

3D TIME-RESOLVED PIV MEASUREMENT IN A FRANCIS TURBINE DRAFT TUBE

AUTHORS

Sylvain Tridon (PhD student)
LEGI INPG CNRS - 1025 rue de la piscine - 38041 Grenoble, France
(+33)4 76 82 50 78 – sylvain.tridon@legi.inpg.fr

Gabriel Dan Ciocan (Project Manager)
ALSTOM HYDRO France – 82 av. Léon Blum – 38000 Grenoble, France
(+33)4 76 39 35 35 – gabriel.ciocan@power.alstom.com

Stéphane Barre (Researcher)
LEGI INPG CNRS - 1025 rue de la piscine - 38041 Grenoble, France
(+33)4 76 82 70 27 – stephane.barre@legi.inpg.fr

Laurent Tomas (Hydraulic Design Expert)
ALSTOM HYDRO France – 82 av. Léon Blum – 38000 Grenoble, France
(+33)4 76 39 33 75 – laurent.tomas@power.alstom.com

ABSTRACT

The draft tube of a hydraulic turbine is the component where the flow exiting the runner is decelerated, thereby converting the excess of kinetic energy into static pressure. In the case of machine refurbishment of an existing power plant, most of the time only the runner and the guide vanes are currently modified. For financial and safety reasons, the spiral casing and the draft tube are seldom redesigned, even if these components present some undesirable behaviour. In some cases, the installation of an upgraded runner leads to a peculiar and undesirable efficiency drop as the discharge is increased above the best efficiency point value. It is found to be related to a corresponding sudden variation in the draft tube pressure recovery coefficient at the same discharge.

Three different scenarios could explain the efficiency drop. The first one proposes a global instability of the flow downstream the runner. The second one supposes that the vortex system issued from the runner propagates until the elbow. The third and last one involves an interaction of the principal vortex with the secondary flows due to elbow and diffuser. A Francis turbine model in which the efficiency drop occurred has been installed on the CREMHyG Laboratory test rig and instrumented for an experimental campaign. The measurements focus on the swirl produced by the constant pitch turbine runner and further ingested by the draft tube.

Development of recent experimental technique allows to measure the detailed vertical flow field at the runner outlet of a turbine model. The experiments include LDV and PIV measurements in the draft tube cone downstream the runner and discharge measurements in each channel of the draft tube. The PIV measurements realized with two fast cameras give us the full unsteady flow velocity field in a plane. The use of a time-resolved PIV system is necessary because of the unsteadiness of the vortex breakdown phenomena which has to be described by the present measurements. This is a first application of the time resolved PIV in hydraulic machinery. This extensive experimental investigation of the draft tube flow aimed at elucidating the swirling flow evolution up to the turbine outlet as well as the phenomena that led to the peculiar sudden drop in the turbine efficiency.

KEY WORD: draft tube, efficiency drop, pressure recovery coefficient, PIV, LDV
1 INTRODUCTION

The FLINDT project (Flow Investigation in a Draft Tube) initiated in 1997 is the source of several interesting articles. Experimental measurement and numerical simulations were made at the EPFL (Ecole Polytechnique Federal de Lausanne) with a draft tube that presents an efficiency drop in order to understand it. The aim was also to establish a bank of experimental measurements for project participants. The velocity profiles and six independent components of the Reynolds tensor were measured at different locations in the geometry for three different operating points. In an engineering perspective, the numerical calculations made by Mauri [1] on a structured mesh with 300 000 nodes well correspond to the experimental data. Avellan [2] attributes the differences to the k-ε model that is known not to be adapted to this kind of flow. In terms of inlet conditions, he noted that the closest results to the experimental data are obtained using a turbulent length of 0.2 percent of the inlet diameter. The velocity profiles imposed on boundary conditions are derived from experimental data.

Mauri [1] carried out during his thesis a numerical study of the FLINDT draft tube and attributed the efficiency drop to a global instability underscored by Susan-Resiga – see next paragraph - leading to a Werlé-Legendre vortex [3]. For us, the interaction between the main flow and the secondary flow caused by the elbow leads to an outbreak and a seat in rows of wall friction. The focus then becomes the origin point of a tube vorticity stretching in one canal. The symmetry of the flow is then broken and a blocking effect occurs, forcing the acceleration of the flow in the second canal.

Susan-Resiga and al [4] have developed a theoretical model based on an analysis of the velocity profile at the draft tube inlet. The authors show that an analytical representation may be set on experimental data. It is achieved by superposing two counter-rotational Batchelor vortices and a solid body vortex in the center. A non-viscous stability analysis, applied to these profiles, assuming a hypothesis of fully developed flow, show that there is a critical discharge factor leading to a flow instability well-correlated with the efficiency drop.

Another important source of information on the calculation of draft tube are all the results obtained during the three workshops Turbine-99 [5] that occurred in 1999, 2001 and 2005. The objective of this research conducted in Sweden is to assess and advance the state of the art in turbine flow simulation. The draft tube studied in this context does not present efficiency drop near the best operating point but modelling it still a major challenge encountered in practice. We note among the results showed that it was essential to specify the speed profiles input, including the radial velocity, which is unfortunately not available experimentally.

Along with the workshops Turbine-99, Cervantes [6] presents a numerical study on the impact of the boundary conditions on the calculation. The results found in his doctoral thesis presented in 2003 are obtained with the k-ε turbulence model. The author used a factorial design approach to determine that the factor having the greatest influence on the recovery factor is the radial velocity input. He also concluded that the length characteristic of turbulence, assumed constant throughout the radius of entry, and surface roughness do not have major effects. But, Cervantes acknowledges that the range of y+ found in its simulations is very large, probably because of the low amount of element in his mesh. Another interesting contribution is the way in which the author considers the radial velocity under the runner of the turbine [7]. It uses an iterative process based on the Squire Long equation to estimate the radial velocity. This approach requires the use of experimental data of axial and tangential velocity at inlet and outlet of the calculation domain. It is also difficult to judge the outcome because Cervantes did not succeed to converge some simulations where the deduced velocity profile is imposed as an upstream boundary condition. This work is very interesting but it is not excluded that it may be affected by a mesh dependence results problem.
2 METHODOLOGY

2.1 Francis Turbine Model Presentation

The investigated case corresponds to the scale model of a Francis turbine of high specific speed, \( v = 0.55 \) \((n_q = 86)\). The scale model supplied by ALSTOM is installed on the test rig of the CREMHyG and the tests are carried out according to the IEC 60193 International Standards [8]. The turbine model has a spiral casing of double curvature type with a stay ring of 10 stay vanes, a distributor made of 20 guide vanes, a 19 blades runner of a 365mm outlet diameter, and a symmetric elbow draft tube with one pier.

The global “efficiency” of the draft tube is quantified using his static pressure recovery coefficient, defined as:

\[
\chi = \frac{(P/\rho + gz)_{out} - (P/\rho + gz)_{ref}}{Q^2/2A^2_{ref}}
\]

Figure 1: Efficiency break off obtained by increasing and decreasing the discharge.

By increasing the discharge (see Figure 1) the efficiency presents an important drop of more than one percent very close to the best operating point. This break-off presents a hysteresis effect; by decreasing the discharge the drop is less important. It is found to be produced by a corresponding drop in the draft tube pressure recovery. The perfect similarity between the two profiles on Figure 1 confirms the idea that the accident occurred in the draft tube. The six operating points where full velocity measurements are performed on the survey section are selected around the efficiency drop. The first four are performed under the nominal head and the last two under 75% of the nominal head. They are detailed in the table 1 and the first four are marked by a square on Figure 1.
Table 1: Six operating points near efficiency drop selected for full velocity measurements

<table>
<thead>
<tr>
<th>Pf</th>
<th>Accident</th>
<th>Discharge coef. $\varphi$</th>
<th>Energy coef. $\psi$</th>
<th>Speed [rpm]</th>
<th>$H/H_{nom}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>No</td>
<td>0.368</td>
<td>1.18</td>
<td>955.7</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>No</td>
<td>0.374</td>
<td>1.18</td>
<td>955.7</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>Yes</td>
<td>0.380</td>
<td>1.18</td>
<td>955.7</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>Yes</td>
<td>0.390</td>
<td>1.18</td>
<td>955.7</td>
<td>1</td>
</tr>
<tr>
<td>5</td>
<td>No</td>
<td>0.374</td>
<td>1.18</td>
<td>827.6</td>
<td>0.75</td>
</tr>
<tr>
<td>6</td>
<td>Yes</td>
<td>0.390</td>
<td>1.18</td>
<td>827.6</td>
<td>0.75</td>
</tr>
</tbody>
</table>

2.2 Laser Doppler Velocimetry Instrumentation

Figure 2: Sketch of the francis turbine model for the flow survey section in the cone

The 2D velocity profile survey is performed by the LDV measurement method on three diameters at 120° in the cone - see Figure 3 and Figure 4 - and near the draft tube outlet to estimate the discharge repartition in each side of the draft tube. The experimental data used in this paper were obtained with a two-component probe Laser Doppler Anemometer (LDA), using back-scattered light and transmission by optical fibber, with a 5 W argon-ion laser source. The main characteristics of the optical system for the measurement under the runner and in the draft tube are detailed in table 2.

<table>
<thead>
<tr>
<th>Optical Characteristics</th>
<th>Under the Runner</th>
<th>Draft Tube</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wave lengths [µm]</td>
<td>488/514.5</td>
<td>488/514.5</td>
</tr>
<tr>
<td>Focal length [mm]</td>
<td>400</td>
<td>600</td>
</tr>
<tr>
<td>Probe diameter [mm]</td>
<td>60</td>
<td>120</td>
</tr>
<tr>
<td>Beam spacing probe [mm]</td>
<td>38.5</td>
<td>75</td>
</tr>
<tr>
<td>Fringe spacing [µm]</td>
<td>5.41</td>
<td>4.07</td>
</tr>
<tr>
<td>Beam half-angle [°]</td>
<td>2.725</td>
<td>3.624</td>
</tr>
<tr>
<td>Measuring volume dx,dy [mm]</td>
<td>0.218</td>
<td>0.090</td>
</tr>
<tr>
<td>Measuring volume dz [mm]</td>
<td>4.6</td>
<td>1.4</td>
</tr>
</tbody>
</table>

Table 2: main characteristics of the LDV optical system
Spherical silver-coated glass particles are introduced in the test rig flow. These particles are hollowed in order to match the water density and are able to follow flow fluctuations frequency up to 5 kHz [9]. The mean diameter of these particles is 10 μm. Three cylindrical glass optical windows with plane and parallel faces are placed at 60° with their axes 115 mm under the runner which correspond to 0.63 runner outlet radius $R_{ref}$ (Figure 2). They are used as interfaces and exempt rotate the cone between each azimuthal direction. The geometrical reference position of the measurements is obtained by positioning the laser beams on the windows faces with accuracy better than 0.05 mm. Both axial and circumferential components of the velocity are measured. The uncertainties of the velocity measurements are estimated to be 2% of the measured value; see Ciocan and al. [10].

2.3 Particle Image Velocimetry Instrumentation

The 3D instantaneous velocity field in the cone is investigated with a Dantec M.T. 3D PIV system, which consists of a double-pulsed laser, two double-frame cameras, and a processor unit for the acquisition synchronization and the vectors detection by cross correlation. The illuminating system is composed of two laser units with Neodymium-doped Yttrium Aluminium Garnet crystals (Nd-YAG), each delivering a short impulse of 10 ns and 15 mJ energy at 1 kHz frequency. Thus the time interval between two successive impulses is fixed to 100μs according to have approximately a 10 pixels displacement in the axial direction. The output laser beam of 532 nm is transformed into a sheet of 4 mm width and 25° divergence.

Two Nanosense cameras with a resolution of 1280×1024 pixels are used for 200×200mm² investigation area. The cameras are placed in a stereoscopic configuration, focused on the laser-sheet, synchronized with the two pulses. They capture the position of seeding particles of ~10μm diameter by detecting their scattered light.

For the optical access, the cone is manufactured in Polymethyl methacrylate (PPMA) with a refractive index of 1.4, equipped with a narrow window for the laser’s access and two large symmetric windows for the cameras access, having a flat external surface for
minimizing the optical distortions. The corresponding two-dimensional vector maps, obtained from each camera by a fast Fourier transform-based algorithm, are combined in order to obtain the out-of-plane component, characterizing the displacement in the laser-sheet width.

The correlation between the local image coordinates and real space coordinates is realized through a third order optical transfer matrix, which includes the correction of distortions due to different refractive indices in the optical path and to the oblique position of the cameras. The calibration relation is obtained acquiring images of a calibration target with equally spaced markers, moved in five transversal positions in order to have volume information. The target displacement in the measurement area, with accuracy within the narrow limits of 0.01 mm in translation and 0.1 deg in rotation, insured a good calibration quality ; Iliescu and al. [11,12]. The overall uncertainty of the PIV 3D velocity fields is 3% of the mean velocity value (see Figure 5 who shows the calibration set-up).

![Image: Calibration setup for 3D PIV measurements](image)

3 RESULTS

Throughout this paper the velocity is made dimensionless by the runner outlet velocity \((\text{runner angular speed} \times \text{runner outlet radius})\), and lengths are made dimensionless with respect to the runner outlet radius \(R_{ref}\) (Fig. 1).

3.1 Velocity Profile Under The Runner

The LDV velocity measurements provide the axial and tangential velocity component in the cone under the runner on three diameters at 120° or six radii at 60°. By observing these velocity profiles it is obvious that the flow under the runner is not axisymmetric because of the small extent of the draft tube cone and thus the elbow influence. Therefore, we calculate the velocity field on a complete section by a cubic spline interpolation between the six azimuthal values corresponding to the six radii. PIV measurements lead us to obtain after post-processing the whole 3D instantaneous velocity fields at 1 KHz or 10 Hz depending on the experiment performed.

Figure 6 shows a 3D visualisation of the axial and tangential velocity obtained by the interpolated LDV measurements under the runner for the first four operating points (N°1 to 4 as defined in Table 1). The axial velocity is represented on the vertical axis and the tangential component is described by the colorscale.

Figure 7 shows the juxtaposition of three tangential velocity mean field averaged from 3000 instantaneous 3D fields acquired by PIV for operating point N°4 and azimuthal direction N°3 (V3). The vortex core axis tends to the left side which corresponds to the elbow side.
Figure 6: Axial and Tangential Velocity under the runner - PF1 to PF4

Figure 7: Tangential velocity spatial repartition for point N°4

Figure 8 shows PIV profiles obtained for eight vertical stations ranging for z=0.35 \( R_{\text{ref}} \) to z=0.73 \( R_{\text{ref}} \) for operating point N°4. LDV and PIV results are compared at station z=0.66 \( R_{\text{ref}} \) for the longitudinal and tangential velocity component. A quite good agreement is found.

The overall aspect of the mean velocity profiles does not depend much on the operating point chosen. Each operating point exhibits a two counter rotating vortex structure. One vortex is centered very close the cone axis and extends roughly until 0.5 \( R_{\text{ref}} \) (see tangential velocity on Figure 8). The tangential velocity repartition for this vortex shows that it is counter rotating with respect to the runner and that it seems to be close to a Burger’s vortex type. The other vortex concerns the external part of the flow where the velocity evolves quasi linearly with the radius above 0.5 \( R_{\text{ref}} \). This one is co-rotating with respect to the runner and corresponds to the swirling rate introduced by the runner tangential velocity field. It has to be noted that for most part of the flow (±0.2 \( R_{\text{ref}} < r < ±0.6 \( R_{\text{ref}} \)), the tangential velocity is almost negligible giving a very low swirling rate for this part of the flow. This is not surprising because all of the four operating point studied here are very close to the optimum operating point of the runner.
Figure 9 shows the evolution of the three components of the velocity field obtained by PIV for operating points N°1 to 4 as described in table N°1. Results are displayed for \( z=0.66R_{\text{ref}} \). The main effect of the "accident" is to increase the vorticity of the main eddy at the center of the flow duct. This vorticity is drastically increased across the "accident" itself (from points N°2 to 3) and is then amplified from points N°3 to 4 when leaving the "accident" zone. Other results obtained on fluctuating wall pressure in the cone, not presented in this paper, showed that point N°4 corresponds to an unstable situation.

Concerning these fluctuations, their maximal amplitude is 1.5% of the total head pressure with a time period of 30 to 50 seconds. Then, the increase in mean vortex activity obtained at point N°4 must be further investigated in order to verify if it resulted from a stationary or a non-stationary effect.

The external vortex seems to be almost unaffected by the "accident". It is difficult, at the present stage of the study, to explain clearly the "accident" dynamics from mean velocity fields obtained in the conical diffuser. It appears that most of the changes on mean flow fields must occur more...
downstream probably close to the elbow. Then, a close investigation of the elbow flow both from numerical simulations and experimental approach seems necessary to describe accurately the "accident" dynamics.

3.2 Discharge Repartition Between Channels

Figure 10 shows the velocity field in each channel near the draft tube outlet obtained by LDV measurements through a 60 points measuring grid of 5 cm × 4 cm grid spacing in horizontal and vertical direction respectively. It permits us to calculate by simple integration the discharge repartition in each channel of the draft tube which appears to be non homogeneous. Indeed, for the operating point N°3 corresponding to the accident, the major part of the discharge passes through the right channel. This behaviour corresponds to the unbalance of the flow rate found in the FLINDT project [13]. In this study the blockage effect occurs in the left channel because the turbine is counter-clockwise rotating.

![Figure 10: Velocity contours in each channel - PF2 and PF3 - Dimensionless velocity in colorscale](image)

4 CONCLUSIONS

An experimental investigation including LDV and PIV measurements has been performed in the conical diffuser and the draft tube of a Francis turbine. The measurements were obtained at six operating points near the “best efficiency point”. Particularly, 3D time-resolved velocity fields were obtained by mean of PIV technique.

The one percent efficiency drop called “accident” was obtained without any ambiguity and with a good repeatability. It was in close correlation with the sudden fall of pressure recovery coefficient certainly linked to drastic changes in the draft tube flow structure.

At first, an experimental investigation was performed to describe mean velocity field in the conical diffuser. The main effect of the “accident” is to increase the vorticity of the main eddy at the center of the flow duct. This vorticity is drastically increased across the "accident" itself and is then amplified when leaving the "accident" zone. At the present stage of the study it is not clear if it is a real mean vorticity increase or an effect due to the strong unsteadiness exhibited by the flow for this operating point. More investigations are needed and will be performed in the near future to explain this issue. Hopefully the unsteady analysis of the 3D PIV velocity fields will help us too.

The discharge repartition between the two draft tube channels has been investigated with a 2D LDV system. Measurements show that the accident leads to a strong unbalance of the flow rate repartition between the two channels. At this stage of the study we could not
establish a direct link between the swirling flow at the runner outlet and the flow rate unbalance at the draft tube outlet. But it seems clear that a main change in the flow structure is present in the elbow itself and has to be described by a future investigation in this flow region.

ACKNOWLEDGMENT

The authors would like to acknowledge ADEME, ALSTOM and DGR for their financial support.

BIBLIOGRAPHICAL REFERENCES