Gautam, "Design and performance evaluation of a pump-as-turbine micro hydro test facility with incorporated inlet flow control", *Renewable Energy* **78**, 1-6 (2015).