

Chapter 2

Reverse Pump Theory

Abstract Users, researchers and engineers require a fundamental knowledge of reverse pump theory. In this chapter first, a complete description of all PAT components is given. Then, the theory of small reverse pumps is described, based on the results of theoretical analyses, experiments and numerical simulations. PAT characteristic and efficiency curves are introduced and, the influence of pump size and rotational speed on such curves is considered. The possibility of using numerical simulations instead of experimental results is discussed. Finally, additional aspects like cavitation and the incidence of pressure surges are presented and different tools, commonly used for PAT sizing and hydropower plant design, are described. The reader will be prepared, by means of the theoretical basis provided, for the comprehension of the hydraulic and mechanical processes connected with a PAT operation.

Around 1930, Thoma started his research on PATs by chance during tests to obtain the characteristic curve of a pump. However, the first complete characteristic curves of a pump working in turbine mode were presented by Knap in 1937 [1]. The hydraulic behavior of a pump, when it rotates as a turbine, has changed significantly. Although, the rotation speed and flow are reverse in turbine mode, the behavior of a PAT is completely different compared with its pump mode. In general, a pump works in turbine mode with higher heads and discharges for the same rotational speed. In the past, the electromechanical characterization of a pump working in turbine mode was limited to pump and storage hydro power plants, with the installed power as large as hundreds of megawatts, as in the case of the Presenzano power plant in Italy, which is equipped with two pump/turbines with 250 MW each. In recent years the interest in the use of small pumps in mini and micro hydro power plants has increased. The behavior of these small PATs is extremely different when compared with the huge pumps of the past, because industrial PATs are generally used without any optimization of the geometry for reverse mode application.

2.1 PAT Components

Depending on the PAT type, the components of each machine are different. However, in general, PAT components are classified in two groups: rotary and stationary. The rotary group consists of the impeller and shaft and the stationary group includes the inlet parts (i.e., the inlet pipe and volute), outlet parts (i.e., the suction pipe and draft tube), and the casing and mechanical parts (i.e., rings, mechanical seals and bearings). However, the main hydraulic parts of a PAT are the impeller, volute and draft tube (Fig. 2.1).

2.1.1 The Volute

Although the volute of a PAT is designed to collect the water from the impeller of a pump, the design method for a pump is quite similar to a Francis turbine volute (i.e.,

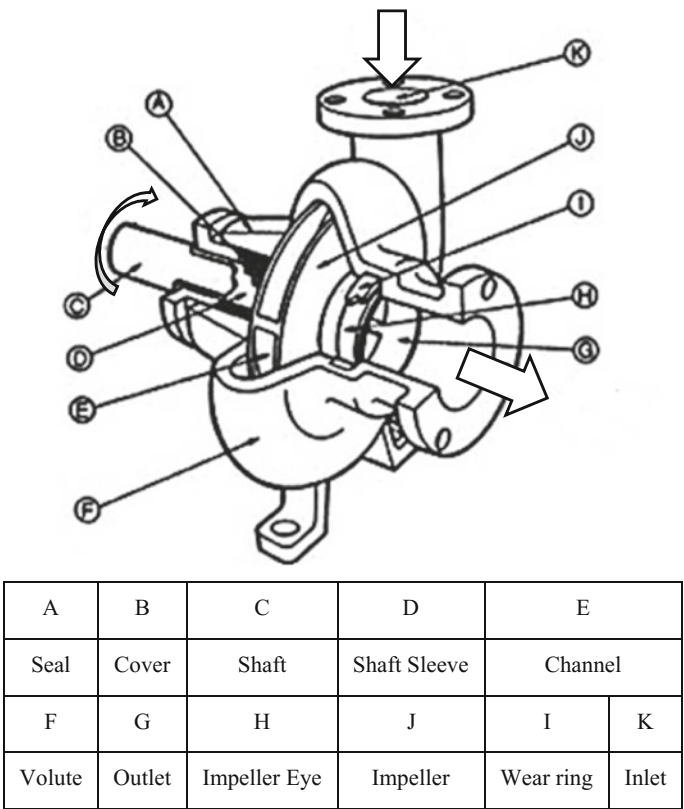
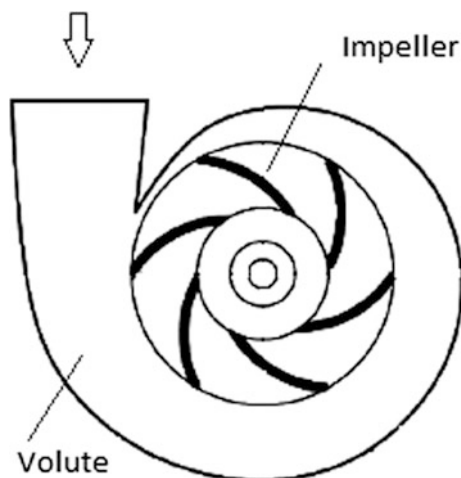


Fig. 2.1 PAT components (www.processindustryforum.com)

Fig. 2.2 Plan view of a PAT
[18]



a constant velocity method). Therefore, when the flow is reverse, the volute of a PAT will work properly as a distributor of the flow surrounding the impeller. However, the most important role of a volute is the balancing of radial pressure and forces. Figure 2.2 shows a typical volute of a PAT.

2.1.2 The Impeller

The impeller, the main component of a PAT, converts the hydraulic energy of the inlet flow to the mechanical shaft. Although the impeller has been designed to pressurize the liquid using the centrifugal force, it can also absorb the flow energy with a reasonable efficiency. The impeller usually has 5–8 blades, with a hydrofoil shape type. An impeller and the details of a blade are presented in Fig. 2.3.

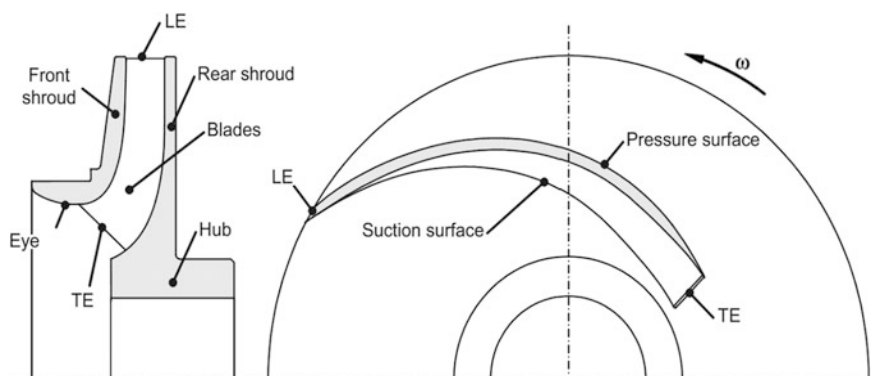
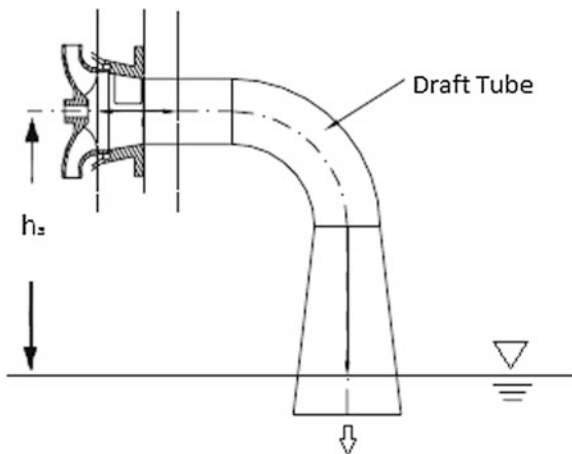


Fig. 2.3 Impeller and details of a blade [19]

Fig. 2.4 Draft tube of a PAT
[20]



2.1.3 The Draft Tube

A small kinetic energy is required at the PAT outflow to minimize energy losses. Using a draft tube, the kinetic energy can be converted into potential energy. Therefore, a draft tube is designed based on the maximum conversion of velocity to pressure (according to the Bernoulli equation). Figure 2.4 shows a typical draft tube of a PAT.

2.2 PAT Theory

When a pump runs as a turbine, the high pressure fluid enters the machine and then leaves after the conversion of the energy has occurred within the impeller. Since the flow direction and the rotational speed are reverse, the velocity triangles of both modes change.

The pump and outlet velocity triangles in direct and reverse modes are shown in Fig. 2.5. Although only the shaft rotation and flow direction are opposite in pump and turbine modes, their velocity triangles are different. Considerations show that the inlet flow angle to the impeller in a PAT is not exactly the same as the outlet flow angle of the impeller in the pump mode. This angle (α_v) is approximately equal to the volute angle. In fact, the volute operates as a guide channel in turbine mode. Assuming the outlet flow angle from the impeller in reverse operation is equal to the impeller inlet flow angle in direct operation (i.e., with no whirl at the impeller exit), the same Euler heads for both modes can be considered [2]:

$$H_{pE} = H_{tE} \quad (2.1)$$

Due to the slip factor related to the finite blade number, the pump and turbine theoretical heads can be written as:

$$H_p'' = \xi_p H_{pE} \quad (2.2)$$

$$H_t'' = \frac{H_{tE}}{\xi_t} \quad (2.3)$$

where ξ_p is the slip factor for the pump operation ($\xi_p < 1$) and ξ_t is the slip factor for the turbine operation, that is approximately equal to 1.0 [2].

Considering the hydraulic efficiency for each mode, Eqs. (2.2) and (2.3) can be rewritten as:

$$H_p = \eta_{hp} H_p'' = \psi \eta_{hp} H_{pE} \quad (2.4)$$

$$H_t = \frac{H_t''}{\eta_{ht}} = \frac{H_{tE}}{\psi \eta_{ht}} \quad (2.5)$$

resulting in:

$$h = \frac{H_t}{H_p} = \frac{\left(\frac{1}{\psi^3}\right)}{\eta_{hp} \eta_{ht}} \quad (2.6)$$

The hydraulic efficiency of a pump at its best efficiency point (BEP) can be computed using its overall efficiency by [2]:

$$\eta \approx \eta_h^{0.5} \quad (2.7)$$

Finally, the turbine and pump head ratio can be summarized as:

$$h = \frac{H_t}{H_p} = \frac{b}{\eta_p^a}, \quad (a > 1.0, \quad b > 1.0) \quad (2.8)$$

Assuming $\eta_p \approx \eta_t$. Therefore, a pump will work in higher heads in turbine mode ($h > 1.0$).

The leakage ratio for reverse and direct operations is [3]:

$$\frac{Q_{lt}}{Q_{lp}} = \sqrt{\frac{H_t}{H_p}} = \frac{b^{0.5}}{\eta_p^{0.5a}} \quad (2.9)$$

In addition, the theoretical flow rate ratio for turbine and pump modes is [3]:

$$\frac{Q_t''}{Q_p''} = \sqrt{\frac{H_t}{H_p}} = \frac{b^{0.5}}{\eta_p^{0.5a}} \quad (2.10)$$

Thus, the actual flow rate ratio is:

$$q = \frac{Q_t}{Q_p} = \frac{Q_t'' + Q_{lt}}{Q_p'' - Q_{lp}} \approx \left(\frac{Q_p'' + Q_{lp}}{Q_p'' - Q_{lp}} \right) \frac{b^{0.5}}{\eta_p^{0.5a}} = c \frac{b^{0.5}}{\eta_p^{0.5a}} \quad (2.11)$$

where $c > 1.0$. Therefore, a pump will work in higher flow rates in turbine mode ($q > 1.0$).

The values of a , b and c may change for various pumps depending on the pump specific speed and size, mechanical tolerance and manufacturing accuracy. To summarize, a PAT will work in higher heads and flow rates in turbine mode.

2.3 Flow Pattern in a Pump and in a PAT

Based on expert literature, it has now become clear that in reverse mode:

- the mechanical operation is very smooth and without noise;
- the maximum efficiency of the reverse mode is approximately equal to the maximum efficiency of the direct mode;
- the output power at its BEP of a PAT is greater than the absorbed power of a pump;
- the head and the flow rate at its BEP in reverse mode are higher than those in direct mode.

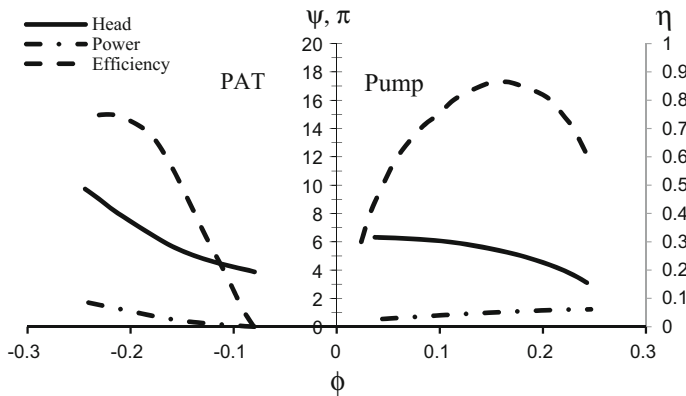
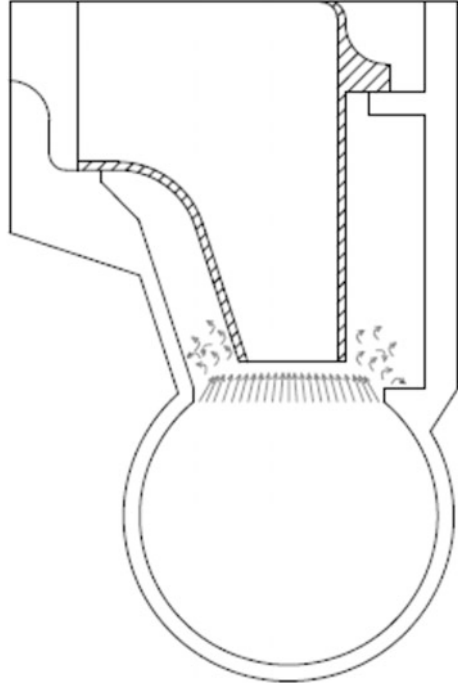


Fig. 2.6 Characteristic curves of a pump in direct and reverse modes [1]

Fig. 2.7 Shock and leakage flow at the entry of an impeller in a PAT [5]



On the basis of these important results, the use of a PAT is an acceptable solution from the technical point of view. Dimensionless characteristic curves of a pump in direct and reverse modes are compared in Fig. 2.6.

Compared to the direct mode, a pump in reverse mode works in higher heads, flow rates and powers, but it still shows at its BEP a good value of efficiency [1]. Therefore, the most challenging question is: why are the maximum efficiencies so high? The pump was not originally designed for reverse working; however, it works at the same efficiency in turbine mode! To answer this question, losses in both modes should be investigated. For instance, how great are the shock losses in an inlet of an impeller in turbine mode compared to those in the pump mode? In fact, the hydraulic losses are seemingly equal in both modes. Shock losses in the pump mode usually occur at low velocities. But, due to the absence of a guide vane in a PAT, these losses are not under control (Fig. 2.7 shows the shock and the leakage flow at the entrance of the impeller in a PAT). In addition, when passing from an increased flow rate in a convergence channel (i.e., in reverse mode) to a divergence channel (i.e., in direct mode), the friction losses increase. Finally, due to a higher pressure of the flow in a PAT, the leakages are also greater. It seems that there is a balancing between the various lead losses at the same maximum hydraulic efficiency (at BEP) in both modes (Fig. 2.8).

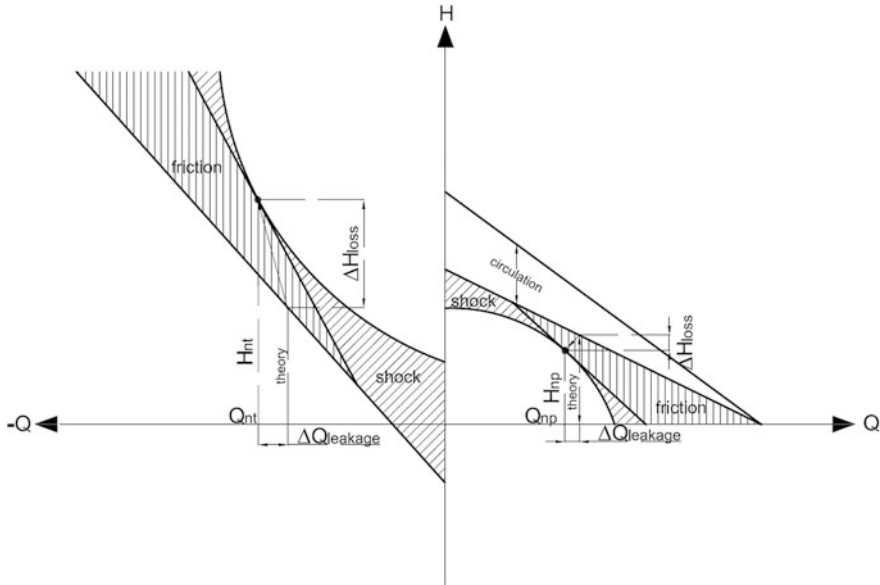


Fig. 2.8 Comparison of losses in direct and reverse modes [9]

Therefore, it is believed that at the BEP of turbine and pump modes:

- leakages in reverse mode are higher than in direct mode;
- losses in the volute of a PAT (the convergence path) are less than those in the pump mode (the divergence path);
- friction losses in the impeller are similar in both modes: on the one hand, they increase due to the higher flow velocities in PATs and, on the other, they decrease due to the convergence path of the flow;
- if we consider the BEP as the shock less point, head–flow rate curves can be divided into two sections: flow rates less than the BEP flow rate and flow rates greater than the BEP flow rate. Shock losses are considerably higher in the right hand side of the curve of Fig. 2.8. Therefore, it is strongly recommended that a PAT must be selected to work in the left hand side of its curve in Fig. 2.8. Low rotational speeds of a PAT are not recommended.

Given these issues, shock losses have an important role in PAT behavior. Therefore, for a certain head and flow rate, selecting a PAT with a higher specific speed is preferred. Stepanoff argued that every good pump can work as a reasonable turbine; however, each good turbine may not be a suitable pump. It is true even for multistage pumps [4].

PATs, although suitable for energy production, are very sensitive. Due to the higher working pressure, the leakages are greater. Therefore, it may be necessary to improve the seals of the machine. In addition, due to the higher radial and axial forces, the life of its bearings is shorter. Table 2.1 shows some of the advantages and disadvantages of PATs.

Table 2.1 Advantages and disadvantages of a PAT operation

Operation	Advantages	Disadvantages
Hydraulic	Reasonable peak efficiency at the BEP	Narrow efficiency curve in off-design points
	Adjusting head-flow rate by impeller trimming or using a specific rotational speed	Higher PAT costs for impeller trimming or velocity variation
Mechanical	Operates without noise	Mechanical seal required, due to higher leakage
	Vibrations are acceptable	Short life cycle of bearings due to higher power
	Capability of using a belt instead of direct coupling to a generator to change the PAT speed	Strengthening of the shaft and coupling required

Last but not least, their lower efficiency, in comparison with conventional turbines, is the major negative point of PATs. Due to the lack of any guide vane, a PAT has a lower efficiency in off-design points in comparison with conventional turbines. Therefore, it is important that the pump geometry is modified for use as a PAT in higher power situations.

2.4 Application of Computational Fluid Dynamics (CFD) to a PAT

2.4.1 Importance of CFD

There are some hydrodynamic and structural fundamental requirements that all hydraulic circuit designs have to respect. The variables that have an influence on the hydraulics, energy efficiency and design are dependent on the local inlet discharge, as well as on the configuration of the circuit elements (i.e., the trash rack, water intakes and all the particular features of the hydraulic circuit, turbines and tail race). All these components, included in the hydraulic circuit, will affect the available head and the turbine discharge. Characteristics which are normally associated with inefficiency and unsafe conditions include separate flow zones, vorticity, macro turbulence intensity, induced vibrations, resonance effects, leakages, ruptures or collapses, pressure fluctuations, cavitation and water column separation, significant friction losses due to shear stress development, huge local pressure gradients, longitudinal swirl vortex and regions of reversed recirculating flow.

Three decades ago, the absence of CFD models led to the use of theoretical analysis to understand the behavior of a PAT. However, due to the one-dimensional (1D) nature of such analysis, presenting a complete flow pattern in a PAT was impossible. In fact, there are many losses, such as secondary flows, that cannot be computed by one-dimensional models. On the other hand, using experimental methods to capture the flow behavior in a PAT is really expensive and laboratory tests, measuring external data such as inlet pressure, outlet pressure, flow rate, rotational speed and power, are limited in the identification of a PAT's characteristic curves.

Nowadays, CFD models are powerful tools for researchers and pump manufacturers for the following purposes:

- simulating the PAT working conditions for a complete set of boundary conditions, whenever only the pump internal geometry is known and experimental tests are incomplete or too expensive;
- optimizing the internal geometry of the PAT to increase its efficiency;
- verifying the influence of the PAT in the network for water-hammer analyses.

When the flow direction completely changes during the turbine mode, the flow enters the volute and then the impeller, leaving the machine through the suction pipe, which is now called the draft tube. Although the geometry of a pump is fixed when rotating as a turbine, the flow pattern is quite different from its normal rotation. Fortunately, CFD analyses can clearly show the flow field and the flow pattern. Some researchers have tested the use of CFD to evaluate a pump working as a turbine [2, 5–7]

In this section, CFD codes, from commercial solvers, are used to demonstrate how a better understanding of the behavior of a PAT can be achieved.

2.4.2 Mathematical Approach

Navier-Stokes equations have a limited number of known analytical solutions, but they are adequate for flow computational models, by means of numerical approaches. CFD models solve Navier-Stokes equations, which are formulations of momentum plus mass and energy conservation laws for fluid flows. The equations are supplemented by fluid state equations defining the nature of the fluid, and by empirical dependencies of fluid density, viscosity and thermal conductivity on temperature.

Equations (2.12) show Navier-Stokes equations for incompressible flows. These equations are based on differential equations of linear momentum for a Newtonian fluid with density and viscosity as constants.

$$\begin{aligned}
 \rho g_x - \frac{dp}{dx} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) &= \rho \frac{du}{dt} \\
 \rho g_y - \frac{dp}{dy} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) &= \rho \frac{dv}{dt} \\
 \rho g_z - \frac{dp}{dz} + \mu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) &= \rho \frac{dw}{dt}
 \end{aligned} \tag{2.12}$$

where: ρ : volumetric mass (kg/m^3); g : acceleration of gravity (m/s^2); p : pressure (Pa); μ : dynamic viscosity (kg/ms); u , v and w : velocity components for each moving fluid particle, depending on x , y and z coordinates for a given t instant (m/s).

The incompressible fluid flow behavior is determined by the velocity and pressure variables and their variations in time and space. In Eq. (2.12), that allow the attainment of pressure and velocity fields, the velocity components at each point x , y and z are vector fields and the pressure at each point is a scalar quantity.

Most of the flows that occur at the hydraulic circuits are turbulent, and CFD models allow the numerical modeling of both laminar and turbulent flow regimes. The turbulent flow occurs for high values of the Reynolds number, given by the Eq. (2.13):

$$Re = \frac{\rho U d}{\mu} \quad (2.13)$$

where: d : is the pipe diameter (m) and U : is the flow mean velocity (m/s). When the flow is turbulent, the variables present at each instant random fluctuations.

A fluid under low pressures can reach the vapor pressure at the local temperature leading to the formation of vaporous cavities. The fluid undergoes a phase change and cavities filled with vapor fluid and other dissolved gases are formed. When analyzing areas subject to flow conditions that lead to the occurrence of cavitation, the CFD model uses a homogeneous equilibrium model of cavitation in water.

CFD allows many calculations both in steady conditions (i.e., the performance curves of the machine) and in transient flow regimes (i.e., machine stoppage and start-up, valve closure and load-rejection). This kind of calculation can be very complex and the computational complexity increases when dynamic simulations are performed. Thus, great attention is necessary in order to find the minimum number of elements in the 1D or 3D fluid dynamic mesh necessary to obtain reliable results for the numerical PAT simulations.

2.4.3 PAT Geometry Modeling and Mesh Generation

To simulate adequately a PAT, the generation of a reliable geometry is very important. The use of CAD/CAM software allows a reasonable model of a real pump. The geometry should be derived according to the exact dimensions of the prototype. The flow field is split into five component parts: the volute, impeller, front and back chambers and outlet pipe, as shown in Fig. 2.9. This separation allows each mesh to be generated individually and tailored to the flow requirements in each particular component. To obtain a relatively stable inlet and outlet flow, the pipe diameter should be extended four or five times in the PAT inlet and outlet sections.

As the motion of the impeller blades relative to the stationary volute is crucial, the analysis must involve multiple frames of reference. The volute and the outlet pipe are set in the stationary frame and the impeller is set in the rotary frame. The interfaces between the two stationary components, the rotary and stationary components, are set to general grid and rotor-stator interface, respectively.

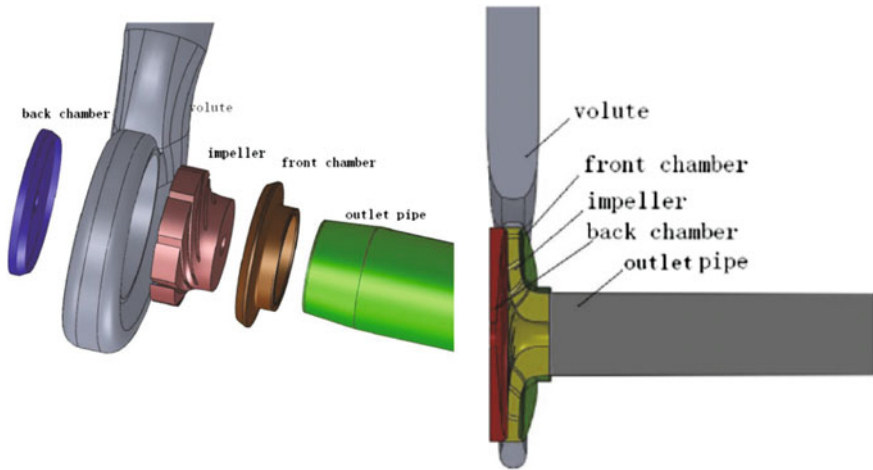


Fig. 2.9 CAD/CAM geometry of a PAT [2, 21]

In the process of numerical simulation, all domains except the leakage through the balancing holes can be included. Therefore, the simulated efficiency includes volumetric loss through the wear ring and flow loss caused by disc friction. The volumetric loss caused by the balancing holes are usually neglected.

Different turbulence models are available for the simulation a PAT. However, in the authors' experience, $k-\varepsilon$ RNG and SST $k-\omega$ are the best models to evaluate Reynolds Stresses in Navier-Stokes equations. The fluid properties must be considered in accordance with the site information. For instance, ideal water at 25 °C can be assumed. All the wall surface roughness within the control volume should be selected based on the real pump.

Some boundary condition strategies are available. However, one recommended strategy is to use the static pressure for the inlet and the mass flow rate for the outlet [2]. By changing the mass flow rate, the performance curves of the PAT can be achieved.

All numerical simulations face the necessity of making a compromise between results accuracy and CPU time. In the CFD of a PAT, the primary factors increasing calculation times are the small dimensions of the wear rings compared to the dimensions of the computational domain and the time variability of the process for the rotary frame of the impeller.

Several mesh models can be used to generate structured or unstructured grids for each component. The possibility of using a variable grid dimension in the computational domain is exploited to reduce the total number of elements without increasing the mesh size in the wear ring zone. Due to the complexity of generating a structured mesh based on the geometry, an unstructured grid can be generated for the volute. In all the other regions, the use of a structured mesh is generally preferred, especially for the impeller and for the inlet and the outlet pipes. In some cases, due to the small size of the elements in the wear ring zone, an unstructured

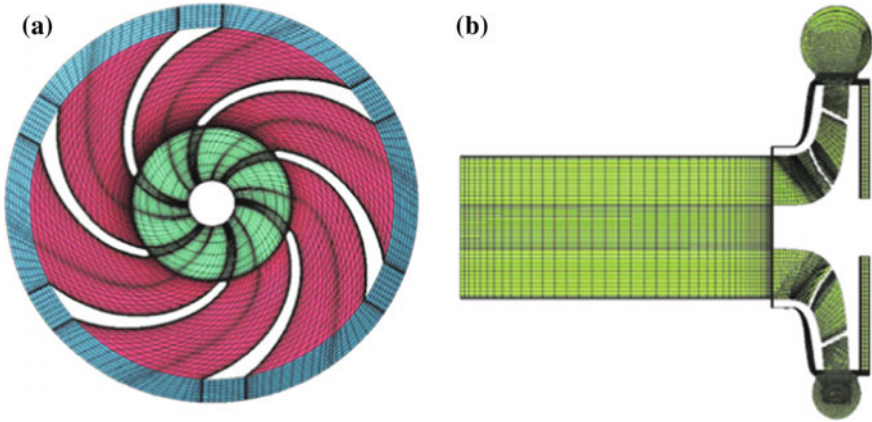


Fig. 2.10 A sample generated mesh for a PAT [2]: **a** Impeller, **b** Cut by the draft tube and impeller

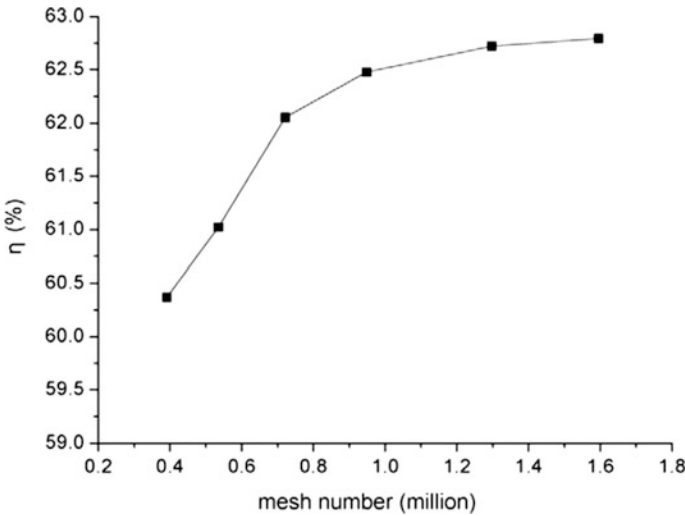


Fig. 2.11 Mesh independency curve with a structured grid [2]

grid is used in that section to reduce the time required to set up the model. Figure 2.10 gives a general view of a structured sample mesh.

A grid independent test of the turbine mode's performance should be performed with both kinds of grid. Figures 2.11 and 2.12 show a sample of this strategy for structured and unstructured grid, respectively. As shown in Fig. 2.11, the effect of the refinement of a structured grid is an increase in the apparent machine efficiency, approaching the experimental value. Reducing the mesh size, the narrowest passages of the pump are better modeled, as the internal leakages are lower, and, additionally, the computed efficiency is increased. When the mesh numbers are around one million, the variation of efficiency is about 0.5%, as indicated. The same

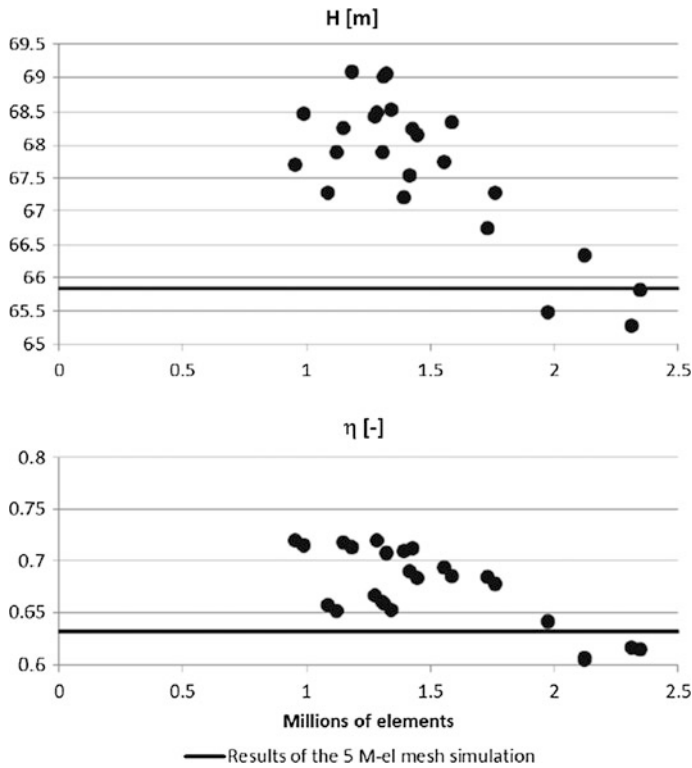


Fig. 2.12 Mesh independency curve with an unstructured grid [6]

behavior is observed in Fig. 2.12, where the results of many simulations with an SST $k-\omega$ turbulence model on 24 meshes of different minimum element sizes are plotted. The head increases with the decrease in the number of elements, and the maximum error in relation 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 spreading of the CFD results is determined by the uncontrolled mesh geometry of the unstructured grid.

Special care should be taken in the machine inlet pipe mesh generation, in order to ensure that the mesh between the interface of the inlet pipe and the front chamber leakage is almost the same. In the case of multistage pumps, a minimum number of three stages has to be simulated. Thus, the domain of the middle stage is far from the boundaries and its fluid dynamic field is reliable.

y^+ takes into account the effect of the wall on the shear stress. Reynolds models aim to identify laminar sub-layers that can be used to calculate the height of the first near-wall cell based on a specific y^+ value in order to determine the effect of wall friction more accurately. An appropriate value of y^+ has to be selected based on the turbulence model and wall-treatment method for a robust model simulation.

The second aspect that could have an influence on the CFD results is the kind of flow field variability considered in the numerical problem. Both steady and transient flow simulations can be performed in order to obtain the performance curves of a turbo-machine. The first type of simulation solves the steady state problem for a fixed position of the impeller, while the second considers different positions of the impeller for different time instants.

In the transient case, the solution is obtained as an average of the different transient solutions. Obviously, steady state calculations significantly reduce the computation time, but appreciable differences have been observed between steady state solutions and the experimental results.

2.4.4 CFD Solutions for a PAT

Based on CFD models, a simulation of the hydrodynamic flow in a PAT can be performed [8]. The computational analysis of each test aims to:

- obtain the distribution of descriptive physical parameters of the hydrodynamic flow in planes that intersect the representative geometric model of a PAT;
- make a comparison between the velocity diagrams obtained experimentally, in the field, and computationally.

In accordance with Sect. 2.4.2, in order to make a computer simulation of the flow in the PAT and its installation, it is necessary to build a representative geometric model by using a CAD system or a solid work which is then imported into the CFD model (Fig. 2.13). The geometric model constructed results from the meeting of several independent solid components, of which the most important are the rotor and the volute.

The rotational behavior induced in the flow by the impeller rotational speed and the shape of the respective vanes is observed in Fig. 2.14a. This variation results from the centrifugal force induced by the rotor rotation.

The flow enters the volute and focuses radially on the rotor, which imposes a speed of rotation. In turn, when the rotation of the rotor continuously varies the

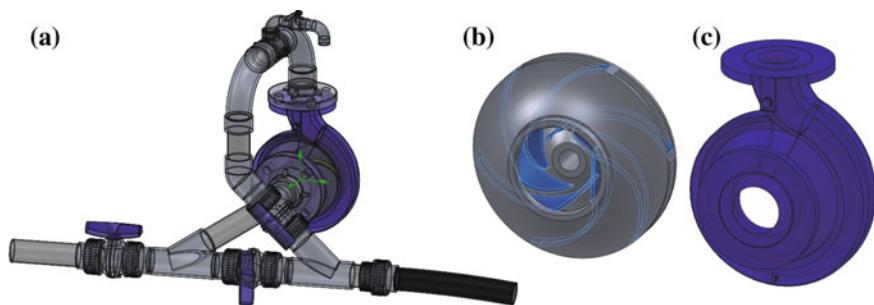


Fig. 2.13 PAT model: **a** Complete geometric model, **b** the PAT rotor, and **c** the volute [8]

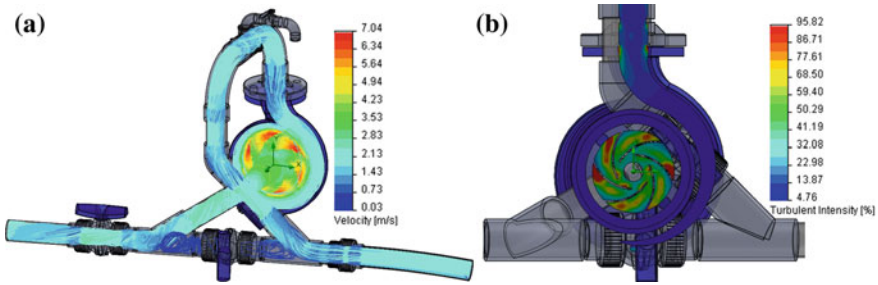


Fig. 2.14 Flow condition: **a** Stream lines (m/s) along the geometric model, and **b** the turbulence intensity distribution (%) in a longitudinal plane of the rotor [8]

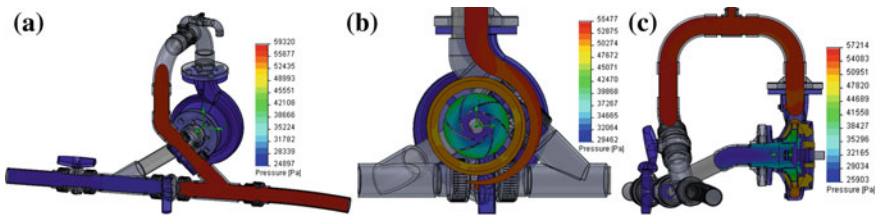


Fig. 2.15 Static pressure distribution: **a** a longitudinal plane at the ball valves, **b** a longitudinal plane of the rotor, and **c** in a plane transverse to the rotor [8]

direction of the flow along the passage through the rotor, consequently, the outlet flow direction is changed. The continuous variation of the direction of the flow gives rise to a centrifugal force, which has the effect of spacing the flow rotor rotation axis concentrating it on the periphery. The presence of this hydrodynamic behavior justifies the higher flow velocity values which occur at the rotor periphery. The pair with the highest tangential velocity values also occurs on the periphery of the rotor values with the greatest turbulence intensity, as seen in Fig. 2.14b. The rotation of the rotor also induces the increase in the flow turbulence intensity.

Since the PAT converts mechanical to electrical energy, the pressure that exists in the PAT downstream of the rotor is lower than the pressure upstream (Fig. 2.15a), creating the available head. The differential pressure reflects the net head for each turbine's operating conditions. Figure 2.15b shows a reduction in the pressure from upstream to downstream and along the flow passage in the rotor. It shows the extraction of energy from flow pressure, promoted by the rotor.

When a vortex appears in the rotor's outlet and extends downstream, associated with the turbulence and hydrodynamic instability, pressure fluctuations and loss of efficiency occur. Figure 2.15b, c show lower values of static pressure, respectively along the rotor axis and at the axis of the diffuser duct, which is the core of the vortex that is formed at the output of the downstream rotor. The pressure reduction characteristic of the cores of the vortices that occurs downstream of the rotors can lead to cavitation. Under certain operating conditions, the lower values of pressure

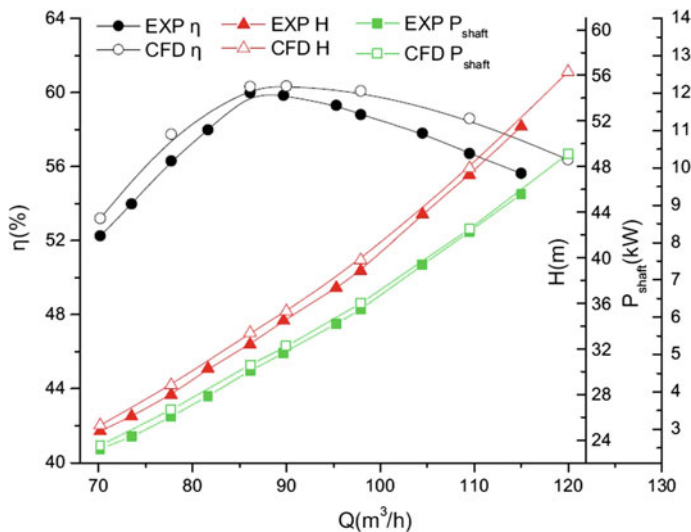


Fig. 2.16 Characteristic curves of a PAT obtained by CFD and experimentation [2]

occurring along the rotor axis (Fig. 2.15b) and along the axis of the draft tube (Fig. 2.15c) are significantly higher than the saturation pressure water vapor. Therefore, no vapor bubbles or cavitation occurs in this case.

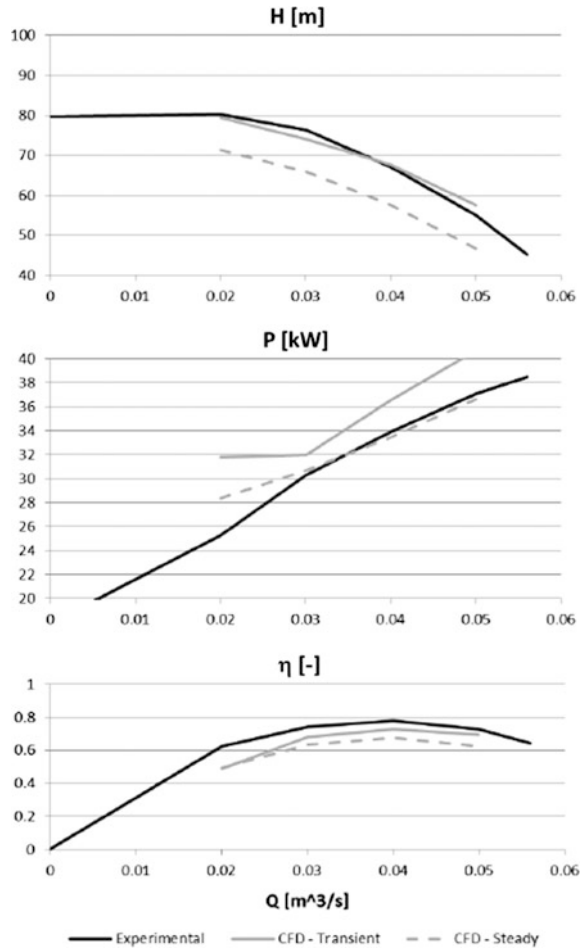
Figure 2.16 shows a comparison of the characteristic curves of a PAT obtained by CFD and experiments with a single stage pump.

In Fig. 2.17 a similar comparison is performed for a multistage pump, including the difference between the characteristic curves of a PAT obtained by CFD, in steady and unsteady state simulations, and experiments. The better performance of the unsteady state simulations is evident. The most important ability of CFD models is the capability of fluid flow analyses. Flow parameters such as velocity and pressure can be easily studied with many available methods of presenting the results. 2D and 3D color contours can be used to show the parameter variety from inlet to outlet. Figure 2.18 shows a sample velocity contour in a PAT. Other contours can also be depicted, if necessary. In addition, some extra stream flows can be used to detect the appearance of secondary flows.

Additionally, knowledge of the pressure distribution in a PAT can be useful to locate the point of minimum pressure. In the presence of low pressure at the PAT outflow this point is a potential source of cavitation. Figure 2.19 shows the pressure distribution in a multistage pump, for two different mesh resolutions.

Figure 2.20 shows the hydraulic head drop distribution in the different zones of a PAT: the volute, gap between volute and impeller, impeller and outlet pipe. It is clear that the impeller is the main component of a PAT for the efficiency. Additionally, losses in the gap between the impeller and the case can have an important role. Therefore, it is recommended to make efforts possible to improve the efficiency of a PAT by refining the impeller and gap head drops.

Fig. 2.17 Comparison of steady and unsteady state conditions [6]



2.5 Cavitation in a PAT

A major limitation in any hydraulic machinery operation is the phenomenon of cavitation. Whenever, in the suction pipes of pumps and turbines, the local pressure maybe drops to the values of the vapor pressure of the liquid, the liquid phase can rapidly convert to the vapor phase. Pump manufacturers normally determine the cavitation limitation of their products by NPSH¹ (Fig. 2.21), directly looped by the location of the pump installation above the tail water (Fig. 2.4):

¹Net Positive Suction Head.

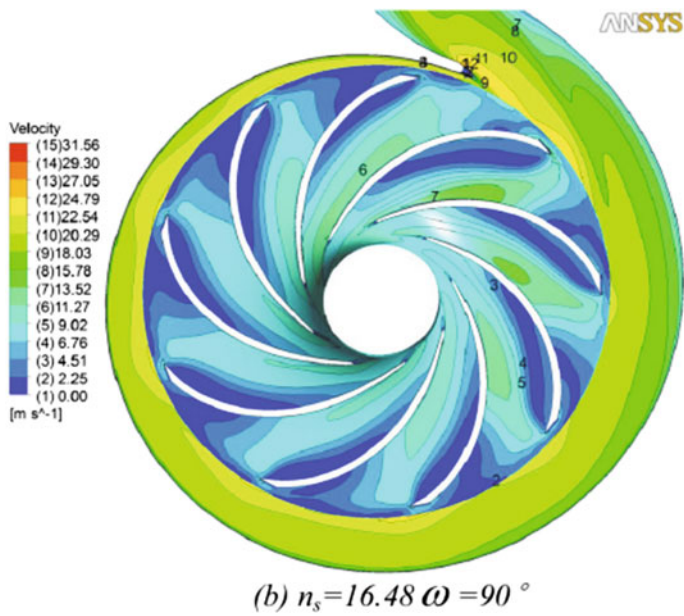


Fig. 2.18 Velocity contour in a PAT [21]

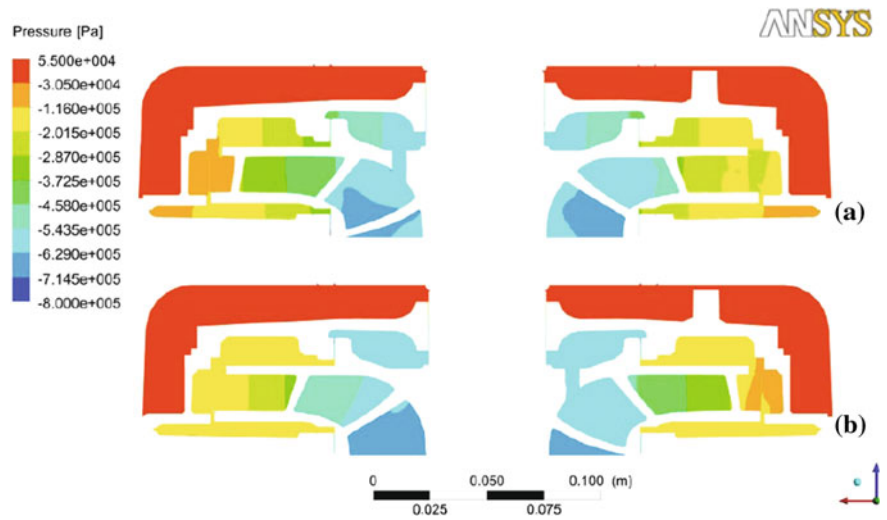


Fig. 2.19 Comparison between pressure fields: **a** a higher mesh resolution; and **b** a coarse mesh [6]

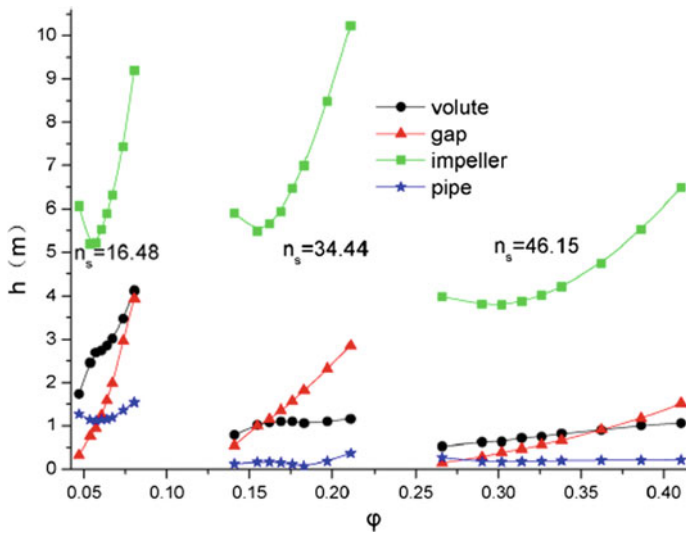
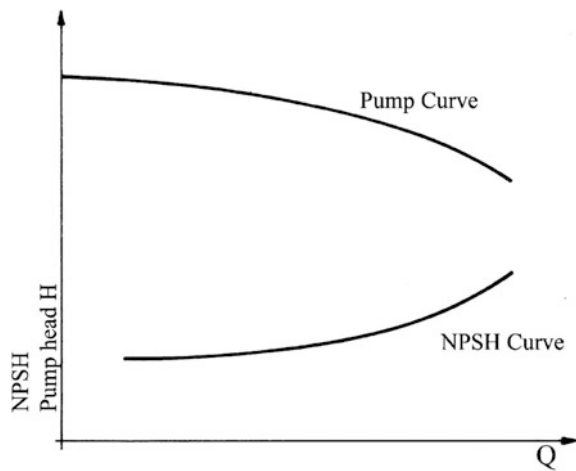


Fig. 2.20 Loss distribution in different zones of a PAT [21]

Fig. 2.21 Typical NPSH curve of a pump



$$h_{s,max} = \frac{P_a}{\rho g} - \left(\frac{P_v}{\rho g} + NPSH_p + h_f \right) \quad (2.14)$$

where $h_{s,max}$ (m) is the maximum suction head, P_a (Pa) is the atmosphere pressure, P_v (Pa) is the water vapor pressure and h_f (m) is the hydraulic loss in the suction pipe. $NPSH_p$ (m) is the Net Positive Suction Head, available from a pump catalogue (Fig. 2.21).

Considering P_v for a normal water temperature (3 kPa for 25 °C), and P_a for an average height of the site location (101 kPa sea level) and assuming 0.5 m for losses in the suction pipe, $h_{s,max}$ can be estimated by:

$$h_{s,max_p} \approx 9.0 - NPSH_p \quad (2.15)$$

Providentially, $h_{s,max}$ for turbines is less than that for pumps. As a PAT works at a higher pressure than a pump, $h_{s,maxPAT} < h_{s,max_p}$ for each corresponding rated point in direct and reverse modes [9]. A PAT may work at a higher rotational speed than its direct mode. Using affinity (or similarity) laws in turbo-machinery, the Eq. (2.15) will be modified to:

$$h_{s,maxPAT} = 9.0 - NPSH_p \left(\frac{N_{PAT}}{N_p} \right)^2 \quad (2.16)$$

where, N_{PAT} and N_p are the rotational speeds of reverse and direct modes, respectively.

2.6 Pressure Surge Analysis in a PAT

One of the main reasons for the still limited use of PATs in water supply networks (WSN) is due to doubts about the behavior of this electromechanical device during the lifetime of the WSN. The WSN technical management is responsible for preventing any worsening in the running of the water service. This attitude is quite comprehensible because water distribution is the main target of water companies and a suspension of the service could lead to penalties and to customer dissatisfaction.

Three aspects that have to be considered in micro hydropower design are represented by: (i) the effect on the WSN of the power plant start and stop procedures; (ii) the effects on the WSN of power plant load rejection or power failures; and (iii) the behavior of the PAT in the presence of pressure surges reaching the power plant node from different points of the network.

A power plant failure is possible on account of either a PAT failure or an electric grid failure. Any PAT failure could be of an electrical or mechanical type. The most serious situation is represented by an abrupt PAT stop or by the occurrence of a runaway or overspeeding of the machine due to a sudden black-out, namely a total load-rejection.

It is important to understand that experimental results show differences in the dynamic response of a PAT and a pressure relief valve (PRV) in the presence of similar flow conditions (Fig. 2.22). In addition, the wave period is also different and there is a fast damping of pressure oscillations in the presence of a PRV, with higher extreme pressures than with a PAT.

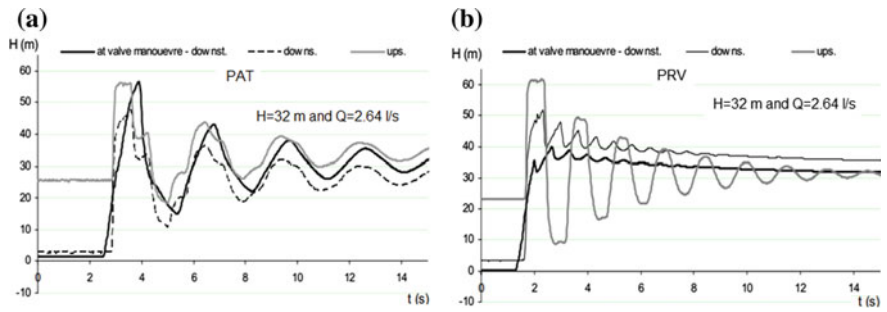


Fig. 2.22 Dynamic response of **a** a PAT and **b** a PRV [22]

Table 2.2 Maximum allowable $\Delta H/H_0$ as a function of the design head

H_0 (m)	Allowable maximum $\Delta H/H_0$
>100	0.15–0.20
100–20	0.25–0.35
<20	0.50

The hydraulic transient effects will strongly depend on the overall hydro-system characteristics, including the number of units and the type of PAT. In all cases the main objective is to prevent any serious damage to the penstocks or other pressurized pipes, as well as to any other components of the hydro-system. Regarding structural penstock protection both the maximum and minimum transient pressures need to be controlled in order to avoid the possibility that: (1) the allowable maximum pressure is exceeded, thereby preventing any breakdown or pipe burst; and (2) the sub-atmospheric pressure is increased, thereby preventing any cavitation phenomena and water column separation effects, as well as a potential pipe wall buckling event.

Typically, the allowable maximum relative transient head variations $\Delta H/H_0$ will depend on the design head, the range and dependency, as shown in Table 2.2.

In all cases, the downsurge, or minimum transient pressure, should also be properly considered in order that the pipe system is protected against atmospheric pressures and vacuum formation (e.g., water column separation).

Any disturbance induced in the flow will propagate as a wave with a finite relative velocity, known as the elastic wave celerity, whose value will depend on the elastic properties of both the fluid and the pipe according to [10]

$$c = \sqrt{\frac{K}{\rho[1 + (K/E)\Psi]}} \tag{2.17}$$

where E—Young’s modulus of elasticity of the pipe wall; K—fluid bulk modulus of elasticity (2×10^3 MPa for water); Ψ —parameter that depends upon the pipe structural constraints; ρ —fluid specific mass (1000 kg/m^3 for water).

In small hydro schemes, when equipped with small inertia turbines, the parameterization of water hammer effects might allow a better characterization of

the dynamic behavior of these turbo-machines, assisting in the choice of the most appropriate solutions.

Several numerical methods are available for the prediction of maximum and minimum pressures in each network point in the presence of unsteady state conditions [11]. Numerical simulations are commonly based on the one-dimensional momentum and continuity equations:

$$g \frac{\partial H}{\partial x} + \frac{\partial H}{\partial t} + \frac{f}{2D} V|V| + k \left(\frac{\partial V}{\partial t} + \text{sgn} \left(V \frac{\partial V}{\partial x} \right) c \frac{\partial V}{\partial x} \right) = 0 \quad (2.18)$$

$$\frac{\partial H}{\partial t} + \frac{c^2}{g} \frac{\partial V}{\partial x} = 0 \quad (2.19)$$

where V is the pipe cross average velocity, H is the piezometric head and c is the pressure wave celerity. The last term in the momentum equation is used for highly deformable pipe materials, producing a variability of wave celerity and a reduction of the pressure peaks during the transient [12, 13]. The WSN interior points could be solved using the method of characteristics (MOC). MOC transforms these equations into a pair of ordinary differential equations valid along the characteristic lines in the (x, t) plane. These equations can be replaced by algebraic equations that are solved together with the other basic equations of the computational model. MOC is the most common numerical scheme used to solve the transient problem in a discrete number of pipelines nodes, based on the solution of Eqs. (2.18) and (2.19), in the WSN initial flow conditions and in the WSN boundary conditions.

In the presence of a PAT in the hydropower node, generating a pressure surge or reacting to a pressure wave generated in a different node of the network, a few alternatives are available for transient calculations. In particular, the main alternatives concern the schematization of the electro mechanical device.

During normal steady-state operating conditions the speed of a turbine is constant, but it rises quickly when the load of the turbine is rejected. During the closure of a flow control valve, the speed of the turbine rises to a maximum value, under runaway conditions, that cannot surpass the allowable value fixed by the PAT manufacturer.

With reaction turbines, as PATs are, the runner speed changes the turbine discharge and the overspeed conditions also influence the hydraulic circuit [14]. The transient overspeed can induce an additional source, a water-hammer wave, especially in relation to PATs with small N values. A complete computer analysis should be performed based on the PAT characteristic equations, the complete hydraulic Eqs. (2.18) and (2.19) and the equation of the rotating masses of each unit:

$$T_H - T_R = I \frac{d\omega}{dt} \quad (2.20)$$

in which T_H is the transient hydraulic torque (N m), T_R is the resistant or electrical torque (N m), I is the rotating mass inertia (kg m^2) and ω is the angular rotating speed (rad/s).

The moment of inertia of a pump reflects its resistance to variations of the wheel while rotating on its shaft. Pumps with high rotating masses have higher values of inertia: they take longer to reach the rated speed during start-up and to slow down and lose power in the case of a planned or accidental shut off [10]. The inertia of the PAT is given by the sum of the inertia of the pump and motor and the following expressions (2.21) and (2.22) are proposed [10, 11]:

$$I_{\text{pump}} = 912 \left(\frac{P}{N} \right)^{1.435} \quad (2.21)$$

$$I_{\text{motor}} = 118 \left(\frac{P}{N} \right)^{1.48} \quad (2.22)$$

in which P is the power (kW); N is the rotational speed (rpm); and I is the inertia of the rotating mass (kg m^2).

Due to the need to test all the different failure scenarios, experimental or numerical PAT characteristic curves could be used in the PAT node as internal conditions between two network branches. This method can be considered fully one-dimensional, treating the fluid mass in the PAT as a single node. In a more complex procedure [6, 7, 15] a new computational scheme is proposed coupling CFD and MOC. In Fig. 2.23 the compatibility conditions between the two computational schemes are reported: the coupling was performed by modeling two pipe branches in CFD calculations, at the inlet and outlet of the PAT, for flow regulation, and by imposing compatibility equations between CFD and MOC at the extremes of the branches.

Differences are observed when running a CFD incompressible (rigid) or compressible fluid (elastic) model [7]. In Figs. 2.24 and 2.25 time-head plots are shown

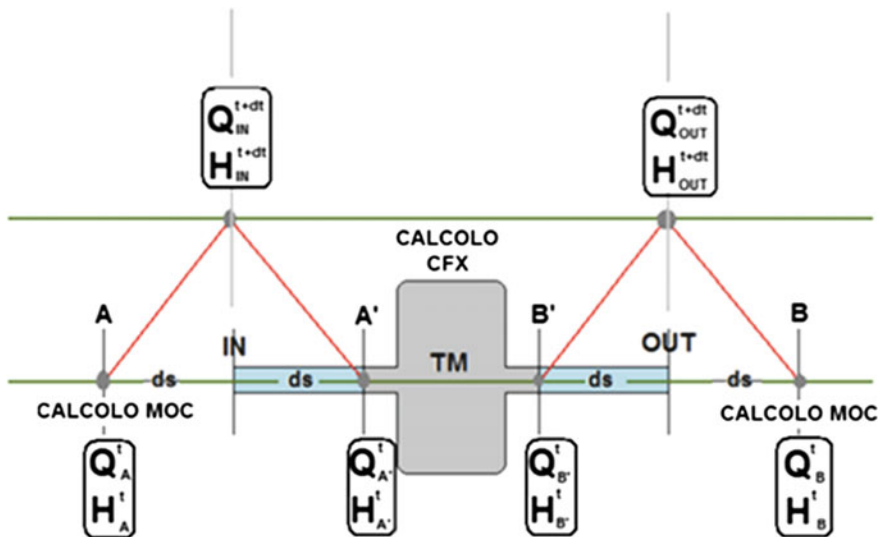


Fig. 2.23 Compatibility conditions between CFD and MOC

Fig. 2.24 Head variation after a sudden discharge variation (rigid model) [4]

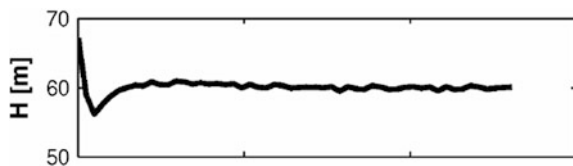
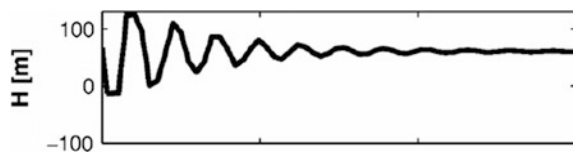


Fig. 2.25 Head variation after a sudden discharge variation (elastic model) [7]



after a sudden finite variation of discharge, from 35 to 30 l/s, with a rigid and elastic model, respectively. According to the first 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. The results of the elastic model are significantly different. 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 take a longer time to be dissipated.

The coupled CFD + MOC model, which was validated on experimental data at the Instituto Superior, Tecnico University of Lisbon (Portugal), was acquired on the simple hydraulic scheme of Fig. 2.26 [16]. Tests were performed for a sudden closure of the downstream valve. With a dumping factor $k = 0.34$ in Eq. (2.18) accounting for the viscoelastic behavior of the pipe system, a good agreement between the experimental and numerical piezometric head immediately upstream the regulation valve was observed (Fig. 2.27).

On the same hydraulic scheme, but with a different PAT and pipeline size, some comparisons have been made between the transient results obtained by using a one-dimensional model (MOC + one dimensional characteristic curve; CrCv in the

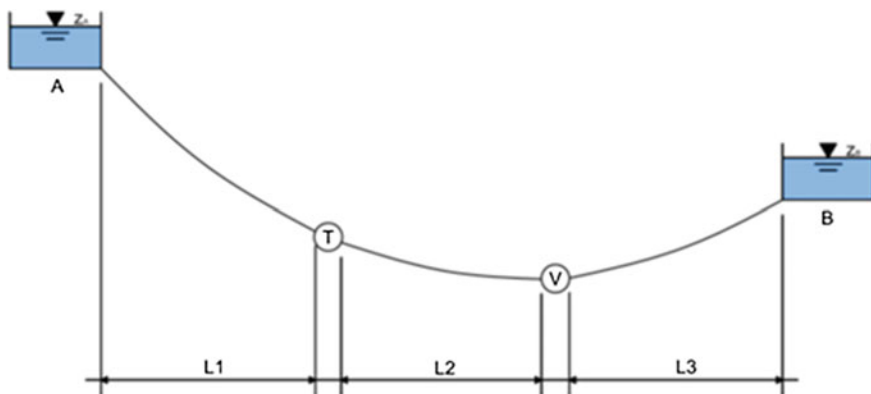


Fig. 2.26 Hydraulic scheme for water hammer analysis

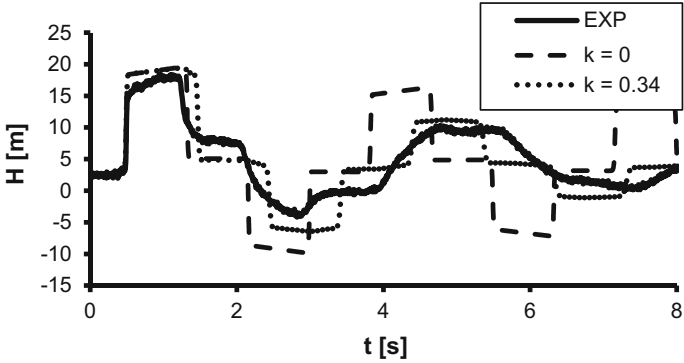


Fig. 2.27 Model and experimental heads at the downstream valve

following figures), the steady PAT characteristic curves obtained by a Suter parameters calculation, or the coupled CFD + MOC model (CFD). The calculations were performed for three different closure times of the stroking valves, $T_c = 0$ s (Case I), $T_c = 0.15$ s = 0.75τ (Case II) and $T_c = 0.45$ s = 1.125τ (Case III). In Figs. 2.28, 2.29 and 2.30, for Case I, Case II and Case III, respectively, the head

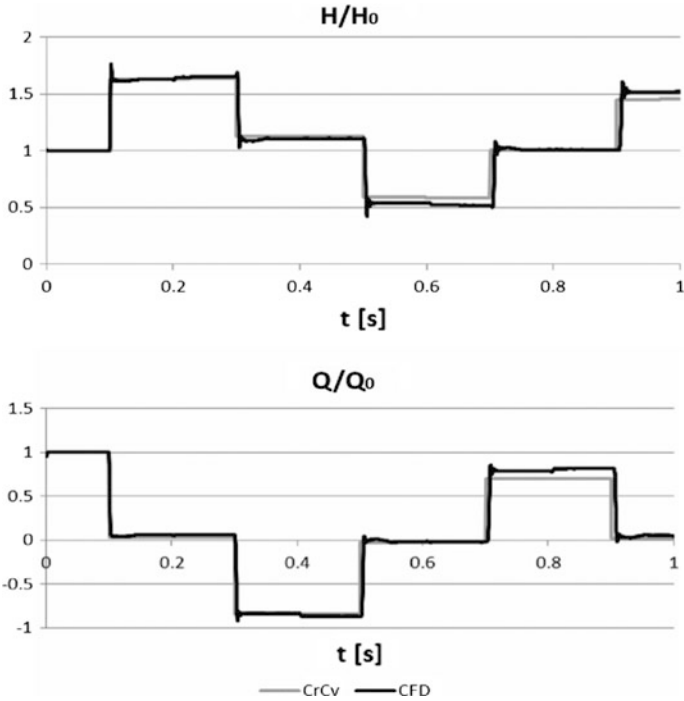


Fig. 2.28 Head and discharge in Case I [6]

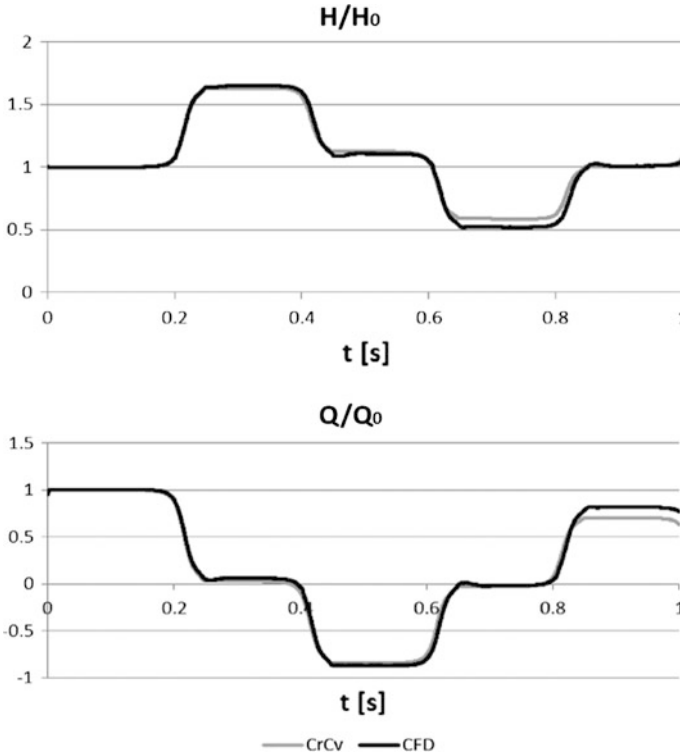


Fig. 2.29 Head and discharge in Case II [6]

and the discharge downstream the PAT, divided by the initial values, are plotted. Some appreciable differences appear in the calculated values of discharge and head, especially when the flow reverses. These differences are greater when the valve closure is faster and in Fig. 2.28 some spikes in both the head and discharge diagrams are visible. These spikes are generated by the response of the rotating machine to the water hammer waves and they are greater when the wave is steeper.

Similar results were found by Ismaier and Schlücker in 2009 [17], in experiments on the interaction between water hammer waves and pumps. They showed that the rotating machine generates pulsations which interact with the water hammer phenomenon and that the pulsation effect increases with the rotating speed. While the usual 1-D models that use the steady characteristic curves do not allow a simulation of the impulse response of the machine, the CFD + MOC model is a reliable method to predict this phenomenon.

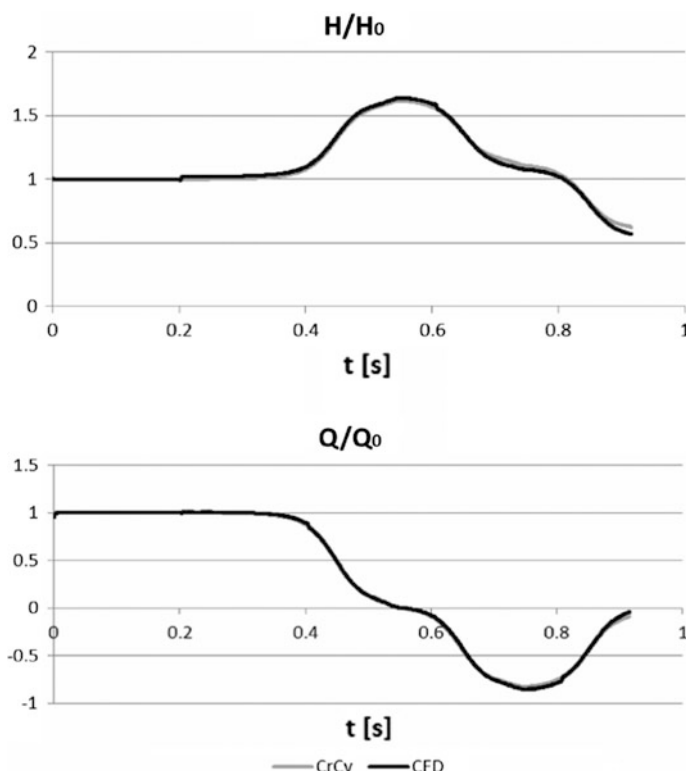


Fig. 2.30 Head and discharge in Case III [6]

Finally, in Fig. 2.31 the net head-discharge curves of a PAT are plotted as calculated by the coupled CFD + MOC model in unsteady conditions, and the difference discrepancies between these curves and the steady characteristic curve are clearly noticeable.

The control of transients and the dynamic behavior of hydropower plants are fundamental for the design and exploitation, in order to guarantee a safe and reliable solution. The effectiveness of new design criteria based upon computational techniques should allow an analysis of the global behavior in order to identify possible operational constraints, from the beginning of the design and exploitation process, in particular situations that are vital for the total WSN safety.

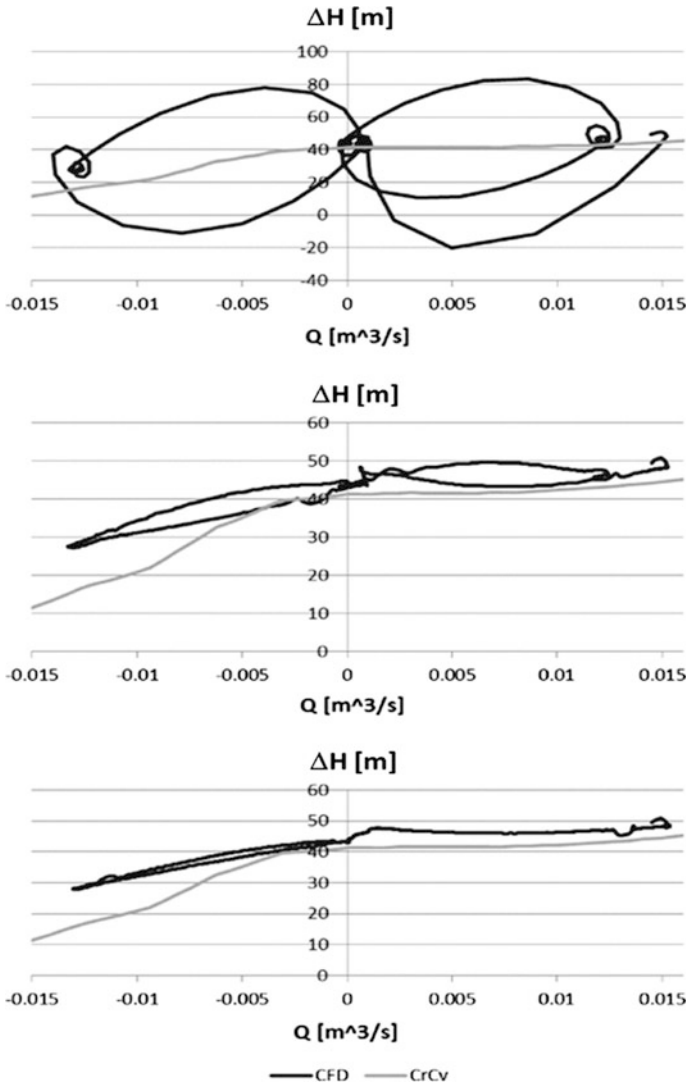


Fig. 2.31 Net head-discharge curves of a PAT under unsteady conditions in Cases I, II, and III, respectively

References

1. S. Derakhshan, A. Nourbakhsh, Experimental study of characteristic curves of centrifugal pumps working as turbines in different specific speeds. *Exp. Therm. Fluid Sci.* **32**(3), 800–807 (2008)
2. S.-S. Yang, S. Derakhshan, F.-Y. Kong, Theoretical, numerical and experimental prediction of pump as turbine performance. *Renew. Energy* **48**, 507–513 (2012)

3. A. Williams, *Pumps as Turbines used with Induction Generators for Stand-alone Micro-hydroelectric Power Plants* (Nottingham Trent University, UK, 1992)
4. A. Nourbakhsh, S. Derakhshan, E. Javidpour, A. Riasi, Centrifugal & axial pumps used as turbines in small hydropower stations, in *Hidroenergia 2010: International Congress on Small Hydropower International Conference and Exhibition on Small Hydropower* (2010), pp. 16–19
5. S. Derakhshan, A. Nourbakhsh, Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation. *Exp. Therm. Fluid Sci.* **32**(8), 1620–1627 (2008)
6. A. Carravetta, O. Fecarotta, H. Ramos, Numerical simulation on pump as turbine: mesh reliability and performance concerns, in *2011 International Conference on Clean Electrical Power (ICCEP)* (2011), pp. 169–174
7. O. Fecarotta, A. Carravetta, H. Ramos, CFD and comparisons for a pump as turbine: mesh reliability and performance concerns. *Int J Energy Env.* **2**(1), 39–48 (2011)
8. A. Pereira, H. Ramos, CFD for hydrodynamic efficiency and design optimization of key elements of SHP. *Int. J. Energy* **1**(6), 937–952 (2010)
9. J.-M. Chapallaz, P. Eichenberger, G. Fischer, Mini Hydro Power Group., and German Appropriate Technology Exchange., in *Manual on Pumps Used as Turbines* (Vieweg, 1992)
10. H. Ramos (Editor), Guidelines for design of small hydropower plants. WREAN (Western Regional Energy Agency and Network) and DED (Department of Economic Development—Energy Division), Belfast, North Ireland, ISBN 972-96346-4-5 (2000)
11. E. Wylie, V. Streeter, in *Fluid Transient in Systems* (Englewood Cliffs, NJ 7632: Prentice Hall, 1993)
12. B. Brunone, U.M. Golia, M. Greco, Effects of two-dimensionality on pipe transients modeling. *J. Hydraul. Eng.* **121**(12), 906–912 (1995)
13. G. Pezzinga, Evaluation of unsteady flow resistances by Quasi-2D or 1D models. *J. Hydraul. Eng.* **126**(10), 778–785 (2000)
14. H. Ramos, *Simulation and Control of Hydrotransients at Small Hydroelectric Power Plants* (Technical University of Lisbon, Portugal, 1995)
15. X.X. Zhang, Y.G. Cheng, L.S. Xia, J.D. Yang, Dynamic characteristics of a pump-turbine during hydraulic transients of a model pumped-storage system: 3D CFD simulation. *IOP Conf. Ser. Earth Environ. Sci.* **22**(3), 32030 (2014)
16. O. Fecarotta, Impiego Di Pompe Come Turbine: calcolo fluidodinamico per l'analisi prestazionale e l'inserimento in rete (2011)
17. A. Ismaier, E. Schlücker, Fluid dynamic interaction between water hammer and centrifugal pumps. *Nucl. Eng. Des.* **239**(12), 3151–3154 (2009)
18. A. Nourbakhsh, A. Jaumotte, C. Hirsch, H. Parizi, in *Turbopumps and Pumping Systems* (Springer Science & Business Media, 2007)
19. J.F. Gülich, *Centrifugal Pumps* (Springer, Berlin, 2008)
20. P. Singh, *Optimization of the Internal Hydraulic and of System Design in Pumps as Turbines with Field Implementation and Evaluation* (2005)
21. S.-S. Yang, F.-Y. Kong, H. Chen, X.-H. Su, Effects of blade wrap angle influencing a pump as turbine. *J. Fluids Eng.* **134**(6), 61102 (2012)
22. H. Ramos, A.B. Almeida, Dynamic orifice model on waterhammer analysis of high or medium heads of small hydropower schemes. *J. Hydraul. Res.* **39**(4), 429–436 (2001)

<http://www.springer.com/978-3-319-67506-0>

Pumps as Turbines

Fundamentals and Applications

Carravetta, A.; Derakhshan Houreh, S.; Ramos, H.M.

2018, XV, 218 p. 209 illus., 120 illus. in color.,

Hardcover

ISBN: 978-3-319-67506-0