Free surface simulations of intakes for low head machines

Simon Weissenberger
Andritz Hydro GmbH
P.O.Box 28
4031 Linz
Austria

Josef Steinmassl
Andritz Hydro GmbH
P.O.Box 28
4031 Linz
Austria

Peter Grafenberger
Andritz Hydro GmbH
P.O.Box 28
4031 Linz
Austria

Introduction

For low head axial turbines the influence of the intake design on the performance is significant due to a relatively small submergence of the intake below the free surface on one side and high discharge rates on the other side. Common methods for design and optimization of hydropower intakes are based on experience, guidelines and physical model testing. The procedure to achieve an optimal, uniform approaching flow towards the turbine units without disturbing flow patterns such as air entraining vortices can become time-consuming and expensive. Therefore especially for small hydro power projects with low budgets there is a demand for a reliable and economic way to predict and optimize the approaching flow.

During the last decade much effort has been put into the development of methods for the simulation of the transition from open channel flow to pressure flow in order to accelerate the intake design process. On the topic of the detection of visible vortices most research has been done in the field of pump intakes (Tomoyoshi Okamura et al. 2007). In recent years the focus has been set to simulations of intake flow using free surface methods in Computational Fluid Dynamics (CFD). The major advantages of CFD are its flexibility with respect to geometry modifications and the possibility of a detailed investigation of the fluid flow structure, since all velocity components and the pressure field are accessible within the whole simulation domain. Many CFD studies have been performed with the water surface modelled as a fixed free slip wall. Only a few studies included a full modelling of the free surface. For most of these studies only qualitative comparisons with measurement data have been done. The complexity of free surface modelling can become very high because of the use of multiphase models, large simulation domains, fine grids and highly sophisticated turbulence modelling (LES) rather than the standard RANS models (k-ε-model). But even for complex simulations using the inhomogeneous VOF multiphase model and the SST turbulence model with curvature correction implemented by Langtry and Menter the quantitative comparison with measured data shows that the CFD significantly underestimates the rate of rotation of the vortex near the free surface (France Suerich-Gulick et al.) while vortices are reproduced qualitatively well by CFD.

Furthermore CFD has shown to be a valuable part in a multidisciplinary approach for the optimization of hydraulic design of a penstock intake using numerical and physical modelling (Marc Villeneuve et al. 2009). Even if for certain design aspects numerical modelling can already be seen as an alternative to physical modelling it should be principally employed complementary to scale models.

The successful application of CFD in the design of penstock intakes encouraged to check if a similar methodology would be possible for the optimization of low head axial turbine intakes. This paper presents a qualitative comparison of test and simulation results of CFD simulations of a complete scale model which was tested for the Lower Saint Anthony Falls StrafloMatrix™ project.

Also the benefit of free surface CFD for projects of this kind will be addressed. After presenting the motivation for this paper (1), a short presentation of the investigated project follows (2) and a description of the physical scale model testing is given (3). The CFD model and solution process is described (4) and finally the results are discussed (5).

1. Background

All low head axial turbine projects have in common that the best efficiency and longest operating life of the power plant is achieved with an undisturbed, almost uniform approaching flow towards the units. To accomplish these operating conditions, an analysis of the open channel flow in the upstream region of the power plant has to be performed. If indicated, refurbishments of the structures have to be executed.

HYDROMATRIX® turbines are a member of the family of low head axial turbines. They are small unregulated units equipped with synchronous or asynchronous generators and usually installed in groups into already existing dams, locks or gates which originally have been used for solely for irrigation, shipping ore similar purposes. But there are also HYDROMATRIX® projects where the complete power plant structures are designed from scratch.

Since the only way to regulate the discharge of a HYDROMATRIX® power plant is by taking single units in and out of operation, several different operating regimes have to be investigated. For this purpose a physical model is optimal but a full bay physical model test is costly.

Therefore there is high demand for an option to accelerate physical model testing, respectively to pre-optimize the intake design in an early stage of the development process. Such a reliable design tool is very interesting for all kinds of low head hydropower projects, especially for small projects where a scale model test would exceed the budget.

CFD has already become a very important tool in the hydraulic design of turbines and their components such as spiral casing, draft tube, guide vanes, etc. Continuous improvements of the commercial CFD codes with respect to multiphase flow on one side and increasing computational power on the other side make it also a natural candidate to simulate the transition from free surface flow in front of the power house to pressure flow inside the turbines. To keep the tool useful for real projects some basic requirements have to be met:

- The location of CFD predicted vortices have to coincide with the location of the appearance of vortices at the physical scale model
- There has to be a correlation between the simulated vortex strength and air entrainment at the physical model
- The simulation shall be able to be performed with appropriate effort regarding the use of resources, computational time and license costs.

To ensure the first two requirements a transient simulation using very fine meshes, highly sophisticated turbulence modelling and an inhomogeneous free surface model would be perfect. The third requirement makes a transient simulation and the use of an inhomogeneous free surface model almost impossible, especially if a fine grid shall be used throughout the whole simulation domain. But a fine mesh is definitely required to accurately predict the appearance of vortices.

The Lower St. Anthony Falls hydro power project is one example for a project taking advantage of an existing dam. Full bay model tests were performed to investigate the approaching flow. The well documented results of these model tests formed the basis of the attempt to consider ways and means to support future intake design by exploiting CFD simulations presented in this paper.

2. Lower St. Anthony Falls project description

The Lower St. Anthony Falls (LSAF) StrafloMatrix™ project is a 10 MW low-head hydropower project located at the Lower St. Anthony Falls Lock and Dam in Minneapolis, Minnesota, USA. It consists of a row of eight Modules. Each Module consists of two vertically arranged StrafloMatrix™ Turbine-Generator units (TG-units) and has a rated generating capacity of 1.237 kW at a gross head of 7.62m and a Module discharge of 21.95m³/s. The Modules are installed in front of a newly constructed concrete bulkhead structure located in the existing auxiliary lock. Challenges of the structural design of this project have been addressed in a paper by Melvin Koleber et al. 2009. The Modules can be separately lifted up above the water level for maintenance or during flood conditions. The draft tube gates may be in open or closed position depending on the available discharge. The maximum hydraulic capacity of the power plant is about 175.6m³/s. Flows exceeding the hydraulic capacity of the Modules are discharged through the existing dam gates located adjacent to the auxiliary lock.

3. Physical Model Test description

In preparation for the LSAF hydropower project hydraulic model testing was performed at the University of Technology Vienna in 2007. A full bay model with a scale of 1:15 was used to investigate the approaching flow conditions and the flood discharge capacity of the auxiliary lock with and without the power plant. The main objectives of the first task were to identify all hydraulic effects of the power facility in terms of the generation of swirls and vortices and the detection of possible air entrainment due to surface vortices in all potential combinations of TG-unit operating regimes. The model was run on the basis of Froude's law of similitude which requires a model scale greater than 1:20 and sufficient high water temperature. For the detection of possible air entraining vortices in the prototype the tests were also performed with 10% increased flow rates ($1.1 * Q_{Froude}$) in order to amplify vortex formation phenomena.

The tests were performed using a plexiglass model of the auxiliary lock at the LSAF Locks and Dam facility. The complete model setup consisted of a rectangular inflow chamber to calm the water connected with the forebay by means of a stilling device to obtain uniform approaching flow conditions to the actual model of the auxiliary lock and the implemented power-plant. The model of the auxiliary lock itself had a width of 1.15 m a length of 4 m and a height of 1.50 m and its walls were manufactured with transparent plexiglass panels to achieve a good visibility for the flow observation. A submerged Tainter gate was partially rebuilt in a simplified form and mounted in fully opened position. The turbines were modelled as pipes with a diameter of 0.088 m which corresponds to the prototype diameter of 1.32 m.

For the execution of the model tests lengths, discharge and velocities were scaled according to Froude's similarity law.

Length, pressure head	$L_{model} = 0.066667 * L_{nature}$
Discharge Q	$Q_{model} = 0.0011476 * Q_{nature}$
Velocity v	$V_{model} = 0.258199 * v_{nature}$

Table 1 Conversion factors to a scale model of 1:15

Reynolds number and Weber number were high enough to have no significant impact on vortex formation with air entrainment due to scale effects. The test were performed with quasi steady inflow conditions with opened and with closed draft tube gates over a time period of 10 to 30 minutes. The model was tested with constant flow discharges and various headwater (HWL) and tail-water (TWL) levels. The model discharge was measured with a magnetic inductive flow-meter in the water supply system of the laboratory. The water head-water level was read off in front of the matrix-powerhouse and the tail-water level was metered directly behind the powerhouse. The flow patterns were visualized using different methods such as dye injection. The main test cases were documented with a camcorder.

Seven different configurations of units in operation were tested with different discharges and water levels. The paper will concentrate on two different flow cases. Furthermore for each flow case simulations of the initial design and a modified geometry were done. Flow case 1 is the main flow case with all sixteen units in operation and flow case 2 with eight units at the left side of the bulkhead structure in operation. The flow cases to be presented in this paper were selected so that the main flow case which is symmetric and one asymmetric flow case were captured. The findings of the comparison of CFD simulation results and physical model test observations do not depend on a particular flow case.

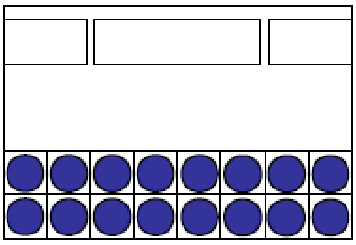

Fig. 1 Flow Case 1

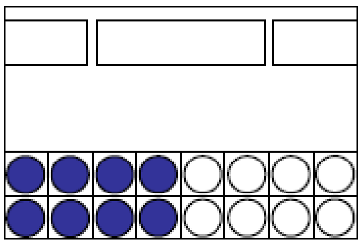

Fig. 2 Flow Case 2

The model tests of flow case 1 did not show a completely uniform approaching flow. Relatively strong stable surface vortices in front of both corners upstream of the power house with air entrainment occurred at all tested head-water levels. Since these vortices most probably would have affected the performance of the most lateral units, a solution to minimize the disturbances had to be found.

The solution to this problem turned out to be a floating barrier in front of the bulkhead. This floating barrier changed the flow pattern in such a way that an almost uniform approaching flow was achieved.

The tests of the asymmetrical flow case 2 showed one strong air entraining vortex in the corner in front of the operating units. The air entrainment fluctuated between the upper and the lower left units.

4. Simulation setup and procedure

To simulate the LSAF physical model test with CFD the complete testing domain was modelled with CAD. For the simulations including the floating barrier, the floating barrier had been modelled as a straight pipe fixed at the simulated head-water level across the auxiliary lock. An unstructured tetrahedral mesh with refinements close to surfaces with high curvature and the outlets was generated with the commercial software ANSYS ICEM CFD.

To get the unabridged version of this paper please contact:

ANDRITZ HYDRO GmbH
Lunzerstrasse 78
A-4031 Linz, Austria

Phone: +43 (732) 6986-0
Fax: +43 (732) 6980-2554
Hydromatrix@andritz.com