New Challenges in Pelton Research

Mirjam Sick
Helmut Keck
VA TECH ESCHER WYSS LTD
P.O. BOX
CH-8023 Zurich
Switzerland

Gérard Vulioud
Etienne Parkinson
VA TECH Hydro Vevey
P.O. BOX
CH-1800 Vevey
Switzerland

Introduction

The completion of three new Pelton turbine units for the Bieudron hydro plant (extension of the CleusonDixence scheme), each with a net head of 1869 m and a rated turbine output of 423 MW, has completely redefined the standard for comparison of technical achievements in Pelton turbine schemes. The Bieudron turbines are not only the most powerful ever produced with the highest ever net head, see figure 1, but have also been used to break new ground in many aspects of the simulation of the mechanical and hydraulic design of Pelton turbine systems. This paper reviews some of the recent breakthroughs in mechanical design, already published in several technical articles (1, 2, and 3) and describes in more detail some more recent breakthroughs in flow simulation based on the Bieudron Pelton turbines.

The rapid development of flow simulation for water turbines (4, 5 and 6) has already led to substantial improvements in the design and performance of Kaplan and Francis turbines. This technology has not been extensively used in Pelton turbines, but has reached a stage of maturity where simulations of the flow in the distributor and the flow in the nozzles of Pelton turbines can be routinely carried out. The flow in the jet and the moving buckets are more difficult to simulate, as they involve a jet flow with a free surface and the unsteady interaction of the jet with the buckets. Such simulations are now possible but, as the methods have not yet been widely validated, the simulations have to be considered to be an area of active research and development. The need to develop accurate simulation methods for the flow in the buckets has been driven by the desire for better tools to estimate the distributed water jet loading on the buckets to aid the mechanical aspects of the new turbine designs. For the Bieudron turbines, the design method was changed from a relatively simple and highly conservative model of the water jet loading on the buckets to more accurate Finite Element methods based on hydraulic rig test data for the bucket loading. This allowed substantial improvements to be made in the mechanical design and has promoted the development of CFD simulations for the definition of the water-jet loading of the buckets. Accurate simulation models allow better physical understanding of the forces involved and allow more accurate transfer of know-how derived from one design to another one.

1. Mechanical Design and FE analysis

Bieudron represents a new milestone in the mechanical design of Pelton turbines as both the power and the head are larger than for any other Pelton turbine ever built, see figure 1. The decision to go ahead in this project relied on the ability to safely extrapolate earlier mechanical design experience on smaller turbines to the Bieudron machines. For this a clear understanding of the physical and mechanical aspects of turbine design is required, and Finite Element Analysis (FEA) simulation tools need to be applied appropriately to assess the critical design features. In Bieudron all the turbine parts underwent rigorous analysis with FEA methods.

For example, the spherical valves, the spiral case bifurcations, and the distributor manifold of Bieudron all underwent a complete FEA study to optimise their design (ref. 1). The simulations were ultimately validated by strain gauge measurements on the prototype machine during the shop testing and good agreement was found between predictions and measurements. This demonstrates the high quality of the FEA methods, shows that proper quality procedures were in place for their use, and also demonstrates the high level of know how in the process of the design optimisation – higher stresses than intended would have meant that the risk of failure was too high and lower stresses would have indicated a sub-optimal design with potential of further cost optimisation.
The traditional method to determine the stress in a Pelton bucket is a stress analysis based on classical beam theory (treating the bucket as a point-loaded beam clamped at its root). This is known to deliver conservative results and so leads to very reliable mechanical designs for the normal range of Pelton turbines shown in figure 1 (ref. 1). In view of the higher loading level of the Bieudron hydro plant, it seemed to be insufficient to rely on this simplified model of the jet – bucket interaction for the mechanical design of the bucket as this may have led to an overly conservative design. For Bieudron a completely new analysis technique was developed and applied for the first time. Experimental tests were carried out on a model turbine in which the miniature built-in pressure transducers were incorporated into the buckets to derive the actual distribution of the pressure load seen by the buckets. This distribution was used in place of an assumed point-loading to provide the mechanical loading conditions for an FE analysis in the mechanical design. The stress levels in the runner and the buckets derived from this extensive FE analysis were examined to ensure that the cyclic loads would not lead to high cycle or low cycle fatigue in operation with the chosen runner material. Strain gauge measurements carried out on the model turbine allowed FEA calculations of the stress levels and bending loads on the buckets derived from the pressure in the buckets to be validated, see fig. 2.

The change from a relatively simple and highly conservative model of the water jet loading on the buckets to more accurate techniques which are based on hydraulic rig test data for the bucket loading has allowed substantial improvements to be made in the mechanical design. The development of CFD simulations as a replacement for the hydraulic test rig data for the definition of the water-jet loading of the buckets will be the next step in this
process of improved mechanical design. Accurate simulation models will allow better physical understanding of the forces involved and allow more accurate transfer of know-how derived from one design to another one.

2. Hydraulic Design and CFD methods

The hydraulic design of the Bieudron power plant was also a novelty in that several components, such as the nozzles and the distributor, were based on hydraulic design experience from two separate companies, Hydro Vevey and Sulzer Hydro, now united in a single company VA TECH. The new company is able to draw on an even wider range of experience based on the joint Pelton turbine references.

The hydraulic design of Bieudron was based on classical empirical methods for Pelton turbine design derived from analysis of test data on extensive turbine prototype and model tests. Within VATech, the design of the components in this successful machine design is now also being used as a test case for validating the use of Computational Fluid Dynamics (CFD) in the design of Pelton machines.

Three different commercial CFD codes (CFX-TASCflow (7), CFX-4 (8) and FLOW-3D (9)) have been used for different parts of this work, making use of the best features of each code for studying different aspects of the complex features of the flow. The fact that three separate codes have been used demonstrates that such simulations are at the forefront of simulation technology, as no single commercial code is currently able to deal with all aspects of the flow.

2.1 Distributor

The analysis of the flow within Pelton distributors has been the subject of CFD investigations for only three years (ref. 10, ref. 11). Simulations of the distributor components were initiated before simulations of the bucket and jets as the CFD codes and application techniques in use for the components of Francis and Kaplan turbines could be directly applied with no major difficulties to the distributor. The main issue that had to be solved was the efficient generation of computational grids for these complex components. The development of parameterised design data in the form of CAD solid models for the distributor considerably eased the grid generation process, as the CAD data was directly imported into the ICEM-hexa (see 12) and a block-structured grid could be derived.

Figure 3: Computational grid of a 5 nozzle Pelton distributor
The flow within the distributor of a Pelton turbine is of importance for the performance of the turbine, firstly because of the losses produced within the distributor itself, but more importantly because any flow disturbances due to secondary flows and separations arising in the distributor can have a determining influence on the quality of the jet leaving the nozzle. The optimisation of the distributor geometry is therefore one of the most important steps on the way towards optimal Pelton turbine designs.

**Computational Method**

The flow field within the distributor itself can be simulated by conventional CFD methods for which extensive experience is available from other turbines: the Reynolds averaged Navier Stokes equations are solved by a Finite Volume technique using a standard turbulence model, such as the standard k-ε model or its Kato-Laundner modification. The result is a prediction of the steady state flow field. Details of simulations of this type carried out with the commercial CFD code CFX-TASCflow are described in ref. 7.

**Results**

Within VA TECH there have now been several thorough CFD studies of Pelton distributor flow, including the Bieudron distributor. The CFD calculations have been validated against measurements made in our test rigs (pressure measurements and flow visualisation) and good agreement has been found between the test data and the numerical results.

![Figure 4: Velocity distribution in the plane of symmetry](image)

Generally the results of the CFD studies show that the flow within the distributor suffers from flow separation within the bifurcations, see fig. 4, and the flow is distorted by secondary flow in the bends. The flow studies have made it possible to improve the geometry of the bifurcations such that the separation zones could be reduced and, as a result, the losses within the distributor could also be reduced, but these losses were found to be relatively unimportant compared to the losses of the nozzle. The influence of the non-uniformities in the distributor flow on the quality of the jet from the nozzle turned out to be more important.

**2.2 Nozzle**

**Computational Method**

The flow field within the nozzles can also be handled by the standard CFD codes which have been applied for the flow simulation within the distributor but two special problems have to be dealt with. Firstly, the highly accelerated flow in the nozzle leads to unrealistic generation of turbulent kinetic energy when the standard k-ε turbulence model is used. The weakness of the standard the k-ε turbulence model in this respect is well documented
for both stagnation point flows and for highly accelerated flows (13) and is related to \((\partial u / \partial x)^2\). The Kato-Lauder modification of the k-\(\varepsilon\) turbulence model is found to be necessary to avoid the unrealistic production of turbulent kinetic energy at the stagnation point of the central bulb and in the acceleration zone in the nozzle. Secondly, the outlet position of the computational domain and the appropriate outlet boundary condition are not easy to define. The jet exits the nozzle into the surrounding air, which means that a single phase flow simulation is no longer valid downstream of the nozzle exit. Examination of this effect by the use of several different outlet boundary positions and different outlet boundary conditions have demonstrated that the simulation results in the near outlet flow field are highly sensitive to this.

Therefore the flow within the nozzle has been calculated by applying two different computational methods. The first calculation is done in one computational domain with the distributor using a single-phase CFD code as described above. The second calculation of the nozzle is done with a multiphase CFD code by using the results of the first calculation at the inlet of the nozzle. Here the computational domain includes the nozzle and the region between nozzle and bucket as well.

**Results**

The velocity field at nozzle outlet is determined by the flow features within the distributor. It was found that the jet quality is affected by the non-uniformities of the velocities, by the swirl in the flow and by the distribution of the turbulent kinetic energy. This is clearly illustrated for the extreme case of a five nozzle distributor operating with only one nozzle open. When the last nozzle of the distributor is open the flow field is fairly uniform and the efficiency is high. But when only the second nozzle is operating the flow field within the distributor is disturbed by a separation bubble at the bifurcation, by the high curvature of the bifurcation and also a separation at the gusset blade which leads to a strong swirl structure with high turbulent kinetic energy at nozzle outlet, see fig. 5. The relevance of this can be seen from the fact that the measured efficiency for the latter case is 0.5 % lower than for the first case.

![Figure 5: Turbulent kinetic energy and streamlines at nozzle outlet (only nozzle 2 operating)](image)

**2.3 Jet**

With the free surface flow simulation of the Pelton jet we cannot rely on the experience from Kaplan and Francis turbines but need to start a new era of flow simulation and hydraulic design of Pelton turbines. The free surface simulations of the jet, and particularly the location of the separation of the jet from the nozzle outer wall, require special fluid dynamic models. The volume of fluid (VOF) method is used to model the two phase, and additional models are needed to reconstruct the fluid surface and to take into account the cavitation or ventilation of the flow at the nozzle exit. The accuracy of these models is of crucial importance for the point of separation and the shape of the jet. As there was no experience how to correctly simulate the free jet flow two leading CFD codes with free surface capability, were validated with experimental data, see ref. 14.

**Experiments**

It was quickly realised that for the validation of the CFD simulations no suitable test data were available to provide accurate measurements of flows with a free surface. Two series of validation experiments were initiated. First, a test rig was designed at the EPFL involving a jet impinging at different angles onto a flat plate, whereby measurements of the surface static pressure on the plate are available for validation with CFD simulations. (ref.
Secondly a model of a pelton turbine nozzle was investigated in a test rig at Sulzer Innotec and LDA measurements of the velocities in various cross sections and at various positions along the jet were carried out. LDA measurements in Pelton jets are particularly difficult because the rough unsteady surface of the water jet disturbs the laser beams. A new method was invented to measure free jet flow involving a small glass prism which was brought up close to the jet so that it just touched the surface of the jet and provided an undisturbed path for the laser beams into the flow and for the doppler shifted beams out of the flow, ref. 16.

**Computational Method**

Two different multiphase codes, Flow-3D of Flow Science Inc. and CFX-4 of AEA, were used for the calculation of the nozzle-jet flow and were validated against the test data mentioned above. Both codes use the volume of fluid method to calculate the free surface.

The Navier-Stokes Code Flow-3D uses a fixed grid of rectangular elements and only calculates the flow field of the liquid phase. These two features allow CFD calculations to be carried out very quickly and with low costs of CPU. The non body-fitted computational grid has advantages with regard to ease of grid generation but can lead to a wavy or uneven representation of surfaces that are not orientated along the grid lines.

The CFX-4 code is tested on a structured multiblock grid which is body fitted. This code not only simulates the flow of the liquid phase but also the flow of the air which generally leads to higher computational time.

**Results**

The comparison between measurement and the CFD simulations is carried out at three cross sections of the jet. Figure 6 shows the curves of the measured velocity compared to the calculated ones. Both codes have their advantages and disadvantages for this case. The velocity distributions within the jet predicted by Flow-3D are not as good as those of CFX-4 results, whereas the CFX4 simulations are less precise with regard to the exact position of the jet surface. Being aware that these simulations are the first of this type and there is still work to be done with respect to the modelling of the separation of the water jet from the surface of the nozzle or with respect to the grid effects, we were nevertheless encouraged to go ahead and simulate the flow of the jet – bucket interaction.

![Pelton Nozzle In 1012, s/smax=70%](image)

Figure 6: Comparison of measured flow velocities and predicted flow velocities at 3 different cross sections of the jet. CFX4 simulation and Flow3D simulation

**2.4 Bucket**

The simulations of the moving bucket and its interaction with the jet require not only CFD models of the 3D turbulent viscous flow but also special models to deal with unsteady flows in a rotating frame of reference, moving sources, and free surfaces. In the Spring of 2000 Flow-3D was probably the only commercial CFD code capable of this kind of computation and, together with Sulzer Innotec, we used it for the first ever simulation of the Pelton jet-bucket flow in a rotating frame of reference.

In parallel investigations of the stationary free surface flow of the jet into a non-rotating bucket have been performed, see ref. 15 and 17.

**Computational Method**

The unsteady, Reynolds-averaged Navier Stokes equations are solved in the rotating frame of reference with the jet entrance simulated as a moving mass source. The velocity profile, position and motion of the jet in the
ing frame of reference can be prescribed. This allows the velocity profile resulting from the nozzle-jet calculation to be imposed as the inlet boundary condition for the jet-bucket simulation. As mentioned above the water surface is simulated by means of the volume of fluid method.

Figure 7: Jet bucket interaction at angle position –16°, photo of test rig (left), CFD simulation (right)

Figure 8: Jet bucket interaction at angle position –24°, photo of test rig (left), CFD simulation (right)
The calculations presented here are carried out on a computational grid of 1,500,000 nodes. The computational domain contains four half buckets, as the flow field is assumed to be symmetric. For these calculations a LES turbulence model turned out to be most successful. Each calculation took about 14 h CPU time on a Compaq Workstation with a single DEC alpha processor.

Results
The calculated flow field was firstly compared with photographs of a model turbine in a test rig at different time steps or bucket positions respectively. The direction of the jet and the interaction of the jet with the bucket is well predicted by the CFD simulation, see fig. 7 and fig. 8, but there are still some problems of the CFD simulation with a proper reconstruction of the free surface of the jet. The pictures also show the non body-fitted structure of the computational grid which leads to a rough appearance of the bucket and the jet.

The calculated pressure distribution on the bucket surface from the CFD simulation shows some differences to the measured pressure in the model turbine. In figure 9 the comparison of the measured and calculated pressure distribution in the bucket is shown for two different angle positions. The left hand picture is related to the situation when nearly the whole of the jet is just hitting the edge of the bucket, see fig. 7. The prediction shows the maximum pressure in the right position, but the measurements cannot show this pressure distribution because it is not possible to mount pressure transducers on the edge of the bucket.

A similar problem with the information which can be derived from the measurements can be seen for the angle position -24 in the right hand side of fig. 9. The measured pressure distribution is an interpolation of 19 measurement points marked by red circles in the middle of fig. 9. The region of high pressure gradients as indicated by both the measurement and the calculation is not resolved sufficiently by the measurement which makes it difficult to judge the quality of the numerically predicted results. The pressure distribution which results from the calculation shows the effect of a skewed jet impinging on a surface far stronger than the measurements and the pressure maximum is situated a little bit nearer to the entrance region of the bucket.

Figure 9: comparison of the pressure distribution for two different angle positions, measurement left, simulation right.

The comparison between the calculated and the measured results show an encouraging good agreement on one hand but also show the need for some more validation of the CFD calculation. The simulation shows physical effects which the measurements cannot show because of the position of the pressure transducers. Knowing the numerically predicted flow field it is now possible to define experiments more precisely and to hereby validate the CFD methods.

1. Conclusion

The major technological break-through which enabled the outstanding Pelton plant in project Bieudron to be realised was derived mainly from more sophisticated methods in stress analysis and mechanical design for Pelton turbines. Pressure measurements in the rotating bucket were combined with FE analysis and substantial empirical knowledge, to extend the validity of existing models and to improve the physical understanding of the engineers.

This improvement in simulation technology has now been extended to the hydraulic analysis of the Pelton turbine flow. The simulations of all the components presented in this paper represent a milestone in the use of flow simulation methods for Pelton machines. All the CFD simulations presented have been validated against measurements, many of which were made during the development of the Bieudron turbines. The validated tools will
allow the next generation of Pelton turbines to be designed making use of a combination of empirical know-how from previous experience and an improved physical understanding of the complex flow. It is now possible – for the first time – to simulate velocities and the pressure in a whole Pelton turbine and thereby to optimise the hydraulic profiles with respect to the hydraulic efficiency on one hand and on basis of the calculated pressure loading with respect to the mechanical design on the other hand.

Acknowledgements
The authors would like to thank the many engineers who have been involved in the research and development described in this paper. Special thanks go for the Sulzer Innotec, Sulzer Markets and Technology Ltd. for the work of Felix Muggli and Sebastian Hirschberg on the Flow-3D simulations of the bucket-jet interaction presented here, to Zhengji Zhang for his innovative tests in Pelton jet flows and to Michael Casey for his careful editing and correction of the English text in an early draft of this paper. A special note of appreciation is also due to the late Jack Rettich, whose skill and expertise in measurement methods and data acquisition enabled the pressure field in the model rotating Bieudron Pelton buckets to be obtained.

References
12 ICEM CFD Version 4.0, 1999. ICEM CFD Engineering, Berkeley, CA, USA.


Authors

Mirjam Sick is responsible for CFD tools development within VA TECH ESCHER WYSS Ltd. She obtained her Ph. D. in numerical simulation with CFD from the University of Karlsruhe, and joined Sulzer Innotec in 1995 where she was first a research engineer and then project leader for CFD projects involving stage simulations of Francis turbines, Kaplan turbines and axial compressors. Since joining VA TECH (formerly Sulzer Hydro) she has further pursued the use of CFD in new applications, such as Pelton buckets.

Helmut Keck, PhD, is manager of technology and R&D at VA TECH ESCHER WYSS Ltd. Having been with Sulzer Escher Wyss and Sulzer Hydro for more than 20 years, he has been involved in the project Bieudron from the early beginning in various tasks and functions.

Gerald Vullioud, Mech.Eng. graduated from EPF Lausanne, Head of the Research & Development department of VA TECH HYDRO VEVEY Ltd. He is an internationally well known expert in the field of Pelton R&D and provided key contributions to the development and design of the Bieudron turbines.

Etienne Parkinson studied hydraulic engineering in Grenoble (France) and followed his studies with a PhD in Computational Fluid Dynamics at the Ecole Centrale de Lyon (France). Following 5 years as a research engineer at EPFL-IMHEF in Lausanne (Switzerland), he joined VA TECH HYDRO where he is now active in hydraulic research & development.