IAHR
24th Symposium on Hydraulic Machinery and Systems
OCTOBER 27-31, FOZ DO IGUASSU

SURFACE ROUGHNESS IMPACT ON FRANCIS TURBINE PERFORMANCES AND PREDICTION OF EFFICIENCY STEP UP

[1]PIERRE MARUZEWSKI, VLAD HASMATUCHI, HENRI-PASCAL MOMBELLI
[2]DANNY BURGGRAEVE, JACOB IOSFIN, PETER FINNEGAN, [1]FRANCOIS AVELLAN,
[1]EPFL – LMH, Avenue de Cour 33 bis, CH-1007 Lausanne, Switzerland
[2]BC Hydro, Vancouver, Canada
+41 21 693 2510, +41 21 693 3554, pierre.maruzewski@epfl.ch

ABSTRACT

In the process of turbine modernizations, the investigation of the influences of water passage roughness on radial flow machine performance is crucial and validates the efficiency step up between reduced scale model and prototype. This study presents the specific losses per component of a Francis turbine, which are estimated by CFD simulation. Simulations are performed for different water passage surface roughness heights, which represents the equivalent sand grain roughness height.

As a result, the boundary layer logarithmic velocity profile still exists for rough walls, but moves closer to the wall. Consequently, the wall friction depends not only on roughness height but also on its shape and distribution. The specific losses are determined by CFD numerical simulations for every component of the prototype, taking into account its own specific sand grain roughness height. The model efficiency step up between reduced scale model and prototype value is finally computed by the assessment of specific losses on prototype and by evaluating specific losses for a reduced scale model with smooth walls. Furthermore, surveys of rough walls of each component were performed during the geometry recovery on the prototype and comparisons are made with experimental data from the EPFL Laboratory for Hydraulic Machines reduced scale model measurements.

This study underlines that if rough walls are considered, the CFD approach estimates well the local friction loss coefficient. It is clear that by considering sand grain roughness heights in CFD simulations, it forms a significant part of the global performance estimation. The availability of the efficiency field measurements provides a unique opportunity to assess the CFD method in view of a systematic approach for turbine modernization step up evaluation. Moreover, this paper states that CFD is a very promising tool for future evaluation of turbine performance transposition from the model scale to the prototype.

KEY WORD: Francis Turbine, Model & Prototype Testing, CFD Simulation, Efficiency Step Up

INTRODUCTION

In the process of turbine modernizations, the investigation of the influences of water passage roughness on radial flow machine performance is crucial and validates the efficiency step up between reduced scale model and prototype.

In the past, Kurokawa studied theoretically and experimentally the roughness effects on the three dimensional boundary layer flow along an enclosed rotating disk [1]. In 1997, IEC TC4 WG18 group, convened by Tanaka, initiated its work. The WG18 IEC working group decided to revise and upgrade the scale effect formula in the existing IEC code 60995. In 2000, it incorporated the new formula for reaction turbines and pumps [2]. In 2004, Tanaka established a scale effect formula for axial flow machines [3]. At the same time, Nichtawitz emphasized that three different friction coefficients, as a function of Reynolds number and roughness, were necessary to determine the specific losses of hydraulic machines [4]. Recently, Krishnamachar and Fay synthesized analytical procedures with practical data and provided a reasonably simple computational method to obtain realistic estimation for roughness effects on the optimum efficiency of Francis turbines [5].

The present work underlines that if rough walls are considered, the CFD approach estimates well the local friction loss coefficient with respect to the values provided by Churchill's formula [6]. It is clear that imposing sand grain roughness heights as part of the wall condition in CFD simulations makes up a significant part of the global performance estimation and prediction of efficiency step up according to the methodology depicted in Figure 1.

The scope of the present paper is to present the methodology of prediction for the efficiency step up based on CFD numerical simulation taking into account the turbine component wall roughness and to validate this step up with respect to the available experimental results obtained from both field and model tests. First the BC Hydro Francis turbine case study and EPFL base line tests are presented. The following section describes the numerical model, the investigated simulations, as well as the corresponding validations. The numerical results of the impact of surface roughness on Francis turbine performances are presented in the results section. Before the conclusions, the prediction of efficiency step up by CFD simulation under roughness consideration is assessed.

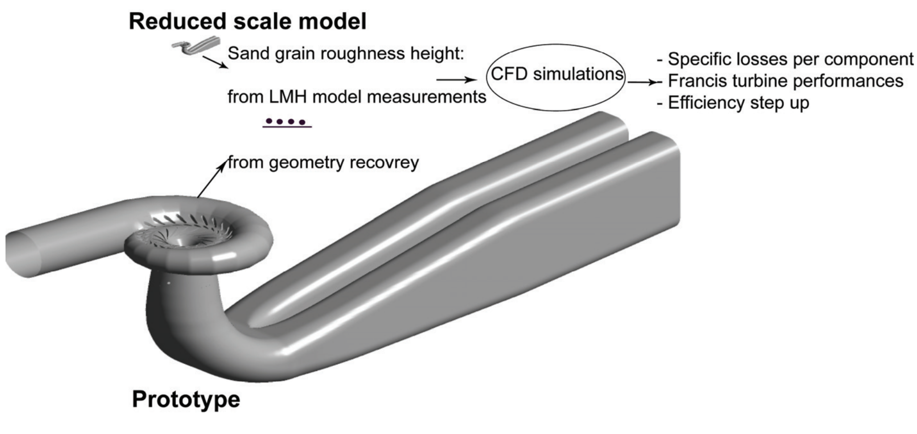

Figure 1: Methodology for the prediction of efficiency step up.

CASE STUDY: THE GORDON MERRITT SHRUM GENERATING STATION

Williston Lake, situated North-East of Vancouver, is the largest reservoir in British Colombia. The WAC Bennett Dam from British Columbia Hydro, BC Hydro, is the controlling structure for the Reservoir. The dam is 186 m high and 2'068 m long along its crest. This dam and Gordon

Merritt Shrum, GMS, generating station were constructed between 1961 and 1968. GMS houses 10 generating units that have a combined maximum output of 2’730 MW. The GMS turbines consist of medium head hydraulic Francis turbines. The case study is about turbines 1 to 5, commissioned in 1968 see Figure 2; these turbines have a rated power of 265 MW under 161 m head.

Figure 2: Computing domain of the GMS Unit 2 Francis turbine.

The EPFL was contracted by BC Hydro to carry out the geometry recovery of GMS Unit 2, in 2006, in the frame of a refurbishment project, [7]. EPFL performed a reverse engineering process which involves measuring several blades of GMS Unit 2 runner and then reconstructing it as a 3D CAD model. Each blade geometry is defined by 1’350 points using a 3D FARO© arm. Finally, a reduced scale model homologous to the prototype was manufactured and tested. Furthermore, during the geometry recovery, an evaluation of the roughness of each turbine component was performed as well. The corresponding estimated sand grain roughness heights were around hundred microns, and were not uniformly distributed. These roughness height values are used as wall roughness conditions for the CFD numerical simulation of the prototype.

GMS Unit 1 to 5 turbine model tests were performed at the EPFL PF3 test rig in 2008. The evolution of model efficiency and of prototype mechanical power is represented in

Figure *3* for the prototype net head of 161 m.

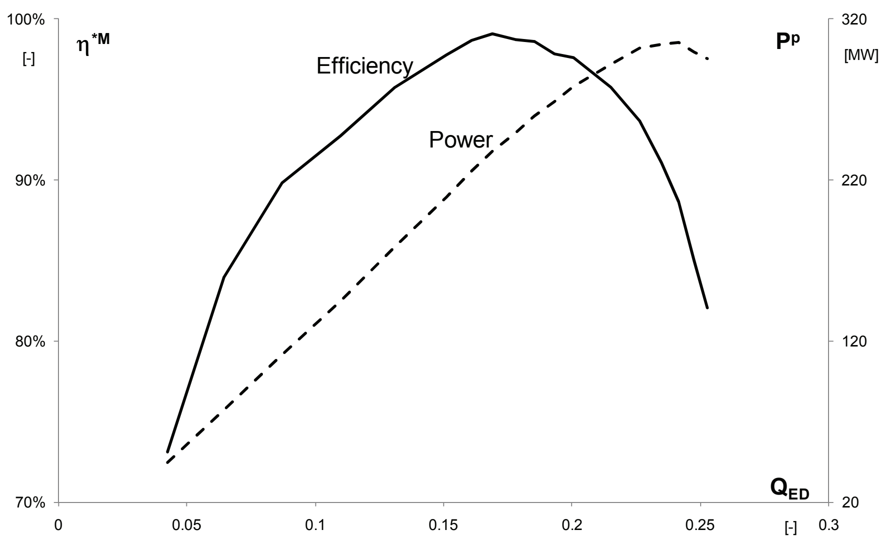

Figure 3: Model reduced efficiency and rated power for a net head of 161 m as a function the discharge factor Q_{ED} [7].

CFD NUMERICAL SIMULATIONS

The CFD numerical simulations are carried out with the ANSYS CFX-11 software, [7]. the unsteady Reynolds averaged Navier-Stokes, URANS, equations are solved with two additional equations for modeling the turbulence. Moreover, CFX-11 features the capability for the user to impose rough walls for the treatment of local friction condition provided that the k-ε turbulence model is selected. For rough walls, the boundary layer logarithmic velocity profile still exists, but moves closer to the walls. The roughness height specified is the equivalent sand grain roughness height. This is either exactly equal to the real roughness height of the surface under consideration or a function of the arithmetic mean roughness, R_a. Therefore, the appropriate equivalent sand grain roughness height must be defined according to White [8] or Schlichting [9]. Roughness effects are accounted for by modifying the expression for u^+ as follows:

$$u^+ = \frac{1}{\kappa}\ln\frac{y^*}{1+0.3k^+} \qquad k^+ = y_R\frac{\rho}{\mu}u^* \qquad u^* = C_\mu^{1/4}k^{1/2}$$

With u^+ representing the near wall velocity, κ the von Karman's constant, y^* the scalable dimensionless distance from the wall, μ the dynamic viscosity, ρ the density, C_μ being equal to 0.09, k the turbulent kinetic energy and y_R the equivalent sand grain roughness height.

Specific energy losses and efficiency predictions are performed by the hydraulic power dissipation breakdown from the numerical simulations on the computing domain $V_{sc+v+o+b+d}$, represented in Figure 4.

The available hydraulic power, P_h, is the sum of the mechanical power generated by the runner, P_t with the hydraulic power P_r dissipated by the flow in each component of the turbine.

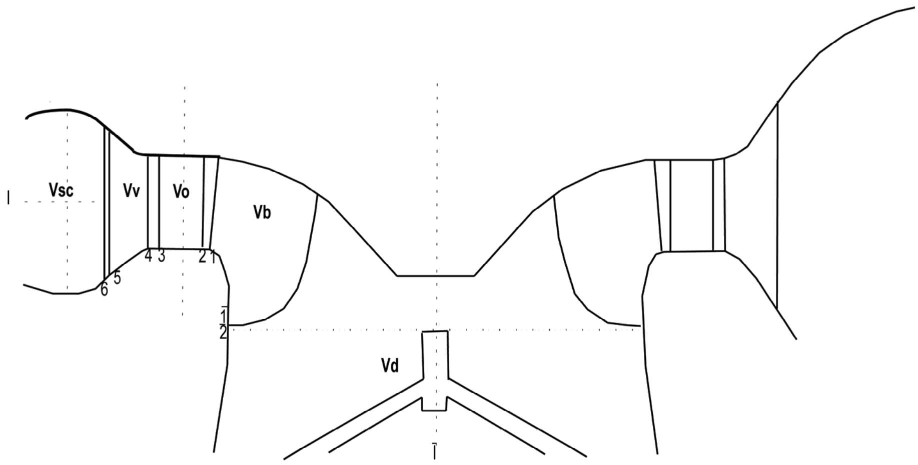

Figure 4: Nomenclature of BC Hydro Unit 2:
-sc, the spiral casing; -v the stay vane;
-o the guide vanes; -b the runner; -d the draft tube.

The following relations stands form the power flow balance in the computing domain:

$$P_h = P_r + P_t = \rho Q(gH_I - gH_{\bar{I}})$$

$$P_r = P_{rsc} + P_{rv} + P_{ro} + P_{rb} + P_{rd} + P_{rq}$$

$$P_t = \omega T_t = \rho Q_t E_t$$

Where T_t is the torque exerted by the flow on the runner flow passage, E_t the transferred specific energy, Q_t the mass flow through the blades of the runner, and Q the mass flow through a component of the hydraulic turbine.

The dissipated power, P_{ri}, within each i^th turbine component is defined by:

$$P_{ri} = \rho Q E_{ri} = \rho Q(gH_{Ii} - gH_{\bar{I}i})$$

Where E_{ri} represents the specific hydraulic energy loss in the i^{th} component.

The specific pressure energy, the specific potential energy and the specific kinetic energy are contributing to the specific hydraulic energy of a fluid particle as it follows

$$\frac{p}{\rho}+gZ+\frac{C^2}{2}$$

Then the average specific hydraulic energy gH_j of a fluid in section A_j is defined by

$$gH_j=\frac{1}{Q}\int_{A_j}\left[\frac{p}{\rho}+gZ+\frac{C^2}{2}\right]\vec{C}\cdot\vec{n}dA$$

Furthermore, the present power balance does not take into account the P_{rq}, the leakage power losses, and the disk friction power losses. Then, the balance of power within the runner reduced to

$$P_{hb}=P_t+P_{rb}=\rho Q(gH_1-gH_{\bar{1}})$$

and from Avellan [11], P_{hb}, P_t and P_{rb} are defined by

$$P_{hb}=-\int_{A_1\cup A_{\bar{1}}}\rho(\vec{C}\cdot\vec{U})\vec{C}\cdot\vec{n}dA-\int_{A_1\cup A_{\bar{1}}}\rho\left[\left\{2\nu\bar{\bar{D}}+\frac{\bar{\bar{\tau}}_t}{\rho}\right\}\cdot\vec{U}\right]\cdot\vec{n}dA+\int_V\rho\left\{2\nu\bar{\bar{D}}:\bar{\bar{D}}+\frac{1}{\rho}\bar{\bar{\tau}}_t:\bar{\bar{D}}\right\}dV$$

$$P_t=-\int_{A_1\cup A_{\bar{1}}}\rho(\vec{C}\cdot\vec{U})\vec{C}\cdot\vec{n}dA+\int_{A_1\cup A_{\bar{1}}}\left(\bar{\bar{\tau}}+\bar{\bar{\tau}}_t\right)\vec{n}\cdot\vec{U}dA$$

$$P_{rb}=\int_V\rho\left\{2\nu\bar{\bar{D}}:\bar{\bar{D}}+\frac{1}{\rho}\bar{\bar{\tau}}_t:\bar{\bar{D}}\right\}dV$$

The hydraulic power balance in the computing domain could be defined by the sum of specific energy losses as

$$\rho QE=\rho Q\left(E_{rsc}+E_{rv}+E_{ro}+E_t+E_{rb}+E_{rd}\right)\quad \text{with } E=E_t+\sum E_{ri}$$

The specific hydraulic energy efficiency, η_e, of the hydraulic turbine is defined by

$$\eta_e=\frac{E_t}{E_t+\Sigma E_{ri}}=1-\frac{\Sigma E_{ri}}{E_t+\Sigma E_{ri}}$$

The computing domain includes all the machine components, i.e. the spiral casing, the stay vanes, the guide vanes, the runner and the draft tube with a tripod in the draft tube cone. The fluid-fluid interface with stage option is imposed at the interface between the guide vanes and the runner domains and between the runner and the draft tube cone domains as well. In the simulations, walls are considered as rough and then the specific sand grain roughness heights per component are taken into account. The characteristics of the mesh for the computing domain $V_{sc+v+o+b+d}$ are summarized in Table 1 for the both cases, the prototype, p, and the reduced scale model, m.

Table 1: Mesh characteristics of the computing domain Vsc+v+o+b+d.

Component	**Number of Nodes**	**Sand Roughness Height**	**Inlet**	**Outlet**	**Topology**
Spiral casing	640’071	p: 1.5 10^{-4} m m: 7.2 10^{-7} m	Net head p: 161mWC m: 30mWC	Relative pressure equal to 0 Pa	Structured
Stay vanes	810’450	p: 1.0 10^{-4} m m: 1.6 10^{-7} m			
Guide vanes	1’073’088	p: 1.25 10^{-5} m m: 2.8 10^{-7} m			
Runner	1’757’273	p: 2.5 10^{-5} m m: 5.8 10^{-7} m			
Draft tube	928’268	p: 5.0 10^{-5} m m: 7.0 10^{-7} m			
Vsc+v+o+b+d	5’209’150	-			

The Reynolds number, Re, increases from 5.7 10^6 for reduced scale model to 1.8 10^8 for prototype.

Two cases of wall conditions for numerical simulations on the prototype are investigated. In one case, the sand grain roughness height is considered as a linear function of the arithmetical mean roughness, R_a, by the following relation:

$y_R = k \cdot R_a$, where $k = 0.1,1..6$ and $2.5\ \mu\text{m} \le y_R \le 150\ \mu\text{m}$

In the other case, real sand grain roughness heights, taken from the geometry recovery and described in Table 1, are applied to each prototype component. Moreover, 2 severe roughness heights conditions are also applied.

The quality of grids is estimated by evaluating the sensitivity of the pressure coefficient C_p versus three kinds of grids defined from the coarsest to the finest. The properties and the error of these grids applied to the draft tube are presented in Table 2. Then, all the following results are performed on medium grid with an error of ±0.8% for predicting C_p value.

Table 2: Mesh characteristics of coarse, medium and fine grids for the draft tube.

Cp Definition	**Grids**	**Number of Nodes**	**Sensitivity of Cp**
$C_p = 2\dfrac{p - p_{tinlet}}{\rho C_{m\bar{1}e}^2}$	coarse	395’208	3.4%
	medium	928’268	0.8%
	fine	1’773’384	0.0%

INFLUENCE OF ROUGHNESS ON FRANCIS TURBINE EFFICIENCY AND STEP UP PREDICTION

Regarding the ratio of specific losses over specific hydraulic energy, CFD numerical simulations allow for the definition of the limit between “admissible roughness” and “hydraulically smooth” wall as described by Nichtawitz and Tanaka in 2004,[4]. They have first defined the lower limit of roughness height and called it "*Admissible Roughness*". All the walls with a roughness height under this limit is considered as "*Hydraulically Smooth*", as shown in Figure 5.

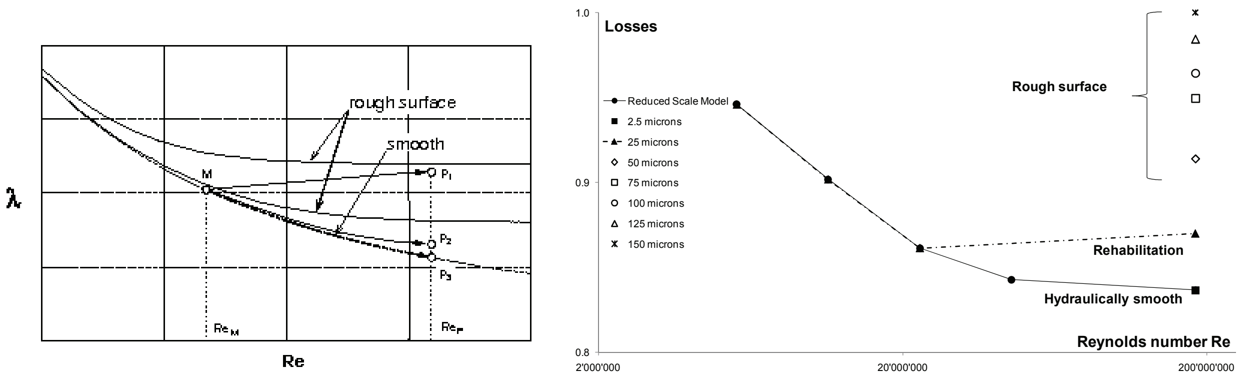

Figure 5: Definition of hydraulically smooth surface and rough surface: on the left from Nichtawitz's analysis [4]and on the right from CFD simulations.

The influence the wall roughness height on the specific losses prediction is then investigated by increasing the sand grain roughness height from 2.5 micrometers to 150 microns. According to the Figure 5, the wall of the prototype can be considered as hydraulically smooth for sand grain roughness heights within 25 microns, and as very rough for sand grain roughness heights greater than 25 microns. Furthermore, the power plant operator considers that after sanding and painting an existing rough prototype, the resulting sand grain roughness height becomes equivalent to a R_a value of 25 microns; this value being characteristic of the rehabilitated machine roughness.

For all solid walls of the prototype, the impact of surface roughness on turbine performance is carried out by linearly increasing the sand grain roughness heights from 2.5 microns to 150 microns, see Figure 6. First, over prediction of efficiency values at the best operating point and for low discharge are observed if hydraulically smooth surfaces are considered. These over predictions have already been noticed when simulations with smooth walls are performed [12] with respect to the values provided by the Churchill's empirical loss formula [6]. Then, results in Figure 6 show that the evolution of efficiency versus the sand grain roughness height and versus the discharge does not decrease as the roughness increases.

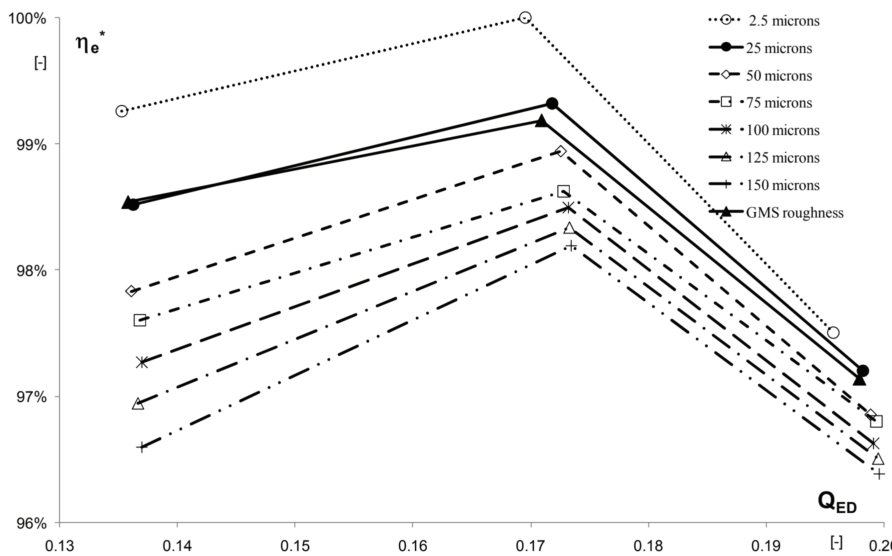

Figure 6: Surface roughness impact on the GMS turbine energy efficiency.

In the second case of simulations with the real sand grain roughness height per component of real prototype, the major difficulty consists of finding the correct CFD value for the roughness height. In fact, on the real prototype, roughness is not uniformly distributed on the component walls. Some pieces of walls are smooth while others are not. However, for the CFD numerical simulations, a uniform sand roughness height should be specified to the CFX-11. According to this constraint, it is decided to impose an averaged value of the real sand grain roughness height per component as shown in Table 1. Then, the GMS efficiency with real sand roughness height per component is equivalent to a prototype with a uniform sand grain roughness height of 25 microns.

Moreover, the impact of surface roughness on specific losses per component is presented in Table 3 for the best efficiency operating point. Three cases are investigated:

- ideal smooth walls;
- slightly rough walls with a uniform sand grain roughness height of 10 microns;
- and finally a value corresponding to the measured GMS walls roughness.

The second case corresponds to a prototype condition after rehabilitation by sandblasting and painting. Performing CFD simulations with ideal smooth walls over predicts results around 1%. However, rehabilitating a rough machine allows a decrease of 0.45% of total specific losses. Finally, the analysis of the results made apparent that it is more beneficial to rehabilitate the spiral casing, the guide vanes and the runner, than the stay vanes or the draft tube.

Table 3: Surface roughness impact on specific losses per component.

Prototype wall roughness status	**Spiral Casing [%]**	**Stay Vanes [%]**	**Guide Vanes [%]**	**Runner [%]**	**Draft Tube [%]**	**Total Losses [%]**
Smooth	0.17	1.05	1.87	1.76	0.86	5.71
Rehabilitation 10μm	0.18	1.13	2.55	1.91	0.97	6.74
Rough GMS	0.29	1.17	2.66	2.01	1.06	7.19
Saving After Rehabilitation	0.11	0.04	0.11	0.10	0.09	0.45

The efficiency step up between reduced scale model and prototype is finally predicted by the analysis of the specific hydraulic energy losses on the prototype and on a reduced scale model. Then, comparisons are made with the experimental data of the reduced model base line tests as presented in Figure 7.

The CFD numerical simulations over predict the efficiency for low discharge and under predict it for high discharge. In fact, considering the breakdown of specific energy losses per component and comparing the resulting efficiency with the efficiency scale established by Osterwalder,[13], it is clear that disk friction losses and volumetric losses take a significant part for low discharge. If the CFD numerical prototype efficiency is corrected by taking into account these losses, estimated to 2% the CFD numerical and experimental efficiency curves are in a good agreement, see Figure 8.

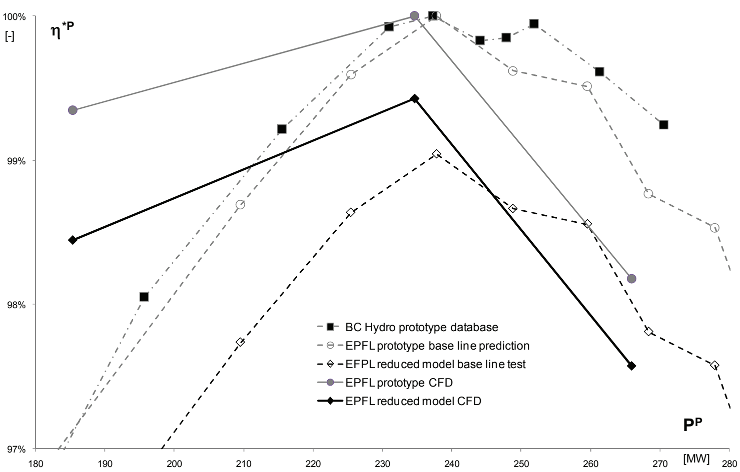

Figure 7: Prediction of efficiency step up as a function the discharge factor Q_{ED}.

Furthermore, the experimental efficiency of the reduced scale model allows predicting the uniform efficiency step up about 0.89%. But, for CFD results under real roughness conditions, the numerical efficiency step up reaches 0.50% at the best efficiency operating point, see Table 4.

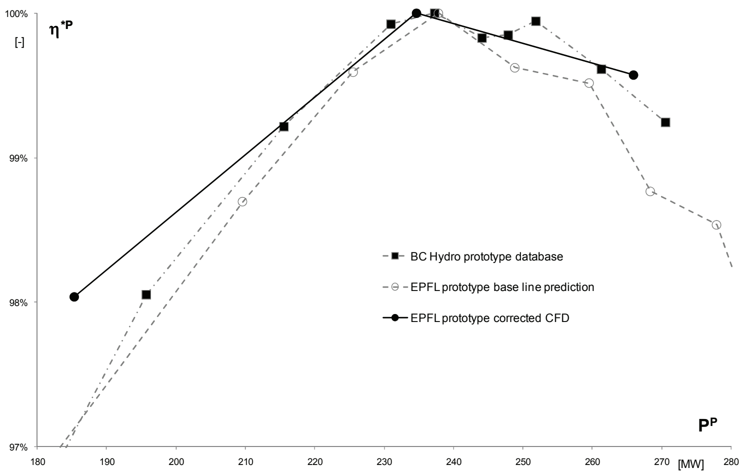

Figure 8: Corrected energy efficiency by taking account disk and gap losses.

Table 4: Wall roughness impact on efficiency step up.

Cases	Efficiency Step Up
Base line tests to rough GMS	+0.89%
CFD reduced model to rough GMS	+0.50%
CFD reduced model to rehabilitation 10μm	+0.97%
CFD reduced model to smooth GMS	+1.98%

The analysis of the impact of roughness height increases on prediction of the efficiency step up is then performed for the best efficiency operating point; see Figure 9 and 10. An efficiency step up scale versus uniform sand grain roughness height allows understanding the impact of wall roughness on the efficiency step up, Figure 9. The horizontal axis describes the uniform sand grain roughness height y_R from 0 microns, representing a smooth wall to 150 microns, meaning a strong rough wall. The real rough GMS prototype efficiency step up is compared to the rehabilitated cases, 10 and 25 microns sand grain roughness heights, and to the most severe rough walls, from 0.5 to 1 mm sand grain roughness heights, Figure 9.

Firstly, it is observed that the efficiency step up takes positive and negative values. A positive efficiency step up means that the impact of surface roughness remains weak to keep acceptable performances. Whereas, a negative efficiency step up shows that the impact of roughness is so high that the performances considerably decrease.

Moreover, considering an ideal smooth wall prototype with y_R close to 0 microns, the efficiency step up is almost +2%. For the hydraulically smooth case with y_R around 2.5 microns, the value of efficiency step up is around +1.25%. Finally, for a severe roughness case, the efficiency step up negatively decreases to -0.45%, see Figure 9. In this last figure, it is shown how the GMS Unit 2 can be evolved from existing situation to rehabilitation or to stronger roughness impact.

CONCLUSION

This study underlines that if rough walls are considered, the CFD approach is well predicting the total and local specific hydraulic energy losses. It is clear that modeling sand grain roughness heights in CFD predictions makes a significant impact on the global performance estimation. By this way, it is stated that CFD is a very promising tool for future evaluation of turbine performances. Moreover, the availability of the efficiency field measurements provides a unique opportunity to assess the CFD method in view of a systematic approach for turbine modernization step up evaluation.

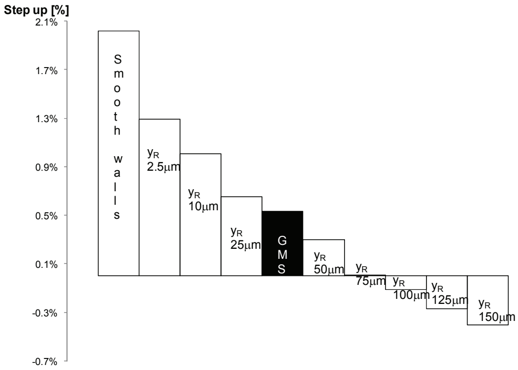

Figure 9 Efficiency step up versus sand grain roughness height.

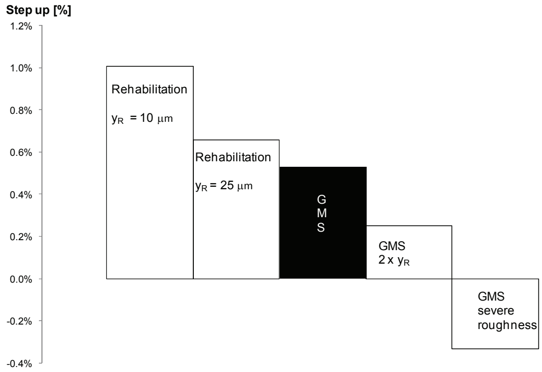

Figure 10: Efficiency step up versus sand grain roughness heights for GMS unit 2.

Next features of the prediction of efficiency step up by CFD simulations are:

- taking account the auxiliary water passages as friction disk and labyrinth;
- taking account the leakage loss;
- taking account the mechanical power losses in the bearings and shaft seal;
- using real sand grain roughness heights of newly built machines from manufacturers databases;
- Enlarging the scope of this methodology to other hydraulic machines.

ACKNOWLEDGEMENT

The authors would like to express sincere gratitude to the staff of the EPFL Laboratory for Hydraulic Machines for its support and advices to well lead this work.

BIBLIOGRAPHICAL REFERENCES

[1] KUROKAWA J., TOYOKURA T., SHINJO M., MATSUO K., December, 1978. *Roughness effects on the flow along an enclosed rotating disk*, bullet. of JSME, Vol.21, No.162, pp.1725-1732

[2] TANAKA H., NICHTAWITZ A., 2000. *New scale effect formula being studied for future IEC code*, IAHR Symposium, Charlotte

[3] TANAKA H., NICHTAWITZ A., June 29-July 2, 2004. *Scale effect formula for future IEC code its theoretical background and features*, IAHR Symposium, Stockholm

[4] NICHTAWITZ A., TANAKA H., June 29-July 2, 2004. *Derivation of formulae for future IEC code on scale effects,* IAHR Symposium, Stockholm

[5] KRISHNAMACHAR P., FAY A., 15 May 2007. *The effect of surface roughness,* International Water Power and Dam Construction, www.waterpowermagazine.com

[6] CHURCHILL S.W., 1988. *Viscous Flows. The practical use of theory.* Butterworth Ser Chem Engng, ISBN 0-409-95185-4

[7] EPFL, November, 2006. *British Columbia Hydro, Geometry recover of Francis turbine*, technical report, Lausanne, Switzerland

[8] LECHNER R., MENTER F. R., December 2008, *Treatment of rough wall on CFX-11*, ANSYS technical report, Germany

[9] WHITE M., 1979. *Viscous Fluid flow*, Mac Graw-Hill

[10] SCHLICHTING H., 1979. *Boundary layer*, Mac Graw Hill

[11] AVELLAN F., 2005. *Cours de turbomachines hydrauliques, équations des turbomachines.* Cours, EPFL

[12] MARUZEWSKI P., AVELLAN F., January, 2008. *Roughness analysis*, EPFL technical report, Lausanne, Switzerland

[13] OSTERWALDER J., HIPPE L., 1984. *Guidelines for efficiency scaling process of hydraulic turbomachines with different technical roughnesses of low passages,* Jo. Hydraulic Research, Vol. 22, No. 2, pp. 77-102