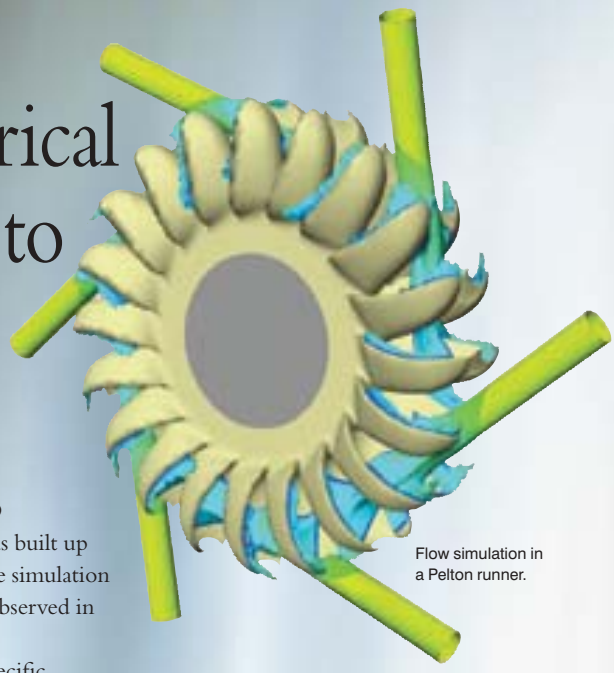
Developments in numerical flow simulation applied to Pelton turbines

by Etienne Parkinson, VA TECH HYDRO Ltd., Switzerland



Flow simulation in a Pelton runner.

Flow patterns in a Pelton runner.

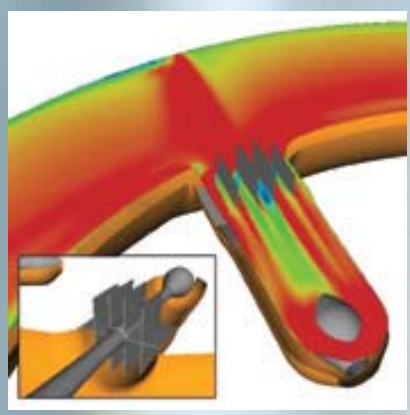
Thanks to developments in simulation methods over the last twenty years, the optimisation of the hydraulic design of reaction-type water turbines (Francis and Kaplan turbines) is now primarily carried out by means of numerical flow simulation. In the design process, numerical flow simulation provides the engineer with a rapid and reliable tool to obtain a better understanding of the flow and to quickly examine various design options in order to select the optimum configuration.

A similar move towards the use of simulation methods is now taking place in the field of Pelton turbines. Until now, the flow in Pelton turbines has not been analysed in such detail as the flow in Francis and Kaplan turbines. One of the reasons for this is that the flow patterns and the hydraulic losses are very difficult to observe and quantify by experiments. This is due to the very complex flow processes that occur in Pelton turbines, which include pressure losses, secondary flows, jets, film flow, free surfaces, spray formation, ventilation losses, unsteadiness, and complex interaction between the components.

Following a systematic step-by-step process with an optimum use of numerical and experimental developments, using CFX-5, VA TECH HYDRO (www.vatech-hydro.com) has built up wide ranging expertise in the simulation of unsteady free surfaces as observed in Pelton turbines.

Jet visualisation of a Pelton turbine in hydraulic laboratory.

With the aid of many specific experimental developments designed for validation of free surface flows conducted with our research partners and the strong interactive collaboration with the CFX group, VA TECH HYDRO has now achieved a major breakthrough in the simulation of unsteady free surfaces. These simulations include, among many other applications, the validated numerical flow simulation in a rotating bucket, and the correct prediction of the shape of a water jet following a bend pipe.

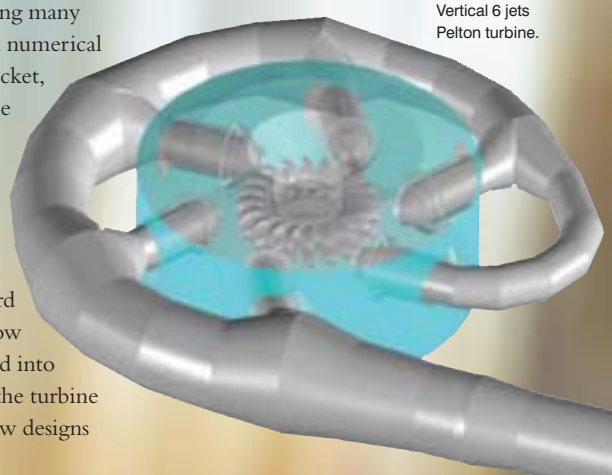


CFX simulation of flow field in a bifurcation.

There are still numerous CFD challenges in the simulation of unsteady Pelton turbine flows. Every step forward brings a new insight into the flow patterns, which is then translated into design and analysis to improve the turbine performance. This is true for new designs but also for refurbishment projects where only some of the turbine components are replaced. A valid analysis, both qualitative and quantitative, of the existing components and their interactions is a key for the success of such projects.

> 'CFX-5 analysis of Pelton turbines is helping to improve their performance.'

CFX-5 Computational Fluid Dynamics software is helping to make possible the analysis of Pelton turbines in order to improve performance.



Vertical 6 jets Pelton turbine.