IMPROVING MEASUREMENTS IN POWER PLANTS USING NUMERICAL FLOW SIMULATION

T. Staubli, S. Hug

Abstract: Numerical flow simulation provides valuable information on flow conditions prevailing in Hydro Power Plants. Such information may be used in combination with onsite measurements in various ways. For static pressure measurements, current meter measurements, the pressure time method or the acoustic transit time measurements, the integration uncertainty can be estimated and the measurements eventually be corrected. Optimum positioning and alignment of transducers can be carried out on the basis of the simulated flow fields. The paper presents selected examples of cases where numerical flow simulation showed to be useful.

1 Introduction

Measurements in hydro power plants should be carried out in accordance with international or national standards, e.g. IEC60041 [1], ASME PTC18 [2], ISO 3354 [3]. However, when planning the tests one often is confronted with the situation that conditions to be fulfilled do not fully agree with the requirements listed in the standards. The following questions arise in such cases:

- Are measurements still possible and what is the effect on measurement uncertainty?
- Are there means to improve the accuracy the measurements?

Standards propose boundary conditions to be fulfilled, e.g. required distances from upstream or downstream disturbances needed in case of flow measurements or straight pipe sections for the use of the pressure time method.

In the following, typical cases will be discussed where numerical flow simulations (CFD) can successfully be used in order to improve measurements in power plants. Some selected examples will also be given.

1.1 Current meter measurements

IEC60041, section 10.2, deals with flow rate measurements using propeller type current meters. For close conduits and regular flow conditions this standard refers to ISO 3354 [3]. For measurements in short intakes an increased measurement uncertainty is assumed. A penstock is defined as short if the straight length is less than 25 diameters, as stated in section 10.2.4.

The uncertainty may rise due to an increased integration error since the velocity profiles deviates from ideal, fully developed velocity distributions. Furthermore, increased uncertainty arises from an unknown incidence of the flow with respect to the
current meter axis or from local or global swirling flow, which might increase or decrease the speed of rotation of the current meters. For short intakes a CFD study will provide information for a best possible alignment of the current meters and for estimation of possible integration errors. For long conduits with downstream or upstream disturbing elements from the measurement section, CFD will also provide useful information. Summarizing CFD may be useful for:
- Optimize the alignment of current meters in case of oblique flow,
- Estimation of the uncertainty due to integration,
- Analysing the influence of swirling flows on the current meter readings,
- Quantification of the blockage effect of the mounting structure.

1.2 Acoustic transit time measurement

It is foreseen to include the acoustic transit time method (ATT) in next version of the standard IEC60041 as a primary method for discharge measurements. However, the method most likely will be restricted to circular or rectangular cross sections and to cases of fully developed velocity distributions without major disturbances from upstream or downstream elements, such as piers, bends, bifurcations or valves. As an example, the ASME PTC18 code requires no disturbing element at least ten diameters upstream and three diameters downstream from the measurement section. A CFD study might here be useful to estimate the influence of disturbing elements upstream or downstream of the flow meter installation, since the distance is not the only important factor but also the type of disturbance. From the predicted degree of the disturbance of the velocity distribution of a specific installation a minimum number of acoustic paths can be determined which are necessary in order to attain a certain maximum integration error. For installations, deviating from the recommendations given in the standards (short intakes, converging sections or non-circular flow cross sections) best possible orientation, positions and weightings of the acoustic paths can be predicted on the basis of CFD results. In case of heavily disturbed flow within the measurement cross section, the uncertainty due to integration for the chosen integration method (Gauss-Legendre, Gauss-Jacobi, OWICS, OWISS) can be estimated [4]. Transducers used for the acoustic transit time measurements protrude in general to a certain degree into the flow cross section. Such a protrusion effect reduces on one hand the acoustic path length and on the other hand disturbs locally the flow field around the transducers. Both effects may eventually cancel or be negligible for large pipe diameters. The quantitative influence of the protrusion effect on the flow rate measurements can only be determined by means of CFD simulations [5].

1.3 Pressure time measurement

According to IEC60041, section 10.4.2.1, the conduit shall be straight and have a constant cross-section within the measuring reach and not present any significant irregularity. The distance between the two measuring sections shall not be less than 10 m. Another rule, found in the literature, is that this distance should be two times of the pipe diameter if the pipe diameter is more than five meters. CFD allows to deter-
mine the equivalent value of the geometry factor for a measuring in a penstock segment with non-standard geometry, as described by Adamkowski [6].

1.4 Pressure measurement

Regarding the choice of the pressure measurement section, the following statements can be found in IEC60041, section 11.4.1: “Special attention must be given to the location of the measurement section. There should be a minimum of disturbance to the flow. Sections where the velocity pattern is distorted by an elbow, valve or other flow disturbances outside of the hydraulic machine should be replaced if possible by other measurement sections with better flow conditions. The measurement section should preferably be arranged in a straight conduit section (which may also be slightly convergent or divergent) extending three diameters upstream and two diameters downstream from the measurement section and free from any water extraction or injection active during the test. Closed branches shall be more than five times their diameter away from the measurement section.”

Especially in older power plants the existing pressure taps often do not fulfil these requirements stated in IEC60041 and, in addition, plant owners are reluctant to drill additional pressure taps in their penstocks. Accordingly one has to live with the given conditions. CFD allows here to estimate the increased uncertainty of the measurement or to apply a correction if all parties agree upon. An example will be given in section 3.3.

1.5 Thermodynamic measurement

Major uncertainty of a thermodynamic efficiency measurement originate from the un-explored energy distribution at inlet and outlet measurement sections, as e.g. described recently by Ramdal [7]. Here CFD may help to estimate the influence of energy distributions and to find the best positioning of the temperature probes. For instance, it does not make sense to mount probes on the inner radius downstream of a bend since all boundary layer fluid is collected there leading to increased local temperatures [8].

2 CFD simulation

When applying CFD to improve measurements in power plants it is essential that the simulation fulfil minimum quality criteria. A report of CFD simulations should allow tracing back all essential steps of the simulations.

2.1 Geometrical model for CFD

Basis of a reliable CFD simulation is an adequate model of the geometry. This comprises:
- Definition of the simulation domain (inlet, outlet).
- The geometry must be modelled in a way that the flow predictions encompass the major influences on velocity distributions.
- Drawings of plants are often incomplete or inaccurate. Geometrical checks are necessary or as-built and approved drawings should be used.
- Simplification of the geometry is allowed (e.g. inlet trash rack, bulkhead gate slots), assuming the influence of these simplifications on velocity distributions is negligible.
- To enable a good meshing it is essential to set the geometrical tolerances of the CAD model to a minimum. This is especially important for transitions between different cross-sections or for edges so that the boundary layer can be accurately projected onto the geometry.
- Export of the geometry from the CAD software in a suitable format to the used meshing program.

2.2 Numerical models

In general, steady state simulations will be sufficient to predict time averaged velocity distributions. However, it should be noted that bend flows, wake flows, and swirling flows are inherently unsteady.

Concerning the turbulence modelling, the SST (shear stress transport) turbulence model in general provides satisfying results. Dissipation is sometimes overestimated with the SST model, leading to secondary flows which are damped out within short distances. LES, DES (large/detached eddy) models are also well suited for conduit flow predictions.

The high resolution advection scheme assures good convergence for most simulations. High resolution locally adapts the scheme of the equation solver. In areas with high local gradients the solver uses a first order scheme and in areas with low gradients, it is solved in general with a second order scheme. Using the high resolution scheme ensures in most cases a stable solver process.

Simulations with long and/or complicated geometries (cross-section changes, bifurcation, surge chambers, one of several turbines in operation, etc.) can first be simulated with an upwind scheme so that the high resolution solution converges in a next step.

3 Case studies

3.1 Current meter measurements: Kootenay canal simulations

In October, 2009 a series of comparative intake flow method tests were conducted by the Centre for Energy Advancement through Technological Innovation (CEATI) at the Kootenay Canal, British Columbia, Canada, under the direction of the American Society of Mechanical Engineers (ASME) Performance Test Code (PTC) 18 Hydraulic Turbines and Pump-Turbines [10] [11].

Current meter measurements resulted in an average 1.1 percent, higher flow rate compared to the reference measurement and all other applied flow rate measurements of the comparative tests.

A CFD study performed by the Hochschule Luzern showed for all current meters an incidence angle of the flow with respect to the current meter axis of in between 6 and 7 degrees due to a lightly oblique flow in the measurement section. From calibration with oblique flow it is known that angles in between 6 and 7 degrees may cause higher readings of the velocities - even for compensated current meters - of the order of magnitude of one percent.
A correction after the measurements is not possible anymore; though a correct alignment of the current meters to the flow during the installation would most likely have improved the accuracy of measurement considerably.

![Current meter setup at the short intake at Kootenay canal](image)

**Fig. 1. Current meter setup at the short intake at Kootenay canal**

### 3.2 Acoustic transit time measurements: HPP Aratiatia

Goal of the CFD study for the HPP Aratiatia, New Zealand, was to predict the uncertainty of an 8-path, two plane, ATT discharge measurement in advance of the installation. Justification for this study came from the heavily disturbed velocity distributions, which were expected. In this power plant three turbines are fed from a common surge tank and are operated in parallel or individually.

After installation of the instrumentation measured path velocities were compared to the simulated path velocities, which were evaluated from the CFD data. A comparison of the axial and cross velocity components on the individual paths resulting from the CFD simulations and the measurement shows good agreement. From this agreement, it can be concluded that the shape of the disturbed velocity distributions as well as the secondary flow is well predicted by the simulation and accordingly also the integration uncertainty. The analysis of the uncertainty on the basis of the CFD simulations showed that the integration uncertainty remains below 1.3 % for the chosen 8 path, two plane configuration, even for the worst operating condition.

The simulation domain extends from a four diameter straight section in the intake tunnel down to the spiral casings of the 3 turbines as shown in figure 2. Downstream of the surge tank the penstock separates into three pipes. After the trifurcation, the
rectangular cross-section changes to the circular cross section which remains constant downstream to the measuring plane.

Figure 2 depicts the simulated flow surfaces of the HPP Aratiatia. The simulated flow measurement sections (marked as ATT) are located in the lower part of each penstock. In the following only the results of penstock 1 and test 1 will be presented. More details can be found in [9].

Three different operating conditions were analysed:
- Test 1: 3x90 m³/s (all three turbines in operation)
- Test 2: 2x90 m³/s (turbine 1 and 2 in operation, turbine 3 shutdown)
- Test 3: 1x50 m³/s (turbine 1 in operation, turbine 2 and 3 shutdown)

The straight intake tunnel of the power plant is about 50 diameters long with a constant circular cross section. The range of the Reynolds numbers lies between Re=7·10⁶ and 4.3·10⁷ for the investigated operating points. Consequently, a fully developed velocity profile can be assumed at the inlet to the simulation domain. This profile is calculated beforehand for each of the given flow rates in a separate simulation with a 2 diameter short straight section with translationally periodic boundary conditions. These velocity distributions as well as the turbulence quantities are then used as inlet conditions for the main simulations.

The number of outlets varies from 1 to 3 outlets depending on the operating conditions. The mass flow is set for each of the outlets. Then the outlet mass flows are linked to the mass flow at the inlet in order to satisfy the mass flow balance.

The free surface of the surge tank is defined as a free slip wall. This means that the water level is constant and the water has no friction at this boundary.
The figure 4 shows the axial velocity (left) and the transversal velocity (right) in the layers (elevations of the acoustic paths) 1 to 4. These velocities are mean values of the two crossed paths on each layer. The maximum deviation of the mean axial velocity between the measured and the CFD results of a layer amounts to less than 10% for all tested cases, the maximum deviation for test 1 was 3%. The deviations of the transversal velocity are larger than the deviations of the axial velocity components. The sign of the transversal velocity components is well predicted. The transversal velocity components are slightly over predicted by CFD. The cross-flow of the ideal velocity distribution is obviously zero. For all 3 tests, the highest transversal velocities are found on layer 1.
3.3 Pressure measurements: HPP Soazza

The pressure taps upstream of the individual injectors of the horizontal axis Pelton turbines in the HPP Soazza are positioned at rather unfortunate location, in a sense that the static wall pressure measurement are done in a cross section with accelerated flow and furthermore with a radial pressure gradient.

In order to estimate the influence of this position on the pressure measurements a CFD study was carried out.

Figure 5 shows the simulation domain, as well as the two planes where the pressure distributions were evaluated. The position of plane A is located just upstream from the internal servo motor body. The position plane B corresponds to the cross-section with the pressure taps. The pressure distribution in plane A is dominantly influenced by the upstream bend flow. On the inner radius the pressures are about 15000 kPa lower compared to the outer radius. Maximum pressure differences in the cross section are of the order of 20 kPa. The pressure distribution in plane B is dominated by the acceleration of the flow around the servo motor body. The blockage of the flow leads on the body to increase pressures. From these findings of the CFD study the static pressure was corrected by 3.1 kPa at nominal flow rate, being a negligible correction of about 0.05% of the head in this case.

![Fig. 5. CFD simulation domain with the locations of plane A and plane B (plane of pressure taps)]
4 Conclusions

For all situations where considerable disturbances may occur in the flow fields, CFD is an appropriate tool to judge whether these disturbances will influence the measured quantities in a specific HPP. Such quantities could be path velocities of an ATT measurement, the speed of rotation of current meters or static wall pressure measurements, to give some examples. The benefits of a simulation can be a prediction of the measurement uncertainty, eventually a statement on the acceptability of measurements, or, in the best case, a correction of the measured quantities.

CFD is especially useful for ATT measurements in case of short intakes. On the basis of CFD best positions and the weights of the individual acoustic paths can be determined.

The case study of the short intake of Kootenay Canal demonstrates that propeller type current meters should be aligned to the flow direction in order to reduce the uncertainty of the measurement.

The flow in the penstocks of the hydro power plant Aratiatia has been simulated for 3 operating points. Due to the trifurcation, the cross-section change and the bends, the velocity profile at the ATT measurement section is heavily disturbed and a cross-flow prevails. Uneven distributions and cross-flow, both, affect the accuracy of flow rate integration. Simulations of the axial and cross-flow distributions agree well with the measurements, from what can be concluded that the numerically predicted integration uncertainty is correct.

For low head power plants the inlet cross-section with the pressure taps often is close to a bend and the static pressure measurement might be influenced by the secondary flow which is induced by the bend. The magnitude of this influence can be estimated by CFD. For low head plants such an influence might be non-negligible. In the case study of Soazza where the static pressure readings are considerably influenced by a bend flow and the flow acceleration around the internal servomotor of the injector this influence is clearly negligible because of the high head of the power plant.
References

[9] Hug, S., Staubli, T., Gruber, P., Comparison of measured path velocities with numerical simulations for heavily disturbed velocity distributions, IGHEM 2012, Trondheim

Author(s)

Dr. Prof. Thomas STAUBLI & Silvan HUG
Hochschule Luzern
CC Fluid Mechanics & Hydro Machines
Technikumstrasse 21, CH-6048 Horw, Switzerland
Phone: +41 41 349 35 52
E-mail: thomas.staubli@hslu.ch

Thomas Staubli graduated in Mechanical Engineering from the Swiss Federal Institute of Technology (ETH) in Zürich. After two years of post-doctoral research in the field of flow induced vibration at Lehigh University, Pennsylvania, he worked in experimental fluid mechanics at Sulzer Hydro (now Andritz Hydro) in Zürich. He then headed the Hydromachinery Laboratory at the ETH Zürich. During this period he directed research projects in the field of hydraulic machinery. Since 1996 he is professor for Fluid Mechanics and Hydro Machines at the Hochschule Luzern.

Silvan Hug graduated in mechanical engineering from Hochschule Luzern – Technik & Architektur. Since 2010 he is research assistant at the CC Fluid Mechanics and Hydro Machines.