Numerical analysis of a measured efficiency hysteresis on a bulb turbine model

S Houde¹, A Carrier¹, J D Buron¹ and C Deschênes¹

¹ Laval University, 1065 Avenue de la Médecine, Québec, QC.G1V 0A6, Canada

E-mail: sebastien.houde@gmc.ulaval.ca

Abstract. Within the framework of the BulbT project, simulations were performed to understand the origin of a measured hysteresis on the efficiency hill chart of a bulb turbine model. This hysteresis is associated with a sharp drop of efficiency located at slightly higher discharge than the best efficiency operating condition. It appears as a variation in the turbine performance whether an operating condition located in the efficiency drop is reached from a lower or a higher discharge. This hysteresis was reproduced numerically using Reynolds Averaged Navier Stokes (RANS) simulations. The paper presents the experimental results, the numerical methodology and a comprehensive analysis of the simulations to shed light on this interesting phenomenon.

1. Introduction

Being high specific speed and low head turbines, bulb units performance are strongly influenced by the draft tube flow which can account for up to 40% of the recovered energy. Hence, the prediction of the flow in the draft tube of bulb unit is of paramount importance to assess the turbine performance at the design stages. Reynolds Averaged Navier Stokes (RANS) simulations are a commonly used strategy to perform global studies of turbine performance [1][2][3][4][5]. RANS simulations are a compromise where the mean flow physics, besides turbulence, is averaged in time. Hence, there is always some doubt pertaining to the validity of RANS draft tube simulations since the draft tube flow, even around best efficiency conditions, is unsteady and can undergo non-periodic large scale motions [7] [8].

Setting itself in the global trend of combining flow measurements and numerical simulations to investigate flow dynamics in hydraulic turbine, the BulbT project, underway in Laval University, regroups the major hydro-turbine manufacturers with Hydro-Québec and the Canadian government to study the idiosyncrasies of bulb turbine flows. In the course of investigating through RANS simulations a sharp drop in efficiency on the BulbT turbine model, it was found that the solution in the draft tube at the threshold of the drop exhibited non-uniqueness. Specifically, depending on the initial conditions, the draft tube flow was dominated either by a recirculation bubble in the core flow or by a large separation at the wall of the diffuser. This behaviour is coherent with a measured hysteresis on the efficiency curve of the model [9].

This paper presents the result of different RANS simulations that were used in order to understand the origin of the hysteresis, its repeatability and its impact on the performance prediction. The first part presents the BulbT turbine hill chart with the measured hysteresis and the numerical methodology.
Then results from simulations of the complete turbine, from the water intake to the draft tube outlet, are analyzed to illustrate the need to account for the hysteresis in the performance prediction. Finally, the results from draft tube only simulations are presented in order to shed some light on the origin of the hysteresis.

2. BulbT turbine

2.1. Performance

The 34 cm throat diameter bulb model used in the BulbT project is of classical configuration (Figure 1a) with 4 adjustable runner blades, 16 guide vanes, a sunken bulb casing with an upper and lower pier and a two part draft tube. The draft tube has an average diverging angle of 10.25°. A conical diffusing section is used for the first third of its length while the remaining part provides a transition from a circular to a rectangular exit section with the main divergence in the horizontal plane. The rotation axis of the turbine, the Z axis on Figure 1a, is oriented downstream, toward the draft tube exit. Hence, a negative Z velocity could be considered as counter flowing.

![Figure 1: a) BulbT turbine assembly. b) BulbT operating conditions for flow investigation.](image)

It was decided early in the project that to study the ability of numerical simulations to predict the performance of the turbine, it was not necessary to perform flow measurements at different runner blade openings. Hence, although the turbine hill charts were made with 3 different blade angles, 5 operating conditions (OP1-2-3-4-5) for the final flow measurements were selected (Figure 1b) at a runner angle of 80% of the full opening, which is 20% higher than the theoretical best efficiency angle.

The efficiency curve associated with those points clearly shows a sharp drop in performance close to the best operating condition (OP2). This drop is also associated with a marked drop in power output and a sharp increase in draft tube losses. Wool tuft observations and pressure measurements throughout the turbine confirm that a change in the draft tube flow is associated with the performance drop. At OP2, the flow in the draft tube shows no detectable wall separations while at OP4 a large separation is present, on one side of the draft tube, that extents from the exit plane up to part of the conical diffuser. Pressure measurements at different sections of the draft tube tend to indicate an increase in the dissymmetry of the flow from OP2 to OP4 which is coherent with the presence of a flow separation on a preferential side [8].
2.2. Hysteresis

Early in the project, considering that flow measurements in different parts of the turbine were to be performed over many months, it was decided to study the repeatability of selected operating conditions. The repeatability is necessary to perform kinematic comparison of each measurement sets and forgo the use of multiple operating conditions for numerical simulations. During the repeatability study, it was observed that within the efficiency drop, namely around OP3, the performance of the turbine showed an hysteresis (Figure 2) associated with reaching the operating condition from a lower or higher guide vanes opening [9]. Experimentally, it was not deemed relevant at this stage in the BulbT project to study the exact nature of the hysteresis. Nevertheless, some hypothesis on the nature of this performance hysteresis can be made based on the fact that at OP3, wool tuft observation in the draft tube showed that the wall separation, observed at OP4, was not continuously present, the draft tube flow appeared to be switching randomly between two states: attached or detached wall flow. The time spent in either of the two states might be affected by the flow rate from which the condition is reached.

3. Numerical methodology

To study the turbine performance and the hysteresis, two types of simulations were used. Performance curves were evaluated with complete domain simulations, coupling all the turbine components from the intake to the draft tube. In order to isolate the origin of the hysteresis, simulations were also performed with only the draft tube. This section gives an overview of the numerical methodology describing each simulations geometry, mesh, boundary conditions and solver parameter.

3.1. Geometries, meshes, flow solver

The most commonly performed flow simulations in hydraulic turbines rely on the RANS formulation with some eddy viscosity based turbulence model. RANS are attractive tools for engineers, providing quickly, adequate performance estimates and a detailed enough description of the flow enabling some measure of optimization. Nevertheless, they do suffer many limitations, to name a few, the reliance, most of the time, on isotropic representation of turbulence through semi-empirical models, the addition of empirical corrections to those models (e.g. Kato-Lauder production correction, curvature correction) and too often the use of wall models to tackle the inner part of the boundary layers. Nevertheless, no new simulation technology appears, in the near horizon, able to replace it.

All the geometries used for the simulations were derived from the manufacturer engineering drawings, except for those for the runner and the draft tube, which were measured with a laser based CMM.
(Coordinate Measuring Machine) with a precision of about 0.05mm. The four runner blades were measured and preliminary simulations quantified the effect of the minor blade differences on the performance prediction [4]. Geometry simplifications were introduced in the model, such as the exclusion of the guide vanes gaps. The runner model used in this study includes the runner tip gaps.

Hexahedral structured meshes were used in this project. ANSYS ICEMCFD Hexa was used to mesh the intake, the runner and the draft tube, while Numeca AutoGRID was used for the guide vanes. All meshes were checked for grid independence [4] and were built with strict quality criterion where all internal angles are above 20°, all determinants above 0.5 and with maximum aspect ratio of 300. The expansion factors are all below 1.3, even between the different blocks where they are controlled manually to prevent sudden jumps in size. The first element heights at the walls were all set to yield Y+ between 20-200.

ANSYS CFX 14.5 was the flow solver used in the present study. Most of the RANS simulations were performed using k-ε turbulence model with scalable wall functions. The advection scheme is an upwind formulation with diffusion correction, corresponding in CFX to a Blend Factor of 0.75, yielding close to a second order accuracy. For multi-domain simulations (distributor, runner, draft tube), a STAGE mixing plane interface was used to perform stationary-to-rotating domains flow quantities exchanges. Simply put, the stage interface will circumferentially average the upstream velocity field and transfer it to the downstream component while the downstream pressure field is transferred to the upstream component with a mix of circumferential averaging and some measure of adaptation of the upstream conditions. This procedure is well adapted for time averaged simulations and widely used for such calculations in turbo machinery. Based on prior experiences, the final solution residuals, i.e. the level of imbalance of the discretized RANS equations for each solution loops, was typically set at 5x10⁻⁵. This level of residual is critical for the draft tube flow.

3.2. Boundary conditions

The inlet boundary conditions are set to represent the measurements (Figure 3). For the complete simulations, the inflow is either the measured flow rate or a total pressure corresponding to a constant net head. Some tests were done with a modified flow rate adjusted so the runner would yield the measured torque within 0.1%. The turbulent quantities at the inlet were set to represent an expected low turbulence level coming from the test bench settling tank. The turbulent intensity was set to 1% while the ratio of eddy to kinetic viscosity was set to 10. Simulations with higher levels representing what was measured through LDV within the intake channels showed that entrance turbulence level has almost no impact on RANS predicted turbine performance. For the draft tube only simulations, the inlet velocity profiles are extracted from the complete simulations.

![Figure 3: Boundary conditions for complete domains simulations.](image-url)
The exit condition in all cases is an average pressure applied on a straight prolongation of the draft tube. The walls of that prolongation are slip walls while all the other walls in the simulations are no-slip walls with no rugosity.

3.3. Initial conditions

Generally, most well-converged RANS simulations are independent of the initial conditions. The initial guess provided to the solver will mainly affect the initial residual level and convergence rate consequently affecting the solution time. But, knowing the existence of the closing/opening hysteresis on the BulbT turbine, it was deemed necessary to control the effect of the initial solution on the simulations results. Hence, for the complete domain simulations, a procedure was established to mimic the effect of either closing or opening the guide vanes based on using respectively a converged higher or lower flow rate solution as initial guess.

3.4. Performance evaluation

The numerical efficiency is calculated using the same IEC 60193 procedures used on the test bench, with the pressure taps located at exactly the same positions. The losses in the different components are evaluated numerically by the difference in the mass flow averaged of the total pressure between the inlet and the outlet of each component. The draft tube recovery coefficient \( C_E \) is evaluated by:

\[
C_E = \frac{P_{\text{out}} - P_{\text{in}}}{\frac{1}{2} \rho |u_{\text{in}}|^2}
\]

Where \( P_{\text{out}} \) is the average pressure in the entrance and exit planes and \( |u_{\text{in}}| \) is the module of the entrance velocity field.

4. Performance curve with complete domain simulations

For the complete domain simulations using the measured flow rate as inflow conditions, Figure 4a shows that the efficiency curves obtained from the Opening and Closing initial conditions are an almost exact match except for OP3. This difference at OP3 does change significantly the analysis one would make about the best operating condition. The curve using the “Opening” initial conditions sets the best efficiency at the wrong operating condition, OP3 instead of OP2, but do yield the sharp drop in efficiency. The curve based on the “Closing” initial condition does predict the location of the best efficiency but fails to capture adequately the drop. The torque coming from both sets of initial conditions is identical (Figure 4b). Looking at the losses in the different turbine components (Figure 4c) it is clear that the difference at OP3 stems from the draft tube flow, the closing simulations yielding higher losses. This translates into a higher recovery coefficient for the opening simulations (Figure 4d) at OP3. The existence of BulbT numerical hysteresis was confirmed independently by Vu et al. [12].

Figure 5 presents iso-contours of negative “\( V_z \)” velocity at OP3 which can be considered as representative of counter flowing regions. The “Opening” simulation shows the presence of a large recirculation bubble within the core flow extending from the hub to the entrance of the trumpet. The “Closing” simulation on the other hand exhibit a large separation starting from the wall of the conical diffuser and extending to the draft tube exit. Those two results are coherent with simulation at other operating conditions: OP1-OP2 simulations show a recirculation bubble under the hub while simulations at OP4-5 show large separation at the wall. Analysis of Figure 4c and Figure 5 indicates...
that the presence of the large recirculation bubble for the “Opening” simulations is limiting the effective area of the conical diffuser, its performance relative to the closing simulation being slightly inferior, but within the trumpet, the absence of separation in the “Opening” simulation yield a higher recovery coefficient at the draft tube exit. It is clear at this point that the draft tube flow can settle on two different solutions depending on the initial condition.

Figure 4: RANS simulations results with “Closing” and “Opening” initial conditions a) Efficiency $\eta/\eta_{opt}$, b) Torque $T/T_{opt}$, c) Component losses, d) Draft tube recovery coefficient ($C_e$).

Figure 5: Topology of the draft tube flow for the “Opening” and “Closing” simulations. Isocontours of negative axial velocity ($V_z$) illustrating counter-flows are in blue.
The occurrence of this hysteresis at OP3 is coherent with the measurements presented in Figure 2. Furthermore, it is known from wool tuft observations that within the drop (around OP3) the wall separation is intermittent, thus indicating that the flow can exist in both states (separated and attached). Only measurements within the core flow are lacking to confirm the existence of the large recirculation bubble.

If neither the closing nor the opening simulations yield an efficiency curve matching the experimental measurements, the average of both curves gives qualitatively a good match (Figure 6a-b). The average curve predicts the best efficiency at OP2 and the drop position and amplitude between OP3 and OP4. So basically, given the right flow rate and rotation speed, and accounting for the difference between the constructed geometries and the CAD model used in the simulations, RANS simulations, used with proper initial conditions, delivered an accurate relative prediction of the BulbT turbine performance (torque and losses distribution).

![Figure 6](image)

**Figure 6:** Comparison of the average performance of “Closing” and “Opening” simulations with 3 measurements campaign data: a) Efficiency $\eta/\eta_{opt}$, b) Torque $T/T_{opt}$.

5. Investigation into the origin of the hysteresis

The essential question that remains is what is the origin of the numerical hysteresis? Kinematic comparison for “Opening” and “Closing” simulations at OP3 of each component (intake, distributor, runner, draft tube) reveals that beside the difference in draft tube flow, the main difference in the velocity field appears at the exit of the runner (Figure 7). In absolute term, for the 3 velocity components, the differences between the velocity profiles at draft tube inlet are between 0.001 and 0.01 m/s concentrated mainly around the hub. Analysis at all the other simulated operating conditions (OP-1-2-4-5) showed that the differences in the velocity fields throughout the turbine are around $10^{-5}$ m/s indicating that the differences at the runner exit at OP3 are indeed significant.

![Figure 7](image)

**Figure 7:** Velocity field differences at draft tube inlet (Opening – Closing). From left to right: $V_z, V_\theta, V_r$. 
There are many hypotheses on how those differences can affect the draft tube flow, two appeared obvious at first sight. The first one is the slight diminution of the axial velocity near the shroud in “Closing” simulations, compensated by a more significant increase at the hub, which may accentuate the momentum losses on the draft tube walls leading to flow separation downstream. Another one is related to the significant changes both in \( V_r \) and \( V_z \) around the hub that may change the stability of the flow, inhibiting some type of vortex breakdown occurring under the hub in “Opening” simulations that lead to the formation of the large recirculation bubble. This recirculation bubble does accelerate the flow close to the walls so theoretically helping energizing the boundary layer in the first part of the draft tube and minimizing the likelihood of a separation for “Opening” simulations.

In order to validate or dispel those hypotheses, 12 simulations were made using only the draft tube and the axisymmetric velocity profiles coming from the runner. Two simulations were made using directly the two velocity profiles from the runner with closing and opening initial conditions (\( DTOP, \) \( DTCL \)). The next 10 simulations were made by inverting one by one the 5 velocity profile components (\( V_\theta, V_r, V_z, TKE, \varepsilon \)). Those simulations were all started with the same initial solution (\( u=v=w=TKE=\varepsilon=0 \)).

| Simulations | Inlet condition | \( C_\alpha(\%) \) | Topology           |
|-------------|-----------------|-------------------|--------------------|
| DTOP        | Opening         | 84.7              | Bubble             |
| DTCL        | Closing         | 81.2              | Wall separation    |
| DTOP-VZ     | Opening - \( V_z \) closing | 85.2              | Bubble             |
| DTCL-VZ     | Closing - \( V_z \) opening | 81.0              | Wall separation    |
| DTOP-VQ     | Opening - \( V_q \) closing | 84.6              | Bubble             |
| DTCL-VQ     | Closing - \( V_q \) opening | 80.8              | Wall separation    |
| DTOP-VR     | Opening - \( V_r \) closing | 85.2              | Bubble             |
| DTCL-VR     | Closing - \( V_r \) opening | 85.2              | Bubble             |
| DTOP-TKE    | Opening - TKE closing | 84.7              | Bubble             |
| DTCL-TKE    | Closing - TKE opening | 80.8              | Wall separation    |
| DTOP-\( \varepsilon \) | Opening - \( \varepsilon \) closing | 84.6              | Bubble             |
| DTCL-\( \varepsilon \) | Closing - \( \varepsilon \) opening | 80.9              | Wall separation    |

Table 1: Performance and topology of draft tube only simulations.

Table 1 gives an overview of the results. Simulations DTOP and DTCL do confirm that the change in velocity profiles at the runner exit from the “Closing” and “Opening” simulations are indeed responsible for the difference in flow behaviour. Both simulations show the same performance and topology from their full machine counterpart. Simulations \( DTOP-VQ, DTCL-VQ, DTOP-VZ, DTCL-VZ, DTOP-TKE, DTCL-TKE, DTOP-\( \varepsilon \), DTCL-\( \varepsilon \) do not exhibit any change in topology. This tends to invalidate the first hypothesis about the effect of the axial flow variation close to the wall. Simulations DTOP-VR and DTCL-VR do show an inversion in the flow topology. Hence the change in the radial velocity is the one influencing the hysteresis of the flow.

Considering that the major difference in the radial velocity between the “Closing” and “Opening” simulations appears in the vicinity of the hub, it might be reasonable at this point to assume that the stability of the flow under the hub at OP3 is close to a critical value line. Studies on the stability of vortical flow do confirm that hysteresis can exist in the presence of vortex breakdown and it is generally postulated that the recirculation bubble under the hub of a hydraulic turbine is a type of
vortex breakdown [10] [11]. Further studies are underway at the LAMH to confirm this hypothesis, add results with turbulence models not relying on wall functions and clearly identify where in the runner those differences in velocity profiles originates.

6. Conclusion

This paper has shown that hysteresis associated with a change in initial condition can affect the relative performance prediction of hydraulic turbine relying on RANS equations with $k$-$\varepsilon$ turbulence modelling. This conclusion is based on a comparison between the measured performance on a bulb turbine model and numerical simulations. The measured model performance had some idiosyncrasies with a sharp drop in performance right after the best efficiency condition and an hysteresis within that sharp drop that was put to light when the same conditions was reached from a higher or a lower flow rate.

In order to match the main feature of the measured efficiency curve (sharp drop and location of the best $\eta$), two sets of simulations were averaged. Those two sets were obtained by changing the initial conditions of the simulations by using a converged solution from either a higher or a lower flow rate, thus mimicking opening and closing the turbine guide vanes for a fixed head. For the simulations at the beginning of the drop, the simulations exhibited a hysteresis in the draft tube flow that translated into two different solutions, either a recirculation bubble under the runner hub or a separation at the draft tube walls.

Acknowledgments

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, 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.

Nomenclature

- $D$ Throat diameter of the turbine [m]
- $H$ Total head [m]
- $T$ Torque [Nm]
- $C_{ex}$ Draft tube recovery coefficient [%]
- $\eta$ Hydraulic efficiency
- $Q$ Flow rate [m$^3$/s]
- $Q_{opt}$ Unit discharge = $Q/(D^2H^{1/2})$
- $opt$ Index referring to the point near best efficiency point
- $V_r, V_\theta, V_z$ Radial, tangential and axial velocity component [m/s]
- $k, \text{TKE}$ Turbulent kinetic energy [m$^2$/s$^2$]
- $\varepsilon$ Turbulent eddy dissipation [m$^2$/s$^3$]

References

[1] Vu T C and Retieb S 2002 Accuracy assessment of current CFD tools to predict hydraulic turbine efficiency hill chart 21st IAHR Symp. on Hydraulic Machinery and Systems (Lausanne, Switzerland)

[2] Houde S, Giroux A-M 2005 Recent developments and new challenges in the use of computational fluid dynamics at Hydro-Québec HYDRO (Villach, Austria)

[3] Vu T C, Koller M, Gauthier M and Deschênes C 2011 Flow simulation and efficiency hill chart prediction for a propeller turbine International Journal of Fluid Machinery and Systems 4(2) 243-254
[4] Guénette V, Houde S, Ciodan D G, Dumas G, Huang J and Deschênes C 2012 Numerical prediction of a bulb turbine performance hill chart through RANS simulations IOP Conf. Series: Earth Environ. Sci 15 032007.

[5] Vincent P B, Dumas G, Houde S 2009 Comparison of URANS and DES Predictions for Hydroturbine Draft Tube 17th Annual Conference of the CFD Society of Canada (Ottawa, Canada)

[6] Mauri S, Kueny J-L and Avellan F 2004 Welé-legendre separation in a hydraulic turbine draft tube Journal of Fluids Engineering 126 976–980

[7] Gagnon J-M, Flemming F, Qian R, Deschênes C and Coulson S 2010 Experimental and numerical investigations of inlet boundary conditions for a propeller turbine draft tube ASME- FEDSM (Montreal, Canada)

[8] 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 (Montreal, Canada)

[9] 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 (Montreal, Canada)

[10] Shtern V, Hussain R and 1999 Collapse, symmetry breaking, and hysteresis in swirling flow Annual Review of Fluid Mechanics 31

[11] Escudier M 1987 Confined Vortices in Flow Machinery Annual Review of Fluid Mechanics 19

[12] Vu T C , Gauthier M, Nennemann B, Wallimann H and Deschênes C 2014 CFD analysis of a Bulb turbine and validation with measurements from the BulbT project 27th IAHR Symp. on Hydraulic Machinery and Systems (Montreal, Canada)