Experimental investigation of the draft tube inlet flow of a bulb turbine

J Vuillemard¹, V Aeschlimann¹, R Fraser¹, S Lemay¹ and C Deschênes¹

¹Hydraulic Machine Laboratory (LAMH), Laval University, Department of Mechanical Engineering, Pavillon Adrien-Pouliot, Québec G1V 0A6, Canada

E-mail: julien.vuillemard.1@ulaval.ca

Abstract. In the BulbT project framework, a bulb turbine model was studied with a strongly diverging draft tube. At high discharge, flow separation occurs in the draft tube correlated to significant efficiency and power drops. In this context, a focus was put on the draft tube inlet flow conditions. Actually, a precise inlet flow velocity field is required for comparison and validation purposes with CFD simulation. This paper presents different laser Doppler velocimetry (LDV) measurements at the draft tube inlet and their analysis. The LDV was setup to measure the axial and circumferential velocity on a radius under the runner and a diameter under the hub. A method was developed to perform indirect measurement of the mean radial velocity component. Five operating conditions were studied to correlate the inlet flow to the separation in the draft tube. Mean velocities, fluctuations and frequencies allowed characterizing the flow. Using this experimental database, the flow structure was characterized. Phase averaged velocities based on the runner position allowed detecting the runner blade wakes. The velocity gradients induced by the blade tip vortices were captured. The guide vane wakes was also detected at the draft tube inlet. The recirculation in the hub wake was observed.

1. Introduction

Hydraulic turbine draft tubes present large flow instabilities associated with flow separation and backflow. These behaviors occur mainly in high discharge operating regimes, which is when the highest power production is expected. These instabilities might have an important impact on the power production and the turbine lifetime. The flow behavior inside the draft tube is highly dependent of the inlet flow condition. Several experimental studies have been conducted on model of Kaplan [1][2], propeller [3][4] or Francis [5][6] types of turbine. The late development in hydraulic turbine CFD research has also shown how the draft tube performance prediction is sensitive to the draft tube inlet velocity profile.

Being part of the BulbT project, the present study was an opportunity to measure the velocity profile at the inlet draft tube of a bulb turbine and to provide a new test case. One particularity of those turbines is that there are no ninety degree bents between the guide vanes and the draft tube inlet and in the draft tube, contrary to other low head turbine as propeller for example.

This paper presents LDV measurements at the inlet of a bulb turbine draft tube model. Five operating points are investigated at high discharge where a large flow separation develops in the draft tube [7][8]. Diverse flow phenomena are detected and characterized from the analysis of the mean velocity fields and fluctuations. Observations are performed at two locations, right downstream the...
runner blades and downstream the runner hub. Their comparison highlights the evolution of the inlet velocity profile. Finally, a method to calculate the radial velocity is presented.

2. Experimental setup

2.1. Test bench and bulb turbine model

The measurements were performed at Laval University Hydraulic Machine Laboratory (LAMH) within the framework of the Consortium on Hydraulic Machines involving hydraulic turbine manufacturers, electric utilities and Canadian government agencies [9]. LAMH's test rig consists of a hydraulic closed-loop that can accommodate axial and radial turbomachine models. The test rig is powered by a pump with a maximal flow rate of 1 m$^3$/s and a maximal head of 50 m. The turbine operates with rotational speed up to 2000 rpm and a maximal net power output of 225 kW. A vacuum pump is installed on the downstream tank to control the relative pressure in the test loop to perform cavitation studies. The test rig is set up to perform measurements according to the IEC 60193 standard.

The turbine model which is currently under investigation at the LAMH is a four bladed bulb turbine. A section view of the model is shown in figure 1. The inlet is composed of a bulb held with two stay vanes (upper and lower piers) and of a distributor aligning 16 guide vanes. The first portion of the draft tube is conical with an opening half angle of 10.25° and made of transparent PMMA to allow optical access to the runner and diffuser flow. The second portion is a transition from a circular to a rectangular section.

![Figure 1. Left: View of the model bulb turbine. Right: Zoom on the runner.](image)

2.2. Operating conditions

One of the goals of the BulbT project is to investigate the efficiency dropout near a best operating point at high discharge. Five points were chosen with this consideration for a runner blades angle fixed at 30.2° [10]. Figure 2 presents the efficiency hill chart of the turbine with the selected operating points at N11/N11_{BEP} = 1.135 (OP1 to OP5).

2.3. Reference scales

The reference length is the runner shroud radius ($R_{Ref}$) and the reference velocity ($C_{Ref}$) is defined by the equation (1) where $Q$ is the flow rate at OP2.

$$C_{Ref} = \frac{Q}{\pi R_{Ref}^2}$$ (1)
2.4. Measurement sections

To investigate the flow evolution, two sections were chosen to perform LDV measurements: one radius slightly downstream the runner (axis A) and one complete diameter downstream the runner hub (axis B) as shown in figure 3. Two coordinate systems, cartesian and cylindrical, used in this paper are presented in this figure 3, x-axis correspond to $\theta = 0^\circ$. Axis A is located at $z/R_{Ref} = 0.37$ and axis B at 0.94.

The radial velocity was measured on axis C at the azimuth 315° and same z coordinate than axis A, as shown in figure 4. The shroud radii of each axis are $R_A$, $R_B$ and $R_C$ where $R_A = R_C$. Due to geometric constraints, optical access size and LDV requirements, the radial velocity was measured at axis C only on a part of the radius, from $r/R_C = 0.64$ to 0.75.

2.5. LDV setup

The LDV system is a two-components with four-beams operating in backward scatter on-axis collection mode. There are two LDA (laser Doppler Anemometer) inside the probe head and each LDA uses two laser beams at the same wavelength to measure one component of velocity. The probe diameter measures 60 mm with the front lens having a focal length of 400 mm. The measurement volume of the crossing beams in the water is $0.1 \times 3.4$ mm in size. Bragg-cell shifting at 40 MHz is
used to resolve directional ambiguity on a 5W Argon-ion laser. The particles used as a reflective medium are silver coated hollow glass spheres. Their diameter is 10 µm and the density 1.4 g/cm³ allowing them to follow the flow fluctuations up to 12 kHz.

To measure the axial and circumferential velocities on axes A and B, the LDV probe was mounted on a double axis displacement system as shown in figure 5. The measurement grid was determined to catch the high velocity gradient near the cone wall and in the hub region. The spatial sampling varied from $\Delta r/R_{Ref} = 0.009$ to 0.035, with a total of 41 positions at axis A and 89 at axis B. To measure the radial velocity, the LDV probe was inclined at an angle of 45° and placed as shown in figure 6. The motors were mounted to allow an oblique displacement of the probe. For this setup, a different lens was used. The focal length was about 300 mm and the measurement volume size of the crossing beams is $0.09 \times 1.9$ mm.

The velocity signal can be interpreted by equation (2).

$$u(t) = \bar{u} + \tilde{u} + u'$$  \hspace{1cm} (2)

where $\bar{u}$ is the mean velocity, $\tilde{u}$ is the velocity fluctuations due to the runner also known as the phase average velocity and $u'$ is the random fluctuation.

The runner angular position was acquired simultaneously with the LDV measurements allowing the phase averaged velocities and the random fluctuations to be separated. The runner angular position was divided into 360 bins of 1.25° width. This means that the velocities were assign to a bin every degree with 20% overlap. The sample size was 120000 per location corresponding to about 400 per bin. The accuracy of the phase averaged velocity was estimated to 0.05 m/s.

3. Measurement results

3.1. Axial and tangential velocity measurements

The figure 7 displays the axial and circumferential mean velocities for axes A and B for each operating point. At both axis A and B, the velocity profiles present the same characteristics. The axial velocity increases with the operating point which corresponds to the flow rate increase. As imposed by the area expansion, the axial velocity decreases between the two axes. At axis B, the axial velocity profile shows a backflow downstream the hub ($r/R_B < 0.1$).

The circumferential velocity profile contains counter-rotating zones. Three zones are clearly identified at axis B: the outer flow, from the cone wall with a negative velocity as the runner rotation, the backflow under the hub, which is also co-rotating with the runner and finally, a counter-rotating zone is present in-between. At axis A, the outer flow is also co-rotating with the runner and the inner flow is counter-rotating. Furthermore, as the hub wall imposes its velocity, a third thin zone, co-rotating, is present in the boundary layer but was not measured.
The same trends are observed for the five studied operating conditions. The major noticeable difference due to the flow rate increase (i.e. from OP1 to OP5) is the positive shift of the circumferential velocity profile. This shift enlarges the counter rotating zone at both axis A and B. The flow rate increase is associated with the opening of the guide vane which reduces the swirl at the runner inlet. An evolution is also noticed in the hub wake, the negative circumferential velocity doubled from OP1 to OP5, leading to larger velocity gradients at the border of the backflow zone.

3.2. Phase average velocities and observations

Isosurfaces of the phase average axial and circumferential velocities for axis A and B at OP1 are shown in figure 8. The measurement position is defined by the radial position and runner angle. The color levels show the velocity magnitude normalized by the reference velocity. As mentioned in section 2.5, this representation of the phase averaged velocities should not be interpreted as the mean velocities of the entire 360° cross-section. The four blade wakes are noticeable on each velocity fields. Those are more bent and diffused on axis B due to the surrounding flow.

Lemay et al. [8][9] performed velocity field analysis using LDV measurement inside the inter-blade channel of the present bulb turbine model. In this study, two counter-rotating structures have been identified on the suction side of the runner blades and close to the shroud wall. These blade tip vortices are induced by the flow in the blade tip gap. In the present configuration, it was found that the flat optical access at axis A modifies the original conical hydraulic profile so the blade tip vortices might be altered near the wall at axis A and azimuth 0°. However, at axis B azimuth 180°, the hydraulic profile is exempt of modification because it was measured from the other side of the cone (figure 5). In figure 8 (bottom right, Axis B azimuth 180°), the circumferential velocity field presents...
the highest gradients near the wall at the tip of the blade wake. This pattern is similar to the one observed inside the inter-blade channels and associated to the blade tip vortices in [11][12]. This confirms the presence of the blade tip vortices at the draft tube inlet.

On the axial velocity profile at axis A, there is a presence of wave shape between the blade wakes. Those waves are characteristic of the guide vane wakes in this bulb turbine model as demonstrated in [11][12]. There are 16 guide vane wakes distributed on a 360° cross section, because only a 1D radial profile at one azimuth is measured, only 2 guide vane wakes are detected along this profile as explained in [12]. These wakes dissipate slowly from the distributor to the draft tube. Their tracks are smoother on axis B than on axis A.

Figure 8. Phase average velocity isosurfaces for OP1. Top left: phase average axial velocity on axis A. Top right: phase average axial velocity on axis B. Bottom left: phase average circumferential velocity on axis A. Bottom right: phase average circumferential velocity on axis B.

Figure 9. Sketches of the phase averaged flow patterns observed at axis A and B.
The runner blade wakes, the blade tip vortices and the guide vanes wakes compose a complex flow pattern sketched in figure 9. Similar flow features were measured for the five studied operating conditions. One minor difference is the radial position of the guide vane wakes, which is in accordance with the guide vane opening. For example, figure 10 displays the phase average axial velocity on axis A for OP1 and OP4 at a runner angle of 60°. The velocity drop is associated with the guide vane wake and it shifts between OP1 and OP4 as indicated by the arrows.

**Figure 10.** Phase average axial velocity at axis A and a runner angular position of 60° for OP1 and OP4.

The velocity standard deviation for OP1. Top left: axial component, axis A. Top right: axial component, axis B. Bottom left: circumferential component, axis A. Bottom right: circumferential component, axis B.

**Figure 11.**
3.3. Phase average fluctuations and observations

In this paper, the random velocity fluctuations are represented by the velocity standard deviation. Figure 11 shows the axial velocity standard deviation ($\sigma_z$ and $\sigma_\theta$) for axis A and B at OP1. The highest fluctuations are located in the blade wakes and in the blade tip vortices at axis A. At axis B, the highest fluctuations are present at the border of the backflow downstream the hub ($r/R_B = 0.1$), where the highest mean velocity gradients were measured.

The guide vane wakes present another noticeable pattern out of the background turbulence level. Those wakes fade rapidly as shown by the fluctuations at axis B. It is interesting to look at the movement of those wakes. Starting for the pressure side of the runner blade wakes, the guide vane wakes go to the suction side with a radial movement inward the radius. This movement is in accordance to the radial velocity field presented in section 4. The guide vane wakes follow the surrounding flow. Figure 12 shows the axial fluctuation and the phase average velocity at axis A, runner angle 60°, OP1. As shown by the dotted lines, the minimum velocity induced by the guide vane wake corresponds to a fluctuation peak from the standard deviation. The wakes induce velocity gradients and fluctuations that lead to energy dissipation and losses.

3.4. Radial velocity measurements

Figure 13 presents the radial velocity measured on a partial radius at azimuth 315° at axis C. It shows that the radial velocity increases with the radius for all operating points. Also, there is a negative offset on the velocity when opening the guide vane from OP1 to OP5.

The phase average values and fluctuations of the radial velocity are shown in figure 14 for OP1. The runner blade wake is present and the velocity values switch sign from positive to negative after its passage. The standard deviation shows that a guide vane wake is perceptible on the radial direction.
4. Radial velocity estimation

4.1. Methodology
A method was developed to estimate the radial velocity on axis A since its measurement was not possible over the entire radius with the present LDV setup. The idea is to resolve the equation of continuity (3) using its discrete form (4) in a cylindrical coordinate system.

\[
\frac{1}{r} \frac{\partial (rC_r)}{\partial r} + \frac{1}{r} \frac{\partial C_\theta}{\partial \theta} + \frac{\partial C_z}{\partial z} = 0
\]  

(3)

\[
\frac{1}{r_{i,j,k}} \left( C_{r_{i+1,j,k}} - C_{r_{i-1,j,k}} \right) + \frac{1}{\theta_{i,j+1,k}} \left( C_{\theta_{i,j+1,k}} - C_{\theta_{i,j-1,k}} \right) + \frac{1}{Z_{i,j+1,k}} \left( C_{Z_{i,j+1,k}} - C_{Z_{i,j-1,k}} \right) = 0
\]  

(4)

Two additional radii were measured to estimate the axial gradient (third term of eq. (3) and (4)). One was upstream the axis A and one downstream with \( \Delta z/R_{\text{Ref}} = 0.018 \), as shown in figure 15. The mean velocity profile is assumed to be axisymmetric, then the circumferential velocity gradient is null (i.e. second term of eq. (3) and (4)). Now with an estimation of the second and third terms, the first term can be evaluated. Then, the radial velocity is calculated by performing a numerical integration.

The highest velocity gradients on the mean velocity fields are in the radial direction and located near the walls. Accordingly, the measurement grid resolution was refined in the radial direction near the wall (\( \Delta r/R_{\text{Ref}} = 0.009 \)) and larger at mid-span (\( \Delta r/R_{\text{Ref}} = 0.035 \)). The algorithm was tested on a representative mean velocity data from CFD simulations. The method was proven to be accurate showing differences lower than 0.01 \( C_{\text{Ref}} \) between the estimated radial velocity and the CFD reference case.

4.2. Initial condition
To initiate the radial velocity calculus, an input condition is required. The zero velocity at the wall is not used because the high gradients present at the walls would amplify the errors and can affect the entire set of values calculated. To avoid this effect, the radial velocity measurements at Axis C were used. To use the radial velocity at another azimuth requires the velocity field to be axisymmetric. It is supported here with the comparison of the axial velocity shown in figure 16. The difference between the velocities at azimuth 0° and 315° is about 1%, which confirms the quasi-axisymmetric hypothesis.

4.3. Computed Radial velocity
Figure 17 presents the estimated radial velocity profile for OP1. The estimation matches the measurements in the overlapping zone. This is also true for the other operating conditions.
The figure 18 shows the mean radial velocity estimation for each operating point. The trend is similar for each of them. From the hub till about \( r/R_C = 0.8 \), the flow moves toward the hub with a negative radial velocity. Further than \( r/R_C = 0.8 \), the flow moves toward the cone wall. Finally, the largest differences between the operating conditions are near the walls, the radial velocity magnitude increasing with the flow rate increase (i.e. opening the guide vanes from OP1 to OP5).

**Figure 15.** Additional radii near axis A location.  
**Figure 16.** Comparison of the mean axial velocity for azimuth 0° and azimuth 315°.

**Figure 17.** Mean radial velocity estimated and measured on axis C for OP1.  
**Figure 18.** Mean radial velocity estimate for every operating points.

5. Conclusion

Hydraulic turbine draft tubes present large flow instabilities associated with flow separation and backflow. The flow behavior inside the draft tube is highly dependent of the inlet flow condition. The present study brings a new test case with a bulb turbine where the straight alignment from the distributor to the draft tube outlet is conductive to guide vane wake propagation.

Velocity fields were measured using a LDV two-component system at the runner outlet and right downstream the runner hub. A second setup allowed measuring the radial velocity in a small portion of the radius. A method was developed to perform indirect measurement of the mean radial velocity component on the complete radius. The mean velocity fields, the periodic fluctuations induced by the runner blades and the turbulent fluctuations were analysed. Five operation conditions were tested at high discharge associated with the development of a large flow separation in the draft tube.

The analysis revealed the complex pattern of the velocity field at the draft tube inlet. The mean velocity field is composed of three coaxial rotating zones with the inner one presenting a backflow under the runner hub. The runner blade wakes induce periodic fluctuations associated with large
turbulent fluctuations. The blade tip vortices were detected at the draft tube inlet. The guide vane wakes were detected at the runner outlet for the first time on a hydraulic turbine model. They alter the mean velocity field and add turbulent fluctuations. Their locations oscillated radially with a period of one runner blade passage. No abrupt flow evolution was noticed with the large flow separation that develops downstream in the draft tube according to the five studied operating conditions.

This study is part of a test case for validation of numerical simulation strategies and also provides realistic inlet boundary conditions to initiate draft tube flow simulations. Further work will be conducted jointly with CFD development and analysis.

Acknowledgement
The authors would like to thank the participants of the Consortium on Hydraulic Machines for their support and contribution to this research project: ALSTOM Renewable Power Canada Inc., Andritz Hydro LTD, Hydro-Quebec, Laval University, NRCan, Voith Hydro Inc. Our gratitude goes as well to the Canadian Natural Sciences and Engineering Research Council who provided funding for this research.

References
[1] Tridon S, Barre S, Ciocan G D and Tomas L 2010 Experimental analysis of the swirling flow in a Francis turbine draft tube: Focus on radial velocity component determination Eur. J. Mech. - B/Fluids 29 (4) 321–335
[2] Cervantes M J, Andersson U and Lövgren H M 2010 Turbine-99 unsteady simulations – Validation IOP Conference Series: Earth and Environmental Science 12 012014
[3] Gagnon J-M, Aeschlimann V, Houde S, Flemming F, Coulson S and Deschênes C 2012 Experimental Investigation of Draft Tube Inlet Velocity Field of a Propeller Turbine J. Fluids Eng. 134 (10) 101102
[4] Gouin P, Deschênes C, Iliescu M and Ciocan G D 2009 Experimental Investigation of Draft Tube Flow of an Axial Turbine by Laser Doppler Velocimetry 3rd IAHR Int. Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems (Brno, Czech Republic)
[5] Susan-Resiga R, Ciocan G D, Anton I and Avellan F 2006 Analysis of the Swirling Flow Downstream a Francis Turbine Runner J. Fluids Eng. 128 (1) 177
[6] Ciocan G D, Iliescu M S, Vu T C, Nenemann B and Avellan F 2007 Experimental Study and Numerical Simulation of the FLINDT Draft Tube Rotating Vortex J. Fluids Eng. 129 (2) 146
[7] Duquesne P, Maciel Y, Aeschlimann V, Ciocan G D and Deschênes C 2014 Power break off in a bulb turbine: wall pressure sensor investigation 27th IAHR Symp. on Hydraulic Machinery and Systems (Montréal, Canada)
[8] Duquesne P, Maciel Y, Ciocan G D and Deschênes C 2014 Flow separation in a straight draft tube, particle image velocimetry 27th IAHR Symp. on Hydraulic Machinery and Systems (Montréal, Canada)
[9] Deschênes C, Ciocan G D, De Henau V, Flemming F, Huang J, Koller M, Naime F A, Page M, Qian R and Vu T 2010 General overview of the AxialT project: A partnership for low head turbine developments IOP Conf. Ser. Earth Environ. Sci. 12 012043
[10] Duquesne P, Fraser R, Maciel Y, Aeschlimann V and Deschênes C 2014 Draft tube flow phenomena across the bulb turbine hill chart 27th IAHR Symp. on Hydraulic Machinery and Systems (Montréal, Canada)
[11] Lemay S 2014 Étude expérimentale de l’écoulement dans le canal inter- aube d’une turbine de type bulbe (master thesis) Université Laval (Québec, Canada) 121-127
[12] Lemay S, Fraser R, Ciocan GD, Aeschlimann V and Deschênes C 2014 Flow field study in a bulb turbine runner using LDV and endoscopic S-PIV measurements 27th IAHR Symp. on Hydraulic Machinery and Systems (Montréal, Canada)