CFD analysis of a bulb turbine and validation with measurements from the BulbT project

T C Vu¹, M Gauthier¹, B Nennemann¹, H Wallimann² and C Deschênes³

¹ Andritz Hydro Ltd., 6100 Transcanadienne, Pointe Claire, QC. H9R 1B9, Canada
² Andritz Hydro AG, Hardstrasse 319, 8021 Zürich, Switzerland
³ Laval University, 1065 Avenue de la Médecine, Québec, QC. G1V 0A6, Canada

E-mail: thi.vu@andritz.com

Abstract. In the present paper both steady and unsteady CFD analysis were performed to investigate the flow behavior in a bulb turbine. A study was carried out to assess the effect of runner tip and hub gap size on the turbine performance. Also special attention was paid to the turbine power break off at full load condition as well as the hysteresis phenomena on the turbine efficiency curve occurring in the same region. The unsteady calculations were performed only for some operating points of interest near the power break off region. The numerical results were compared with experimental data from the BulbT project.

1. Introduction

A bulb turbine is a Kaplan type design with a horizontal axis. Its fully axial design with a straight draft tube results in a reduction in turbine size and in a shallower excavation, leading to less civil works. The double regulation of the bulb turbine allows installation for low head power plants (up to 30m).

The BulbT project, started in 2011 at the “Laboratoire de Machines Hydrauliques” (LAMH) of Laval University in Québec City, Canada, aims to investigate the flow phenomena in a bulb turbine. The project was centered on flow measurements in a turbine model at the LAMH and on numerical analysis performed by industry partners and LAMH students. A 0.34m throat diameter scaled model has been installed on the LAMH test rig. The turbine model consists of a large intake, a distributor with a vertical pier and 16 axial guide vanes, a 4 blade runner and a draft tube containing a conical diffuser and a circular to rectangular section diffuser. The turbine assembly is shown in figure 1 with different measurement sections for experimental investigation. During the course of the investigation, different measurement techniques were used to gather data, such as a Coordinates Measurement Machine (CMM) to recover the geometry of the turbine components, a High speed camera (HSC) for qualitative flow observation, Laser Doppler Velocimetry (LDV), Particle Image Velocimetry (PIV) and Embedded Pressure Sensors (EPS) and strain gage measuring the wall pressure and the strain on the blades. The different measurement techniques used in the BulbT components are listed in table 1. More detail information on the measurement techniques can be found in references [1] - [7].

Efficiency hill chart measurement along with tuft and cavitation visualizations was performed for the turbine. Preliminary measurements of the bulb turbine with the original draft tube having a 6° cone angle showed a smooth efficiency hill chart with no efficiency drop off at full load near the best efficiency point (BEP). A new draft tube with a larger cone angle of 10.25° was used instead, producing abrupt efficiency drop and power break off at full load near the BEP. This new draft tube was adopted for subsequent experimental investigation.
In the present paper both steady and unsteady CFD analysis were performed to study the bulb turbine flow behavior. The steady state computation was used to predict the turbine efficiency curve. Special attention was paid to the turbine power break off at full load condition as well as the hysteresis phenomena on the turbine efficiency curve. Also, a comprehensive numerical investigation was carried out to study the effect of some runner geometric parameters on the turbine performance such as the size of runner hub and tip gaps and the runner blade geometry with and without consideration of the fillet on the hub side. The unsteady calculations were performed only at the BEP. The numerical results are compared to phase-averaged LDV velocity measurements in the inter-blade channel and ensemble-averaged LDV velocity data in the draft tube cone.

**Table 1.** BulbT measurements techniques.

| Section                        | Measurement technique |
|--------------------------------|-----------------------|
| 1. Inlet                       | LDV (2D)              |
| 2. Distributor                 |                       |
| 3. Inter-blade channel         | LDV (1D)              |
| 4. Conical diffuser            | PIV (2D)              |
| 5. Circular to rectangular diffuser | LDV²                  |

² Measurements yet to be done

**Figure 1.** BulbT turbine assembly with measurement sections [1].

2. Problem setup

2.1. BulbT runner blade geometries

The BulbT runner geometry was recovered by a CMM 6 axis Faro arm mounted with a laser scanner at LAMH. The runner blade geometries and hub geometry were provided in IGES files. It was found that the geometry of the four blades was slightly different from each other. At the nominal blade tilt angle of 30.20°, the four blades have measured tilt angles respectively of 30.260°, 30.180°, 30.175° and 30.170°. The blade geometries were entered in our in-house runner blade geometry design tool using the provided geometry files. Since different results were obtained with individual runner blade geometry during our preliminary CFD analysis, an “average blade” geometry was created by averaging the profiles of four individual blades. The obtained averaged blade, with a tilt angle of 30.16°, was then compared with the original blades in order to determine the geometry deviation of each blade versus the averaged one. Blade 1 to blade 4 had a maximum deviation respectively of 0.38%Dth, 0.20% Dth, 0.24% Dth and 0.29% Dth, far beyond the IEC maximum allowable geometry deviation of ±0.1% Dth. In the previous LAMH AxialT project, using an average blade for a runner having 6 different blade geometries was justified and provided very good CFD results comparing with experimental data ([9] and [10]). There were two sets runner geometry in the BulbT database providing blade runner geometry with or without the fillet at the junction of blade and hub runner. These two sets of runner blade geometries were both entered into our in-house CAD. Figure 2 shows the difference in the blade profile on the hub between the 2 sets of blade geometries. The runner geometry with fillet was used for the CFD study in this paper.
2.2. BulbT runner gap profile
The runner gap profiles at the hub and shroud sides, as shown in figure 3, were defined by assuming a minimum and maximum blade opening at 5° and 37.5°. The model runner tip gap had a minimum value of 0.055% $D_{th}$ at the blade axis plane. In order to study the effect of the runner gap size on the turbine performance, the gap profiles at runner shroud were also defined with minimum gap value respectively at 0.03% $D_{th}$, 0.10% $D_{th}$ and 0.20% $D_{th}$.

2.3. Computational flow domain and CFD setting
The computational flow domain for CFD simulation for the entire bulb turbine model, as shown in figure 4, comprised the intake including the pier, 16 guide vanes, one runner blade flow passage and the draft tube. Grid generation for the intake including the pier was made with the commercial grid generator Hexpress/Hybrid from Numeca providing unstructured hexahedral elements with mesh refinement near the walls. For other components of the turbine, guide vane, runner and draft tube, the grid generation was made with in-house automatic mesh generators providing structured H- and O-type hexahedral meshes. The gap configurations due to the runner tip clearance were taken care of by the mesh generator. The generated meshes were intended to be used with k-ε turbulence model which requires a $y+$ value varying from 30 to 100 for the first node near solid wall. Typical mesh sizes are 1.5M nodes for the intake, 200k nodes for one guide vane flow passage, 500k nodes for one runner blade flow passage and 500k nodes for the draft tube.
the flow domain of the entire bulb turbine as “Full Turbine” and the flow without the intake as “Reduced Domain”.

2.3.1. Steady state computation. The commercial flow solver ANSYS CFX v14.0 is used for performing the flow analysis. Steady-state time-discretization with a constant pseudo-time step and the so called ‘high-resolution’ space-discretization (mostly 2nd-order-accurate) has been applied. Turbulence is modelled by the standard k-ε model. GGI interface was prescribed for the connections between the intake and guide vane domain. A “stage-type” mixing plane is applied to the interfaces between the rotating and non-rotating components.

2.3.2. Unsteady computation. The unsteady flow simulations were run with the “Full Turbine” configuration, including the full runner with 4 blade passages. The standard k-ε turbulence model was also used for the turbulence modelling.

The unsteady calculations were initialized from steady state simulations with frozen rotor interfaces between rotating and stationary components. Afterwards, the unsteady calculation with rotor-stator interfaces between rotating and stationary components is launched, with the timestep interval of first 5° of rotation per timestep and finally of 1° rotation per timestep.

User monitors are used to extract the velocities from the unsteady CFD simulations. The positions of the user monitors match the LDV measurement locations. For velocity measurements in the inter-blade channel, the user monitor positions are recalculated at every timestep. In this way, the user monitors are fixed in the stationary frame. However, this capability is only implanted in ANSYS CFX v15, and so all unsteady computations were run with this version of CFX.

3. Prediction of the efficiency curve and power breakoff with reduced flow domain

Steady state computation was performed to predict the bulb turbine efficiency curve for the speed coefficient \( N_{11} \) of 170. Figure 6 shows the experimental turbine efficiency versus flow coefficient \( Q_{11} \), obtained from three measurement campaigns. The turbine efficiency was normalized to the efficiency at BEP and the \( Q_{11} \) was normalized to \( Q_{11} \) at BEP. During preliminary test campaigns, hysteresis phenomena were observed for the efficiency measurement at the vicinity of the power breakoff region where the measured efficiency depended on the guide vane opening direction. The runner efficiency reached a lower value when measured with the guide vane openings going from a large to smaller angle value. In order to avoid this dependency, for each measurement of a specific operating condition, the guide vane was set at the assigned opening position before the machine startup and the measurement was made after about one hour of warm up. The rotational speed of the turbine was kept constant while the turbine head was fixed at 4m with a variation of +0.15% to -0.22% between each point of measurement, giving a variation of \( N_{11} \) between 169.87 rpm to 170.78 rpm. This explains the observed dispersion of the experimental data, although the uncertainty of the turbine efficiency measurement was estimated at \( \pm 0.2\% \).

Computation for the turbine efficiency was performed for a range of guide vane openings from 55° to 69°, with small increment of 0.5° opening near the BEP and the power breakoff region (the guide vanes are completely closed at 0° and are completely opened at 90°). Preliminary CFD simulation indicated that the hysteresis phenomena were also observed with the numerical solution. At the break off region, the final flow solution was influenced by an initial solution coming from a previous solution of a guide vane in the opening or closing direction. To prevent this influence, the results presented in the paper were obtained with a calculation initialized from a zero flow field defined in CFX-Solver.

For the efficiency curve computation, special attention was paid to the draft tube mesh generation and a draft tube mesh sensitivity study was carried out. The generated hexahedral draft tube mesh had O mesh type on the draft tube shell and around runner hub and H mesh type in the core flow. For the O mesh layer, the distance of the first node to the solid wall was kept constant with a mesh expansion ration below 1.2. The mesh control in the O mesh layer was kept constant in the normal direction to
the solid wall for the four generated mesh sizes, respectively of 220k, 500k, 900k and 2M nodes. The mesh size was varied only in the core flow (H mesh region) and in the through flow direction. Figure 6 shows the predicted efficiency curves obtained with the four draft tube mesh sizes. The shape of the efficiency curve and especially the position of the efficiency break off were well predicted with the four draft tube meshes. Finer mesh yielded a lower efficiency curve and a sharper shape at the BEP. Coarser mesh tended to slightly shift the BEP position to the left and predicted a higher efficiency at part load condition.

The bulb turbine energy loss was broken down into individual component losses as shown in figure 7. It showed that the draft tube loss controlled essentially the shape of the efficiency curve. The solution with finer mesh produced more losses in the draft tube. We did not obtain a mesh convergence in this study. This could be explained by the limitation of the turbulence modeling with standard k-ε and/or by the use of steady state computation in such instable draft tube flow conditions. Further numerical study is required for better understanding of the flow behavior in this bulb draft tube. The 500k node draft tube mesh gave the best match with the experimental data and was used subsequently for others geometry parameter studies.

The hysteresis of the solution depending on the initial solutions coming from a previous solution of a guide vane in the opening or closing direction was investigated. With all draft tube mesh sizes, the efficiency curve obtained with initial flow field of smaller guide vane opening position was found to be the same as the one obtained with initial zero flow field. Different results were obtained with initial flow field in the inverse direction (starting from larger guide vane opening position). This hysteresis behavior obtained with the 500k draft tube mesh solution can be seen in figure 6 and figure 7.

**Figure 6.** BulbT turbine efficiency as function of the size of the draft tube mesh.

**Figure 7.** BulbT component losses as function of the size of the draft tube mesh.

**Figure 8.** BulbT turbine efficiency – Influence of fillet and hub gap.
The efficiency curve computation was also performed for the runner blade geometry without fillets at hub (see figure 2) and also for runner blade without gap at the hub side. As shown in figure 8, there was no significant difference on the turbine efficiency comparing to the one obtained with the original geometry. The same study for the BulbT runner without gap at hub was made in reference [8] and it was found little difference in the turbine efficiency in comparison with the original runner geometry with hub gap.

4. Study on the effect of the runner tip gap with reduced flow domain
A comprehensive CFD study on the effect of the runner tip gap size on the turbine performance was performed. The reference gap size of the actual BulbT runner was at 0.055%D_n. For the study, the runner gap was reduced to about half of the reference value (0.03%D_n) then increased to about twice and quadruple of the reference gap size (0.10%D_n & 0.20%D_n). Preliminary computations performed for a single operating condition (BEP) with fixed runner flow rate were made for the four blade geometries. Comparing the obtained runner torques with the reference runner blade torque, there was a gain of +0.8% for the runner blade with smaller tip gap (0.03%D_2) and there was diminution of the computed runner torque for runner blades with larger gap size, respectively of -1.2% and -3.8%. Then the effect of runner tip gap on the runner performance was assessed over a large range of operating conditions. As seen in figure 9, at part load condition, the trend of variation of the efficiency of runner with different gap sizes corresponded to the results obtained from the preliminary computation for the single BEP. Comparing to the reference runner, the runner with small gap (0.03%D_n) had the BEP position shifted slightly to the left and the power break off occurred earlier. The behavior of the 0.10%D_n tip gap runner was quite similar to the reference runner but with a less sharp drop of efficiency at the power break off. The runner with the largest tip gap (0.20%D_n) had a relatively less efficient performance curve compared to the reference runner but there was no abrupt power break off after the BEP. It is likely that larger runner tip gap provides more kinetic energy at the draft tube wall, preventing an early flow separation in the draft tube. The plotted efficiency curve versus the power coefficient P_11 as seen in figure 10 shows that predicted power for the reference runner matched very well with experimental data and again, the largest tip gap runner provided an extra of about 2% on the obtained power.

Figure 9. BulbT turbine efficiency vs Q_11 as function of the runner tip gap size.

Figure 10. BulbT turbine efficiency vs P_11 as function of the runner tip gap size.

5. Unsteady CFD simulations vs. phase-averaged LDV measurements in inter-blade channel
We compared results from the unsteady computation and the phase-averaged LDV measurements in the inter-blade channel. The operating point for this comparison corresponds to the BEP for the measurement campaign #1 (OP2, guide vane opening = 63.3°). The LDV measurements were made at the blade axis plane. The measurements are organized by azimuths numbered according to angular
position (see figure 11). The runner reference position for both the phase-averaged lab data and the CFD results is defined as the position of the trailing edge of blade #1 at the trailing edge relative to the positive Y axis, in the runner rotation direction (towards the positive X axis). An example is shown in figure 12.

![Figure 11. LDV measurement axes in the inter-blade channel.](image1)

![Figure 12. Runner reference position, seen from downstream [1].](image2)

![Figure 13. Location of inter-blade channel points along Az 0.](image3)

We compare axial and tangential velocity components extracted from the user monitors of the last simulated runner rotation in the unsteady CFD simulations to phase-averaged LDV measurements taken in the inter-blade channel. For brevity, we only present results along a single azimuth, Az 0, at points near the outer wall, near the passage center and near the hub. These points are shown in figure 13. The axial and tangential velocities are compared in figure 14.

In all cases, the phase-averaged LDV data (thick, solid lines) is plotted along with the uncertainty on the phase-averaged values (thin solid lines). The gaps visible in the lab data corresponds to the passage of the blades and are highlighted in grey on the figures. The extracted CFD data in these intervals are invalid and can be discarded.

For the axial velocity in figure 14, there is overall a good agreement between the phase-averaged LDV data and the unsteady CFD axial velocity data. For the measurement points near the center of the flow and near the hub (figure 14 (c) and (e)), there is an almost exact match between the CFD and the lab data. Near the outer walls (figure 14 (a)) we see that the CFD under-predicts the axial velocity relative to the phase-averaged axial velocity. Nonetheless, the CFD predictions fall within the uncertainty and still reproduce the general shape of the LDV data.

For the tangential velocity in figure 14, we again see a good overall match between the LDV data and the CFD data, although for the tangential velocity the CFD has a very small over-prediction relative to the LDV data.

Finally, we can note that the uncertainty on the lab data is quite significant for the points nearest to the wall: there is a significant margin between the phase-averaged velocity and the uncertainty lines in figure 14 (a) and (b). For the other points, the uncertainty is very small and the uncertainty lines cannot be distinguished from the phase-averaged axial velocity lines.
6. CFD simulations vs. ensemble-averaged velocity profiles in the draft tube cone

In the draft tube cone, we compared the ensemble-averaged LDV velocity profiles to velocity profiles extracted from steady-state and unsteady CFD simulations. The operating point for this comparison corresponds to the BEP for the measurement campaign #4 (OP2, guide vane opening = 62.6°). In order to match the conditions on the test stand as closely as possible, the steady state simulations were run with the same flow rate as measured at OP2 for the measurement campaign #4. For the steady-state simulations, the velocities in the draft tube are extracted by area-averaging on circular bands created on planes 4A and 4B (example shown in figure 15).

For unsteady simulations, the velocity profiles are calculated by averaging the velocity user monitors over one runner rotation. The uncertainty for the ensemble-averaged lab data is available but is not included in the figures below, because the uncertainty is so small that error bars would be smaller than the size of the points representing the ensemble-averaged lab data and would needlessly add clutter to the figures.

For the axial velocity (figure 16 (a) and figure 16(c)), there is a very good correspondence between the ensemble-averaged LDV data and both the steady state and unsteady CFD results. One notable discrepancy is the axial velocity under the hub (figure 16 (c)), where only the reduced domain steady-state CFD shows a good match to the lab data. The unsteady CFD over-predicts the axial velocity under the hub, while the steady-state full domain CFD actually predicts a little bit of reverse flow under the hub.
For tangential velocities (figure 16 (b) and (d)), the CFD simulations under-predict the ensemble-averaged LDV tangential velocity for most of the width of the DT cone (radius > 0.07 m). Nearer to the hub, there is a region of counter-rotating flow seen in the LDV data as well as in the CFD results. At plane 4A (figure 16 (b)), the steady-state reduced domain over-estimates the width of the counter-flow region, while the full domain simulations (both steady-state and unsteady) predict a more narrow counter-flow region than measured. At plane 4B (figure 16 (d)), we can note that, for the region under the hub, only the unsteady CFD predictions are close to the ensemble-averaged tangential velocities. The steady-state simulations (both full and reduced domain) predict peak tangential velocities near radius = 0.02 m, which are not visible in the lab data.

7. Conclusion

In the present paper both steady and unsteady CFD analysis were performed to investigate the challenging flow behavior of a bulb turbine with a very large draft tube cone angle. The CFD steady state flow analysis predicted successfully the turbine efficiency curve with the power break off at full
load as well as the hysteresis behavior of the flow at the same region. For this particular bulb runner, numerical solutions showed no difference between two blade geometries with or without the fillet at the junction of blade at runner hub and between two blade geometry with or without gap at hub. The parameter study on the effect of the size of runner tip gap showed that larger gap size could attenuate the power break off and improve the turbine performance at full load.

The unsteady flow analysis with full turbine flow domain was performed to study the bulb turbine flow behavior at the BEP. The numerical results matched well with the LDV phase-averaged velocity measurements in the inter-blade channel and with the ensemble-averaged velocity profiles in the draft tube cone.

The validation work in this paper covered a small portion of the experimental data available in the BulbT database. More investigation will be performed for other operating conditions at part load and full load.

Acknowledgments

The in-house mesh generators used for the study were developed by project Adopt, a joint R&D project between École Polytechnique de Montréal and Andritz Hydro Ltd. The authors would like to thank the participants on 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] Consortium on Hydraulic Machines 2014 BulbT Database Guide (Quebec City: Laval University – Laboratory on Hydraulic Machines)

[2] Lemay S Étude expérimentale de l’écoulement dans le canal inter-aube d’une turbine de type bulb, master thesis, Laval University, 2014

[3] Longchamp Q Analyse expérimentale et numérique de l’écoulement dans le canal d’entrée d’un modèle de turbine bulb, master thesis, Laval University, 2014

[4] Taraud J-P Recouvrement de géométries complexes et applications pour l’étude d’une turbine hydraulique de type, master thesis, Laval University, 2014

[5] Lemay S, Fraser R, Ciocan G D, 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 (Montreal, Canada)

[6] Vuillemard J, Aeschlimann V, Fraser R, Lemay S and Deschênes C 2014 Experimental investigation of the draft tube inlet flow of a bulb turbine 27th IAHR Symp. on Hydraulic Machinery and Systems (Montreal, Canada)

[7] Deschênes C, Houde S, Aeschlimann V, Fraser R, Ciocan G D 2014 Modern challenges for flow investigations in model hydraulic turbines on classical test rig 27th IAHR Symp. on Hydraulic Machinery and Systems (Montreal, Canada)

[8] Guénette V, Houde S, Ciodan D G, Dumas G, Huang J, & Deschênes C 2012 Numerical prediction of a bulb turbine performance hill chart through RANS simulations 26th IAHR Symp. on Hydraulic Machinery and Systems (Beijing, China) (Also in IOP Conf. Series: Earth Environ. Sci. 15 (2012) 032007)

[9] Vu T C, Koller M, Gauthier M and Deschênes C 2010 Flow simulation and efficiency hill chart prediction for a Propeller turbine, 25th IAHR Symp. On Hydraulic Machinery and Systems (Timisoara, Romania) (Also in IOP Conf. Series: Earth Environ. Sci. 12 (2010) 012040)

[10] Vu T C, Gauthier M, Nennemann B, Koller, M and Deschênes C 2012 Flow simulation for a propeller turbine with different runner blade geometries, 26th IAHR Symp. On Hydraulic Machinery and Systems (Beijing, China) (Also in IOP Conf. Series: Earth Environ. Sci. 15 (2012) 012040)