Head losses prediction and analysis in a bulb turbine draft tube under different operating conditions using unsteady simulations

S Wilhelm 1,2, G Balarac 1, O Métais 1 and C Séguin 2

1 Grenoble-INP/CNRS/UFJ-Grenoble 1, LEGI UMR 5519, Grenoble, F-38041, France
2 GE Renewable Energy, Hydro, 82 avenue Léon Blum, Grenoble, 38041, France
E-mail: Sylvia.Wilhelm@legi.grenoble-inp.fr

Abstract.
Flow prediction in a bulb turbine draft tube is conducted for two operating points using Unsteady RANS (URANS) simulations and Large Eddy Simulations (LES). The inlet boundary condition of the draft tube calculation is a rotating two dimensional velocity profile exported from a RANS guide vane-runner calculation. Numerical results are compared with experimental data in order to validate the flow field and head losses prediction. Velocity profiles prediction is improved with LES in the center of the draft tube compared to URANS results. Moreover, more complex flow structures are obtained with LES. A local analysis of the predicted flow field using the energy balance in the draft tube is then introduced in order to detect the hydrodynamic instabilities responsible for head losses in the draft tube. In particular, the production of turbulent kinetic energy next to the draft tube wall and in the central vortex structure is found to be responsible for a large part of the mean kinetic energy dissipation in the draft tube and thus for head losses. This analysis is used in order to understand the differences in head losses for different operating points. The numerical methodology could then be improved thanks to an in-depth understanding of the local flow topology.

1. Introduction
Bulb turbines are double regulated turbines particularly adapted for low head and high flow rate sites [1]. However, head losses in draft tubes of bulb turbines may represent high energy losses in comparison with the low head under which they operate [2]. Head losses evaluation in a draft tube is thus primordial and can be done experimentally on test platform but also numerically by CFD which is less expensive, more flexible and leads to a more complete description of the flow in the draft tube than experimental measurements. For this purpose, two equations turbulence models are usually used in steady RANS simulations for draft tube flow prediction in industry since these models are robust and lead to affordable computational cost. However, since the flow in a draft tube is highly turbulent and unsteady, it may not be well predicted using steady RANS simulations [3]. Advanced turbulence models with higher computational cost, like SAS or LES, can be used to predict the complex turbulent structures governing the flow in draft tubes [4, 5] which may contribute to head losses. Moreover, inlet boundary conditions of draft tube calculations are highly influencing the flow prediction in the draft tube both in terms of velocity profiles and turbulence quantities [6–10]. In particular, taking into account the flow from the hub and shroud gaps is important [11, 12].

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd
In this context, the skills of GE Renewable Energy in hydraulic machinery on the one hand, and of the LEGI (Laboratory of Geophysical and Industrial Flows) in Large Eddy Simulations (LES) on the other, are pooled in order to reach a deep understanding of the flow in draft tubes. Unsteady RANS (URANS) calculations and Large Eddy Simulations (LES) are compared with experimental data to evaluate the reliability of each computational method. Unsteady inlet boundary conditions are defined which take into account unsteady phenomena from the runner. Based on the simulations results, a physical analysis of the flow is performed in order to determine the phenomena underlying the head losses generation. Two operating points of the turbine are considered in this paper.

2. Numerical methodology and turbulence modeling

2.1. Computational domain and operating points

The draft tube of a bulb turbine studied in the present work is shown in figure 1. The runner tip is included in the computational domain. The numerical outlet is moved away from the real outlet of the draft tube in order to avoid outlet boundary condition’s influence on the flow prediction in the draft tube.

![Figure 1: Whole domain of a bulb turbine (a) and computational domain (b)](image)

Two operating conditions of the turbine are considered in this paper corresponding to the best efficiency point of the turbine (OP1) and a high load point (OP2) whose features are described in table 1. Both are on-cam points of the turbine, which means that the relation between the net head \( H_n \), the blade angle \( \alpha \) and the distributor opening \( \gamma \) ensures optimal performance for a given flow rate \( Q \) [13].

| Operating point | \( H_n/H_n,OP1 \) | \( Q/Q_{OP1} \) | \( \gamma/\gamma_{OP1} \) | \( \alpha/\alpha_{OP1} \) | \( \eta/\eta_{OP1} \) |
|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|
| OP1             | 1.00            | 1.00            | 1.00            | 1.00            | 1.00            |
| OP2             | 1.00            | 1.64            | 1.26            | 1.58            | 0.98            |

2.2. Inlet boundary conditions definition

In steady RANS simulations, axisymmetric velocity profiles are defined at the draft tube inlet, thanks to azimuthal averaging or stage averaging [14] between the runner and the draft tube, assuming a high mixing of the flow at the runner outlet (see figure 2a). However, due to the small number of blades in bulb turbines, the field at the runner outlet is inhomogeneous in azimuthal direction between two successive blade passages [15]. In this study, a RANS simulation of one guide vane - runner blade passage is conducted for each considered operating point in order to obtain two dimensional inlet velocity profiles for the draft tube simulations (see figure 2b). These profiles are imposed at the inlet of the draft tube without azimuthal averaging and rotated to take into account the unsteadiness from the runner. In this
way, the dominant unsteadiness from the runner are considered at the draft tube inlet such as hub and gap vortices and blade wakes.

![Axisymmetric velocity profile](image1.png) ![2D velocity profile](image2.png)

Figure 2: Axisymmetric velocity profile (a) and 2D velocity profile (b) (non-dimensional) at the draft tube inlet

![Axial (Vz), tangential (Vu) and radial (Vr) velocity profiles](image3.png)

Figure 3: Axial (Vz), tangential (Vu) and radial (Vr) velocity profiles at the draft tube inlet for OP1 (——) and OP2 (— — —)

The azimuthal averaged inlet velocity profiles for OP1 and OP2 are compared in figure 3. The axial velocity is higher at OP2 than at OP1 corresponding to the higher flow rate under which the turbine operates at OP2. A larger velocity gradient from the center \((R/R_{\text{max}} = 0.28)\) to the external wall \((R/R_{\text{max}} = 1)\) of the draft tube is observed for the tangential and radial velocity profiles at OP2 than at OP1. This is a feature of the high load point which leads to negative tangential velocity in the center of the draft tube initiating a counter-rotating vortex rope in the draft tube [16].

2.3. URANS simulations

The \(k-\omega\) SST turbulence model available in the commercial flow solver ANSYS CFX v.14.5 is used for the URANS calculations of the draft tube flow. This model is the most appropriate among two equations turbulence models to deal with the high adverse pressure gradients in draft tubes [9, 17]. Transport equations for \(k\) and \(\omega\) are thus resolved, with inlet boundary conditions for these quantities that are classically imposed according to best practice guidelines [9]. In this paper, the same principle as for the velocity profiles (see section 2.2) is used to impose \(k\) and \(\omega\) profiles from the RANS simulation of the guide vane and runner at the draft tube inlet. As outlet boundary condition, an averaged static pressure of 0 is imposed at the outlet. No-slip condition is imposed on the stationary walls and the runner tip is a rotating wall at the runner rotation speed. These simulations have been compared with simulations including guide vane, runner and draft tube with transient rotor-stator interface [14] between the runner and the draft tube. The results were very similar so that the flow in the draft tube has very small influence on the runner flow field and both calculations can thus be decoupled. This enables us to conduct several draft tube simulations with only one runner calculation, for a mesh convergence study for example. A block-structured hexahedral mesh with two million of nodes is used giving a \(y^+\) mean value of 5 (maximum value of \(y^+\) is 10).

The advection terms are discretized using the second order upwind scheme in the momentum equation and the first order in the turbulence equations. The second order backward Euler scheme is used for the transient term in the Navier-Stokes equations. The time step corresponds to five degree of runner rotation per time step and the total simulation time is 4 s which corresponds to approximately 8 flow passages through the draft tube. Flow statistics are calculated on 1 s which is found to be sufficient for statistical convergence.
Compared to classical RANS simulations, in this paper, the flow modeling is improved due to unsteady RANS simulations and a better description of the inlet boundary conditions. However, the inlet turbulent quantities remain uncertain and have a high influence on the flow prediction [9, 10].

2.4. Large Eddy Simulations (LES)
LES computations are carried out with the YALES2 incompressible fractional-step solver with a finite-volume formulation with numerical schemes of 4th order in time and space [18]. The dynamic Smagorinsky subgrid-scale model is used [19]. No turbulent inlet quantities are necessary for resolution of the equations in LES. Thanks to the rotation of the inlet velocity profiles, instabilities developed without adding inflow perturbations. The same boundary conditions as in URANS simulations are used for the walls.

A tetrahedral mesh composed of 4.7 million nodes is used and a y+ ranging from 10 to 20 is ensured by prisms layers at the wall. Mesh influence study has shown that this mesh is refined enough for the flow prediction in the center of the draft tube. In order to assess the validity of the near wall resolution, comparison with calculations conducted on the same mesh with the wall-layer model of Duprat et al. [20] has been done. This has shown only small differences in velocity profiles located near the wall and in head losses. Results obtained without wall function are thus sufficient for the study presented in this manuscript. Note that having a y+ near 1 would require hundreds of millions of nodes but could improve the head losses prediction. This is currently under study. The time scheme being explicit, the time step is calculated to ensure a CFL number smaller than 0.9 in the whole domain and corresponds to 0.07 degrees of runner rotation. The simulation is stopped when the velocity profiles are stabilized and flow statistics are calculated until convergence. Around 4 flow passages through the draft tube are calculated.

3. Velocity profiles prediction in the draft tube
For the validation of the numerical simulations, experimental axial and tangential velocity profiles, measured by means of 2D-LDV (Laser Doppler Velocimetry), are available at three stations in the draft tube shown in figure 4: stations A and B in the cone and station C in the diffuser. The IEC plane is also defined in figure 4 which is used for head losses estimation in the draft tube according to the IEC standard [21].

![Figure 4: Experimental measurement stations for velocity profiles and head losses](image)

3.1. Validation of the simulations for the best efficiency point OP1
The predicted velocity profiles with LES and URANS calculations for the best efficiency point (OP1) are compared with the experimental profiles at the three measured stations in the draft tube in figures 5a and 5b. According to the experimental velocity profiles at stations A and B, in the center of the draft tube ($R/R_{max}$ around zero) a zone of low axial velocity exists which is associated with a high increase of the tangential velocity from the center of the draft tube up to $R/R_{max} = 0.1$. This correspond to the central vortex structure in the draft tube which can be observed using the Q criterion as vortex identification criterion [22] in figure 5c. More vortex structures are resolved with LES than with URANS simulations, in particular parietal turbulent structures are observed in LES. Blade and hub vortices are also observed...
next to the external and tip walls. The central vortex structure has negative vorticity along $z$ axis which means that it rotates in the same direction of rotation than the runner. The velocity profiles are well predicted by URANS and LES in stations A and B. In particular, the low axial velocity region in the draft tube center is well captured by LES. The flow prediction has to be improved in station C.

![Figure 5: Non-dimensional axial (a) and tangential (b) velocity profiles (Vz, Vu) versus the non-dimensional radial position, (——) URANS, (- - - -) LES and (+) experimental data; and vortical structures (c) predicted by URANS (top) and LES (bottom) for OP1](image)

3.2. Validation of the simulation for the high load point OP2

The numerical velocity profiles obtained for the high load point OP2 are also compared with experimental measurements in figures 6a and 6b. Velocity profiles obtained with URANS and LES are again similar in stations A and B although the prediction is better in the center of the draft tube using LES. Velocity profiles prediction has to be improved in station C. Like for OP1, a zone of low axial velocity is observed in the center of the draft tube but is now associated to negative values of tangential velocity. This corresponds to a counter-rotating vortex rope according to the runner rotation which is observed in figure 6c.

4. Head losses prediction in the draft tube

4.1. Definition of head losses

In case of LES and URANS simulations, time averaged values (operator $\langle \rangle$) of modified static pressure $\langle P^* \rangle$ and kinetic energy $\frac{1}{2} \langle \nu_i \nu_i \rangle$ are used to calculate the head losses $\Delta H$ in the draft tube between the inlet and the IEC plane (defined in figure 4). Note that the operator $\bar{a}$ corresponds to the Reynolds averaged value of $a$ in case of a URANS calculation and to the filtered value of $a$ in a LES case. The head losses are then computed as:

$$\Delta H = \frac{1}{gQ} \int \int_{S_{in}} \left( \frac{\langle P^* \rangle}{\rho g} + \frac{1}{2g} \langle \nu_i \nu_i \rangle \right) g\langle \nu_z \rangle dS - \frac{1}{gQ} \int \int_{S_{IEC}} \left( \frac{\langle P^* \rangle}{\rho g} + \frac{1}{2g} \langle \nu_i \nu_i \rangle \right) g\langle \nu_z \rangle dS$$

(1)

where $g$ is the acceleration due to gravity, $\rho$ is the fluid density, $\nu$ is the fluid kinematic viscosity and $u_i$ is the velocity in $x_i$ direction. $S_{in}$ and $S_{IEC}$ are respectively the inlet and the IEC plane of the draft tube.

However, under experimental conditions it is not possible to have access to the velocity and pressure profiles in the IEC plane. The head losses are thus measured according to the IEC international standard [21] using the following formula:
\[ \Delta H_{IEC} = \frac{1}{g Q} \iint_{S_{in}} \left( \frac{\langle p \rangle}{\rho g} + \frac{1}{2g} \langle u_i \rangle \langle u_i \rangle \right) g \langle u_z \rangle dS - \frac{1}{\rho g} \left( \frac{1}{2} \rho \left( \frac{Q}{S_{IEC}} \right)^2 + \langle p_{IEC} \rangle \right) \]

where \( \langle p_{IEC} \rangle \) is the mean static pressure over 8 wall pressure sensors in the IEC plane. The pressure in the IEC plane is thus evaluated with the wall pressure assuming that there is no pressure gradient at the wall. Moreover, the flow is supposed to be uniform and axial in the IEC plane so that the kinetic energy is calculated with the bulk velocity \( \frac{Q}{S_{IEC}} \). The IEC losses may differ from the real head losses defined in equation (1) if the flow is not uniform and still rotating in the IEC plane. However, the IEC loss is the quantity of interest for hydraulic machines which permits comparison between measurements and numerical results.

The IEC losses obtained with URANS and LES calculations are compared with experimental measurements in figure 7. IEC losses are measured between the inlet and the IEC plane (figure 7a) and between station A and the IEC plane (figure 7b). The IEC losses prediction is better between station A and the IEC plane than between the inlet and the IEC plane for both URANS and LES. This means that the head losses prediction between the inlet and station A is poor. This may be due to the proximity of the inlet boundary condition and the transitional flow between the inlet and station A. The IEC losses predicted by URANS and LES are lower than the experimental IEC losses in particular for OP2. The head losses prediction is thus more difficult under high load conditions than under optimal conditions. However, the prediction is better with URANS than LES, which is in contradiction with the velocity profiles prediction. For further understanding, a local analysis of head losses generation is performed based on transport equation of mean kinetic energy.

4.2. Physical analysis of head losses in the draft tube

The expression for the head losses in the draft tube (equation (1)) can be obtained from the time averaged kinetic energy balance applied to the draft tube of volume \( V \) and surface \( S \) of outer-pointing normal \( n_j \) which writes:
(a) Inlet - IEC plane

\[ gQ \Delta H = \int \int \left( \frac{P^*}{\rho g} + \frac{1}{2g} \langle u_i \rangle \langle u_i \rangle \right) g \langle u \rangle dS - \int \int \left( \frac{P^*}{\rho g} + \frac{1}{2g} \langle u_i \rangle \langle u_i \rangle \right) g \langle u \rangle dS \]

\[ = - \int \int \left( 2 \langle u_i \rangle \langle v_i, S_{ij} \rangle n_j \right) dS + \int \int \langle u_i \rangle \langle u_i' u_j' \rangle n_j dS \]

\[ + \int \int \left( 2 \langle v_i \rangle \langle S_{ij} \rangle \right) dV + \int \int \left( -\langle u_i' u_j' \rangle \langle S_{ij} \rangle \right) dV + \int \int \left( 2 \langle v_i S_{ij} \rangle \langle S_{ij} \rangle \right) dV \]  

where \( u_i \) is the velocity in \( x_i \) direction with \( u_i = \langle u_i \rangle + u_i' \), and \( S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \) is the shear stress tensor. \( v_i \) corresponds to the turbulent viscosity in case of a URANS calculation or to the eddy viscosity in case of LES.

According to equation (3), head losses are influenced by four terms which are studied in order to understand and locate the head losses in the draft tube. The terms containing the viscosity \( v_i \) are dependent on the model used for the simulation: \( k-\omega \) SST model for the URANS calculations and the dynamic Smagorinsky model for the LES. Terms (I) and (II) correspond to diffusion of mean kinetic energy. Term (III) corresponds to the viscous dissipation of mean kinetic energy. Term (IV) and (V) are responsible for the transfer of mean kinetic energy to the turbulent flow i.e. for turbulent kinetic energy production which is either resolved (IV) or modeled (V). This turbulent kinetic energy is then dissipated by the small scales of turbulence through viscous dissipation [23].

4.2.1. Physical phenomena responsible for head losses The terms of equation (3) are calculated in URANS and LES results for OP1 and OP2. First of all, surface integrals in equation (3) are negligible in front of volume integrals in the energy balance. We note \( P_M = \int \int \int 2 \langle v_i S_{ij} \rangle \langle S_{ij} \rangle dV \) and \( P_R = \int \int \int -\langle u_i' u_j' \rangle \langle S_{ij} \rangle dV \) the turbulent kinetic energy production which is respectively modeled and
resolved. \( D = \int \int \nu (\langle S_{ij} \rangle \langle S_{ij} \rangle) dV \) is the viscous dissipation of mean kinetic energy. Note that the viscous dissipation by the small scales of turbulence does not appear explicitly but implicitly through both production terms.

The contribution of \( P_R \), \( P_M \) and \( D \) in the head loss \((gQ \Delta H)\) are compared for the URANS simulation and LES of OP1 in figure 8 (results are the same for OP2). Head loss are mainly due to turbulent production represented by \( P_M \) and \( P_R \). The contribution of the viscous dissipation \( D \) is negligible. In the URANS simulation, the turbulent production is principally modeled in \( P_M \) and depends on the turbulent viscosity \( \nu_t \) since the turbulent flow is modeled through the turbulence model. On the contrary, in LES the resolved part of the turbulent production \( P_R \) is dominating so that the head loss prediction is less dependent on the turbulent model than in URANS. However, a part of the head loss is still modeled through \( P_M \) but can be reduced by refining the mesh in LES so that a larger part of the turbulent flow would be resolved.

\[ D = \int \int \nu (\langle S_{ij} \rangle \langle S_{ij} \rangle) dV \]

Figure 8: Contribution of \( P_R \), \( P_M \) and \( D \) to the head loss \((gQ \Delta H)\) for OP1 in URANS and LES results

### 4.2.2. Localization of head losses in the draft tube

In order to locate the head losses generation, the distribution of the dominating terms in equation (3), \( P_R \) and \( P_M \), in the draft tube are shown in figure 9 for OP1 and in figure 10 for OP2. High values of \( P_R \) and \( P_M \) are found in the center of the draft tube, due to turbulent production in the vortex rope, and next to the cone wall in all cases. In URANS results for OP1 and OP2, turbulent production also occurs next to the diffuser wall. This is linked with the axial velocity gradients observed in URANS simulations next to the diffuser wall, which are higher than in LES and in experimental measurements (see figures 5 and 6). This leads to production \( P_M \) which is not realistic and may explain the higher IEC losses obtained with URANS than with LES in figure 7. On the other hand, these velocity gradients are underestimated in LES so that no turbulent production occurs next the to diffuser wall. This explains the underestimation of head losses with LES (see figure 7). The zones where \( P_M \) is non-negligible in LES, especially the draft tube center, are those where the mesh should be refined in order to reduce the part of the modeled turbulent flow and thus the influence of the turbulence model on the head losses prediction. The difference in head losses between OP1 and OP2 seems to be due to more turbulent production in the vortex rope at OP2 than at OP1, especially in the diffuser.

In order to better understand the differences of head losses between OP1 and OP2, the head losses evolutions along the draft tube are shown in figure 11 for URANS and LES results. Head losses evolutions are also decomposed into two regions: near wall region including the head losses in the vicinity of the external wall and central region including the head losses in the vortex rope. Head losses in the central region are higher at OP2 than at OP1 according the contours of \( P_R \) and \( P_M \) in figures 9 and 10. However, in all cases, head losses in the near wall region are dominating the head losses in the central region. The differences in head losses between OP1 and OP2 are thus mainly due to more turbulent production next to the wall at OP2 than at OP1. In order to improve the head losses prediction, especially under high load conditions, the flow prediction next to the wall and in the diffuser of the draft tube has to be improved.
5. Conclusion

In this paper, Unsteady RANS (URANS) and Large Eddy Simulations (LES) were performed in a bulb turbine draft tube under two operating conditions with two objectives: (i) to evaluate the reliability of the numerical approaches and (ii) to better understand the origin of the head losses in the draft tube. Accurate inlet boundary conditions accounting for the unsteadiness from the runner were defined.

The velocity profiles prediction in the draft tube is correct but should be improved in the diffuser of the draft tube and next to the wall. The central zone of low axial velocity is better captured with LES than with URANS. Moreover, LES exhibit more complex structures than URANS since a part of the turbulent flow is explicitly resolved in LES. In particular, a central vortex rope is observed, with a specific pattern for each operating point.

Both URANS and LES computations under-predict the head losses in the draft tube. According to the energy balance in the draft tube, head losses occurs mainly in the vortex rope and next to the wall due to turbulent production. It is found that the head losses prediction in URANS simulations is highly dependent on the turbulence modeling. On the other hand, the head losses are mainly controlled by the resolved structures in LES and only a small part of the head losses is dependent on the modeled flow. The modeled part acts mainly in the vortex rope in LES and can be reduced by refining the mesh. An improvement of the head losses prediction can thus be expected by refining the mesh in LES while improvement of URANS prediction would need a complex tuning of the model constants. Nevertheless, LES results could provide important indications to specifically adapt RANS models to obtain better flow...
prediction in the draft tube with reasonable computational costs. Thanks to this analysis, it is possible to locate and understand the head losses generation in the draft tube. This could be used during a design process of draft tube to limit head losses generation.

Acknowledgments

The authors would like to thank GE Renewable Energy, Hydro for the financial support and contribution to this project. Vincent Moureau and Ghislain Lartigue from the CORIA lab, and the SUCCESS scientific group are acknowledged for providing the YALES2 code. The laboratory LEGI is part of Labex Tec21 (ANR11LABX30). Computations presented in this paper were performed using HPC resources from GENCI-IDRIS (Grant No. 2012-020611) and CIMENT infrastructure (supported by CPER07,13 CIRA and ANR-10-EQPX-29-01).

References

[1] Fonkenell J 2003 La Houille Blanche 27–31
[2] Gubin M F 1973 Draft tubes of hydro-electric stations (Published for the Bureau of Reclamation, US Dept. of the Interior and National Science Foundation, Washington, DC by Amerind Pub. Co.)
[3] Ruprecht A, Helmrich T, Aschbrenner T and Scherer T 2002 Simulation of vortex rope in a turbine draft tube Proceedings of 22nd IAHR Symposium on Hydraulic Machinery and Systems pp 9–12
[4] Paik J, Sotiropoulos F and Sale M J 2005 Journal of hydraulic engineering 131 441–456
[5] Duprat C 2010 Simulation numérique instationnaire des écoulements turbulent dans les diffuseur de centrales hydrauliques en vue de l’amélioration des performances Ph.D. thesis Institut National Polytechnique de Grenoble-INPG
[6] Gagnon J M, Flemming F, Qian R, Deschenes C and Coulson S 2010 Experimental and numerical investigations of inlet boundary conditions for a propeller turbine draft tube ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting collocated with 8th International Conference on Nanochannels, Microchannels, and Minichannels (American Society of Mechanical Engineers) pp 2227–2236
[7] De Henau V, Payette F, Savourin M, Deschênes C, Gagnon J and Gouin P 2010 Computational study of a low head draft tube and validation with experimental data IOP Conference Series: Earth and Environmental Science vol 12 (IOP Publishing) p 012084
[8] Houde S, Carrier A, Buron J and Deschênes C 2014 Numerical analysis of a measured efficiency hysteresis on a bulb turbine model IOP Conference Series: Earth and Environmental Science vol 22 (IOP Publishing) p 022009
[9] Payette F A 2008 Simulation de l’écoulement turbulent dans les aspirateurs de turbines hydrauliques: Impact des paramètres de modélisation Ph.D. thesis Université Laval
[10] Brugiè re O 2015 Fiabilité et évaluation des incertitudes pour la simulation numérique de la turbulence: application aux machines hydrauliques Ph.D. thesis Grenoble University
[11] Nilsson H and Cervantes M 2012 Effects of inlet boundary conditions, on the computed flow in the turbine-99 draft tube, using openfoam and cfx IOP Conference Series: Earth and Environmental Science vol 15 (IOP Publishing) p 032002
[12] Ortiz E R, Gagnon J M and Deschênes C 2010 Numerical simulation in the runner of a propeller turbine-tip leakage flow and blade tip vortex 18th Annual Conference CFD Society of Canada, CFD 2010
[13] Round G F 2004 Incompressible flow turbomachines: design, selection, applications, and theory (Butterworth-Heinemann)
[14] Keck H and Sick M 2008 Acta Mechanica 201 211–229
[15] Vuillemard J, Aeschlimann V, Fraser R, Lemay S and Deschênes C 2014 Experimental investigation of the draft tube inlet flow of a bulb turbine IOP Conference Series: Earth and Environmental Science vol 22 (IOP Publishing) p 032010
[16] Lemay S 2014 Étude expérimentale de l’écoulement dans le canal inter-aube d’une turbine de type bulbe Ph.D. thesis Université Laval
[17] Jost D, Škerlavaj A and Lipeed J 2012 Numerical flow simulation and efficiency prediction for axial turbines by advanced turbulence models IOP Conference Series: Earth and Environmental Science vol 15 (IOP Publishing) p 062016
[18] Moureau V, Domingo P and Vervisch L 2011 Comptes Rendus Mécanique 339 141–148
[19] Germano M, Piomelli U, Moin P and Cabot W H 1991 Physics of Fluids A: Fluid Dynamics (1989-1993) 3 1760–1765
[20] Duprat C, Balarac G, Métais O, Congedo P M and Brugiè re O 2011 Physics of Fluids (1994-present) 23 015101
[21] 1999 Comission lectrotechnique internationale. cei 60193 - turbines hydrauliques, pompes daccumulation et pompes-turbines - essais de reception sur modle. Norme internationale, Genve, Suisse
[22] Lesieur M, Métais O and Comte P 2005 Large-eddy simulations of turbulence (Cambridge University Press)
[23] Pope S B 2000 Turbulent flows (Cambridge university press)