INTERNATIONAL JOURNAL OF
ENERGY AND ENVIRONMENT

Volume 2, Issue 1, 2011 pp.39-48
Journal homepage: www.IJEE.IEEFoundation.org

CFD and comparisons for a pump as turbine: Mesh reliability and performance concerns

O. Fecarotta[1], A. Carravetta[1], H. M. Ramos[2]

[1] Engineering Faculty - Università degli Studi di Napoli Federico II, Italy.
[2] Civil Engineering Department and CEHIDRO, Instituto Superior Técnico, Technical University of Lisbon, Av. Rovisco Pais, 1049-001, Lisbon, Portugal.

Abstract
The need for saving energy in water supply systems has become one of the main concerns of system managers and it will become more important in a near future. New strategies must be developed and implemented in the major energy consumption systems like those for water supply. In drinking pipe systems the use of Pressure Reducing Valves (PRV) as a dissipative device is the common way to uniform the pressure control through a localised head loss. The use of micro-turbines or pumps operating as turbines seem to be an alternative technical and environmental available solution to either control the pressure as well as to produce energy. Pumps as turbines (PAT) could be a convenient choice, but a deep study of the machine in different operating conditions is necessary in order to prevent the water system from ruptures. This paper shows that semi-empirical parametrical models do not generally predict with precision the behaviour of a pump operating as a turbine, while CFD analysis could be a reliable tool to better understand the interaction between the hydromechanical equipment and the flow behaviour. Nevertheless the CFD calculation difficulty is generally very high and the minimum complexity of the CFD calculation mesh has been investigated, in order to perform faster and reliable simulations. Thus CFD calculations have been carried out to predict the turbine behaviour under different flow conditions and the performance curves have been obtained. Some calculations in unsteady state flow regimes have been led to investigate the response of the machine to a sudden discharge changing, as a preliminary study of the behaviour of a turbomachine installed in a water distribution system under water hammer situations.

Keywords: Pump as turbine, Hydrodynamic behaviour, Low power turbines, CFD analysis, Experimental tests.

1. Introduction
Water distribution systems are usually low-efficiency systems both for energy consumption and water leakages [1]. Furthermore pressure reducing valves (PRV), which cause the mechanical dissipation of hydraulic energy, are often used in order to control the pressure regime in the system as well as to face topographical discontinuities during the pipeline path, to avoid high pressures in the network that can cause ruptures and water leakages.
The use of turbines seems to be a valid and a sustainable alternative to reduce pressure, as well as being a strong opportunity to produce energy. A study of the fluid dynamic behaviour in steady and unsteady

conditions of these machines is basic to understand the interactions between the turbine and the pipeline system.
Some specific kinds of micro-hydro turbines have been purposed [2-4], with devices that face the pressure and the discharge variability, or with an inverter in order to modify the rotational speed with the power. All these solutions are quite expensive, while the use of a pump operating as a turbine (PAT) is a cheap solution and a suitable chance. On the other hand pump manufacturers do not normally provide the characteristic curves of their pumps working as turbines, which are necessary to select the correct PAT for the hydro power plant. One-dimensional methods to be used with the Method of Characteristics were purposed in order to calculate the machine behaviour [5-7], but cannot predict 3D hydrodynamic flow behaviour inside the runner and the dissipative effects associated such as turbulence, vorticity and leakages. Commonly the only way to obtain reliable results is to measure the performance of the machine in its desired sense of application, which would offset a large part of the low-cost advantage of a PAT. This could be due to the lack of fluid dynamic knowledge in this kind of turbo machines and their behaviour. The inefficiency and the unsafe conditions are normally associated to the occurrence of separate flow zones, vorticity, macro turbulence intensity, induced vibrations, resonance effects, leakages which can affect the performance and the security of hydro power plants and pipeline systems.
The CFD technique could be a valid tool to overcome this lack, in order to provide basic information about the flow dynamic field to the machine designer and operating conditions such as to verify the interaction between the machine behaviour and variable conditions of discharge and pressure inside of any water distribution system.

2. Open questions

2.1 Design aspects

The characteristic curves of a turbo machine are necessary in order to select the appropriate turbine in the design process of a hydropower plant. Since the performance curves of the turbo machines are usually provided only for the pumping mode, some parametrical method that predict turbine performance curve have been developed. The Figure 1 compares the results of Suter parameters [6] and Derakhshan [5] with one experimental performance curve for a real PAT operating as a pump and turbine mode. The variation of head, power and efficiency with the discharge are presented in the

Figure 1. The rotational speed N_R=2900 r.p.m., while the specific speed N_S=17.53 (m, m^3/s). The specific speed of a pump is defined as:

$$N_S = N_R \frac{Q^{1/2}}{H^{3/4}} \left(m, \frac{m^3}{s} \right) \tag{1}$$

The efficiency of the pump and of the turbine are defined respectively as:

$$\eta_{pump} = \frac{\rho g \Delta H Q}{M\omega} \tag{2}$$

$$\eta_{turbine} = \frac{M\omega}{\rho g \Delta H Q} \tag{3}$$

where Q is the discharge, M the torque, ΔH the net head, g the gravity acceleration, ρ the water density and ω the rotational speed.
Suter parameters give information for both the turbine and the pump mode for few available pumps, and the Derakhshan model predicts information for the turbine performance curve.

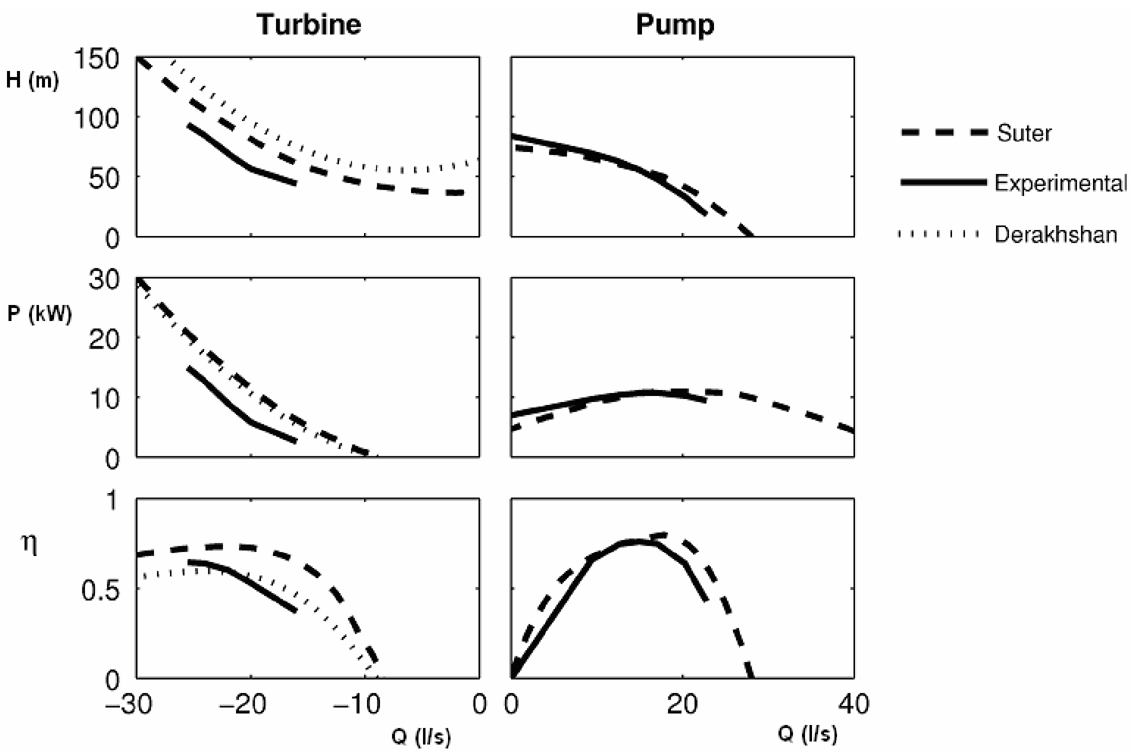

Figure 1. Characteristic curves for pump and turbine mode

These results are quite reliable for the pump mode, but in some cases still have low accuracy in the prediction of the turbine mode for new pumps with small sizes. The head is generally over predicted and the offset is bigger using the Derakhshan model. Also the power is overestimated and consequently the prevision of efficiency is not adequate enough for new small size pumps with non negligible scale effects.

2.2 Practical aspects

The PAT machine response to variable conditions when inserted in a pipeline could be quite different from the response of a PRV. While the pressure downstream the valve can be regulated independently on the discharge by the correct setting of the valve opening, for a PAT [8], that is normally not equipped with a guide vane, a discharge variation causes a head-loss variation imposed by the characteristic curve and the pressure waves pass through the runner inducing high efforts and tensions (Figure 2).

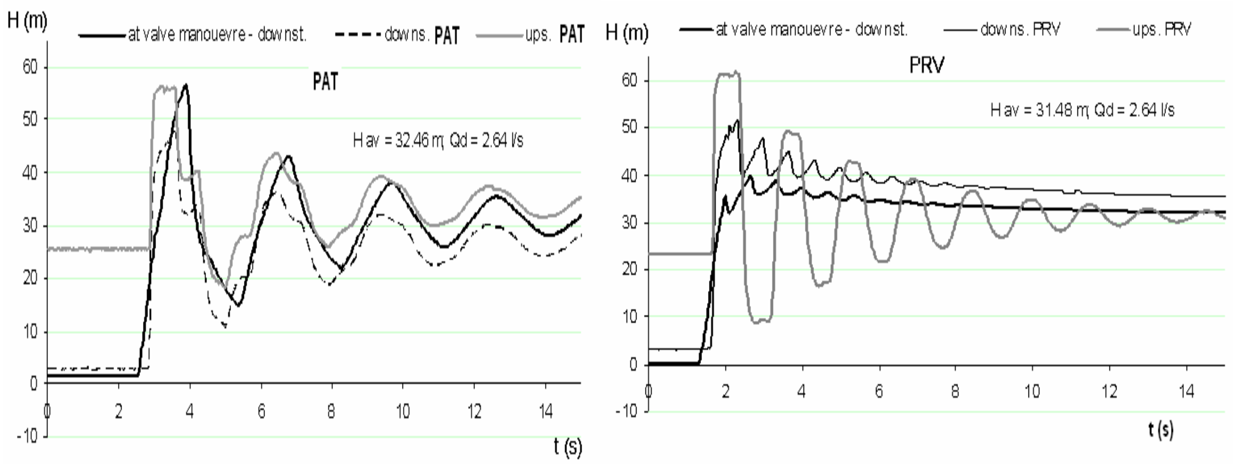

Figure 2. Analysis of the dynamic behaviour through a PRV and a PAT for a fast closure of a pipe downstream valve [8]

Nevertheless the water distribution system designer should be concerned about the PAT behaviour under water-hammer conditions, to preserve the pipeline system from ruptures and inefficiencies. Three kinds of setting should be preliminary studied, in order to understand the behaviour of the machine in transient conditions: runaway or overspeed of the machine due to a sudden black-out, closure of a valve downstream and closure of a valve upstream.
Some results by Ramos et al. are available for the first case and guide vane closure of a turbine, and they were obtained in lab conditions and compared with a reliable one-dimensional calculation. Ramos et al. [7] studied the behaviour of a turbine equipped with a guide vane under runaway conditions and the effects of this behaviour on a penstock of a small hydropower plant with a long penstock. In Figure 3 is presented the good agreement between the modelling and the experimental results being Q_R and H_R the rated conditions for the discharge and net head of the turbine.

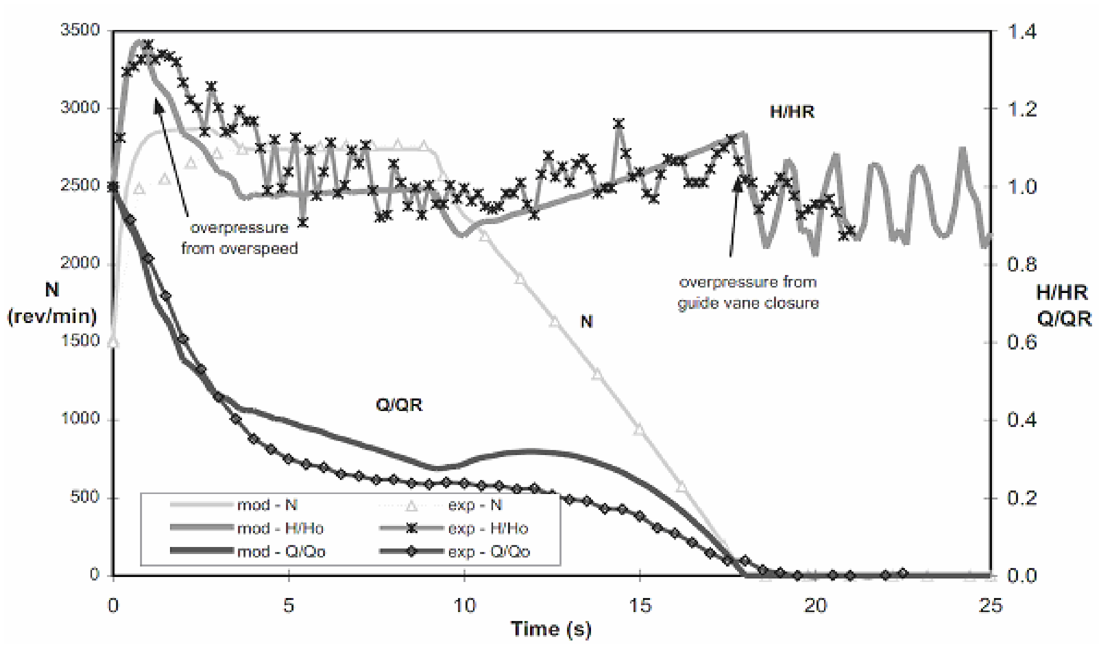

Figure 3. Comparison between experimental data and numerical simulation of a turbine overspeed and guide vane closure in simultaneously for a slow Francis turbine [7]

There is a big lack in the literature about the influence of valves' manoeuvres in water distribution networks with a turbomachine in a middle of a pipeline (valve closure time, discharge time variation with the consumption, pressure time variation) which has different characteristics (specially designed to supply drink water to the population) than a penstock carefully designed to supply water to turbines in best flow conditions to avoid flow disturbances which can affect the machine efficiencies.

3. Numerical results

3.1 Computational fluid dynamics

The Computational Fluid Dynamic allows to perform many calculations both in steady conditions (performance curves of the machine) and in transient flow regimes. This kind of calculations can be very complex, and the computational complexity increases when dynamic simulations are performed. Thus, a first work has been leaded in order to find the minimum number of elements in the 3D fluid dynamic mesh necessary to obtain reliable results for the flow inside the PAT runner. Then the performance curve of a specific PAT model have been computed, both for pump and turbine mode. A preliminary transient calculation has been carried out, in order to obtain a first result about the inertial response of the machine with a sudden discharge finite variation.
The Navier-Stokes equations have been used in order to solve the fluid dynamic field. The turbulence has been modelled with SST-kω model [9]. This turbulence model introduces two differential transport equations in k (turbulent kinetic energy) and ω (turbulent frequency) with some terms which consider the transport of the shear stress, in order to calculate the eddy viscosity μ_T with the following relations:

$$\mu_T = f_\mu c_\mu \rho \frac{k^2}{\varepsilon} \tag{4}$$

$$\omega = \frac{\varepsilon}{k} \tag{5}$$

where ε is the dissipation rate of the turbulent kinetic energy, c_μ a coefficient and f_μ a damping function. This is a widely used model for turbomachine problems.
Both steady and transient flow simulations have been performed in order to obtain the performance curves in a turbo machine. The first type of simulation solves the steady state problem for a fixed position of the impeller, while the second one considers different positions of the impeller for different time instants. Since every point of the characteristic curve is a steady point (fixed discharge, fixed pressure) in the transient case the solution is obtained as an average of the different transient solutions.
A multi-stage pump of Caprari s.p.a. (PM100) has been simulated. In Figure 4 there is the cross-section of the machine where the multi-runners placed in series in order to impulse to a higher elevation.

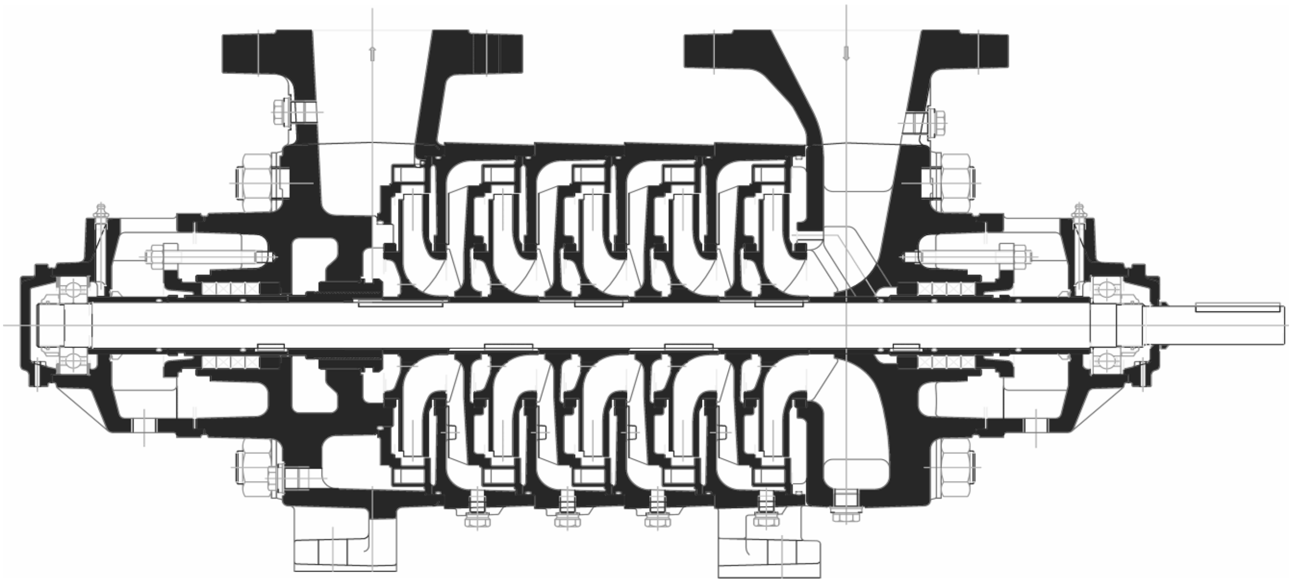

Figure 4. Cross-section of a multi-stage PAT

In Figure 5 CFD results and the comparison with experimental performance curves of head, power and efficiency are showed.
The efficiency is defined as:

$$\eta_{tot} = \frac{\rho g \Delta HQ}{\frac{\rho g \Delta HQ}{\eta_{hidr}\eta_{vol}} + P_{DF}} \tag{6}$$

where

$$\eta_{hidr} = \frac{\Delta H}{\Delta H_{eul}} \tag{7}$$

$$\eta_{vol} = \frac{Q}{Q + Q_l} \tag{8}$$

where P_{DF} is the disk friction dissipated power, ΔH_{eul} the theoretical head based on Euler theory of rotating machines, Q_l the leakage discharge flowing in the small channels of the machine. For the CFD

simulation each stage has been divided in 5 millions elements, and 3 stages have been simulated (about 15 millions elements), in order to keep the domain far from the boundaries.
The accuracy of the CFD simulations is very high, mainly for the transient case, which better simulate the stator-rotor interaction.

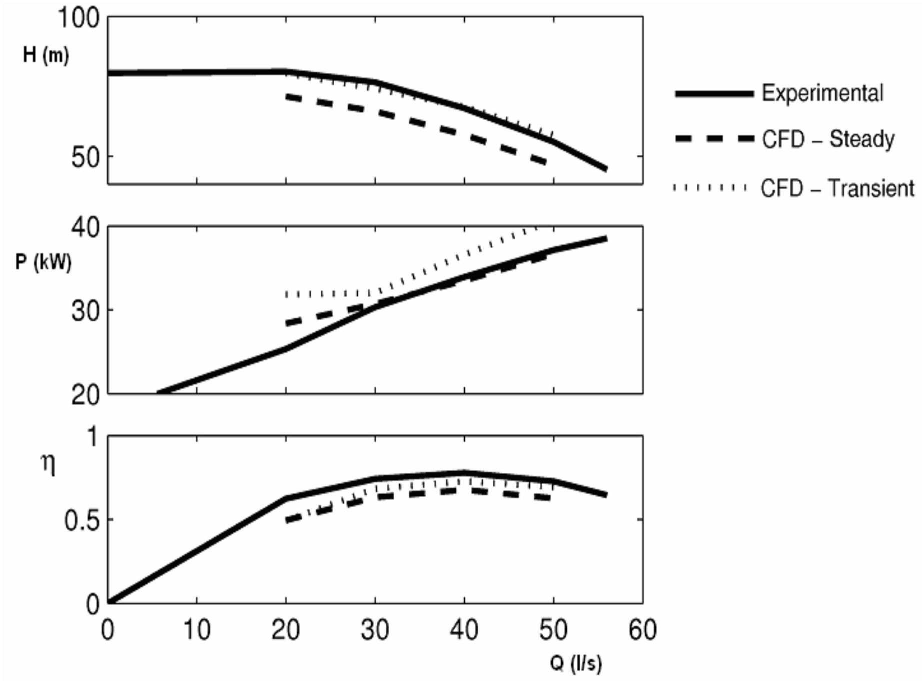

Figure 5. Performance curves in pump mode: comparisons between Experimental and CFD

3.2 Numerical mesh
In order to find the minimum number of elements of the mesh for the calculation, many simulations with 24 different meshes have been performed. The calculation has been leaded with only one stage and some artifices are considered to better simulate the internal leakages. Head and efficiency have been calculated for all simulations in order to compare the results, which are plotted in Figure 6.

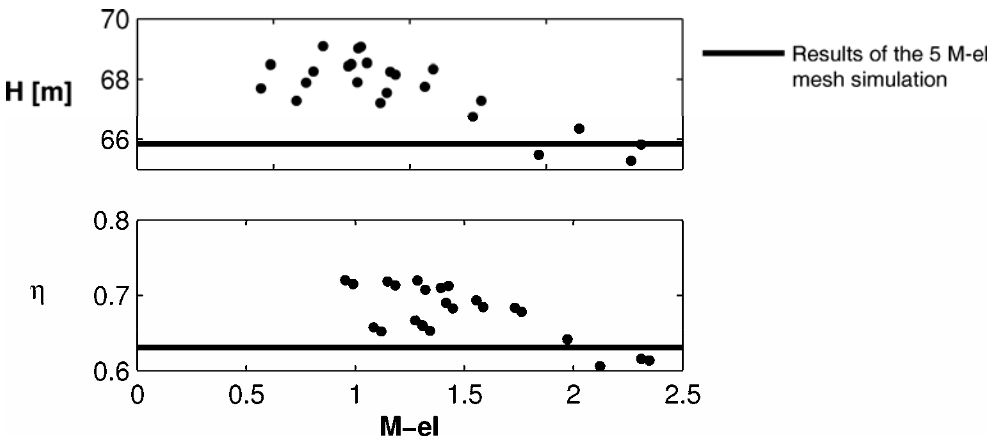

Figure 6. Head and efficiency of the pump with different CFD-meshes

The head increases with the decreasing of the number of elements, and the maximum error respect to the value obtained with the 5 million elements mesh is greater than 7.4%. The efficiency shows the same trend, with a maximum error of 18.8%. The coarser meshes also present a greater dispersion of the data.
This behaviour could be due to the presence of small internal leakages. When the elements of the mesh are bigger, the mesh is coarser and the geometrical accuracy is smaller. Since the narrowest passages are good modelled, the internal leakages are lower and the computed efficiency increases as well as the head.

While the automatic mesh generator can be set up only with some parameters, this effect seems to be bad correlated with the number of elements.
In Figure 7 the characteristic curve obtained with a simplified one-stage mesh (2.4 millions elements) is compared with the one obtained with the 15 million elements mesh. The figure shows that the differences between the two curves are very low, and the maximum errors are of 3.5% for the head and of the 7.4% for the efficiency. Also the best efficiency point (BEP) is the same in the two cases.

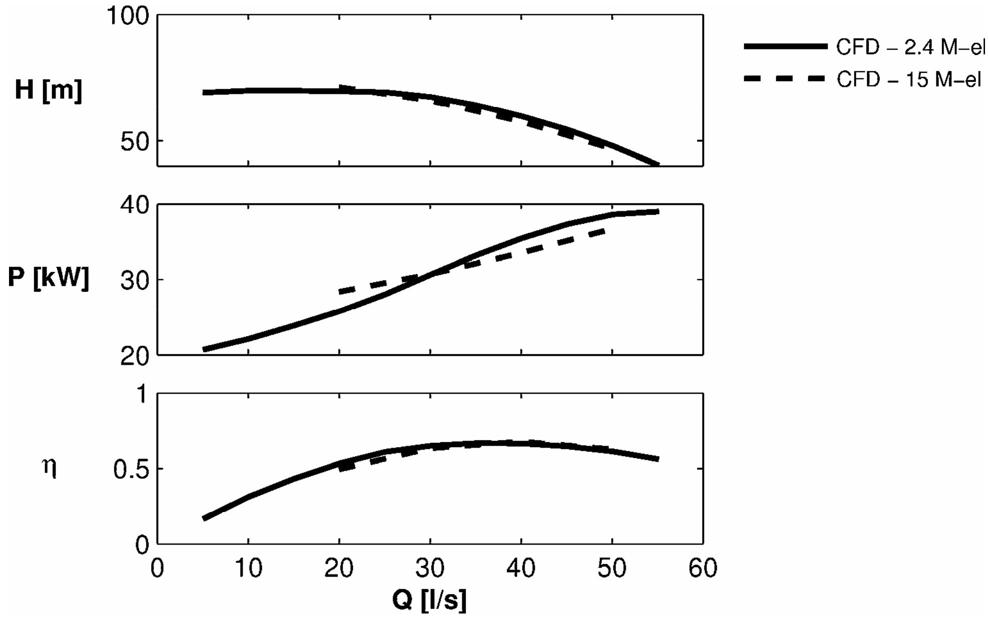

Figure 7. CFD performance curves in pump mode for different number of elements

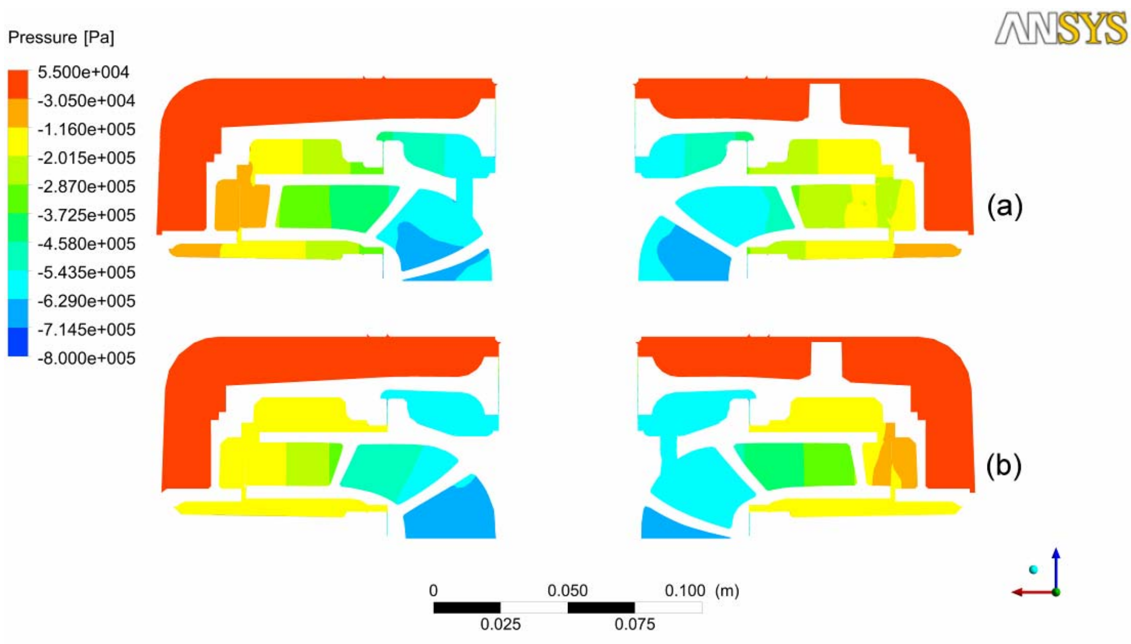

Figure 8. Comparison between pressure fields: (a) higher resolution mesh; (b) coarse mesh

Figure 8 shows that also the differences between the pressure fields inside the runner in both cases are really low. This result demonstrates that with a simplified mesh reliable solutions could be obtained both about performance quantities (head, efficiency, power) and detailed fluid dynamic field.
A reduction of 54% of the number of elements (85% considering the absence of two of three stages) leads to a reduction of 90% of the calculation time and of 84% of the occupied RAM memory. This

advantage can be very useful to simulate the transient state condition, which requires more computational resources, and to build the characteristic curves of multi-stage solutions.
After that the reliability of the simplified mesh has been tested and also the characteristic curve of the machine operating in turbine mode has been produced (Figure 9).

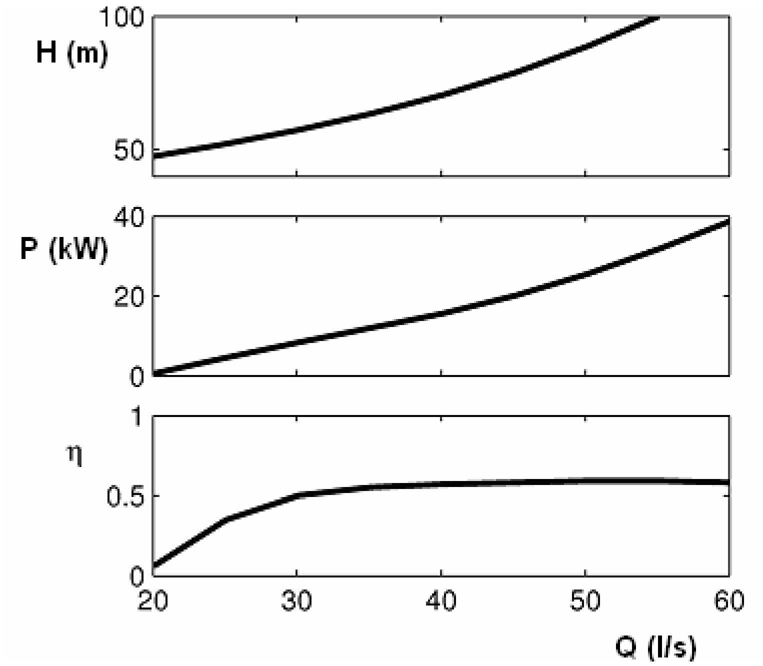

Figure 9. Performance curves in the turbine mode

Figure 9 shows that the discharge value of the BEP is bigger than the one in pump mode, but the best efficiency is almost the same.

3.3 Transient state simulation
A transient state simulation has been developed in order to preliminary evaluate the inertial behaviour of the machine in transient conditions. Two simulations have been led, with uncompressible (rigid) and compressible fluid (elastic) model, respectively. In Figure 10 the time-head plot is showed after a sudden finite variation of discharge, from 35 to 30 l/s with a rigid model. After a sudden initial drop of the head, the machine gradually tends to new steady conditions. These conditions are achieved after 1/3 rounds of the impeller. This time interval is comparable with the time of discharge and head variation in water hammer conditions after, for example, an abrupt valve closing.

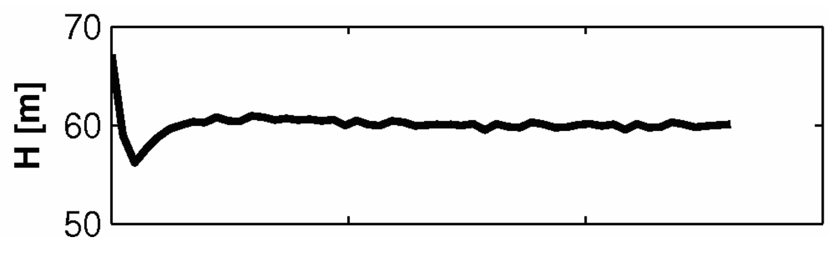

Figure 10. Head variation after a sudden discharge variation (rigid model)

For the elastic model (compressible fluid behaviour) simulation the density of water has been modelled by the following equation [10]:

$$\rho = \rho_0 + \frac{(p - p_0)}{{c_0}^2} \tag{9}$$

where ρ_0=996 kg/m^3 is the density of the water at the reference pressure p_0=1 bar and c_0=1480 m/s the sound speed, assumed constant.
The Figure11 shows the hydrodynamic behaviour of the PAT:

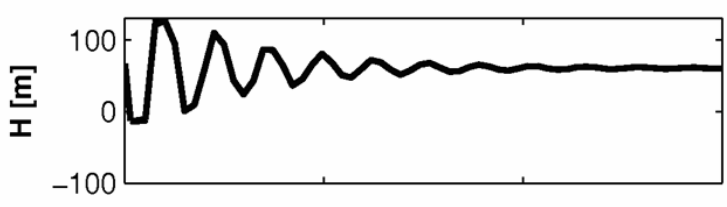

Figure 11. Head variation after a sudden discharge variation (elastic model)

After the variation of the discharge, a pressure wave is originated and reflected, causing severe head fluctuations. These fluctuations are much wider than the maximum upsurge calculated with the rigid model simulation and the effects of the discharge variation takes a longer time to be dissipated.

4. Conclusions

The pressure control in drinking pipe systems has enabled this type of analysis based on the profit of excess available energy that would be dissipated in special head loss devices, such has pressure reducing valves (PRV). Experimental analyses have shown an equivalent behaviour between PAT and PRV for steady state conditions and some important differences under transient conditions, which the PAT allows the pressure waves travelling through the impeller a propagate along the whole pipeline.
Comparisons between CFD simulations and experiments for pumps are analysed. Numerical simulations show that the CFD allows predicting the behaviour of a turbo machine both in pump and in turbine mode. The reliability of the solution depends on the type of flow regime condition (steady vs. transient) and on the number of elements considered for the fluid dynamic mesh.
Transient calculations better simulate the real behaviour of the machine because they better reproduce the interactions between the rotor, the stator and the hydrodynamic conditions.
Several simulations have been compared with different meshes, in order to find the smallest mesh that allows minimizing the resource request and obtaining a reliable result. This work shows which is the number of elements that will influence the solution and the geometrical modelling of the narrow flow passages inside the machine, which is a critical point in the mesh building process.
A preliminary calculation in transient state shows the machine response to a sudden discharge variation, which takes time that is comparable with the time of the variation of hydraulic characteristics in a pipeline under water-hammer conditions.

Acknowledgements

The authors wish to thank to Caprari S.P.A. and especially Eng. Lauro Antipodi, Eng. Francesca Turini and Eng. Franco Strozzi for the support to the research, the International Office of the University of Naples for refunding a part of the research with the“short mobility project”, as well as FCT (PTDC/ECM/65731/2006) and CEHIDRO, the research centre from the Department of Civil Engineering at Instituto Superior Técnico, Lisbon, Portugal, which also supported part of this research work.

References

[1] Araujo, L.S.; Ramos, H.; Coelho, S.T. (2006), Pressure control for Leakage Minimisation in Water Distribution Systems Management. Water Resources Management, NL. ISSN: 0920-4741, Vol. 20, Number 1, February, pp. 133 – 149, 2006.
[2] Ramos, H.; Borga, A. (2000a), “Pump as Turbines: an Unconventional Solution to Energy Production”. Journal URBWAT, Vol. 1, N.? 3, pp. 261-265.
[3] Ramos, H.; Borga, A. (2000b), “Pumps yielding power”. Dam Engineering, Water Power & Dam Construction, Volume X, Issue 4, pp.197-217, UK, ISSN 0-617-00563-X, 2000.
[4] Carravetta A., Giugni M. (2009), Functionality factors in the management and rehabilitation of water networks, in Management of Water Networks, Proceedings of the Conference “Efficient management of water networks. Design and Rehabilitation Techniques”, Franco Angeli Ed. Milano.

[5] Derakhshan, S., Nourbakhsh, A. (2008), Experimental study of characteristic curves of centrifugal pumps working as turbines in different specific speeds, Experimental Thermal and Fluid Science, vol. 32, pp 800-807.
[6] Ramos, H., Almeida, A. B. (2002), Parametric analysis of water-hammer effects in small hydro schemes, Journal of Hydraulic Engineering, ASCE, vol. 128(7), pp. 1-8.
[7] Ramos, H., Almeida, A. B. (2001), Dynamic orifice model on water-hammer analysis of high or medium heads of small hydropower plants, Journal of Hydraulic Research, IAHR, vol. 39(4), pp. 429-436.
[8] Ramos, H., Covas, D., Araujo, L.S. (2004), Hydro potential in drinking pipe systems. Hydro 2004, Porto, Published by The International Journal on Hydropower & Dams, 3.06, Hydropower planning and development: Innovative and challenging case studies. Aqua~Media International, UK, 2004.
[9] Ansys inc., (2006), ANSYS CFX Reference guide, Ansys Inc., Canonsburg.
[10] Yan, J., Koutnik, J., Seidel, U., Hübner, B., Scherer, T., (2009), Compressible Simulation of Rotor-Stator Interaction in a Pump-Turbine, 3rd IAHR international Meeteing of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, October 14-16, 2009, Brno.

Oreste Fecarotta is a Ph.D. student of the Department of hydraulic, Geotechnical and Environmental Engineering of University of Naples. He worked on rheology of iper-concentrated mixtures, turbomachine and Computational Fluid Dynamics.
Email address: oreste.fecarotta@unina.it / oreste.fecarotta@gmail.com

Armando Carravetta has Ph.D. degree in Hydraulic Engineering and is Professor at University of Naples Federico II (Italy) at Department of Hydraulic, Geotechnical and Environmental Engineering. Director of the laboratory of Hydraulics and maritime engineering of the Department. Main research activities: energy efficiency and functionality of pressure water networks, river flow and erosion, groundwater and solute transport in soils. More than sixty papers on National/International Proceedings and on National/International Journals. Owner of national and international patent pending procedures for hydraulic machineries. Author of a manual on hydraulic machineries for pressure irrigation networks.
Co-editor of the Proceedings of IAHR Conference Riverflow2004. Supervisor of several post-doc, PhDs and MSc students.
Email address: armando.carravetta@unina.it Web address: http://www.docenti.unina.it/armando.carravetta

Helena M. Ramos has Ph.D. degree and is Professor at Instituto Superior Técnico (from Technical University of Lisbon - Portugal) at Department of Civil Engineering. Expert in different scientific domains: Hydraulics, Hydrotransients, Hydropower, Pumping Systems, Leakage Control, Energy Efficiency and Renewable Energy Sources, Water Supply, Vulnerability. More than 250 publications being 1 book in Small Hydro, 50 in International Journals with referee and more than one hundred in International Conferences; Supervisor of several post-doc, PhDs and MSc students and author of 8 innovative real solutions in the domain of Civil Engineering - hydropower and hydraulic system control.
Email address: hr@civil.ist.utl.pt / hramos.ist@gmail.com
Web site: https://fenix.ist.utl.pt/homepage/ist12110