Hydraulic turbines—basic principles and state-of-the-art computational fluid dynamics applications

P Drtina* and M Sallaberger
Sulzer Hydro AG, Zürich, Switzerland

Abstract: The present paper discusses the basic principles of hydraulic turbines, with special emphasis on the use of computational fluid dynamics (CFD) as a tool which is being increasingly applied to gain insight into the complex three-dimensional (3D) phenomena occurring in these types of fluid machinery. The basic fluid mechanics is briefly treated for the three main types of hydraulic turbine: Pelton, Francis and axial turbines. From the vast number of applications where CFD has proven to be an important help to the design engineer, two examples have been chosen for a detailed discussion. The first example gives a comparison of experimental data and 3D Euler and 3D Navier–Stokes results for the flow in a Francis runner. The second example highlights the state-of-the-art of predicting the performance of an entire Francis turbine by means of numerical simulation.

Keywords: hydraulic turbines, flow prediction, stage simulation, hill chart, Navier–Stokes and Euler computations

NOTATION

\( C, c \) absolute velocity (m/s)
\( E \) energy per unit mass (m\(^2\)/s\(^2\))
\( g \) gravity (m/s\(^2\))
\( h_{at} \) atmospheric pressure head (m)
\( h_d \) vapour pressure head (m)
\( H \) turbine head (m)
\( H_s \) suction head (m)
\( k \) turbulent kinetic energy (m\(^2\)/s\(^2\))
\( K_e \) normalized velocity
\( K_u \) normalized circumferential velocity
\( K_w \) normalized relative velocity
\( n \) rotational speed (r/min)
\( Q \) flowrate (m\(^3\)/s)
\( R, r \) radius/radial direction
\( T \) torque (N m)
\( U, u \) circumferential velocity (m/s)
\( W, w \) relative velocity (m/s)
\( Z \) axial direction/axis of rotation
\( \alpha \) absolute flow angle (degrees)
\( \beta \) relative flow angle (degrees)
\( \varepsilon \) dissipation rate
\( \xi \) loss coefficient
\( \eta \) efficiency

\( \nu_s \) specific speed
\( \rho \) density (kg/m\(^3\))
\( \varphi \) guide vane opening (deg)
\( \Phi \) flow coefficient
\( \Psi \) head coefficient
\( \omega \) rotational speed (1/s)

Subscripts

1 runner inlet
2 runner outlet
EK leading edge

1 INTRODUCTION

The use of hydraulic turbines for the generation of power has a very strong historical tradition. The first truly effective inward flow reaction turbine was developed and tested by Francis and his collaborators around 1850 in Lowell, Massachusetts [1]. Modern Francis turbines have developed into very different forms from the original, but they all retain the concept of radial inward flow. The modern impulse turbine was also developed in the USA and takes its name from Pelton, who invented the split bucket with a central edge around 1880. The modern Pelton turbine with a double elliptic bucket including a notch for the jet and a needle control for the nozzle was first used around 1900. The axial flow turbine

* Corresponding author: Sulzer Hydro AG, Hardstrasse 319/Postfach, CH-8023 Zürich, Switzerland.
with adjustable runner blades was developed by the Austrian engineer Kaplan in the period from 1910 to 1924.

Hydraulic turbines are not only used to convert hydraulic energy into electricity but also in pumped storage schemes, which is the most efficient large-scale technology available for the storage of electrical energy. Separate pumps and turbines or reversible machines, so-called pump turbines, are used in such schemes. During their long history there has been continuous development of the design of hydraulic turbines, particularly with regard to improvements in efficiency, size, power output and head of water being exploited. Recently, the use of modern techniques like computational fluid dynamics (CFD) for predicting the flow in these machines has brought further substantial improvements in their hydraulic design, in the detailed understanding of the flow and its influence on turbine performance and in the prediction and prevention of cavitation inception. The efficient application of advanced CFD is of great practical importance, as the design of hydraulic turbines is custom-tailored for each project.

Three-dimensional (3D) potential flow codes have been used for about 20 years, but their validity is limited to design point operation and requires a lot of empirical interpretation. 3D Euler codes describe the flow field in turbomachines with all typical vorticity effects, but neglect turbulent and viscous effects. Sulzer Hydro has now applied a 3D Euler code for advanced runner design for 10 years. This was developed together with the Swiss Federal Institute of Technology in Lausanne \[2, 3\] and has been used for over 70 different contracts, allowing wide experience to be gained. For turbine runners with accelerated flow the 3D Euler code is an excellent design tool \[4\]. It is much faster than a Navier–Stokes code and of very high accuracy, as viscous effects are small and confined to thin boundary layers. Nevertheless, the prediction of runner efficiencies requires a comparison between the current 3D Euler result and those of previous designs tested in the laboratory or in the field.

Parallel to the 3D Euler code, 3D Navier–Stokes codes have been applied to the design of components with adverse pressure gradients, i.e. impellers in pump mode \[5\], to the analysis of the losses in turbine components \[6\], to the prediction of turbine hill charts by calculating the flow in complete turbines from the spiral casing to the draft tube, and to the prediction of erosion caused by sand particles \[7\].

In addition to the discussion of the basic fluid dynamics of different kinds of hydraulic turbine, this paper presents two examples where the application of CFD led to a better understanding of complex flow phenomena. An improved understanding usually has a direct impact on the design, resulting in geometrical changes of existing components, the replacement of existing components by a completely new design and/or the use of new materials.

2 BASIC FLUID MECHANICS OF HYDRAULIC TURBINES

2.1 Definitions and parameters

The torque on any turbomachinery rotor can be estimated from the inlet and outlet velocity triangles (Fig. 1). The resulting equation is known as the Euler turbine equation and gives the specific energy transferred by the runner as

$$E = \frac{T\omega}{\rho Q} = U_1 C_1 - U_2 C_2$$  \hspace{1cm} (1)

For hydraulic turbines the degree of reaction is classically defined as the ratio of the static pressure drop across the runner to the static pressure drop across the stage. The Pelton turbine is an impulse stage and has zero reaction with all the pressure drop occurring across the stationary components and no pressure drop across the runner. In reaction stages such as Francis and Kaplan turbines a proportion of the pressure drop occurs in the rotor and a proportion in the stator. Typically, at their design points, a Kaplan turbine has a reaction of around 90 per cent, a Francis turbine of around 75 per cent and a pump turbine of around 50 per cent. At off-design operating points these values change.

The overall efficiency \(\eta_o\) of a turbine is defined as the ratio of the power delivered to the shaft to that available in the water entering the turbine:

$$\eta_o = \frac{T\omega}{\rho g Q H}$$  \hspace{1cm} (2)

where the net head \(H\) is the difference between the total pressure at turbine inlet and turbine outlet.

For pumps and pump turbines, the flow coefficient \(\Phi\) and the head coefficient \(\Psi\) are generally normalized with the rotor blade speed, in a similar manner to other
turbomachines, as

\[ \phi = \frac{Q}{UD^2(\pi/4)} \]  \hspace{1cm} (3)

\[ \psi = \frac{H}{U^2/(2g)} \]  \hspace{1cm} (4)

In hydraulic turbines, the usual performance parameters, such as the flow velocity, \( C \), and the circumferential blade speed, \( U \), are made dimensionless with respect to \( \sqrt{2gH} \):

\[ K_c = \frac{C}{\sqrt{2gH}} \]  \hspace{1cm} (5)

\[ K_u = \frac{U}{\sqrt{2gH}} \]  \hspace{1cm} (6)

The meridional and the circumferential components are denoted by \( K_{cm} \) and \( K_{cu} \). For both pumps and turbines, a useful parameter is the dimensionless specific speed defined as

\[ \nu_s = \frac{\omega}{\pi^{1/2}} \left( \frac{Q}{gH} \right)^{1/4} \frac{\phi^{1/2}}{\psi^{3/4}} \]  \hspace{1cm} (7)

The specific speed is the main parameter in hydraulic turbomachinery and is used for classification of turbines from turbine head, flowrate and speed.

2.2 Types of hydraulic turbomachine

Each type of turbine design can be further classified according to such criteria as:

- **Shaft orientation** → horizontal axis or vertical axis
- **Specific speed** → high, medium or low specific speed
- **Operating head** → high pressure \( 200 \text{ m} < H < 2000 \text{ m} \)
  - → medium pressure \( 20 \text{ m} < H < 200 \text{ m} \)
  - → low pressure \( H < 20 \text{ m} \)
- **Type of regulation** → single, variable stator vanes, e.g. Francis
  - → double, variable runner and stator vanes, e.g. Kaplan or variable needle stroke and variable number of jets
- **Design concepts** → single-stage or multistage
  - → single-volute or double-volute
  - → single-jet or multijet

More details of the regimes of application of the different turbines can be found in Fig. 2 (see also reference [8]). This figure shows which type of turbine is used for different volume flows and head rise and also gives the absolute power output generated for each category. A
further diagram showing the different impeller forms and the head for a variation in specific speed is given in Fig. 3. In general, the number of runner vanes or buckets decreases with increasing specific speed, from about 26 to 18 in a Pelton wheel, from 19 to 11 in a Francis turbine and from 7 to 3 in an axial turbine.

2.3 Pelton turbines

The impulse turbine extracts energy from the water by first converting the available head into kinetic energy in the form of a high-speed jet discharged from the nozzle. All the pressure drop occurs in the nozzle and the runner operates at constant static pressure. The jet is directed on to buckets fixed around the rim of a runner, and these are designed to remove the maximum energy from the water. The power of a given runner may be increased by using more than one jet. A cross-section through a typical vertical unit with six jets is shown in Fig. 4.

The hydrodynamics of the Pelton turbine is simple to understand and a one-dimensional steady analysis based on the velocity triangles provides much insight into the design principles. The Pelton turbine remains, however, the most complicated of all types of hydraulic turbo-machinery with respect to detailed flow analysis, as it involves a highly three-dimensional, viscous, unsteady flow with a free surface on a moving boundary, and this is just on the limit of the capability of the most advanced CFD methods [8, 9].

Figure 5 shows an instantaneous snapshot of the flow interaction between the jet and the buckets of a Pelton runner [10]. It can be seen that the jet acts on several buckets at the same time. Much empirical work has been carried out to determine the shape of the bucket for optimum performance. In general, standard bucket shapes from a family of profiles can be modified for a wide range of applications.

![Fig. 3 Selection diagram for different types of turbine](image)

![Fig. 4 Cross-section through a vertical axis Pelton turbine with six jets](image)
contains a rotating component in the same direction as the runner. For $H > H_{\text{opt}}$, the outlet water rotates against the rotational direction of the runner.

2.4 Francis turbines

The modern Francis turbine utilizes purely radial inlet flow through stationary guide vanes, but the runners are mixed flow devices with a component of the flow in the axial direction. The trend from purely radial inflow device through mixed flow devices to near axial flow devices increases as the specific speed is increased. The flow channel of a modern Francis turbine with a vertical axis is shown in Fig. 7, comprising a spiral inlet case, stay vanes, guide vanes, runner and draft tube.

The spiral case of a Francis turbine is designed such that the velocity distribution in the circumferential direction at the inlet to the stay vanes is uniform and the incidence angle over the height of the stay vanes varies only little. The main function of the stay vanes is to carry the pressure loads in the spiral case and turbine head cover. Their second purpose is to direct the flow towards the adjustable guide vanes with an optimal incidence angle. The adjustable guide vanes are the only device available to control the flow and thus the power output of a Francis turbine. Leakage flow through the gaps between the guide vane tips and facing plates causes efficiency losses and can cause local erosion [7]. The runner (Fig. 8) consists of a crown and band supporting highly curved, three-dimensional sculpted blades. To reduce the leakage flow between the runner and the casing, labyrinth seals at the crown and band are provided. The diffuser downstream of the runner is usually an elbow-type draft tube similar to that of Kaplan turbines. The vortices in the draft tube at off-design conditions often give rise to severe oscillations. A breakdown of the loss distribution within a Francis turbine at

![Fig. 5 Visualization of the flow in a Pelton runner](image)

The velocity triangles of the water entering and leaving the bucket in the middle section of its passage through the jet are shown in Fig. 6. At a single value of the jet velocity or available head, i.e. for $U_2 < U_{2\text{opt}}$ and $H = H_{\text{opt}}$, the optimum exit angle of $\alpha_2 = 90^\circ$ is attained with the minimum kinetic energy in the flow leaving the bucket. For $H < H_{\text{opt}}$ the relative velocity of the flow leaving the bucket $w_2$ is reduced and the outlet water

![Fig. 6 Velocity triangles for analysis of a Pelton bucket](image)
the design point as a function of specific speed $n_s$ is shown in Fig. 9. At off-design conditions the losses attributable to the runner, such as incidence losses, friction inside blade passages and exit swirl losses, strongly increase, as do the losses in the draft tube.

Typical velocity triangles for a mean streamline for low and high $n_s$ turbines are shown in Fig. 10. For very high $n_s$ the flow in the runner is nearly axial, giving $K_u_1 \approx K_u_2$. With decreasing $n_s$ the flow becomes increasingly radial with larger differences between inlet and outlet diameters $D_1$ and $D_2$ and hence between $K_u_1$ and $K_u_2$. The increasing difference between inlet and outlet velocity $U_1$ and $U_2$ explains the increasing head as $n_s$ decreases.

The flow in a Francis runner is a strongly three-dimensional rotational flow. The close proximity of the guide vanes to the highly curved meridional flow channel leads to a non-uniform meridional velocity at stator outlet. This gives rise to a strongly rotational flow at the outlet of the stator vane and a severe three-dimensional flow pattern inside the runner. Therefore only fully three-dimensional methods will provide effective solutions of the flow in a Francis runner. As a result of 3D Euler flow computations the flow inside the blade channel of a Francis runner of high specific speed close to the suction side and close to the pressure side can be obtained (Fig. 11). Typically the flow is roughly aligned with the meridional shape on the suction side of the blade, whereas on the pressure side the flow is forced towards the band. This indicates the strong three-dimensional character and the distinct secondary flow of a Francis runner.
The velocity vectors at the leading edge together with part of the blading are shown in Fig. 12 for the same Francis runner. The absolute and relative velocity components are shown and the variation in the flow in the circumferential direction, especially at the hub, is obvious. The three-dimensional character of the flow is also evident in visualization of the flow on the suction side of the blade in the vicinity of the leading edge at a turbine head greater than the optimum (see Fig. 13). The flow enters the blade passage and then deviates strongly towards the band. While operating at the design flowrate this motion is aligned roughly parallel to the leading edge. Operation at lower flowrates leads to the occurrence of strong vortices which are contained within the blade passage. These interblade vortices may induce pressure pulsations.

2.5 Axial turbines

Axial turbines may be realized in different concepts. The most flexible in terms of variation in flowrate $Q$ and turbine head $H$ is the double-regulated turbine with adjustable stator and rotor blades, the so-called full Kaplan concept. Cheaper but not so flexible concepts are designed as single-regulated turbines. The fixed blade propeller turbine has adjustable stator blades but fixed runner blades. In the so-called semi-Kaplan concept, fixed stator blades and adjustable rotor blades are used.

For classical vertical Kaplan turbines (Fig. 14) the inflow and the outflow of the stator is radial, while the inflow and the outflow of the runner is fully axial. In general, steel scroll cases are used for heads between 30 and 60 m and concrete semispiral casings for heads between 10 and 40 m. The largest Kaplan turbines have runner diameters of up to 10 m. The horizontal bulb turbines (Fig. 15) are designed with a horizontal axis and have the advantage of a more or less straight flow path through the intake and draft tube. The friction losses are considerably lower in these components than in the spiral casing and elbow draft tube of the vertical Kaplan turbine. In the Straflø turbine design (Straflø = straight flow) the turbine and the generator form an integral unit without a driving shaft. The turbine rotor blades are connected to an outer ring which directly carries the generator rotor poles.

The hub–tip ratio ranges from 0.30 for three-blade
Fig. 11 Meridional flow in a Francis runner of high specific speed $n_s$

Fig. 12 Euler results for the flow at the leading edge of a Francis runner
runners up to 0.65 for seven-blade runners. Because of the large $U$ variation over the blade span, the velocity triangles differ strongly between the root and the tip section (see Fig. 16). The profiles at the root section have a large camber and thickness, high turning of the flow and high stresses, while the profiles at the tip section have a small camber and thickness, low turning of the flow and low stresses.

For the stator vanes of a Kaplan turbine a straight profile is used, but bulb turbines are designed with conical vanes. Therefore the swirl at the stator outlet is rotational and the flow in the runner will have a strong three-dimensional character, which is demonstrated in Fig. 17. The velocity vectors at the leading edge and at the trailing edge are given in Fig. 18. Again, fully 3D computation methods will be superior to others. The secondary flows, however, are less pronounced than in Francis turbines, and therefore axial runner design has been based on 2D theory for many decades.

The swirl at the runner outlet strongly influences the performance of the draft tube, which gives the largest contribution to the losses of a low-head turbine. In cases where the draft tube is designed to be close to the stability limit, the swirl at the runner outlet, especially at the tip section, has to be carefully controlled.

2.6 Cavitation

Cavitation occurs in the flow of water when, owing to regions of high-flow velocity, the local static pressure decreases below the vapour pressure and vapour bubbles appear. Cavitation may occur on the blade suction surface in regions of low pressure or at the runner leading edge at off-design operation. For low-head operation, cavitation will be located on the pressure side; for high-head operation it will be on the suction side. Within vortices such as those induced by the tip leakage flow of axial runners or those occurring in the blade passages of a Francis runner at extreme off-design operation and in the draft tube of Francis turbines, cavitation may also occur. In hydraulic turbines the most important
parameter describing cavitation is the cavitation number \( s \) defined by Thoma:

\[
\sigma = \frac{h_{\text{at}} - h_v - H_s}{H_s} \tag{8}
\]

In this dimensionless parameter the pressure margin between local pressure and vapour pressure is normalized by the turbine head. In order to achieve cavitation-free conditions, the value of \( \sigma \) of the plant must be larger than the value of \( \sigma \) at which laboratory tests indicate the onset of cavitation. For Francis turbines the typical value of \( \sigma \) may vary from 0.04 to 0.09 for low specific speed, \( \nu_s \), from 0.09 to 0.20 for medium specific speed and from 0.20 to 0.30 for high specific speed. For Kaplan turbines with seven runner blades the value of \( \sigma \) for cavitation onset is typically 0.3–0.5, with 4–5 blades it may be 0.5–2.0 and for runners with three blades it may vary from 2.0 to 3.0.

The effects of cavitation are harmful, both on performance and on erosion of material. Cavitation erosion is caused by the extremely high pressure peaks that occur during the implosion of cavitation bubbles in the vicinity of a solid surface. Cavitation imposes restrictions on blade loading and blade design. Modern CFD based design methods attempt to avoid cavitation by optimizing the pressure distribution on the blades to avoid areas of high relative velocity [11]. In addition the setting level of the turbine relative to the tailwater can be reduced or
the turbine can be designed with larger diameter and lower flow velocities.

3 GRID GENERATION AND SIMULATION METHOD

After discussion of the basic fluid mechanics of hydraulic turbines and the most important design criteria, the following sections are devoted to the application of modern 3D Navier–Stokes codes, with particular emphasis on the recently developed stage simulation method. In Section 3 the still very time-consuming process of generating an appropriate grid for the complex geometry of a hydraulic turbine is briefly reviewed. Sections 4 and 5 demonstrate the ability of state-of-the-art CFD applications to predict not only flow fields but complete hill charts for Francis turbines.

3.1 Grid generation

Presently there is no unique grid generation approach that perfectly fulfils all requirements imposed by the various components of hydraulic turbines: spiral casings, stay vanes and guide vanes, runners and draft tubes.
As long as the ultimate automatic grid generation tool applicable for any given complex geometry is missing, tools will be used that are specifically adopted to single components.

Owing to the geometrical complexity of most of the hydraulic machinery components under investigation (i.e. spiral casing), these component grids were generated with the help of ICEM–CFD/P³ and ICEM–CFD/HEXA which both deliver high-quality block-structured meshes for geometrically and topologically complex domains (i.e. mixed H,O,C grids, intersecting butterfly grids). In order to avoid highly skewed grid cells the main body of the mesh for a spiral casing (see Section 5) is constructed using a butterfly arrangement of sub-blocks (a central block surrounded by four outer blocks: see inlet section of spiral casing in Fig. 19). To capture all significant flow phenomena the grid resolution has to be sufficiently high. On the other hand the available com-
puter resources impose severe limitations on the mesh size.

Other grids for guide vane passages and draft tubes have been generated by applying the grid generation software TASCgrid which forms part of the CFX–TASCflow software package [12]. An important advantage of applying TASCgrid is its ability to parameterize the basic geometrical description. Thus, it is possible to change the guide vane angle within certain limits by simply changing the value of a single parameter in the input file. For most of the sophisticated grid generation programs (ICEM–CFD included) this is still a challenging problem.

3.2 Discretization method and turbulence model

All calculations discussed below were carried out using the CFX–TASCflow software package. This CFD code solves the 3D Reynolds-averaged Navier–Stokes equations in strong conservative form for structured multiblock grids. The system of transport equations is discretized using a conservative finite element based finite volume method and a second-order accurate skew upwind differencing scheme with physical advection correction. The discretization scheme is second-order accurate in space. Turbulence effects are modelled using the standard k–ε model, the RNG model or the Kato–Launder formulation.

Liquid and subsonic, transonic and supersonic gas flows can be analysed in rotating and stationary coordinate systems as well as in multiple frames of references. For more than 10 years this software has been applied to various fluid flow problems [13, 14]. Details regarding the theoretical basis of the software are reported by Raw [15].

All calculations were performed applying the CFX–TASCflow code versions 2.6 or 2.7 implemented on an IBM RS/6000 and an SGI Power Challenge carrying 12 processors. Some of the calculations have been carried out in parallel mode using up to four processors.

3.3 Stage capability

For the calculations discussed in Sections 4 and 5 the so-called stage method developed by Galpin et al. has been applied [16]. Here the steady state interaction between stationary and rotating components in a turbo-machine is simulated by a mixing plane between the components. Each component is calculated in its own frame of reference and the blade rows can be reduced to single-blade channels with periodic boundaries. The method is based on stage simulation ideas of Denton [17] which were extended by Galpin [16] and installed into TASCflow under the partnership of ASC, Sulzer Hydro and Sulzer Innotec. Details of the validation of the stage capability are described by Sick et al. [18].

4 RUNNER FLOW PREDICTION

4.1 Introduction

The first major breakthrough in the use of CFD methods for hydraulic turbine design allowed Euler methods to be used for design of water turbine components and, combined with suitable design rules, drastically reduced the number of model tests needed for the achievement of a satisfactory design [3]. The relatively simple Euler methods predict the important features of the flow, such as incidence levels at runner inlet, pressure levels at runner outlet (cavitation) and swirl in the draft tube. They are, however, limited in their ability to predict the losses, as viscous forces are neglected. While Section 4 concentrates on the comparison of runner outlet velocity profiles obtained by applying a Euler and a Navier–Stokes code, the prediction of losses and turbine performance will be discussed in Section 5.

This section demonstrates the improvement gained by applying Navier–Stokes codes instead of simple Euler codes. The most significant improvement can be seen by comparing the runner outlet velocity distributions for off-design operating points. In particular for the design of draft tubes a reliable knowledge of the velocity profiles at the draft tube inlet is crucial. The better prediction of velocity profiles is a direct consequence of taking automatically into account the losses when applying a Navier–Stokes flow code. In Section 5 it will be shown that even the loss distribution is predicted very well.

4.2 Predicting Francis turbine runner flow

The test case considered for the verification of the stage interface method is a model Francis turbine of high specific speed (see Fig. 20). The full-scale turbine has a runner diameter of 3.4 m and was designed for a head of 32.7 m at a nominal flowrate of 90 m$^3$/s with a rotational speed of 166 r/min and a specific speed of 422.

The model turbine is a scale model of the original machine with a runner diameter of 0.3 m. The experiments were carried out in the hydraulic turbine test stands of Sulzer Hydro in Zürich. Standard measurement techniques for modern hydraulic practice were used for the derivation of all performance parameters.

The velocity near the draft tube outlet was measured by means of a propeller anemometer. Measurements of the detailed flow velocities were also carried out in the hydraulic turbine test stands of Sulzer Hydro using L2F laser velocimetry. Flow traverse planes were established at the runner inlet and at the runner outlet by insertion of suitable windows in the casing (see Fig. 20). Flow velocities at each of these measurement planes were
obtained at five operating points of the turbine. These operating points were selected to provide a good overview of the flow at the best operating point and at a variety of different off-design conditions.

During the investigations a series of computations with different levels of complexity was carried out. Beginning with simple simulation of the flow in the runner alone and then proceeding with two-component calculations (distributor–runner and runner–draft tube), simulations for the entire hydraulic turbine were finally arrived at, covering all components starting from the inlet of the spiral casing and ending at the draft tube outlet. These full machine simulations were performed applying a two-step procedure: in the first step the spiral casing, including the entire stay vane ring, was simulated; the second step included appropriate passages of the stay vane ring, guide vane ring and the runner as well as the complete draft tube. Thus, the two steps overlapped at the stay vanes. Flow conditions extracted from the first step were used as inlet conditions for the second step. Mixing interfaces were applied between rotating and non-rotating components.

Figure 21 shows the results obtained for the velocity distribution (meridional and swirl component) at the runner outlet for inviscid and viscous cases in comparison with experimental data. All velocity data for the viscous case were extracted from the full stage calculation.

The comparison of the measured and calculated circumferential and meridional velocity components at the runner outlet demonstrates the reliability of the computational method in detail. The good agreement indicates that the component interactions are well predicted at both design and off-design points.

In all operating points the Navier–Stokes solution predicts the velocity profiles for both the swirl component and the meridional component very well. At operating points 1, 2 and 4 the viscous results are clearly in closer agreement with the measurements than the Euler computations. As regards operating points 3 and 5, viscous and inviscid simulation results give comparably good results with respect to the measurements. It is hard to decide which one is closer to reality. Nevertheless, it should be borne in mind that these are part load operating points where measurements are difficult to obtain, especially for small radii.

5 HILL CHART PREDICTION BY STAGE SIMULATIONS

To predict a complete hill chart for a hydraulic turbine the entire machine from the inlet of the spiral casing to the outlet of the draft tube has to be modelled. In the case of a Francis turbine, grids for different guide vane openings have to be generated.

In order to reduce the computational effort, the two-step procedure described in Section 4.2 was applied. This is physically reasonable as the flow in the spiral casing is a function of the Reynolds number only, just as the flow in the penstock or the branch pipe. To determine the losses in the spiral casing over the whole hill chart it is only necessary to calculate the viscous flow in the spiral casing at one operating point. The
Fig. 21 Francis runner. Comparison of non-dimensional meridional velocity, $K_{cm}$, and swirl velocity $K_{cu}$, for five operating points. Operating point 1 is identical to the best efficiency point, while operating points 2–4 reflect different off-design conditions (see sketch in bottom left corner).

losses are then a parabolic function of the volume flow, $Q^2$ (see reference [19]). For the stage simulation it is therefore sensible to calculate the flow in the spiral casing separately (and in one-half of the symmetric casing only) to avoid unnecessary large grids for each operating point.

The outlet of the spiral casing simulation is chosen to be downstream of the stay vanes so that the flow at the inlet to the stay vane cascade is correctly modelled. The boundary conditions for the stage including the distributor, runner and draft tube consist of tangential and radial velocity components (defining the swirl) in a reference plane upstream of the distributor (as calculated in the spiral casing analysis) and average constant pressure in an outlet plane downstream of the draft tube exit (this allows for the typical non-uniformities of the flow at the draft tube exit).

The solution of the first simulation (best efficiency point) is taken as the initial condition for other operating points. This reduces the number of subsequent iteration steps to 100–300 (1–3 days) depending on the operating point.

5.1 Numerical hill chart

The hill chart for a high specific speed Francis turbine has been determined experimentally on a model test rig. The turbine efficiency has been evaluated for more than 200 operating points, which yields a good approximation of the turbine performance for the operating range of interest.

To calculate the turbine efficiency from the numerical simulation data, the following procedure is applied to each operating point:

1. The spiral casing (casing and stay vane ring) and stage
(distributor, runner and draft tube) calculation results are examined together in order to evaluate the total pressure loss over the entire machine.

2. The power delivered by the runner is calculated as the product of rotational speed and torque on the runner. Both pressure and viscous forces acting on the runner surfaces (blades, hub and shroud) contribute to the torque.

3. Efficiencies are determined for each component, $i$, by applying

$$\eta_i = \frac{\dot{p}_{\text{tot,in}} - \dot{p}_{\text{tot,out}}}{\Delta \dot{p}_{\text{tot,em}}}$$

with $\Delta \dot{p}_{\text{tot,em}}$ being the difference in total pressure over the entire machine. Note that for the runner the work done, which is evaluated in step 2, has to be taken into account.

4. Component losses are defined by $\zeta_i = 1 - \eta_i$, and added to give the entire loss appearing in the machine $\zeta_{\text{tot}} = \sum_i \zeta_i$. The resulting overall efficiency is defined by $\eta_{\text{em}} = 1 - \zeta_{\text{tot}}$.

Owing to the high demand in computer memory and CPU time, the number of operating points that could be simulated was restricted. In the present case 14 operating points were chosen, with three different guide vane openings (volume flowrates) and six different heads investigated.

A comparison between experimentally and numerically evaluated hill charts only makes sense if the number and distribution of data points is identical. For this purpose the experimental hill chart shown in Fig. 22 is based only on the experimental data corresponding to the operating points that were also calculated. Figure 23 shows the resulting hill chart based on the 14 operating points from the stage calculations.

The qualitative agreement of both hill charts is impressive. All general features are captured by the simulations. The best efficiency point is identical in the two hill charts and all gradients show a very similar behaviour. Please note that the two scales $K_u$ and $K_m$ are in absolute rather than in relative terms in order to be able to check the correct location of the best efficiency point. Quantitatively there is some discrepancy in the steepness of the gradients, in particular for high values of $K_u$.

A detailed comparison for the three different guide vane openings under investigation can be obtained from Figs 24 to 26. For each guide vane opening the normalized efficiency is plotted as a function of $K_u$. It can be seen that the shape of the efficiency curves for all three guide vane openings is predicted correctly. For small openings the best efficiency point occurs at low $K_u$ values (high head). At large openings the high efficiency region occurs at high $K_u$ values (low head). This typical characteristic of a high specific speed turbine is perfectly represented in the analysis.

Two weaknesses of the present analysis should be mentioned. Firstly, whereas the relative shape of the hill chart is perfectly simulated, the absolute level of the calculated efficiency is about 3 per cent lower than that measured. This is attributed to the use of a coarse grid. A check of one operating point calculated with a finer mesh produces a reduction in friction losses.

Further improvements are expected if the near-wall regions are treated by advanced turbulence formulations such as two-layer and Reynolds stress models. Calculations for individual components have shown that the application of two-layer models increases the accuracy of loss prediction but also increases the computational time.

Secondly, the rate of convergence is rather poor for high $K_u$ values at small guide vane openings. Large
Fig. 23  Hill chart based on 14 numerically obtained efficiency values (● data points; --- constant guide vane opening)

Fig. 24  Turbine efficiency for $\phi = 25^\circ$

Fig. 25  Turbine efficiency for $\phi = 40.5^\circ$

Fig. 26  Turbine efficiency for $\phi = 33^\circ$ (design mass flow)

Separation zones occur in the draft tube in this case. This is probably the reason why the steep gradients of the measured efficiency curves could not be reproduced in the numerical analysis. However, it is known from the model test that the turbine exhibits a very rough and unstable operational behaviour at $K_u \geq 1.5$, especially for small openings. The unstable conditions make it difficult to compare the time-averaged data of strongly unsteady experimental signals with numerical data of a basically steady analysis.

This is confirmed by comparison of measured and predicted diffuser efficiency of the draft tube (see Fig. 27). At high values of $K_u$, where the flow separates, the numerical simulation currently overestimates the off-design performance of the draft tube.
6 CONCLUSION

As is to be expected for turbomachines with such a long history, the design of hydraulic turbines still relies heavily on experience gained in earlier designs. 3D Euler codes provide a systematic tool for the application of this know-how to new designs. Nowadays, however, the use of modern stage simulation methods in 3D viscous Navier–Stokes codes is transforming the analysis of turbine designs. Details of flow separation, loss sources and loss distribution in components, matching of components at design and off-design, and low pressure levels with risk of cavitation are now amenable to analysis with CFD.

In each numerical investigation a significant amount of time has still to be spent with grid generation and grid modification. Often this effort prevents the integration of sophisticated Navier–Stokes simulations into the design procedure. For this reason 3D viscous calculations for complete turbines are not yet a standard design step for a new hydraulic turbine. Advanced automatic grid generation tools are needed that generate high-quality meshes (block-structured hexahedron or mixed elements) on the basis of 3D CAD data for all essential components. Together with the application of parallel computers (parallelized codes are already available), the generation of numerical hill charts will become feasible for standard design tasks.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the helpful discussions with Dr M. V. Casey, Sulzer Innotec, and Dr H. Keck, Sulzer Hydro, during the preparation of this paper. Thanks are also due to Dr M. Sick for her contribution, in particular with the stage calculations.

REFERENCES


4 Sallaberger, M. J. Quasi-three-dimensional and three-dimensional flow calculation in a Francis turbine. IGIT paper 96-GT-38, Birmingham, 1996.


13 Casey, M. V., Borth, J., Dritina, P., Hirt, F., Lang, E., Metzen, G. and Wiss, D. The application of computational modeling to the simulation of environmental, medical and engineering flows. 18th Workshop Proceedings. Speedup J., 1996.


18 Sick, M., Casey, M. V. and Galpin, P. Validation of a stage calculation in a Francis turbine. In Proceedings of 18th IAHR Symposium, 1996.
