CFD ANALYSIS OF THE PERFORMANCE OF A NOVEL IMPELLER FOR A DOUBLE SUCTION CENTRIFUGAL PUMP WORKING AS A TURBINE

T. Capurso[1] *- L. Bergamini*[2] *- S.M. Camporeale*[1] *- B. Fortunato*[1] *- M. Torresi*[1]

[1]Polytechnic University of Bari, Department of Mechanics, Mathematics and Management (DMMM), Via Edoardo Orabona, 4, 70126 Bari, Italy (tommaso.capurso@poliba.it – sergio.camporeale@poliba.it – bernardo.fortunato@poliba.it – marco.torresi@poliba.it)

[2]Nuovo Pignone, Strada Provinciale Modugno Bari 10, 70123 Bari, Italy (lorenzo.bergamini@bhge.com)

ABSTRACT

In this study a novel impeller for double suction centrifugal pump has been studied working as a turbine. The novel impeller, which shows higher performance than conventional ones when used in pump mode, has the number of blades at the outlet of the impeller doubled and a novel shape of the vanes. These characteristics can help to extract energy from the fluid more efficiently if it is employed as a turbine. The higher number of blades improves the flow guidance at the inlet of the runner and the new shape of the vanes reduces the losses inside the channels. To evaluate the performance of the new design, working in reverse mode, both a baseline and the novel impeller have been investigated by means of U-RANS simulations. Furthermore, the rotary stagnation pressure along the channels at their best efficiency points and their characteristic curves have been calculated and compared. Eventually, advantages in terms of power coefficient and space saving have been pointed out by scaling the novel geometry.

KEYWORDS

CENTRIFUGAL PUMP, PUMP AS TURBINE, OPENFOAM, 3D U-RANS, ENERGY SAVING

NOMENCLATURE

p	static pressure [N/m^2]
ρ	density [kg/m^3]
w	relative velocity vector [m/s]
u	peripheral velocity [m/s]
H	head [m]
Q	flow rate [m^3/s]
n_q	specific speed [$rpm\ m^{3/4}/s^{1/2}$]
f_q	number of impeller entries
C	torque [Nm]
p_{st}	stagnation pressure [N/m^2]
Ψ	pressure coefficient [-]

φ flow coefficient [-]

λ power coefficient [-]

P_{RR} disk friction losses [-]

D diameter [m]

n rotational speed [rpm]

INTRODUCTION

World's energy consumption is increasing year by year and much attention is paid to renewable energy sources (such as hydroelectric) in order to replace fossil fuels consumption. In the field of Micro Hydropower Plants (MHP) where turbines are critical technological components, many researchers in the last decades proposed the use of pump as turbine (PaT) (Binama, 2017 – Stefanizzi 2018) and applied them to a real sites (Morabito, 2018). Furthermore, a significant amount of energy in petrochemical plants and water distribution networks is often wasted when Pressure-Reducing Valves (PRV) or other restriction mechanisms are used. Indeed, these devices can be effectively replaced by pumps used as turbines for waste energy recovery.

Furthermore, pumps as turbines are used in combination with renewable energy plants in order to storage energy during off-peak demand due to their intermittency (Morabito, 2017). Thus, they can be used as energy storage systems (ESSs). One of them is the Pumped Hydro Energy Storage (PHES), which is characterized by predictable energy characteristics, long-term reliability and reduced global environmental effects. Moreover, thanks to its storage capacity and flexibility it can support the installation of other intermittent renewable power plants (Ardizzon, 2014 – Rehman, 2015). The use of PaT is strongly suggested because pumps are mass-produced, cheaper than conventional turbines and covers wide range of specific speeds and sizes. Moreover, considering their economic impact, it is well known that their installation guarantees a payback period of two years on average, which is more profitable than conventional hydraulic turbine (Paish, 2002).

Herein a novel impeller designed for low-medium specific speed double suction centrifugal pumps (n_q <60), is studied numerically in reverse mode. The novel geometry has been previously studied as a pump showing an efficiency improvement with respect to a conventional geometry (a baseline impeller with the same specific speed) in the order of 1-2% associated to a slip factor increase, secondary losses reduction and impeller outflow homogeneity improvement (Capurso, 2018 – Bergamini, 2017).

The novel double suction impeller is characterized by a new arrangement of its flow channels, which come up alternately on the same circumferential exit even if they start from the two different sides actually doubling the number of blades at the outlet of the impeller. The new channel arrangement can help in guiding the flow at the inlet of the runner of a PaT in absence of diffuser and their novel shape allows the reduction of the hydraulic losses providing more energy to the shaft.

Numerical analyses of both a baseline and the novel impeller operating in turbine mode have been performed by means of OpenFOAM and solving the 3D U-RANS equations with the k-ω SST model for turbulence closure (Menter, 2003).

Eventually, their characteristic curves have been compared. The novel geometry shows higher values of the flow coefficient and of the global efficiency (+0.83%) at the BEP; specifically the latter improvement is justified by evaluating the disk friction losses and the hydraulic losses inside the vanes. To do this the rotary stagnation pressure variation has been calculated along the channels of the two geometries. The results show that the novel geometry is able to extract energy from the fluid more efficiently. Furthermore, the new geometry can be downsized (-3.5%) to work at the same flow coefficient of the baseline providing both higher power coefficient (+5.53%) and space saving.

NUMERICAL METHODS

In this work, numerical investigations have been performed by means of the open-source CFD code OpenFOAM. The 3D unsteady RANS equations have been chosen in order to take into consideration flow unsteadiness and the interaction between the exit of the volute (where the tongue is placed) and the inlet of the runner. Furthermore, they have been considered adequate in order to model the flow through the pump where quantities (such as head, torque, etc.) have to be considered averaged over a time period short enough with respect to global unsteady phenomena but long enough for statistical significance (Shah, 2013).

In the OpenFOAM package the application *pimpleDyMFoam* has been selected to run transient simulations, including moving meshes, with incompressible flow. This application is based on the PIMPLE algorithm: a combination of the PISO (Pressure Implicit with Splitting of Operator) and SIMPLE (Semi-Implicit Method for Pressure-Linked Equations). The turbulence model applied for the system closure is the k-ω SST (Menter, 2003). This turbulence model is a standard to perform numerical analyses in hydraulic turbo machineries. It automatically uses the k-ω model in the near wall region whereas the k-ε model away from the walls. The k-ω SST model can give an accurate prediction of flow separation explaining its common use for the numerical investigations of flow inside the centrifugal pumps (Bardina, 1997).

Numerical domain and Boundary conditions

The geometry studied in this work reproduces the suction pipe, the discharge pipe and the impeller of a commercial double suction centrifugal pump, property of Nuovo Pignone, with a specific speed $n_q \cong 21$. The specific speed is calculated by means of the most popular definition in Europe:

$$\mathrm{n_q} = \frac{n\sqrt{Q_{BEP}/f_q}}{H^{0.75}} \tag{1}$$

where H is the hydraulic head (m), Q is the flow rate (m^3/s), n the rotational speed (rpm) and f_q is the number of impeller entries. The geometry represented in Figure 1 has been simulated working as a turbine. The runner is a double suction impeller, actually two single suction centrifugal pump impellers with seven blades in a back-to-back configuration, see Figure 2. This kind of centrifugal pump is preferred to single entry pumps since it has superior efficiencies and it minimize the net positive suction head required ($NPSH_R$). Moreover, the volute is double to balance the radial loads on the rotor and allow high-speed operation, also at part load (Gülich, 2008).

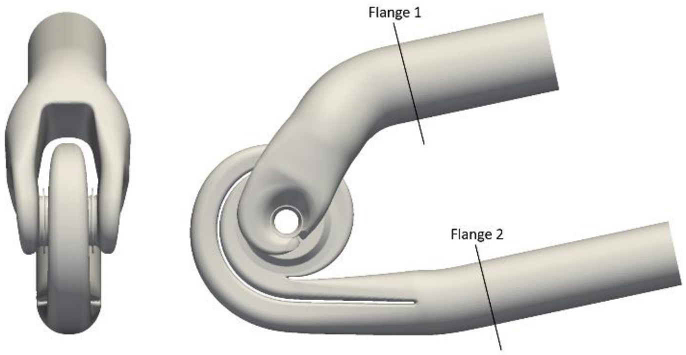

Figure 1: Front and side view of the CAD representing the centrifugal pump.

For all the analyzed cases, the mass flow rate has been imposed at the inlet considering a uniform inlet velocity distribution. Moreover, the value of the mass flow leakage, which flows through the annular seal, has been modeled as exiting from the impeller case and incoming axially upstream the impeller eye with a 45° of swirl with respect to the tangential direction. The leakage has been calculated a priori according to a one-dimensional empirical model because the geometry of the seal has not been included in the computational domain (Gülich, 2008). A uniform pressure distribution has been imposed at the outlet of the domain. At the inlet and outlet of the pump, straight pipes have been added to reduce uncertainties due to boundary conditions. The three parts of the geometry, which have been merged together, communicate each other by means of interfaces. In OpenFOAM the boundary condition used on these interfaces is named *cyclicAMI* (Arbitrary Mesh Interface). Furthermore, for this test case the turbulent intensity has been assumed constant and equal to 3% at the inlet of the domain. A critical aspect has been the definition of the wall roughness in OpenFOAM; the value of the equivalent sand grain roughness is equal to 5.6E-5 *m* and 5.6E-6 *m* respectively for the walls of the stator and rotor parts; the same values are used for all the geometries tested (Adams, 2012). To do this the *nutURoughWallFunction* has been applied to the walls in the *nut* file together with *kqRWallFunction* and *omegaWallFunction*, respectively for k and ω wall boundary conditions.

The numerical method assessment has been done by comparing the characteristic curve (head and efficiency) of the pump calculated either with the commercial code CFX and OpenFOAM (Nilsson, 2006 - Capurso, 2018). A grid refinement study has been performed for the pump configuration. Three different mesh sizes have been generated with the grid generator ICEM CFD® see Table 1. Eventually, a grid made of 11 million of cells has been chosen because of small deviations in the results and a good compromise between the geometry refinement and the computational costs (see Table 1).

Table 1. Different grid refinements with the pump Head calculated at the BEP.

	Coarse	**Medium**	**Fine**
Impeller (10^6)	3	6	10
Suction (10^6)	1	2	4
Discharge (10^6)	2	3	6
Total (10^6)	6	11	20
Head at BEP (*m*)	139.8	140.8	141.0
Time (h) 128 cores	7	18	40

NOVEL GEOMETRY

In this work, a novel impeller geometry proposed by the authors, which can retrofit conventional impellers for double suction centrifugal pumps, has been studied numerically under turbine operating conditions because it is thought to be able to improve the entrance of the flow coming from the volute.

The novel impeller, which is designed by means of a specific 1D code, actually allows one to double the number of blades at the outlet of the impeller in a way that the channel outlets, coming from the two sides, are circumferentially arranged and therefore the flow guidance is increased, see Figure 2. The basic idea behind the novel impeller is its compatibility with the housing of the conventional impeller keeping the same specific speed of the baseline. The channels not only intersect each other without interference but also have a cross sectional area distribution along the center line (*c-line*), which allows a smooth variation of the velocity along the *c-line* inside the channel.

As it will be shown, this results in an increase of the slip factor, which leads to a higher value of the hydraulic head with respect to a conventional configuration with the same impeller outlet diameter. Other results are a higher value of the hydraulic diameter (D_h) and a lower length of the channels (L), which allow a lower amount of losses inside the new channel, hence improving the hydraulic efficiency. This has been highlighted by evaluating the rotary stagnation pressure inside the channel (Capurso, 2018).

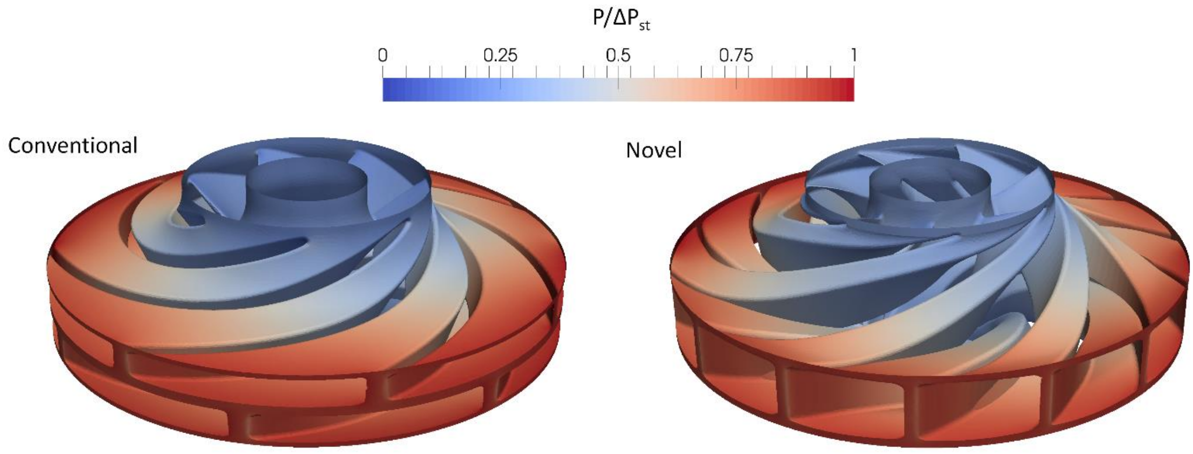

Figure 2: Comparison between the conventional (left) and the novel impeller (right) geometries.

RESULTS

Simulations have been run for 10 complete rotation of the machine and when convergence has been reached, the results have been calculated over the last 3 rotations. The time step has been chosen as a fraction of the time in order to guarantee numerical stability, thus $\Delta t = (T/Z)/\ 256 = 4.5\text{E-}6\ s$ where T is the time that it takes to complete one rotation and Z the number of blades. The mean and the maximum Courant number results equal to 0.03 and 6, respectively.

Turbine mode

Firstly, the characteristic curves of the two geometry have been calculated (Yang 2012 – Derakhshan 2008). In Figure 3 the head, H, and the efficiency, η, of the two machines have been represented by means of dimensionless coefficients whose expressions are defined as follows:

$$\Psi = \frac{2gH}{u_2^2} \tag{2}$$

$$\varphi = \frac{Q}{\pi D^2 u_2/4} \tag{3}$$

$$\eta = \frac{C\omega}{Q\Delta p_{st}} \tag{4}$$

The runaway curve, represented in Figure 3, has been obtained by means of the empirical equations proposed by Gülich (2008).

The novel geometry has been designed in order to guarantee the same specific speed (n_q) of the conventional one (Capurso, 2018). Experimental results have shown that the design process proposed by the authors provides pump impellers with specific speed deviation lower than 3.66% compared to the target one. From the numerical simulations carried out in this work, the specific speeds for the baseline and the novel impeller are equal to 21.88 and 21.08, respectively, with the same error that would have been obtained if the impellers had worked in pump operating mode.

In Figure 3 and 4 we can observe that the novel impeller has a φ in correspondence of the BEP higher than the baseline and close to the BEP the global efficiency is higher (+0.83%) than the baseline. This shall be due to the novel shape of the vanes.

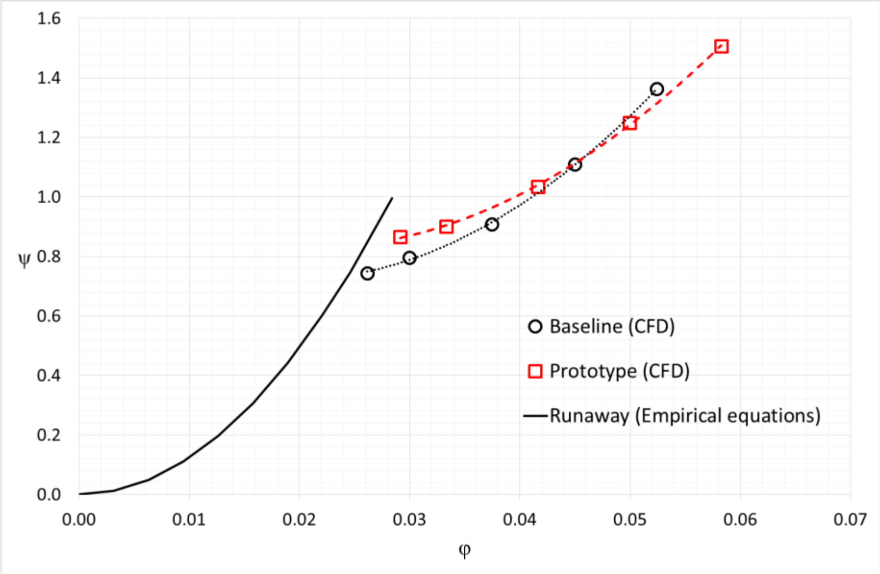

Figure 3: Pressure coefficient (ψ) vs flow coefficient (φ) curves of the two geometries.

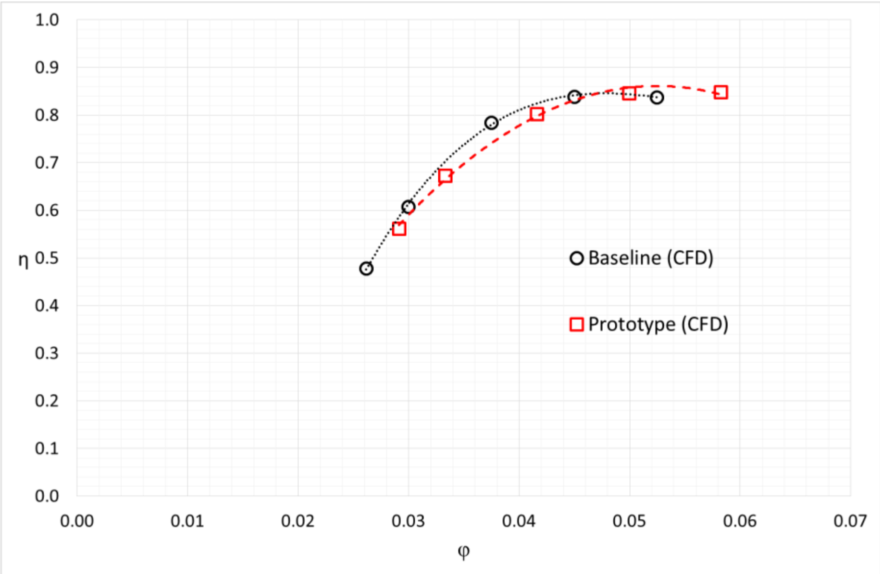

Figure 4: Efficiency (η) vs flow coefficient (φ) curves of the two geometries.

Moreover, the power coefficient (λ) has been evaluated; it is defined as follows:

$$\lambda = \varphi \, \psi \, \eta \tag{5}$$

Looking at Figure 5, the power coefficient of the novel impeller is towards higher flow coefficients.

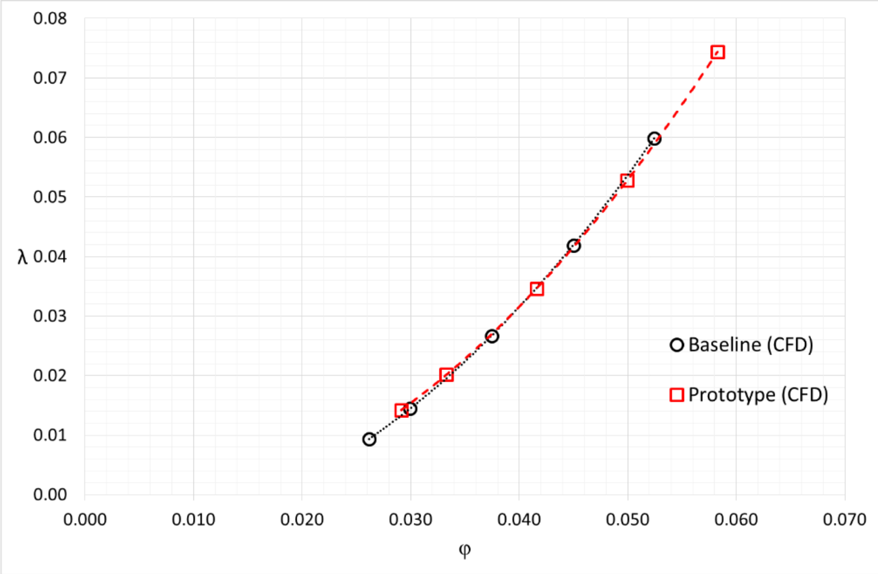

Figure 5: Power coefficient (λ) vs flow coefficient (φ) curves of the two geometries.

Scaled geometry

In order to overlap the novel impeller BEP with the one of the baseline, the former has been downscaled by -3.5%. After downscaling, the two machines have their BEPs placed at $Q/Q_{BEP,\ conv}$ = 1, with the difference that the novel impeller still provides higher head (Figure 5).

This means that the runner is able to extract a larger amount of energy from the fluid than the traditional one, $\Delta H = +4.70\%$.

To overlap the two characteristic curves a further scaling is necessary. In addition to the size scaling a reduction of the rotational speed is needed (-4%). Thus, the new impeller allows to guarantee the same performance of the baseline with size saving and an improvement of the efficiency.

Indeed, it is well known that the power losses due to friction losses is proportional both to the angular velocity and the diameter of the machine in the way described by the equation 6:

$$P_{RR} \propto \omega^3 r^5 \tag{6}$$

Thus, both scaling diameter and angular velocity we obtain a machine, which provides the same performance of the baseline with a remarkable improvement of the efficiency, in particular a disk friction losses reduction equal to -25.96%.

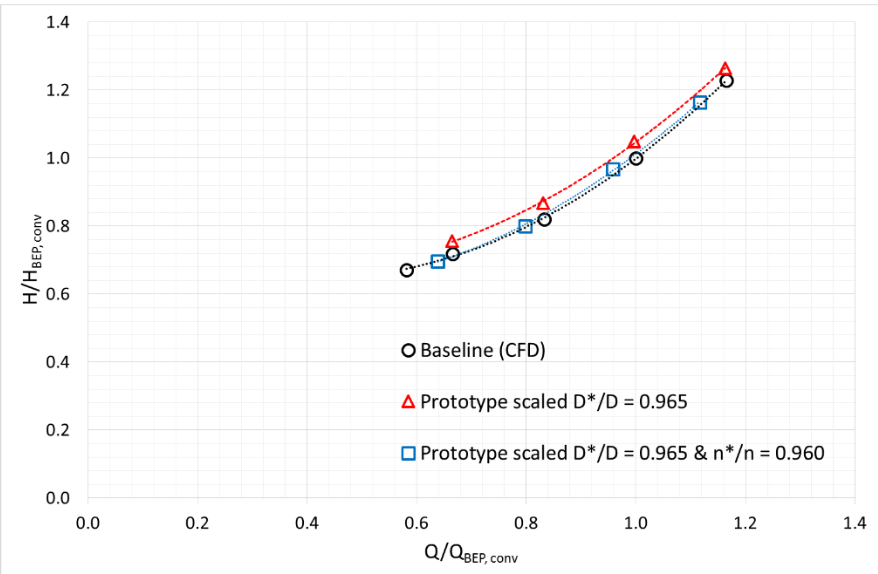

Figure 6: $H/H_{BEP, conv}$ vs $Q/Q_{BEP, conv}$ curves of the baseline, the novel impeller scaled by *D* and the novel impeller scaled by both *D* and *n*.

Moreover, looking at the efficiency curve (Figure 7) the novel geometry is prone to convert fluid energy at BEP as described previously but especially at part loads (+14.9 %). The novel shape of the channel and their width can reduce instabilities at the inlet of the runner at part loads (Xiaohui, 2017) that are responsible of static pressure reduction. Eventually, the evaluation of the power coefficient (λ) shows another important advantage. The novel geometry presents a higher curve over the entire range of flow rate; in particular an increase of 5.53% at the BEP, see Figure 7.

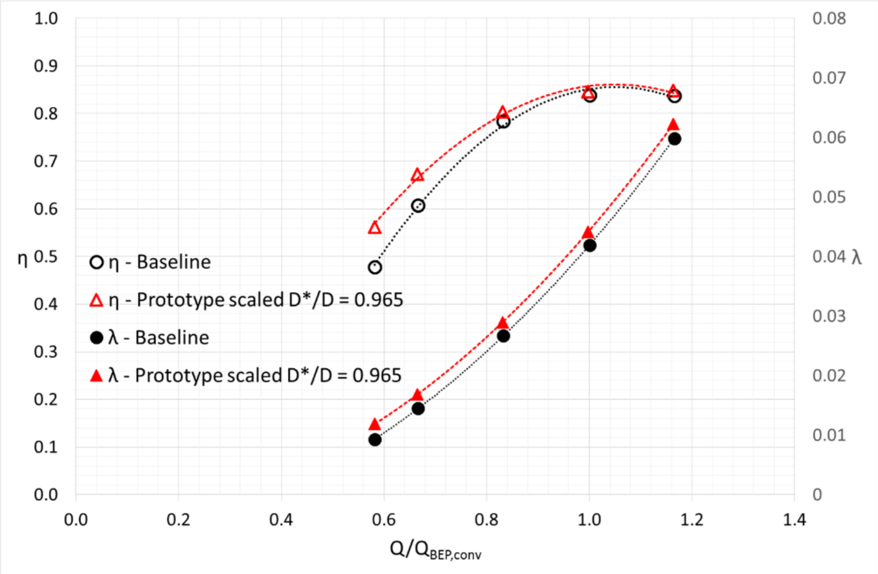

Figure 7: Efficiency (η) and Power coefficient (λ) vs $Q/Q_{BEP, conv}$ curves of the baseline and the novel impeller scaled by *D*.

Comparison of the performance

To further investigate the performance improvement, the rotary stagnation pressure (equation 7) has been calculated inside the channels of the baseline and the novel geometry at their best efficiency points (BEPs). The rotary stagnation pressure has been calculated along the channels as area-weighted integral on iso-surfaces with constant meridional coordinates.

Looking at Figure 7, when the fluid flows through the novel impeller, it is subjected to a smooth reduction of the rotary stagnation pressure, $\Delta p^*/\rho$. This means that the hydraulic efficiency of the new channels is higher than the baseline one. This result also has a positive impact on the global efficiency of the PaT, see Figures 4 and 7.

$$-\mathrm{L_w} = \frac{\Delta p^*}{\rho} = \Delta\left(\frac{p_{abs}}{\rho} + \frac{w^2}{2} - \frac{u^2}{2}\right) \tag{7}$$

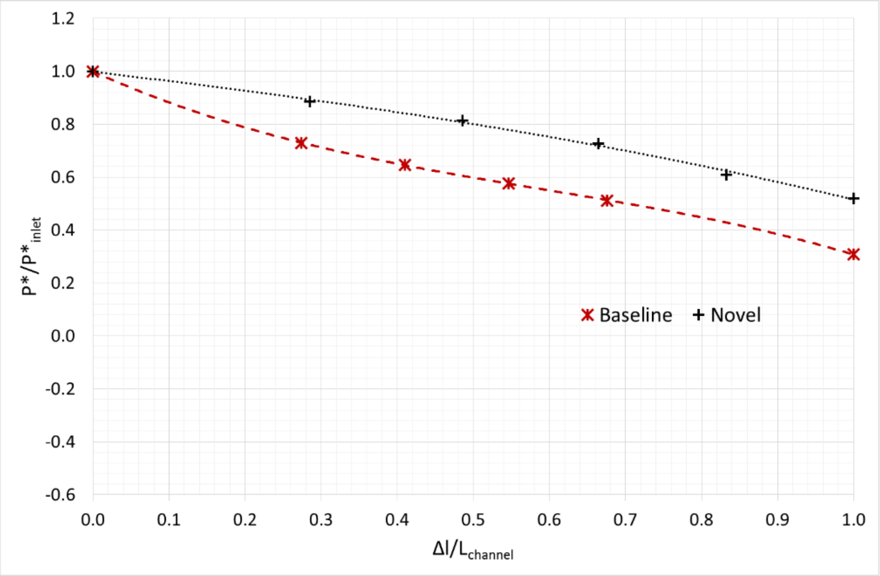

Figure 8: Rotary stagnation pressure inside the channel of the two geometries.

Looking at the efficiency curve at part-loads, the novel impeller not only reduces the losses when it works at the BEP, but also when the fluid at the outlet of the volute is not perfectly guided. Indeed, the novel vanes work better at off-design. Further analysis are suggested to analyze slip phenomenon at the outlet of the novel vanes (Capurso, 2018).

CONCLUSIONS

In this work a comparison of the performance of a baseline and a novel impeller, proposed by the authors to retrofit conventional impellers for double suction centrifugal pump, is presented. The novel geometry has been designed with a new arrangement of the channels with the aim to improve the efficiency and the slip factor in pump operating mode.

Firstly, 3D U-RANS simulations have been run with the open source CFD code OpenFOAM. The characteristic curves of the two machines have been calculated and their specific speeds compared.

From the results analysis a size scaling (-3.5%) is proposed in order to reach the same performance of the baseline geometry. Once done, this size reduction moves the BEP of the novel geometry at the same value of $Q/Q_{BEP,\ conv}$ =1, pointing out that the novel geometry is able to extract excess energy from the fluid ΔH = +4.33%.

Thus, to reach the same performance of the baseline an additional scaling is required. By reducing both the size (D) and the rotational speed (n) an overlapping of the curves of the two machines is possible. This leads to a remarkable reduction of the disk friction losses and an improvement of the global efficiency.

Moreover, the rotary stagnation pressure has been studied inside the two machines working at their BEPs and the fluid flowing inside the novel vanes presents a reduced level of losses from the inlet to the outlet of the runner.

In the future, further analyses of the rotary stagnation pressure will be carried out at part loads in order to investigate the benefits from using the novel geometry in reverse mode. Moreover, velocity profiles at the inlet and at the outlet of the runners will be compared to analyze the behavior of the novel vanes in guiding the fluid exiting from the volute and to quantify slip phenomena at the outlet of the novel runner.

REFERENCES

Binama, M.; Wen-Tao Su; Xiao-Bin Li; Feng-Chen Li; Xian-Zhu Wei; Shi (2017). An Investigation on pump as turbine (PAT) technical aspects for micro hydropower schemes: A state-of-the-art review, Renewable and Sustainable Energy Reviews, Volume 79, Pages 148-179, ISSN 1364-0321.

Morabito A, Steimes J, Bontems O, Al Zohbi G and Hendrick P (2017). Set-up of a pump as turbine use in micro-pumped hydro energy storage: a case of study in Froyennes Belgium. Journal of Physics: Conference Series.

Stefanizzi M., Capurso T., Balacco G., Fortunato B., Camporeale S.M., Torresi M., Preliminary assessment of the installation of a pump as turbine in a real water distribution network, ETC 13th Lousanne, Under review.

Ardizzon G, Cavazzini G and Pavesi G (2014). A new generation of small hydro and pumped-hydro power plants: Advances and future challenges Renew. Sust. Energ. Rev. 31 p 746-761

Shafiqur Rehman, Luai M.Al-Hadhrami, Md. MahbubAlam, Pumped hydro energy storage system: A technological review, 2015, Renewable and Sustainable Energy Reviews Volume 44, April 2015, Pages 586-598

Paish O (2002). Micro-hydropower: Status and prospects PI. Mech. Eng. A-J Pow 216 p 31-40.

Capurso T., Bergamini L., Torresi M., (Under review). Design and CFD performance analysis of a novel impeller for double suction centrifugal pumps. Nuclear Engineering and Design.

Bergamini L, Torresi M, Capurso T. High efficiency double suction impeller - Patent Application – 20180128271

Menter, F. R.; Kuntz, M.; Langtry, R. Ten Years of Industrial Experience with the SST Turbulence Model. Turbulence Heat and Mass Transfer 4. 2003; 4:625-632.

Gülich. J.F. Centrifugal Pumps, 2nd ed.; Springer, 2008.

Huang, P.; Bardina, J.; Coakley, T. (1997). Turbulence Modeling Validation, Testing, and Development, NASA Technical Memorandum, vol. 110446.

Adams, T.; Grant, C.; Watson, H. A Simple Algorithm to Relate Measured Surface Roughness to Equivalent Sand-grain Roughness. International Journal of Mechanical Engineering and Mechatronics, Volume 1, Issue 2 Year 2012 - Pages 66-71.

H. Nilsson Evaluation of OpenFOAM for CFD of turbulent flow in water turbines, 23rd IAHR Symposium, 2006.

Xiaohui Q Study on Slip Phenomenon of Pumps As Turbines in Different Working Conditions 2017 Proceedings of the ASME 2017 Fluids Engineering Division Summer Meeting

S.R.Shah, S.V.Jain, R.N.Patel, V.J.Lakhera, CFD for Centrifugal Pumps: A Review of the State-of-the-Art, Procedia Engineering, Volume 51, 2013, Pages 715-720.

Capurso, T. Stefanizzi, M. Torresi, M. Pascazio, G. Caramia, G. Camporeal, S.M. Fortunato, B. Bergamini, L. How to Improve the Performance Prediction of a Pump as Turbine by Considering the Slip Phenomenon", Proceedings, 2018