Department of Applied Physics and Mechanical Engineering
Division of Fluid Mechanics

Experimental and Numerical Investigation of Axial Turbine Models

Berhanu Mulu Geberkiden
Experimental and Numerical Investigation of Axial Turbine Models

Berhanu Mulu Geberkiden
EXPERIMENTAL AND NUMERICAL INVESTIGATION OF AXIAL TURBINE MODELS

Copyright © Berhanu Mulu Geberkiden (2009). This document is available at

http://www.ltu.se

or by contacting Berhanu Mulu,

Berhanu.Mulu@ltu.se

The document may be freely distributed in its original form including the current author’s name. None of the content may be changed or excluded without permissions from the author.
The research presented in this thesis has been carried out at the Division of Fluid Mechanics, Department of Applied Physics and Mechanical Engineering, Luleå University of Technology, Sweden during the years 2007-2009.

The work was carried out as a part of "Swedish Hydropower Centre – SVC". SVC has been established by the Swedish Energy Agency, Elforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University.

Alstom Hydro Sweden, Andritz Hydro Inepar Sweden AB, E.ON Vattenkraft Sverige, Fortum Generation, Grontmij, Jämtkraft, Jönköping Energi, Mälarenergi, Skellefteå Kraft, Sollefteåforsens, Statoil Lubricants, Sweco VBB, Sweco Energuide, SweMin, Tekniska Verken i Linköping, Vattenfall Research and Development and Vattenfall Vattenkraft, Waprans, VG Power and Öresundskraft are also participating in SVC.

First of all I want to thank my supervisor Dr. Michel Cervantes for his guidance and support during this work, and my assistant supervisor Prof. Staffan Lundstöm for his valuable ideas and encouragement. My heartfelt appreciation also goes to Allan Holmgren and STAFF at Vattenfall Research and Development for all their help and support during the experimental works.

I am indebted to my colleagues at the Division for their interesting discussions and for making the working environment pleasant and fun.

Especially, I owe my deepest gratitude to almighty God and my beloved family and friends.

Hydropower is a versatile renewable source of power generation able to change rapidly operating conditions. Hydropower plants may today work over a larger operating range than designed for due to the introduction of renewable sources of energy and the deregulation of the electricity market. Such operating conditions may involve large stresses and losses due to complex unsteady and transient flow phenomena, which have to be taken into account under design or refurbishment phase. The use of computational fluid dynamics (CFD) in the design and refurbishment process is becoming increasingly popular due to its flexibility, detailed flow description and cost-effectiveness comparing to model testing used since a century in the development of turbines. However, issues have still to be resolved due to the combined flow physics involved in hydropower machines such as partly separated flow at curved surfaces, vortices, unsteadiness, swirl flow, strong adverse pressure gradients, convoluted geometry as well as numerical artefacts. Therefore, experimental data in such complicated systems are required to validate numerical simulations and develop more accurate models.

The first part of this thesis is a numerical investigation of the three-dimensional flow of the axial Hölleforsen model spiral casing and distributor, where the influence of the penstock on the flow is analysed using different turbulence models and inlet boundary conditions. Comparisons with experimental results indicate the importance of the penstock to perform accurate simulation in the present case. Therefore, detailed inlet boundary conditions are necessary to simulate accurately the spiral casing flows if the penstock is not included in the simulation.

The second part of the thesis focuses on an experimental investigation of an axial hydropower turbine model known as Porjus U9. The measurements are part of a project aiming to investigate experimentally the flow in different regions of the machine to build a data bank in order to validate numerical simulations and study scale-up efficiency between model and prototype, since the corresponding prototype is available for similar experiments. The investigation was performed at 3 different working points: part load, best efficiency point and high load. The inlet flow of the spiral casing as well as some sections in the spiral casing and draft tube are investigated with a two components laser Doppler anemometer (LDA). To improve the signal quality and measurement accuracy refractive index matching optical box was mounted on the circular pipe of the spiral casing inlet. LDA result of the mean velocities and corresponding RMS are presented to investigate the flow before the runner and at the inlet of the spiral casing, since the flow is influenced by the existence of a bend before the inlet. The results of the draft tube measurements are also presented. Good quality data are obtained for initial boundary conditions at the inlet of the casing and draft tube cone to perform numerical simulations.
APPENDED PAPERS

Paper A


Paper B


Paper C

Paper A

*Effects of Inlet Boundary Conditions on Spiral Casing Simulation.*

The results of numerical simulations of three-dimensional turbulent viscous flow through the spiral casing and distributor of the Hölleforsen hydraulic turbine (Turbine-99 test case) are presented. Two CAD geometries, with and without the penstock, are analyzed in details to determine the effects of the upstream geometry, i.e. the inlet boundary conditions to the spiral casing. Conservation of mass and momentum equations of the flow are analysed using finite volume methods with the commercial software ANSYS CFX10.0. Standard k-ε with scalable wall function and SST k-ω based turbulence models are applied to study the flow characteristics. Comparisons are made between the numerical simulations with and without the penstock and available experimental results. The numerical results are found to more closely match with the experimental results when the penstock is included in the simulation. Therefore, detailed inlet boundary conditions are necessary to simulate accurately the spiral casing flows if the penstock is not included in the simulation. The numerical simulations also seem to show little sensitivity to the turbulence model.

Paper B

*LDA measurements in a Kaplan spiral casing model.*

This paper presents an experimental investigation of a Kaplan spiral casing turbine model of 1:3.1 scale of a geometrically similar prototype turbine. The prototype is available for similar investigation. The turbine is composed of 6 runner blades, 20 guide vanes and 18 stay vanes and has a runner diameter of 0.5 m. The study aims to investigate the inlet flow of the spiral casing as well as some sections in the spiral casing. The measurements are part of a project aiming to investigate experimentally the flow in different regions of the machine to build a data bank in order to validate numerical simulations and study scale-up efficiency between model and prototype.

A two components laser Doppler anemometry (LDA) apparatus was used to measure the velocity profiles at different locations in the turbine. To improve the signal quality and measurement accuracy refractive index matching optical box was mounted on the circular pipe of the spiral casing.
inlet. Ray tracing technique was used to determine the position of the laser beam intersection and fringes spacing by applying law of refraction and geometric considerations of the laser beams.

The investigations were carried out at three different loads, with constant runner blade angle: best operating point of the turbine and two off-design operating points with the presence of a vortex breakdown. The mean velocities profiles and corresponding RMS at spiral casing before the guide vanes and inlet of the spiral casing are presented for the different loads investigated. Good quality data are obtained for initial boundary conditions at the inlet of the casing to perform numerical simulations.

**Paper C**

*Experimental investigation of a Kaplan model with LDA.*

This paper describes the experimental investigation of a Kaplan turbine model of the Porjus U9 unit. The study aims to investigate the complex unsteady turbulent three-dimensional flow in different regions of the machine to build a data bank in order to validate numerical simulations and study scale-up efficiency between model and prototype. A two components LDA apparatus was used to measure the velocity profiles at different locations in the turbine in coincidence mode. An encoder pulse was used to resolve the measurements angularly. The investigation was carried out at three different loads: best operating point and two off-design operating points (left and right side of the propeller curve). Vortex breakdown was present at both off-design points at the inlet of the draft tube, in the cone. The axial and tangential mean velocity profiles and corresponding RMS at the draft tube cone are presented for the different loads investigated.
EFFECTS OF INLET BOUNDARY CONDITIONS ON SPIRAL CASING SIMULATION
EFFECTS OF INLET BOUNDARY CONDITIONS ON SPIRAL CASING SIMULATION

Berhanu MULU GEBERKIDEN *  
Department of Applied Physics and Mechanical Engineering  
Division of Fluid Mechanics  
Luleå University of Technology

Michel CERVANTES  
Department of Applied Physics and Mechanical Engineering  
Division of Fluid Mechanics  
Luleå University of Technology

*Corresponding author: SE-97187 Luleå, Sweden  
Tel.: +46920491078, +46920492143, E-mail: Berhanu.Mulu@ltu.se

ABSTRACT

The results of numerical simulations of three-dimensional turbulent viscous flow through the spiral casing and distributor of the Hölleforsen hydraulic turbine (Turbine-99 test case) are presented. Two CAD geometries, with and without the penstock, are analyzed in detail to determine the effects of the upstream geometry, i.e. the inlet boundary conditions to the spiral casing. Conservation of mass and momentum equations of the flow are analyzed using finite volume methods with the commercial software ANSYS CFX10.0. Standard k-ε with scalable wall function and SST k-ω based turbulence models are applied to study the flow characteristics. Comparisons are made between the numerical simulations with and without the penstock and available experimental results. The numerical results are found to more closely match with the experimental results when the penstock is included in the simulation. Therefore, detailed inlet boundary conditions are necessary to simulate accurately the spiral casing flows if the penstock is not included in the simulation. The numerical simulations also seem to show little sensitivity to the turbulence model.

KEYWORDS

Initial boundary condition, spiral casing, turbulence model, penstock.

1. INTRODUCTION

The flow in hydropower turbines is very complex with several flow phenomena appearing simultaneously such as: three dimensionality, unsteadiness, separation, swirling flow and turbulence. Due to these circumstances it is highly challenging for computational fluid dynamics (CFD). CFD involves the solution of the governing equations for fluid flow at thousands of discrete points on a computational grid in the flow domain. When appropriately validated, a CFD analysis allows engineers to determine the magnitude, direction and speed of flow at any point in the flow domain. Unlike a physical model, the geometry of the CFD model can be changed on the computer and re-analyzed to explore different options in the project design and in the operational conditions to solve problems involving fluid flow. To predict turbulent flow in complex geometries with computer simulation instead of model experiments can significantly reduce the cost of the project and enhance a profound knowledge of the flow problem. Much attention has been directed to runner and draft tube simulations, while the spiral casing has received minor attention since the losses are minor compared to the runner losses.

The flow in the water intake and the penstock delivers the flow entering the spiral casing, see Fig.1a of the Hölleforsen model. Spiral casing is a major component of reaction turbines. The function of the Spiral casing is to distribute the water as evenly as possible to the stay and guide vanes and then to the turbine runner. The understanding of the water passage through the spiral casing to stay vanes is very important in diminishing the losses of the flow and insures a symmetrical flow to the runner. Also the design of the casing must meet the requirement of the turbine performance in order to attain better overall economic benefit of hydroelectric power plants and to withstand the bursting pressure of maximum headwater plus water hammer.

The water intake and penstock being upstream of the spiral casing, they are responsible for the flow
profile entering the spiral casing. Questions are raised about the inlet boundary conditions at the entrance of the spiral casing due to the presence sometimes of an elbow just upstream; see Fig. 1a. However, most simulations of the spiral casing are generally performed without the water intake and the penstock. For example Carlos Eduardo [12] performed simulation of the Hölleforsen model casing without the water intake and the penstock. The correlation between the simulation and the experimental result available were not optimum.

The present work focuses on simulations of the flow through spiral casing and the distributor (guide vanes and stay vanes) and intends to study the effect of upstream boundary conditions on spiral simulation. The effects of upstream geometry and turbulence model are investigated in details. The Hölleforsen test case used for Turbine-99 (www.turbine-99.org) is used, since an extensive set of experimental data are available in the spiral casing to validate the simulations. The objective is to get boundary conditions for subsequent simulations including the runner and ultimately the entire system.

The current paper is structured as follows. Section 2 presents the test case, the geometry, grid, the boundary conditions, the turbulent models used and the experimental results available. In Section 3 the numerical results are presented with detail comparison and analysis of the deviations with the experiments. Section 4 presents the conclusions of this study.

2. TEST CASE

The full scale unit of the hydropower plant Hölleforsen is situated on the river Indalsälven in Sweden. It consists of three Kaplan turbines units with a total installed capacity of 50 MW at the operational head of 27 m, with a runner diameter of 5.5 m each and a discharge capacity of 230 m³/s per turbine.

The Hölleforsen hydropower plant Kaplan turbine model is known as the Turbine-99 test case. An extensive set of experimental data in the spiral casing and the draft tube are available. Numerical Simulations are therefore performed on the model. It is a 1:11 scale of the prototype turbine with a runner diameter of 500 mm and a runner speed of 595 rpm. The operational condition close to the best efficiency for the scheme at the operational head of 4.5 m, at 60% load is chosen. For this operational condition the volume flow rate is 0.522 m³/s.

2.1. Geometry and grid

The geometry of the model is presented in Fig. 1. Two geometries are available: one with and one without the water intake and the penstock. The stator has 24 guide vanes and 10 stay vanes. All the guide vanes are identical and have a symmetrical outline. In contrast, the stay vanes have unlike profile, see Fig. 2. Marks #1 and #2 show stay vanes with identical profile between each other and the rest of stay vanes have identical profile.

Figure 1: CAD model of the penstock, spiral casing and distributor of the Hölleforsen turbine (a, c, d) with the penstock (b) without the penstock (N.B. all the dimensions are in meters).
The computational grid consists of unstructured hexahedral elements created in ICEM CFD. The grid generation tool is based on a global block topology and generates 3D multi-block structured or unstructured hexahedral volume grids. The block topology model is generated directly on the fundamental CAD geometry. Two high density hexahedral grids with approximate first node wall distances of $y^+ = 50$ and $y^+ = 1$ were generated for each numerical simulations of the computational domain with and without the penstock, see Table 1. Grids A and B were used for the $k-\omega$ model, where grids C and D are used with the SST model. The grid from the inlet spiral casing to the outlet is identical on both cases. The grids have a good quality. Some high $y^+$ values are obtained on the most top part of the spiral casing. This is due to the presence of pinched elements in this region, which produce inadequate values. Pinch elements are hexahedral blocks with one collapse surface or two collapse edges. They do not have any influence on the results. Maximum and average $y^+$ values as well as minimum face angle and number of nodes for the different grids are presented in Table 1.

**Table 1. Characteristics of the grids used.**

<table>
<thead>
<tr>
<th>No. of Node</th>
<th>Max. $y$</th>
<th>Ave. $y$</th>
<th>Min. angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>7,800,768</td>
<td>611.3</td>
<td>25.7</td>
</tr>
<tr>
<td>B</td>
<td>7,499,284</td>
<td>339</td>
<td>24.9</td>
</tr>
<tr>
<td>C</td>
<td>13,801,522</td>
<td>336</td>
<td>1.63</td>
</tr>
<tr>
<td>D</td>
<td>13,356,377</td>
<td>467.7</td>
<td>2.27</td>
</tr>
</tbody>
</table>

Note: A: With the penstock: $y^+ = 50$

B: Without the penstock: $y^+ = 50$

C: With the penstock: $y^+ = 1$

D: Without the penstock: $y^+ = 1$

2.2. Boundary condition

The number and type of boundary conditions must accord with the governing equations of the flow. Different boundary conditions may cause quite different simulation results. Improper sets of boundary conditions may introduce non-realistic results or convergence problems. So orchestrating the boundary conditions for different problems is very essential. Mean time, different variables in the environment may have different boundary conditions according to certain physical problems. Therefore, it is important to set boundary conditions that accurately reflect the real situation to allow you to obtain accurate results.

Inlet boundary condition can be set in a number of ways depending on how we want to specify the conditions and what particular physical models we are using for the simulation. In the current research work the boundary mass flow rate is specified along with the flow direction. The mass flow rate which moves from the supply reservoir towards the intake of the penstock is equivalent to the initial boundary condition, which is specified currently as inlet flow rate 0.522 m$^3$/s. The wall is the most common boundary, encountered in confined fluid flow problems. The velocity of the fluid at the wall boundary is set to zero, no slip boundary condition. The outlet boundary condition can be used where it is known that flow is directed out of the domain. In our case the relative static pressure over the outlet boundary is specified. For all other transport equations the outlet value of the variables is part of the solution.

2.3. Turbulence modeling

Two different types of RANS models, standard $k-\omega$ with scalable wall function and SST $k-\omega$ Based turbulence model are applied to study the flow characteristics. Turbulence models are based on hypotheses about turbulent processes and require empirical input in the form of constants or functions. The most successful computational models for practical engineering purposes are those relating two or more transport equations, because they acquire two quantities to characterize the length and time scales of turbulent processes.

The standard $k-\omega$ model with scalable near wall function is the first model utilized for the different calculations. As a two equation model, it uses an eddy viscosity hypothesis for the Reynolds stresses, which assumes that the turbulent stresses are proportional to the mean velocity gradient and relate them linearly.
\[- \rho \frac{\partial u_i}{\partial t} = \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \rho \delta_{ij} \frac{\partial U_k}{\partial x_i} \right) \]

\[= 2 \mu \rho E_i - \frac{2}{3} \rho k \delta_{ij} \frac{\partial U_k}{\partial x_i} \tag{1} \]

\(\rho\) is the fluid density, \(u_i\) velocity fluctuations, \(U_i\) mean velocity components, \(E_i\) is the mean strain rate tensor, \(\mu\) is the eddy viscosity and \(k\) is the turbulent kinetic energy of the flow. The turbulent eddy viscosity is related to the kinetic energy of the turbulent flow by:

\[\nu_t = \rho C_{\mu} \frac{k^2}{\varepsilon} \tag{2}\]

where \(\varepsilon\) is the turbulent dissipation per unit mass and \(C_{\mu}\) is a constant. The transport equations for the turbulent quantities are:

\[\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho k U_j \right) = \frac{\partial}{\partial x_j} \left( \nu \frac{\partial k}{\partial x_j} \right) - \rho \frac{\partial E_i}{\partial x_i} \cdot \frac{\partial U_k}{\partial x_i} + 
\]

\[+ 2 \mu \rho E_i - \frac{2}{3} \rho k \delta_{ij} \frac{\partial U_k}{\partial x_i} \tag{3}\]

\[\frac{\partial \varepsilon}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho \varepsilon U_j \right) = \frac{\partial}{\partial x_j} \left( \nu \frac{\partial \varepsilon}{\partial x_j} \right) + C_{\mu} \rho \varepsilon \frac{\varepsilon}{k} \frac{\partial U_k}{\partial x_i} \frac{\partial U_k}{\partial x_i} \tag{4}\]

The model closure coefficients for the calculation in the present work are: \(C_{\mu} = 0.09, \sigma_k = 1.00, \sigma_{\varepsilon} = 1.30, C_{\mu} = 1.44\) and \(C_{\varepsilon} = 1.92\).

Shear stress transport model (SST) is the second turbulent model utilized. It is an eddy-viscosity model, which combines the \(k-\varepsilon\) model in the inner boundary layer and \(k-\varepsilon\) model in the outer region. It limits the shear stress in adverse pressure gradient regions. The transport equations for the turbulent quantities after some mathematical manipulations take the form:

\[\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho k U_j \right) = \frac{\partial}{\partial x_j} \left[ \nu \frac{\partial k}{\partial x_j} \right] - \rho \frac{\partial E_i}{\partial x_i} \cdot \frac{\partial U_k}{\partial x_i} + 
\]

\[+ 2 \nu \rho E_i - \beta \rho E_i \omega \tag{5}\]

\[\frac{\partial \varepsilon}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho \varepsilon U_j \right) = \frac{\partial}{\partial x_j} \left[ \nu \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\mu} \rho \varepsilon \frac{k^2}{\varepsilon} \frac{\partial U_k}{\partial x_i} \frac{\partial U_k}{\partial x_i} + 
\]

\[+ \alpha \rho \nu \frac{\partial E_i}{\partial x_i} \omega - \beta \varepsilon \omega^2 \tag{6}\]

where \(\omega\) is the dissipation per unit turbulence kinetic energy \((\omega = \varepsilon / k)\) and the eddy viscosity defined as:

\[\nu_t = \frac{a k}{\max(\gamma, \omega, SF)} \tag{7}\]

where \(F_1\) and \(F_2\) are a blending function and \(S\) is an invariant measure of the strain rate such as:

\[F_1 = \tanh\left( \max\left( \frac{\sqrt{k}}{0.5 \cdot 500}, \frac{4 \sqrt{k}}{0.5 \cdot 500}, \frac{4 \sqrt{k}}{0.5 \cdot 500} \right) \right) \tag{8}\]

\[F_2 = \tanh\left( \max\left( \frac{2 \sqrt{k}}{0.5 \cdot 500}, \frac{4 \sqrt{k}}{0.5 \cdot 500} \right) \right) \tag{9}\]

where \(y\) is the distance to the nearest wall and \(\nu\) is the kinematic viscosity and:

\[C_{D_{\mu_{\text{exp}}}} = \max\left( \frac{2 \rho}{\sigma_{\mu_{\text{exp}}}} V^2 \right) \tag{10}\]

The model constants used for the calculation are:

\[\beta = 0.09, \alpha_1 = 0.556, \alpha_2 = 0.44, \beta = 0.075, \sigma_{\mu_1} = 1.176, \sigma_{\mu_2} = 1, \sigma_{\varepsilon_1} = 2 \text{ and } \sigma_{\varepsilon_2} = 1.168\]

### 2.4. Experimental result

The experimental data used for evaluation of the computation in this work are provided by Håkan Nilsson, Chalmers University of Technology Sweden. The measurements are included in his PhD thesis [5], in which all details of the experimental set up, the LDA technique and how the measurements were performed can be found. However for the expediency of comparison we tried to include a summary of the basic points and results.

Existing experimental data were measured using the laser Doppler anemometer (LDA) technique along a measurement plane at the spiral casing section. The LDA technique uses the Doppler shift effect of reflected light from particles to determine the instantaneous velocity in a single point. The resulting Doppler frequency is proportional to the measured velocity.

The location of the measurements from a top view of the spiral casing and the vertical view of the direction of the measurement grid is shown in Fig. 3. Different regions in the plane were investigated, see Fig. 4. At measurement 1 the velocity normal to the measurement plane and the vertical velocity component (which are tangential and axial velocity components), at measurement 2 the tangential velocity components and the velocity component along the measurement plane (which is the radial velocity components) and at measurement 3 the tangential and axial velocity components were measured.
3. RESULTS

ANSYS CFX10.0, state of the art commercial software is used to perform the RANS simulations in this study. It is based on the finite volume method and has a coupled unstructured solver. To solve the numerical equations, the advection RANS simulations high resolution is used. High resolution advection scheme mean Blend factor values vary throughout the domain based on the local solution field in order to implement a boundedness criterion. For accuracy Blend factor will be close to 1.0 in flow regions with low variable gradients and 0 in areas where the gradients change sharply to prevent over and under estimation and maintain robustness. Setting Blend factor of 0 and 1.0 for the advection scheme is equivalent to using the first order advection scheme and second order differencing for the advection terms, correspondingly [11]. The simulation process started with steady state calculation of the flow. Convergence is achieved for both turbulent models with and without the penstock, by this we mean that the solution of the discretized equations tends to the exact solution as the grid spacing and control volume size is reduced to zero and all the RMS residuals of the momentum, mass and the turbulent equations are drop down below the targeted level.

In order to validate our methodology and to assess the accuracy of the numerical results comparisons with the available experimental data are performed, which is important to see that the numerical results are qualitatively and quantitatively correct before they are further used.

The first comparison is conducted at measurement 1, in the distributor and parts of the spiral casing, see Fig. 4. There, the tangential velocity component increases toward the runner vanes, see Fig. 5, and the axial velocity component increases at the bend before the leading edge of the stay vanes and decreases toward the trailing edge of the guide vanes, see Fig. 6. In both figures, the velocity components are compared with the experimental data and the results obtained from the CFD simulations achieved using both turbulent models with and without the penstock.

From Figure 5, the tangential velocity from the numerical simulations of both turbulent models without the penstock is similar. However, they deviate considerably from the experimental values. The simulations results with penstock of both turbulent models have a tiny variation between each other. They agree quite well with the experiment, the velocity is overestimated before passing the stay vane and underestimated after the passage. The importance of the penstock in the simulation is pointed out.
The axial velocity results from the numerical simulation have all a similar shape independently of the case, see Fig. 6. The SST turbulent model with penstock has a good ballpark figure with the experiment, while the three other cases have a similar shape with an underestimation of the maximum velocity region.

![Figure 6: Axial velocity contour plot at measurement 1 in m/s (a) experiments (b) and (c) are from the SST turbulent model with and without the penstock (d) and (e) are from the k-ε turbulent model with and without the penstock, respectively.](image)

The second comparison is carried out at measurement 2, at the entrance of the distributor, see Fig. 4. In this section the tangential velocity components is examined, see Fig. 7. There, the tangential velocity component increases towards the entrance of the stay vanes and decreases towards the outer vertical wall of the scroll casing.

The result obtained from the k-ε turbulent model with penstock has the best similarity with the experiment at the bottom part of the measuring section. However, the SST turbulent model with penstock shows a better agreement with the experiment weighing against to the other. The outputs from the numerical simulation of both turbulent models without penstock are similar and indicate inverse correlation, see Fig. 7.

![Figure 7: Tangential velocity contour plot at measurement 2 in m/s (a) experiments (b) and (c) are from the SST turbulent model with and without the penstock (d) and (e) are from the k-ε turbulent model with and without the penstock, respectively.](image)

The third comparison is performed for measurement 3, an extension of measurement 1 near the outer wall of the spiral casing, see Fig. 4. In this section, the tangential velocity component increases towards the centre of the runner and decreases around the outer most vertical wall of the spiral casing, see Figure 8. The axial velocity has lower values near the upper wall of the casing and has a highest value at the outer vertical wall towards the lower part of the scroll casing, see Fig. 9.

![Figure 8: Axial velocity contour plot at measurement 3 in m/s (a) experiments (b) and (c) are from the SST turbulent model with and without the penstock (d) and (e) are from the k-ε turbulent model with and without the penstock, respectively.](image)

The second comparison is carried out at measurement 2, at the entrance of the distributor, see Fig. 4. In this section the tangential velocity components is examined, see Fig. 7. There, the tangential velocity component increases towards the entrance of the stay vanes and decreases towards the outer vertical wall of the scroll casing.

The result obtained from the k-ε turbulent model with penstock has the best similarity with the experiment at the bottom part of the measuring section. However, the SST turbulent model with penstock shows a better agreement with the experiment weighing against to the other. The outputs from the numerical simulation of both turbulent models without penstock are similar and indicate inverse correlation, see Fig. 7.

The third comparison is performed for measurement 3, an extension of measurement 1 near the outer wall of the spiral casing, see Fig. 4. In this section, the tangential velocity component increases towards the centre of the runner and decreases around the outer most vertical wall of the spiral casing, see Figure 8. The axial velocity has lower values near the upper wall of the casing and has a highest value at the outer vertical wall towards the lower part of the scroll casing, see Fig. 9.

The second comparison is carried out at measurement 2, at the entrance of the distributor, see Fig. 4. In this section the tangential velocity components is examined, see Fig. 7. There, the tangential velocity component increases towards the entrance of the stay vanes and decreases towards the outer vertical wall of the scroll casing.

The result obtained from the k-ε turbulent model with penstock has the best similarity with the experiment at the bottom part of the measuring section. However, the SST turbulent model with penstock shows a better agreement with the experiment weighing against to the other. The outputs from the numerical simulation of both turbulent models without penstock are similar and indicate inverse correlation, see Fig. 7.

The third comparison is performed for measurement 3, an extension of measurement 1 near the outer wall of the spiral casing, see Fig. 4. In this section, the tangential velocity component increases towards the centre of the runner and decreases around the outer most vertical wall of the spiral casing, see Figure 8. The axial velocity has lower values near the upper wall of the casing and has a highest value at the outer vertical wall towards the lower part of the scroll casing, see Fig. 9.

![Figure 8: Axial velocity contour plot at measurement 3 in m/s (a) experiments (b) and (c) are from the SST turbulent model with and without the penstock (d) and (e) are from the k-ε turbulent model with and without the penstock, respectively.](image)
numerical simulations of the spiral casing are nearly insensitive to the turbulence model chosen, indicating a nearly in viscous flow. However, the SST model presents more accurate results. The correctness of the inlet boundary conditions has a more striking influence on the results. An analysis of the outlet boundary condition should also be conducted.

4. CONCLUSION

Numerical simulations of three-dimensional flow of the Höllforsen spiral casing and distributor with and without penstock have been performed with finite volume method and two turbulence models: k-ε and SST. Comparisons with available experimental results indicate clearly the importance of the penstock to perform accurate simulation. Therefore, detailed inlet boundary conditions are necessary to simulate accurately the spiral casing flows if the penstock is not included in the simulation.

The results found from the numerical simulation show that the turbulent models SST and the k-ε give similar result. However, the SST model performed slightly better.

ACKNOWLEDGEMENTS

The research presented was carried out as a part of “Swedish Hydropower Centre – SVC”. SVC has been established by the Swedish Energy Agency, Eiforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University. www.svc.nu

REFERENCE


Paper B

LDA MEASUREMENTS IN A Kaplan Spiral CASING MODEL
LDA Measurements in a Kaplan Spiral Casing Model

B.G. Mulu¹ and M.J. Cervantes²

Department of Applied Physics and Mechanical Engineering
Division of Fluid Mechanics
Luleå University of Technology
971 87 Luleå, Sweden

¹ phone: + (46) 920 49 10 78, Fax: + (46) 920 49 10 74, E-mail Berhanu.Mulu@ltu.se
² phone: + (46) 920 49 30 14, Fax: + (46) 920 49 10 47, E-mail Michel.Cervantes@ltu.se

ABSTRACT

This paper presents an experimental investigation of a Kaplan spiral casing turbine model of 1:3.1 scale of a geometrically similar prototype turbine. The prototype is available for similar investigation. The turbine is composed of 6 runner blades, 20 guide vanes and 18 stay vanes and has a runner diameter of 0.5 m. The study aims to investigate the inlet flow of the spiral casing as well as some sections in the spiral casing. The measurements are part of a project aiming to investigate experimentally the flow in different regions of the machine to build a data bank in order to validate numerical simulations and study scale-up efficiency between model and prototype.

A two components laser Doppler anemometry (LDA) apparatus was used to measure the velocity profiles at different locations in the turbine. To improve the signal quality and measurement accuracy refractive index matching optical box was mounted on the circular pipe of the spiral casing inlet. Ray tracing technique was used to determine the position of the laser beam intersection and fringes spacing by applying law of refraction and geometric considerations of the laser beams.

The investigation was carried out at three different loads, with constant runner blade angle: best operating point of the turbine and two off-design operating points with the presence of a vortex breakdown. The mean velocities profiles and corresponding RMS at spiral casing before the guide vanes and inlet of the spiral casing are presented for the different loads investigated. Good quality data are obtained for initial boundary conditions at the inlet of the casing to perform numerical simulations.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\tau_0)</td>
<td>Doppler frequency [Hz]</td>
</tr>
<tr>
<td>(L)</td>
<td>Distance from the inner side of the box to the pipe centre [m]</td>
</tr>
<tr>
<td>(n_a), (n_g), (n_w), (n_p)</td>
<td>Refractive index of air, glass, water and Plexiglas, respectively [-]</td>
</tr>
<tr>
<td>(iR)</td>
<td>Inner radius of the circular pipe [m]</td>
</tr>
<tr>
<td>(oR)</td>
<td>Outer radius of the circular pipe [m]</td>
</tr>
<tr>
<td>(r_{fr})</td>
<td>Position of beams intersection with refraction [m]</td>
</tr>
<tr>
<td>(r_{ar})</td>
<td>Position of beams intersection without refraction [m]</td>
</tr>
<tr>
<td>(r^*)</td>
<td>(r/R = ) Normalized radius with the inlet of pipe radius (R) [-]</td>
</tr>
<tr>
<td>(t_p)</td>
<td>(R_o - R_i = ) Thickness of the circular pipe [m]</td>
</tr>
<tr>
<td>(t_g)</td>
<td>Thickness of the glass [m]</td>
</tr>
</tbody>
</table>
t = L - R_o = Distance between pipe outer radius and box inner surface [m]

\( U \) = Axial velocity [m/s]
\( U_r \) = Radial velocity [m/s]
\( U_T \) = Tangential velocity [m/s]
\( U^* \) = \( U / V_T \) = Normalized mean axial velocity [-]
\( U_r^* \) = \( U_r / V_T \) = Normalized mean radial velocity [-]
\( U_T^* \) = \( U_T / V_T \) = Normalized mean tangential velocity [-]
\( u_\delta^* = u_\delta / V_T \), \( u_r^* = u_r / V_T \), \( u_T^* = u_T / V_T \) and \( v^* = v / V_T \) = Normalized RMS [-]

\( V \) = Transversal velocity [m/s]
\( V_{unc} \) = Uncorrected fluid velocity [m/s]
\( V_{cf} \) = Corrected fluid velocity [m/s]

\( V_T \) = \( Q / \pi R_o^2 \) = Bulk velocity, flow rate per area of the inlet pipe [m/s]

\( V^* \) = \( V / V_T \) = Normalized mean transversal velocity [-]

\( W \) = Vertical velocity [m/s]

\( W^* \) = \( W / V_T \) = Normalized mean vertical velocity [-]

\( \Delta \) = Refracted laser beam position during tangential velocity measurement [m]

\( \Delta x \) = Fringes spacing

\( \theta_i \) = Half angle between the laser beams [rad]

\( \phi \) = Half angle of refracted laser beams [rad]

\( \delta, \xi, \text{ and } \zeta \) = Angle between the refracted laser beam and the normal line [rad]

\( \lambda \) = Wavelength [m]

INTRODUCTION

Hydropower is a versatile renewable source of power generation able to change rapidly operating conditions. Hydropower plants may today work over a larger operating range than designed for due to the introduction of renewable sources of energy and the deregulation of the electricity market. Such operating conditions may involve unacceptable large stresses on the system and losses due to complex unsteady flow phenomena. The use of computational fluid dynamics (CFD) in the design and refurbishment process is becoming increasingly popular due to its flexibility, detailed flow description and cost-effectiveness comparing to model testing, usually used in the development of turbines. However, issues have still to be resolved due to the combined flow physics involved in hydropower machines such as the presence of partly separated flow at curved surfaces, vortices, unsteady phenomena, swirl flow, strong pressure gradients, convoluted geometry as well as numerical artefacts. Therefore, experimental data in such complicated systems are required to validate numerical simulations and develop more accurate models.

Several groups have investigated turbines models with advanced measuring techniques in order to validate numerical simulations. The research group at Ecole Polytechnique Federale de Lausanne (Switzerland) and Norwegian University of Science and Technology (Norway), both on a Francis type of turbine, and Laval University (Canada), on a propeller type of turbine, carried out a number of experimental investigations, see Ciocan et al. (2007), Vekve (2002) and Gagnon et al. (2008), respectively. Lulea University of Technology and Vattenfall Research and Development (Sweden) performed also extensive experimental studies on a Kaplan model. The available data bank has served as a benchmark to validate the ability of CFD to predict the flow features and engineering quantities of a draft tube model at three consecutive Turbine-99 workshops (1999, 2001 and 2005), see Andersson (2009). The research group is now focusing on the Porjus U9 model since the corresponding full scale machine is available for similar measurements.
Furthermore, the design of U9 is modern. It is composed of 6 runner blades, 20 guide vanes and 18 stay vanes and has a runner diameter of 1.55 m. The draft tube does not have any sharp heel as the Turbine-99 test case. The model is of 1:3.1 scale of the geometrically similar prototype turbine.

The objectives of U9 model project are to study the phenomenon of a complex unsteady three-dimensional flow caused by its rotor-stator interaction, build a data bank in order to validate numerical simulations and future scale-up studies. The investigation was initially carried out at three different loads: at best operating point and two off-design operating points (left and right side of the propeller curve) with the presence of a vortex breakdown. Three locations were selected for laser Doppler anemometry measurements: inlet of the spiral casing, in the spiral casing before guide vanes and draft tube cone. Measurements at the inlet spiral casing are essential to obtain the necessary boundary conditions to perform numerical simulations, since a bend is present upstream. Inadequate inlet boundary condition is expected to influence the results, see e.g. Mulu and Cervantes (2007). Application of the LDA technique did not present any specific problems since most of the windows are plane except in the spiral casing inlet which is circular. However it is essential to make sure that the optical axis intersects the plane wall at right angle to avoid any optical aberration; for instance the effect of astigmatism, which is associated with the off-axis alignment of LDA probe (Zhang and Eisele 1995, 1996).

Flow measurements in a circular pipe with LDA technique experience difficulties from the laser beam refraction at the pipe surfaces. This is due to the surface curvatures, both inside and outside, of the pipe and the differing refractive indices of the media. This leads to unwanted displacement, rotation, and misalignment of the laser beams resulting in loss of the Doppler signal. However, the signal quality can be considerably improved by matching the refractive index of the fluid to that of the pipe. Eventually with ray tracing technique a correction factor for the position and velocity magnitude can be made with some geometrical consideration and Snell’s law. Previous, researchers tried to overcome this problem with different methods. For instance, Boadway and Karahan (1981) and Bicen (1982), derived a correction factor to adjust the location of the measuring volume and the velocity by ray tracing. The scope of their work was limited to circular pipes in air and they assumed small angle approximation. Durst et al. (1988) used a container with flat walls around the pipe filled with a quiescent matching fluid. The working media had also the same refractive index. Gardavsky and Hrbek (1989) conducted their research work by placing a circular pipe in a rectangular optical box. They derived a series of equations with ray tracing method to determine the position of the laser beam intersection and the fringes spacing, without small angle approximation. Recently, Zhang (2004) has also tried to improve the optical performance by making the outside of the pipe plane without refractive index matching. He performed a comprehensive analysis of ray tracing with small-angle approximation and presented a detailed operating guideline with respect to the shift of the measurement volume, their optical properties and the beam waist dislocation.

The current research was performed by placing the circular pipe in a square optical box filled with a refractive index matching liquid: water. Ray tracing calculations were also performed to correct the position of the measuring volume and velocity using the above mentioned literature.
This paper focuses on the mean results of the spiral casing inlet and in the spiral casing before the guide vanes measurements. The draft tube cone mean measurements results were reported previously, see Mulu and Cervantes (2009).

**EXPERIMENTAL SET UP AND TECHNIQUES**

The full-scale unit of the hydropower plant investigated is situated on the Luleå River in the north of Sweden. The present measurements were performed on the homologous model with a runner diameter $D = 0.5$ m. The turbine model is mounted in the test rig between high-pressure and low-pressure tanks, see Figure 1. The water level is controlled by increasing the absolute pressure in the low-pressure tank to avoid cavitation. The measurements are carried out in a closed loop system. For further description of the power plant and the test rig, see Mulu and Cervantes (2009).

![Figure 1: Test rig with U9 Kaplan turbine model](image)

The investigation have been carried out at three different loads, with constant runner blade angle: at best operating point of the turbine (BEP) and two off-design operating points (left and right side of the propeller curve). The operational net head $H = 7.5$ m and a runner speed $N = 696.3$ rpm were used through out the measurements period. The guide vane angle was 20, 26 and 32° for the chosen operational points left, BEP and right, respectively. The corresponding volume flow rate of the three working conditions are 0.62, 0.71 and 0.76 m$^3$/s, see Mulu and Cervantes (2009).

**Measurement Technique:**

A two-component LDA with an 85 mm optical fibre probe from Dantec was used. The probe uses a backscatter configuration with an upper-lower beam arrangement to measure the velocity components. The basic configuration of the system consists of continuous wave of 20 W Argon-Ion laser, transmitting optics, photodetector and signal processor. To resolve the directional ambiguity a Bragg-cell with a frequency shifting capacity of 40 MHz is used to create the second shifted incident beam of each pair. Two different front lenses of 800 mm and 600 mm focal length were used with respect to the location of the measurement. For 600 mm focal length lens the resulting measuring volume size, based on the $e^2$ Gaussian intensity cut-off point, was estimated to be $2.229 \times$
0.140 mm (length and diameter) and 2.426 × 0.147 mm for both laser beams. Likewise for 800 mm focal length the estimated control volume size was 4.310 × 0.196 mm and 4.088 × 0.186 mm.

The signal analyzer was of the type BSA 57N21 and 35 made by Dantec’s. BSA Flow software with burst mode of spectrum analysis method was used for the data acquisition. The total sampling time was set to 300 s for each measurement point. This corresponds to 20,000-300,000 bursts at each measuring point and is function of the location of the measuring point. The seeding particles used in the investigation are made of Polyamide powder with an average diameter of 5 µm.

Locations of Measurements:

In order to obtain boundary conditions to perform numerical simulations and validate the numerical results before the runner, measurements were made at spiral casing inlet and in the spiral casing before the guide/stay vanes.

The inlet of the turbine model spiral casing is a circular pipe with inner radius of 316 mm. A Plexiglas pipe 290 mm long is mounted between the inlet of the spiral casing and the penstock for optical access, see Figure 2. LDA measurements in circular pipe are challenging due to the surface curvatures. The consequences of these curvatures are a lower signals quality as well as a different measurement volume due to the laser beam refraction at the surfaces. Experience shows that without refractive index matching and/or placing of an optical box outside the pipe the available velocity signal quality and thus the signal rate could be achieved only within a depth of about a third of the pipe diameter. However, if the pipe is placed in an optical box or the outside of the pipe is made plane, high quality velocity signals could be obtained even at a depth of about two-thirds of the pipe diameter. Performing coincident velocity measurements using four laser beams was inconvenient, as the four beams do not intersect at a single point in the flow due to the optical aberration. Thus, measurements of the velocities components were not carried out in coincidence mode. To obtain the full velocity profile across the pipe, measurements were performed from both sides of the pipe. Table 1 presents the location of the profiles measured; P_{y,3} and P_{z,1} are the profiles through the y and z-axis of the circular section, respectively.

In the current study a square optical box is placed around the circular pipe filled with index matching liquid to improve the optical performances. Glass windows are mounted on two sides of the box in order to have a homogenous texture, see Figure 2. To determine the appropriate liquid to fill the box, experiments on a channel flow with a diameter of 100 mm were made. Three different type of liquid were investigated, water (n = 1.33), paraffin oil (1.46) and 80% of sugar water solution (1.49), see Robin Wood. Water and paraffin gave almost the same signal quality but with paraffin the signal was slightly better. The sugar water solution gave the worst result because of the thin film created by the dissolve sugar at the surface of the pipe and box. For small scale at laboratory level paraffin may be preferable however in large scale like the current investigation water might be the best choice, since easier to manage.
Table 1: Location of profiles in the circular section.

<table>
<thead>
<tr>
<th>Profiles</th>
<th>Py_1</th>
<th>Py_2</th>
<th>Py_3</th>
<th>Py_4</th>
<th>Py_5</th>
<th>Pz_1</th>
</tr>
</thead>
<tbody>
<tr>
<td>y (mm)</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>z (mm)</td>
<td>100</td>
<td>50</td>
<td>0</td>
<td>-50</td>
<td>-100</td>
<td>-</td>
</tr>
</tbody>
</table>

Figure 2: Inlet section of the spiral casing: the square glass box is filled with refractive index matching liquid, water.

Figure 3: Location of the windows on the spiral casing and the measurement axis z at the section A-A.
In the spiral casing, two Plexiglas windows are installed on the lower side at the angular position -56.25° (S₁) and -236.25° (S₂) to perform the LDA measurements, see Figure 3. The position of the windows from the central axis of the guide vane to the bottom of the spiral casing is 294.7 and 197.1 mm, respectively. The windows are also placed at the centre of the casing 224.52 mm (S₁) and 74.01 mm (S₂) away from the stay vanes. Here the LDA measurements are straightforward since the windows are plane.

DATA EVALUATION

The total uncertainty in a measurement can be found by combining random (precision) and systematic (biases) errors. Most of the bias errors are small compared to the precision errors and are thus neglected. The bias errors which can be of the same order of magnitude as the precision errors are velocity bias and system noise. This two biases and precision errors are considered; see Mulu and Cervantes (2009).

In LDA system the fluid velocity is directly determined from the fringes spacing of the measurement volume and the Doppler frequency of the scattered light from the particles passing through the measurement volume. The fringes spacing can be calculated from the wavelength of the laser beams and their half angle, if the spacing is assumed to be uniform. A necessary condition for accurate LDA measurements is that the incident laser beams crossing position must coincide with their beam waists. If this condition is not satisfied fringes distortion in the measurement volume may occur and thus the maximum intensity of the laser beams will not be at the measuring volume, resulting in a poor signal-to-noise ratio and non-uniform fringes spacing. This situation might occur when the incident beams travels through different optical media of different refractive indices and the media interfaces are curved surfaces, for instance in pipe flow. Therefore, the difference in refractive indices and the curved surface nature affects the half angle and the intersection point of the beams. Owing to that, LDA measurement in circular pipe needs correction for the fluid velocity and the position of the beams intersection.

In the current work, when the measurements are performed along x, y and z axis the velocity components are refers as the axial, transversal and vertical components or tangential, radial and vertical components.

The formula necessary for determination of velocities and beams intersection position for the measurements performed at the inlet of the spiral casing are presented in appendix.

RESULTS AND DISCUSSION

For three different operating conditions, the LDA results of the mean velocities and RMS are presented to investigate the flow before the runner and at the inlet of the spiral casing, since the flow is influenced by the existence of a bend before the inlet. The measurements carried out at the inlet pipe are corrected for exact location of the laser beams intersection and the velocity according to formula presented in appendix. The location of the measurement volume error arising from the pipe surface curvature lies in the range of 0.02-0.7%. The difference between the incoming ray incident half angle $\theta_1$ and the half angle of refracted laser beams $\Phi$ is less than 0.7%. The bias in the velocity magnitude, due to the refraction of laser beams at curved interfaces, is considerably low, which is due
to the very large inlet pipe diameter comparing to the spacing and diameter of the laser beams.

The spiral casing velocities and RMS are normalized by the bulk velocity $V_T$ obtained from the flow rate and area of the inlet pipe. In spiral casing at section $S_1$ and $S_2$, the positive direction for the axial velocity and radial velocity are defined similarly to the stream flow direction and towards to the centre of the spiral casing, respectively.

The results at the different sections from all guide vanes opening show similar velocity profiles with a different magnitude. The velocity profiles at each section for all guide vane opening collapse to a single profile after normalization with the bulk velocity. This indicates that draft tube does not influence the high pressure flow despite strong unsteady flow phenomena at off design. Therefore, only the result from $\alpha = 26^\circ$ is presented.

![Figure 4: Normalized mean axial velocities and corresponding RMS in the inlet of the spiral casing at three measurement locations.](image)

Normalized mean axial velocities and corresponding RMS in the inlet of the spiral casing at three measurement locations are presented in Figure 4. Flow rate determined from an integration of the measured mean velocity profile is within $\pm 4\%$ compared to the flow rate determined with the test rig flowmeter ($\pm 0.25\%$). The maximum mean axial velocity is observed at the bottom region of the inlet pipe. The presence of the bend in the penstock is pointed out. The velocity is decreasing towards the upper part of the pipe. At central region between $r^* = -0.25$ to 0.25 the velocity decreases when moving upward while maximum velocity regions are spotted at $r^* = \pm (0.5$ to 0.85). This M-shaped character in the velocity distribution is due to the pair of counterrotating Dean vortices, which is know in a circular bend flow. Since, the axial velocity distribution is not uniform in the plane due to lower velocity close to the upper wall, fluid particles with higher velocity are forced to move to the outer side, and those with lower velocity to the centre. This is due to the curvature which causes a positive gradient of the centrifugal force from the centre to the outer wall. This force and the presence of a boundary layer at the wall due to the fluid adhesion to the wall combined are responsible for this kind of flow behaviour. The fluctuating quantity shows inverse trend compare with the mean velocity, when the mean velocity increases the RMS decreases.
Normalized mean axial and transversal velocities and corresponding RMS at measurement location $P_{z_1}$ at the inlet are presented in Figure 5. The average axial velocities measured at $P_{z_1}$ measurement location also confirm that the velocity is increasing towards the bottom region of the pipe. The square points shown on the top two plots of Figure 5 are the corresponding axial velocities and RMS measured at $P_{y_3}$, $P_{y_4}$ and $P_{y_5}$, respectively, indicating a good repeatability of the measurements. The transversal velocity profile indicates the presence of a secondary flow. The fluctuation quantity is larger than the transversal velocity.

Figure 5: Normalized mean axial and transversal velocities and corresponding RMS at measurement location $P_{z_1}$ at the inlet.

Normalized mean tangential and radial velocities and corresponding RMS at spiral casing measuring location $S_I$ and $S_{II}$ for guide vane opening $\alpha = 26^\circ$ are presented in Figure 6. At section $S_I$ the mean tangential velocity has a larger magnitude at the bottom region of the casing and increases toward the middle height of the guide vanes. At section $S_{II}$, the maximum velocity region is observed between the central level of the guide vanes and the upper level of the leading edge of the stay/guide vanes. Above the centre of the guide vanes the velocity at location $S_I$ and $S_{II}$ tend to have similar magnitude. The measured RMS at location $S_I$ is slightly higher than at location $S_{II}$ which indicates that the flow is more turbulent when it enters the spiral casing due to the bend upstream.

The radial velocity measured at location $S_I$ shows the presence of a re-circulation region below the leading edge of the stay/guide vanes. The secondary flow is not observed at location $S_{II}$, the radial velocity is nearly zero in this region. Starting from $z^* = -$
0.41 the radial velocity has a similar magnitude at both locations, indicating an axisymmetric flow entering the distributor. Determination of the flow rate per unit length from an integration of the measured mean radial velocity profile between the lower edge and the centre of the guide vanes for both sections, gives 0.0010 and 0.0011 m²/s at section S₁ and S₁₁, respectively. The RMS results are similar at both locations. The magnitude of the RMS is similar for both radial velocities.

![Figure 6: Normalized mean tangential and radial velocities and corresponding RMS at spiral casing measuring location S₁ and S₁₁ for guide vanes opening $\alpha = 26^\circ$. The results from $\alpha = 20$ and $32^\circ$ are similar. From left to right, the bold lines represent the bottom wall of the spiral casing at S₁ and S₁₁, the lower level of leading edge of the stay/guide vanes, the centre of the guide vanes and the upper level of leading edge of the stay vanes, respectively.](image)

**CONCLUSION**

The flow in a Kaplan spiral has been investigated with laser Doppler anemometry at 3 different working points: part load, BEP and high load. The inlet pipe was placed within a transparent box filled with a fluid of the same index of refraction as the fluid within the pipe to improve the optical distortions. The fringes spacing and location of the measurement volume are corrected by ray tracing technique. The diameter of the inlet pipe is very large comparing to the spacing and diameter of the beams, therefore the results shows that the correction for the velocity magnitude and location is very small.
The mean values of the axial, transversal and radial velocity components are similar independently of the working point when made dimensionless with the bulk velocity. The bend upstream the spiral influences strongly the flow. Maximum velocities are observed at the bottom of the inlet pipe and spiral. Nonetheless, an axis-symmetric flow is found to be delivered to the distributor.

To capture eventual effect of the runner blades in the upstream flow, the velocity field needs to be analyzed angularly in the spiral casing before the stay/guide vanes.

ACKNOWLEDGEMENT

The research presented was carried out as a part of “Swedish Hydropower Centre – SVC”. SVC has been established by the Swedish Energy Agency,Elforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University. (www.svc.nu)

The authors would like to thank the STAFF at Vattenfall Research and Development for all the help and support during the measurements.

BIBLIOGRAPHIC REFERENCES


Mulu, B. G. and Cervantes M. J., 2007, “Effects of Inlet Boundary Conditions on Spiral Casing Simulation”, Proceedings of the 2nd IAHR International Meeting of
the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Timisoara, Romania, pp. 218-224.


APPENDIX A

Axial Velocity Measurements at the Inlet of Spiral

Generally to obtain the best available optical condition for axial velocities measurements: the optical axis should pass through the circular pipe axis and intersect the plane at right angle to avoid any optical aberration. Thus, the refraction of the laser beams lies down in a plane parallel to the pipe axis, see Figure 7. In this case the half angle between the incidents beams remains unchanged. Therefore, the fluid velocity and the position of the beams intersection are directly calculated from Eq. 1 and 2, respectively.

\[
V_{LDA} = \frac{\lambda}{2\sin\theta} \cdot f_0 \tag{1}
\]

\[
r_t = k_1 r_x + (k_1 - k_2)t_y + (k_1 - k_3)t_z + (k_1 - 1)t \tag{2}
\]

Where,

\[
k_1 = n_w \sqrt{\frac{1 - \sin^2\beta}{1 - (n_w/n_p)^2 \sin^2\beta}}, \quad k_2 = \frac{n_w}{n_p} \sqrt{\frac{1 - \sin^2\beta}{1 - (n_w/n_p)^2 \sin^2\beta}} \quad \text{and} \quad k_3 = \frac{n_w}{n_p} \sqrt{\frac{1 - \sin^2\beta}{1 - (n_w/n_p)^2 \sin^2\beta}}
\]

Measurements were also performed at \(b/R_1 = 0.317\) and \(b/R_1 = 0.158\), where \(b\) is the deviation from the pipe axis. Adequate signal rates were obtained at these measurement positions. For moderate deviations, \(b/R_1 < 0.5\) and an identical fluid on either side of the circular pipe, no needs for correction of beams intersection position and fluid velocity are necessary; see Zhang (2004) and Bicen (1982).
Vertical and Transversal Velocity Measurements at the Inlet of Spiral

For measurements of the vertical and transversal velocities, the optical axis was aligned to pass through the pipe axis, i.e., the optical plane (the plane containing both beams) is orthogonal to the pipe axis, see Figure 8. Due to the laser beam refractions on the circular surface of the pipe the accuracy of these velocities measurement should be cautiously examined. The main reason is that the half angle and thus the measurement volume property depend on the local position of the laser beams intersection point. Indeed, this dependency leads to biased estimation of the flow velocity. Ray tracing calculations are necessary to correct the values. The equations are derived by applying law of refraction and geometric considerations of the laser beams. The procedure of the ray tracing equations and derivations are described by Gardavsky and Hrbek (1989). The exact position of the measurement volume after beam refraction can be determined by Eq. 3 or from Gardavsky and Hrbek (1989) by Eq. 14.

\[
\frac{\Delta}{\sin \delta} = \frac{R_w}{\sin \Phi}
\]  

From the refracted laser beams the fringes spacing of the control volume can be calculated by Eq. 4, and thus the transversal velocity can be corrected by Eq. 5, the upper sign hold when \( \varepsilon_i > \phi \) and the lower hold when \( \varepsilon_i < \phi \).

\[
\Delta x = \frac{1}{n_w} \frac{\lambda}{2 \sin(\Phi + \delta + 0 \pm \xi - \zeta)}
\]

\[
V_{ci} = kV_{LDA}
\]

Where, \( k = \frac{1}{n_w \sin(\Phi + \delta + 0 \pm \xi - \zeta)} \)
Figure 8: Ray tracing for measurements of the vertical and transversal velocities components, owing to the symmetry between two laser beams, only one beam is considered.

The value of \( d \) can be determined experimentally or analytically from ray tracing. To calculate the values analytically consider the bottom figure of Figure 8. From the slope of the first ray \( (m_1) \) and the equation of the circle at the intersection point \( (Y, Z) \), two equations can be derived as a function of \( d \):

\[
m_1 = \frac{Z-d}{Y-L}, \quad Z^2 + Y^2 = R_y^2
\]  \hspace{1cm} (6)

\[
Y = \frac{m_1s \pm \sqrt{(m_1s)^2 - (m_1^2 + 1)s^2 - R^2}}{m_1^2 + 1}
\]  \hspace{1cm} (7)
The angle $\varphi$ can be obtained using Snell’s law from the known incoming ray incident angle $\theta_i$:

$$\varphi = \sin^{-1}\left(\frac{n_s}{n_w} \sin \theta_i\right)$$

$$m_1 = \tan \varphi$$

$$s = m_1 L - d$$

Similarly, from the second slope $m_2$ and the equation of the circle at the intersection point $(y', z')$, two additional equations are obtained:

$$m_2 = \frac{z' - Z}{y' - Y}$$

$$y'^2 + z'^2 = R_i^2$$

$$\varepsilon_i = \sin^{-1}\left(\frac{Z}{R_i}\right)$$

The slope $m_2$ is obtained from $\varepsilon_i$ and $\zeta$:

$$m_2 = \tan(\varepsilon_i + \zeta)$$

Using Snell’s Law, $\zeta$ is obtained:

$$\zeta = \sin^{-1}\left(\frac{n_s}{n_w} \sin(\varphi - \varepsilon_i)\right)$$

The first $d$ value can be determined when $z' = 0$ and $y' = R_i$, corresponding to the reference point when the rays lie on the inner wall. Then, one can determine consecutive values of $d$ as a function of the first value as the probe is moved toward the pipe centre.

$$Y - \frac{Z}{m_2} - R_i = 0$$

A similar procedure is applied to correct the vertical velocity for $b/R_i < 0.5$, where the optical axis remains parallel with the pipe axis. If measurements are conducted beyond this limit the above approach for the vertical velocity correction is not be longer true, because the optical axis losses its parallellity with the pipe axis introducing some inclination angle in the control volume.
Paper C

EXPERIMENTAL INVESTIGATION OF A Kaplan model with LDA
Experimental investigation of a Kaplan model with LDA

Berhanu Mulu¹ and Dr. Michel Cervantes²

Luleå University of Technology, Department of Applied Physics and Mechanical Engineering, Division of Fluid Mechanics, SE-97187 Luleå, Sweden, Tel: +46 (0) 920-491078, Email: Berhanu.Mulu@ltu.se¹, Michel.Cervantes@ltu.se²

ABSTRACT

This paper describes the experimental investigation of a Kaplan turbine model of the Porjus U9 unit. The study aims to investigate the complex unsteady turbulent three-dimensional flow in different regions of the machine to build a data bank in order to validate numerical simulations and study scale-up efficiency between model and prototype. A two components LDA apparatus was used to measure the velocity profiles at different locations in the turbine in coincidence mode. An encoder pulse was used to resolve the measurements angularly. The investigation was carried out at three different loads: best operating point and two off-design operating points (left and right side of the propeller curve). Vortex breakdown was present at both off-design points at the inlet of the draft tube, in the cone. The axial and tangential mean velocity profiles and corresponding RMS at the draft tube cone are presented for the different loads investigated.

NOMENCLATURE

- \( V_r = \frac{Q}{\pi R^2} \) Bulk velocity, flow rate per area of the runner
- \( U^* = \frac{U}{V_r} \) Normalized mean axial velocity
- \( V^* = \frac{V}{V_r} \) Normalized mean tangential velocity
- \( u^' = u \frac{V_r}{V_r} \), \( v^* = v \frac{V_r}{V_r} \) Normalized RMS
- \( r^* = \frac{r}{R} \) Normalized radius with the runner radius \( R \)

INTRODUCTION

In the current project three locations were selected to investigate the flow in a Kaplan turbine with the help of laser Doppler anemometry (LDA) technique. In order to obtain boundary conditions to perform numerical simulations and study the flow before the runner, measurements were made at spiral casing inlet and in the spiral casing before the guide vanes. The third location is the draft tube. Previous studies have shown that the flow in a draft tube is complex due to swirling, partly separated flow at curved surfaces, flowing against an opposing pressure gradient and the
geometry; see e.g. Vekve (2002) and Andersson (2008). The measurements were performed in the draft tube cone at four different angular positions at the best efficiency point (BEP) and two off-design points with the presence of a vortex rope. This paper focuses on the mean results of the draft tube measurements.

TEST RIG AND MEASUREMENT METHOD

The full scale unit of the hydropower plant Porjus U9 is situated on the Luleå River in the north of Sweden. The machine is composed of 6 runner blades, 20 guide vanes and 18 stay vanes and has a runner diameter of 1.55 m. The operational head is 55 m with a discharge capacity of 20 m$^3$/s for a maximum power of 10 MW. The present measurements were performed on the homologous model in 1:3.1 scale of the prototype turbine with a runner diameter D = 0.5 m.

Test rig

The test rig is located at the Hydraulic Machinery Laboratory of Vattenfall Research and Development in Älvkarleby. It has been also well described by Marcinkiewicz and Svensson (1994). The archetypical uncertainty in the flow rate measurement and in the total hydraulic efficiency is ± 0.13% and ± 0.20%, respectively. The water level was controlled by increasing the absolute pressure in the low-pressure tank to avoid cavitation. The measurements were carried out in a closed loop system.

Operation conditions

The measurements have been carried out at three different loads: at best operating point of the turbine and two off-design operating points (left and right side of the propeller curve). The operational net head H = 7.5 m, a runner blade angle $\beta = 0.8^o$ and a runner speed N = 696.3 rpm (unit runner speed, $DN/\sqrt{H} = 127.1$) were used throughout the entire period of measurements. The working guide vane angle and the volume flow rate of the three working conditions are summarized in Table 1.

Table 1. Operational condition parameters.

<table>
<thead>
<tr>
<th>Operating point</th>
<th>Left</th>
<th>BEP</th>
<th>Right</th>
</tr>
</thead>
<tbody>
<tr>
<td>Guide vane angle, $\alpha$ (degree)</td>
<td>20</td>
<td>26</td>
<td>32</td>
</tr>
<tr>
<td>Volume flow rate, $Q$ (m$^3$/s)</td>
<td>0.62</td>
<td>0.71</td>
<td>0.76</td>
</tr>
<tr>
<td>Unit flow rate, $Q/D^2 \sqrt{H}$ [-]</td>
<td>0.89</td>
<td>1.03</td>
<td>1.11</td>
</tr>
<tr>
<td>Efficiency, $\eta - \eta_{BEP}$ (%)</td>
<td>-5.2</td>
<td>0.0</td>
<td>-1.0</td>
</tr>
</tbody>
</table>

Measurement method

The measurements presented were made with LDA technique. The system used is a two component set-up from Dantec with an 85 mm optical fibre probe and a front lens of 600 mm focal length. The basic configuration of the system consists of continuous wave of 20 W Argon-ion laser, transmitting optics including a beam.
splitter Bragg-cell, photodetector and signal processor. To resolve the directional ambiguity of the incoming beams were shifted by 40MHz. The resulting measuring volume size, based on the $e^{-2}$ Gaussian intensity cut-off point, was estimated to be $\approx 2.229 \times 0.140$ mm (length and diameter) for the axial velocity component and $2.426 \times 0.147$ mm for the tangential velocity component. Polyamide particles with a mean diameter of 5 μm were used for seeding. The technique uses a backscatter configuration with an upper-lower beam arrangement to measure the velocity components corresponding to the particles pass through the control volume.

The signal analyzer was of the type BSA 57N21 and 35 made by Dantec’s. BSA Flow software with burst mode of spectrum analysis method was used for the data acquisition. The probe was fixed on a three axis traverse system, which was controlled by the software with a possible smallest step of 0.01 mm. The total sampling time was set to be 300 s for each measurement point. This corresponds to 20,000 - 300,000 bursts at each measuring points, function of the location of the measuring point.

The measurements were performed at 4 different windows with angular positions (AP) of 0°, 90°, 180° and 270°, see Figure 1. The radii of the upper and lower circles of the cone are 252.1 mm and 322.58 mm, respectively. The cone angle is 6.1°. Three velocity profiles at section (S) I, II and III were measured for each angular positions and working points. Radial profile I is located below the runner cone in the upper part of the draft tube cone, 334.3 mm below the runner hub centre, for all angular positions. Profile II and III were located in the middle and close to the end of the draft tube cone, respectively. Table 2 presented the exact location of the three profiles for all windows, with a reference datum of runner hub centre.

**Table 2. Location of profile I, II and III from the runner hub centre.**

<table>
<thead>
<tr>
<th>Profiles</th>
<th>Location (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>a (0°)</td>
</tr>
<tr>
<td>I</td>
<td>334.3</td>
</tr>
<tr>
<td>II</td>
<td>492.6</td>
</tr>
<tr>
<td>III</td>
<td>716.2</td>
</tr>
</tbody>
</table>

Figure 1. Windows location at the draft tube cone: a is at 0°, b at 90°, c at 180° and d at 270°. I, II and III are the sections where velocity profiles were measured in each window perpendicular to the draft tube cone wall.
DATA EVALUATION

The total uncertainty in a measurement can be found by combining random (precision) and systematic (biases) errors. Biased errors include errors from laser beam geometry, signal processor bias, seeding bias, calibration uncertainty, probe alignment bias, angular bias, fringe bias, gradient broadening bias, velocity bias and system noise; see Albrecht et al. (2003). Most of the bias errors are very small compared to the precision errors and are thus neglected. The bias errors which can be of the same order of magnitude as the precision errors are velocity bias and system noise. This two biases and precision errors are considered in this paper.

In a uniformly seeded fluid, velocity bias error may occur in LDA measurements when particles with a large velocity travel through the measuring volume more frequently than particles moving with a small velocity. Taking an arithmetic average of these random measurements over a given period of time will therefore result in an overestimated mean velocity; this error was first acknowledged by McLaughlin and Tiederman (1973). The estimated velocity variance may also be biased. To determine statistical moments for each measurement point, velocity and residence time of each particle that pass through the control volume were recorded. To avoid velocity bias, weighting method based on the residence time \( g \) of signal bursts, which is inversely proportional to the modulus of the velocity vector was used. Thus, the time averaged mean velocity component and the RMS are obtained from Eq. 1 and 2, respectively.

\[
\langle U \rangle_i = \frac{\sum_{j=1}^{n} g_j u_{i,j}}{\sum_{j=1}^{n} g_j}
\]

\[
\langle u^2 \rangle_i = \frac{\sum_{j=1}^{n} g_j (u_{i,j} - \langle U \rangle_i)^2}{\sum_{j=1}^{n} g_j}
\]

\[
u_{i,\text{rms}} = \sqrt{\frac{\langle u^2 \rangle_i}{n}}
\]

Where, \( u_{i,j} \) is the j:th particle velocity determined from the corresponding Doppler frequency and \( g_j \) the corresponding residence time. \( n \) is the total number of samples contributing to the value.

Special care was taken in terms of instrumentation and measurement procedures in order to keep noise as low as possible. However, vibration of the test rig induces a small amount of movement of the wall at the measuring sections orthogonal to the probe. The system noise was estimated by a velocity measurement on the surfaces of the test section. The measured noise contribution to the velocities was subtracted from the velocities data.

The precision errors in LDA measurements are data processing errors which result from averaging a finite number of data samples at each measurement point. Currently, this error is estimated by a repeatability test. Owing to turbulence, the velocity being measured does not remain constant during the sampling period. Thus the estimated precision error \( \langle P \rangle \) of the mean values, at the probability of 95%
confidence interval, is calculated by \( P = t \cdot s \), where \( t \) is the coefficient of student t-distribution with the corresponding degrees of freedom and \( s \) is standard deviation of the sample data (Coleman and Steele, 1999).

A great care has been paid to eliminate the error arising from the operational mode of the test rig during measurements and at different operating days. However, the overall estimated error of the operational parameters lies between 0.1 - 0.35% throughout the entire measurement period.

**RESULTS**

The LDA result of the mean velocities and RMS are presented to investigate the flow symmetry below the runner and the flow development in the draft tube cone for the different operating conditions. The velocities and RMS are normalized by the bulk velocity \( V_T \) obtained from the flow rate and area of the runner. The positive direction for the axial velocity and tangential velocity are defined vertically downward through the draft tube cone and in the clockwise direction viewed from the top of runner, respectively.

Figure 2. Normalized mean axial and tangential velocities and corresponding RMS at section I for \( \alpha = 26^\circ \) at 4 angular positions (AP). The bold dash dot-lines represent the lower and upper edges of the runner hub cone and the position of the draft tube cone wall, from left to right.
For all angular positions the mean axial velocity profile at section I has a minimum and maximum below the runner cone. The minimum is in the center while the maximum axial velocity is observed around \( r^* = 0.15 \), see the left plot in Figure 2(a).

The tangential velocity is decreasing linearly from the draft tube cone wall up to \( r^* = 0.4 \), like a solid body, indicating a well functioning runner. From \( r^* = 0.4 \) to nearly the center of the cone the velocity is increasing steadily such as \( V^* = k/r^* \) where \( k \) is constant, indicating a free vortex. Then, the velocity starts to decrease again towards the centre like a forced vortex, see the left plot in Figure 2 (b), like a Rankine vortex.

The RMS values have similar behavior with differences in magnitude. They decrease gradually from the wall up to \( r^* = 0.4 \) and increase towards the center due to the runner cone rotation, see the right plot in Figure 2 (a, b).

Theoretically, the values should be identical at \( r^* = 0 \) for each variable. However, possible misalignment in the LDA probe or the windows may create variations in the control volume position and therefore in the values.

The results show a symmetric flow. The flow symmetry also hold true for off-design points, which velocity profiles are presented in Figure 3.
Figure 3. Normalized mean axial and tangential velocities and RMS at S III

The shape and development of the mean velocities for BEP and overload (highest guide vane angle) at S I, II, and III are similar with a different magnitude, see Figure 3 (a, b, c). The magnitudes of the axial velocity are almost equal close to the draft tube cone wall. However, a significant amount of variation appears towards the centre of the runner. For α = 20°, the axial velocity in the region r* = 0 – 0.6 has a low magnitude. This low magnitude region propagates towards the wall downstream as seen in S II and III. The high velocity region has a nearly constant velocity independently of the section, indicating a low pressure recovery, i.e., the draft tube is not working properly.

For all operating points and sections the mean tangential velocity increases close to the wall, while the mean axial velocity drop off relatively. For α = 20°, the mean tangential velocity decreases towards the centre and behaves like a Rankine vortex. It gets weaker from S I to III, the vortex rope spreads out from the centre. For α = 26°, the solid body rotation from the wall near the middle of the draft tube cone is not altered by the area variation. The Rankine vortex in the region r* = 0 – 0.5 decreases in amplitude. For α = 32°, a contra rotating flow is observed at S I for r* = 0.08 – 0.62, at S II for r* = 0.2 – 0.73 and at S III for r* = 0.3 – 0.8, see Figure 3 (a, b, c).

The RMS for α = 26° and 32° at all sections have similar pattern from the centre to the wall. For α = 20°, the effect of the vortex rope for r* = 0.06 – 0.8 is very high below the runner hub cone and decreases downstream to the draft tube elbow with increment in value towards the wall, see Figure 3 (a, b, c). For the partial load operation the tangential velocity contribution to the RMS is higher than the axial velocity contribution, this is due to the pulsation of the rope, see Figure 3. The RMS values are high at S I close to the runner centre and decrease towards S III which is due to the additional fluctuations of the vortex rope. This effect also propagates in the radial direction to the wall from S I to III where the velocity increases, see Figure 3 (a, b, c). The radius of the vortex rope increases downstream.
CONCLUSION

A Kaplan model has been investigated with laser Doppler technology at 3 different working points. The mean values of the axial and tangential velocity are presented as well as the corresponding RMS values.

The results are similar at BEP and overload, when $\alpha = 26$ and $32^\circ$. The tangential velocity can be decomposed on a forced vortex and a Rankine vortex. The RMS values are important near the runner cone due certainly to its rotation. For the partial load operation the effect of the pulsating frequency of the vortex is significant on the RMS and mean values. A low velocity region is identified below the runner cone.

To investigate the vortex breakdown in detail regarding the forced and free vortex regions, its periodic phenomena and the effect of the blade wakes, a detailed phase resolved analysis will be made.

ACKNOWLEDGEMENT

The research presented was carried out as a part of “Swedish Hydropower Centre – SVC”. SVC has been established by the Swedish Energy Agency, Elforsk and Svenska Kraftnät together with Luleå University of Technology, The Royal Institute of Technology, Chalmers University of Technology and Uppsala University. www.svc.nu

The authors would like to thank the STAFF at Vattenfall Research and Development for all the help and support during the measurements.

REFERENCES


