EXPERIMENTAL AND NUMERICAL ANALYSIS OF FREE SURFACE FLOWS IN A ROTATING BUCKET

Sonia KVICINSKY, ***Jean-Louis KUENY,*** ***François AVELLAN,*** — *Swiss Federal Institute of Technology/LMH Lausanne, Switzerland*

Etienne PARKINSON, — *VA Tech Hydro, Vevey, Switzerland*

ABSTRACT

The aim of this study was to provide a complete experimental and numerical analysis of the flow in a rotating Pelton bucket. For the experimental analysis, several functioning points have been used in order to study differences which may occur in various situations. This analysis will be mainly provided by the pressure distribution in the bucket and will be supported by flow visualisations. Thanks to these results, five distinct zones within the bucket have been isolated each corresponding to different signal characteristics. The numerical simulation has been performed using the two-phase homogeneous method in an original way by considering the relative path of the jet and the relative velocity changing with the rotation of the wheel. These simulations were performed for the same experimental functioning points.

RÉSUMÉ

Le but de cette étude est d'effectuer une analyse expérimentale et numérique complète de l'écoulement dans les augets de turbines Pelton. Pour l'analyse expérimentale, plusieurs points de fonctionnement ont été testés afin d'observer les différences qui peuvent apparaîtrent dans diverses situations. Cette analyse sera effectuée sur la distribution de pression dans l'auget et sera appuyée par des visualisations d'écoulements. Grâce à ces résultats, cinq zones distinctes dans l'auget ont pu être isolées, chacune correspondant à une caractéristique d'un signal différent. La simulation numérique a été effectuée en utilisant la méthode diphasique homogène de façon originale en considérant le cheminement relatif du jet et la vitesse relative variant avec la rotation de la roue. Ces simulations ont été effectuées pour le même point de fonctionnement que les mesures.

NOMENCLATURE

Term	Symbol	Definition
Area	A	m^2
Absolute velocity	$\vec{C}, \vec{c}$	m/s
	$\vec{C}^*$	$\vec{C} / C_{ref}$
Diameter	D	m
Static pressure	p	N/m^2
Pressure coefficient	C_p	$C_p = \frac{p - p_{ref}}{\frac{1}{2}\rho C_{ref}^2}$

Term	Symbol	Definition
Flowrate	Q	m^3/s
Jet radius	r_0	m
Force	$\vec{F}$	N
Jet angle	γ	°
Dynamic viscosity	μ	Ns/m^2
Volume fraction	α	$\alpha_n = \frac{V_n}{\sum_n V_n}$

Term	Symbol	Definition
Head coefficient	ψ	$\psi = \frac{2E}{\omega^2 R_1^2}$
Density	ρ	kg/m^3
Reference to reference values	ref	
Phase number	n	

Term	Symbol	Definition
Discharge coefficient	φ	$\varphi = \frac{Q}{\pi\omega R_1^3}$
Kinetic viscosity	ν	Ns/m^2
Reference to the mixture model	m	
Reference to non dimensional variables	*	

INTRODUCTION

Free surface flows are representative of Pelton turbines. Modeling this kind of flow is difficult, not only due to computational weakness but also because there are two phases to consider. Kvicinsky et al. (Ref. 3) experimentally validated two kinds of mathematical models: the homogeneous model and the Volume Of Fluid model (Hirt, 1988, Ref. 2), implemented in two different industrial codes. They showed in the plane wall case that both were accurate.

As seen with Francis turbines, numerical simulation of free surface flows in Pelton turbines would help to understand the nature of the flow path in the bucket and thus improve the efficiency which is still relatively low. After having modeled the flow path in a non rotating bucket (Kvicinsky et al., 2002, Ref. 4), it is now interesting to numerically and experimentally analyze the flow in a rotating bucket, using the homogeneous mathematical model included in the industrial code CFX4.3® (Ref. 7).

NUMERICAL METHOD IN MODELING FREE SURFACE FLOWS: THE HOMOGENEOUS MODEL

The homogeneous model interprets distinct phases, but the mixture is considered as a whole. Only the mean quantities of the mass source Γ_m, the momentum source M_m and the energy source E_m are considered; the mass diffusion terms and the interfacial mechanical energy can be neglected in the field equation. The volume fractions are still assumed distinct.

Governing equation

The governing equation for the non-isothermal motion of an incompressible and homogeneous fluid is described by three equations:

1. <u>The continuity equation</u>

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot \left(\rho_m \vec{c}_m\right) = 0$$

With: $$\rho_m = \sum_{n=1}^{2} \rho_n \text{ and } \vec{c}_m = \frac{\sum_{n=1}^{2} \rho_n \vec{c}_n}{\rho_m}$$

2. <u>The diffusion equation</u>

The diffusion equation determines the volume fraction of each phase and is written as:

$$\frac{\partial \alpha_n \rho_n}{\partial t} + \nabla \cdot \left(\alpha_n \rho_n \vec{c}_m\right) = \Gamma_n$$

3. The momentum equation

$$\frac{\partial}{\partial t}\left(\rho_m \vec{c}_m\right)+\rho_m\left(\vec{c}_m \cdot \vec{\nabla}\right)\vec{c}_m=-\vec{\nabla}\left(p_m+\rho_m g z\right)+\vec{\nabla}\left(\overline{\overline{\tau}}+\overline{\overline{\tau}}_t+\vec{M}_m\right)$$

With the pressure of the mixture p_m: $\quad p_m=\sum_{n=1}^{2} p_n$

For details concerning two-phase flows theory, one can refer to Ishii (Ref. 1)

NUMERICAL SIMULATION

The calculation is processed with CFX4.3® considering a 3D, viscous and laminar flow. The jet moves on a face named INLET (Fig. 1) and develops itself with the wheel rotation until its interaction with the bucket. The transient movement of the jet velocity profile is set as boundary condition on the inlet face. This ellipse, which corresponds to the intersection between a cylinder (the jet) and a plane (inlet), moves according to the time-dependent angle γ(t). The center of the ellipse is defined such as:

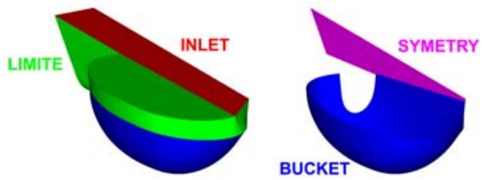

Fig. 1 Boundary condition

$$x_0(t)=0 \qquad y_0(t)=R_1\frac{\cos\left(\gamma(t)+\alpha(t)-\frac{\pi}{2}\right)}{\cos\alpha(t)}-R_p\cos\delta$$

where γ(t) and α(t) can be expressed as follows:

$$\gamma(t)=\gamma_0+\omega dt \qquad \alpha(t)=\alpha_0-\omega dt$$

and dt corresponds to the computation time. The relative velocity of the jet is illustrated in Fig. 2:

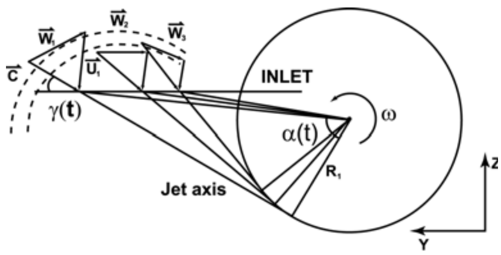

Fig. 2 *Jet path definition and relative velocity evolution*

Mathematically, we have according the Y and Z coordinates:

$$W_y(t)=-C\sin\left(\frac{\pi}{2}-\gamma(t)\right)-U\sin\left(\alpha(t)+\gamma(t)-\frac{\pi}{2}\right)$$

$$W_z(t)=-C\cos\left(\frac{\pi}{2}-\gamma(t)\right)+U\cos\left(\alpha(t)+\gamma(t)-\frac{\pi}{2}\right)$$

However, the method hereby described offers higher possibilities in mesh refinement and optimal adaptation as presented by Janetzky (Ref. 6).

THE UNSTEADY PRESSURE MEASUREMENTS

To measure the unsteady pressure distribution in a rotating Pelton bucket, an onboard acquisition system was used. 32 piezoresistive pressure transducers with a 10 bar range and a D=4.5mm were embedded in three buckets as shown in Fig. 3. This figure shows the position of the transducers as if they would all be in the same bucket. The signal is acquired simultaneously by using 8 acquisition boards of 4 channels each with a sampling frequency of 20 kHz. The A/D resolution is 12 bits and each channel has 64k-samples. The external communication is done using a 4 channel slip ring. Finally the acquisition is controlled via an ARCNET interface card and a sophisticated software developed with LABVIEW. A test rig was provided by our industrial partner VA TECH HYDRO SA. The functioning point for which the results are presented is $\psi_1/\psi_{1^\wedge}=1.08$ and $\varphi_{B2}/\varphi_{B2^\wedge}=0.95$.

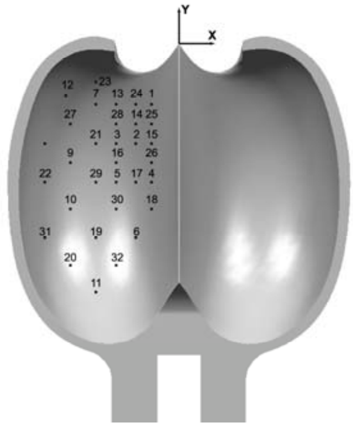

Fig. 3 Transducers's positioning

EXPERIMENTAL RESULTS

The measurements provided for the same functioning point as above can also be animated. Fig. 4 illustrates the evolution of the jet's pressure distribution in the rotating bucket expressed as a pressure coefficient Cp. The maximum value of this Cp is about 0.8, and not 1, because there were no P-ducers close to the stagnation point which is located on the median line. To verify the coherence of our measurements, it is possible with the plane wall theory, to recover the missing pressure. Indeed, by considering the impingement of the jet on successive inclined plates, and according to the wheel position when the strength is maximum, the Cp reaches an excepted value of 0.99.

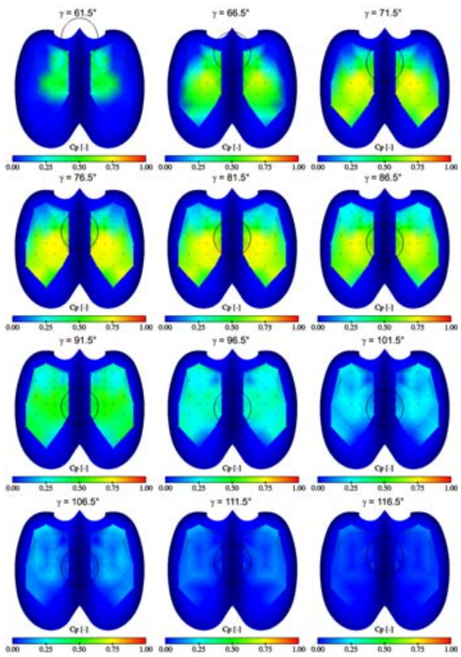

Fig. 4 Experimental pressure distribution

Furthermore, considering the transient averaged pressure signal, five kinds of signals can be isolated, each attached to a region of the bucket as illustrated in Fig. 5. Three of these five signals are described below. Notice that these signals are synchronized with the pictures.

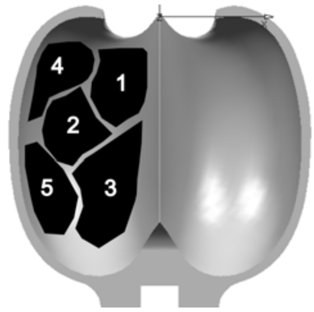

Fig. 5 Zones

Visualization

PD number 1: double pick (Fig. 6)

The jet hits the bucket, this increasing the pressure at a rotation angle of 116° which then diminishes with the rotation of the wheel. Then the jet is cut, but according to Bachman's kinetic analysis (Ref. 3), the remaining sheet of water conserves its initial kinetic energy involving that as the wheel rotates, this sheet of water is driven towards P-ducer number 1, applying a new charge on it and creating this second pick.

PD number 5: signal with a Dirac (Fig. 7)

A Dirac in the signal appears when the jet hits the bucket at each rotation. Because of the under-sampled signal in this zone, its intensity changes. Nevertheless, the intensity value is close to the expected pressure initially applied, and lets us assume that, according to the same Bachman kinetic analysis, each water particle issuing from a jet, possesses a kinetic energy equal to the initial jet. When the water goes into the bucket it creates a local impact like a stagnation point which gives this Dirac shape.

PD number 23: zone 4 (Fig. 8)

This P-ducer is located in the upper-left part of the bucket near the opening. The signal is characterized by a point shape and a belated starting (around 120°) in comparison with the P-ducers 1 and 5. The maximum Cp is at ~147°, that is once when the jet has been cut (134°) proving that the sheet of water keeps a kinetic energy which goes on to the concerned P-ducer as the wheel rotates, applying a non negligible charge.

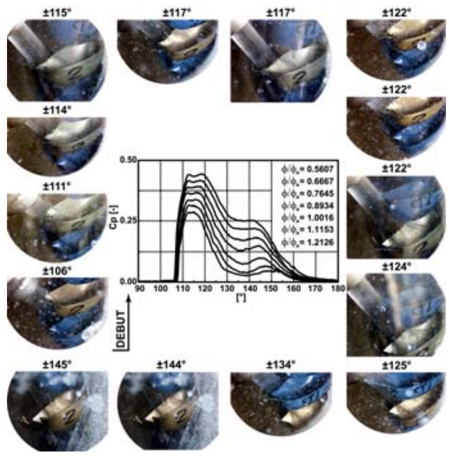

Fig. 6 P-ducer 1 signal: zone 1

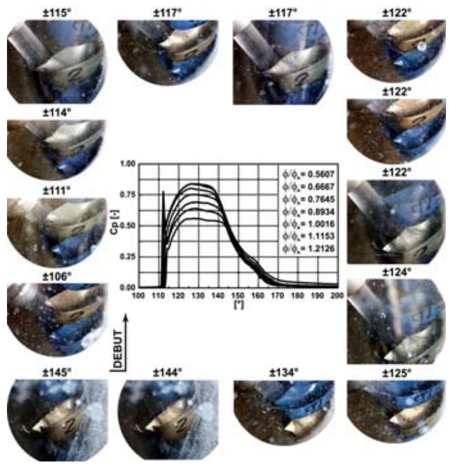

Fig. 7 P-ducer 5 signal: zone 2

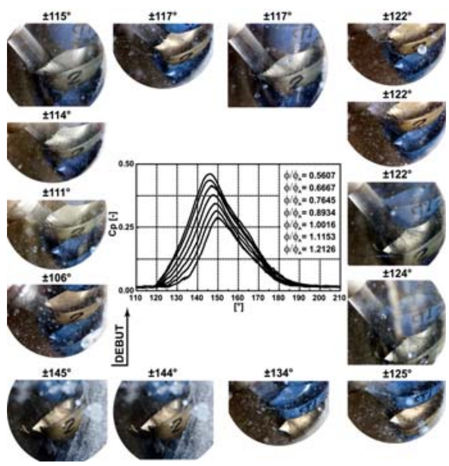

Fig. 8 P-ducer 23 signal: zone 4

NUMERICAL RESULTS

Due to the complexity of this non classical approach and in order to obtain coherent numerical results, we must proceed step by step. By optimizing the computational domain, the results presented in Fig. 9 are obtained. We can see clearly the evolution of the jet trough the domain, with a strong diffusion at the interface. Nevertheless, even if the method seems to be validated, the domain is still not optimized because the intersection of the jet with the median line does not correspond to the physical observations. Actually, we cannot compare these results the experimental ones.

CONCLUSION

The experimental and numerical analysis of free surface flows in a rotating Pelton bucket has been done. Numerically, an original method considering the relative path of the jet has been used to model the flow in a rotating bucket. It has been shown that this method is validated, whereas the volume of control is not well optimized, because the first interaction of the jet with the bucket median line does not correspond with the physical observations. On the other hand the measurements were completely successful, showing in a first phase that the maximum pressure coefficient obtained was coherent with the expectation. Then five kind of signals corresponding to certain parts of the bucket have been isolated. Three of these five signals have been described in this paper emphasizing the Bachman kinetic theory. The future challenges are now to superpose numerical signals with the experimental ones.

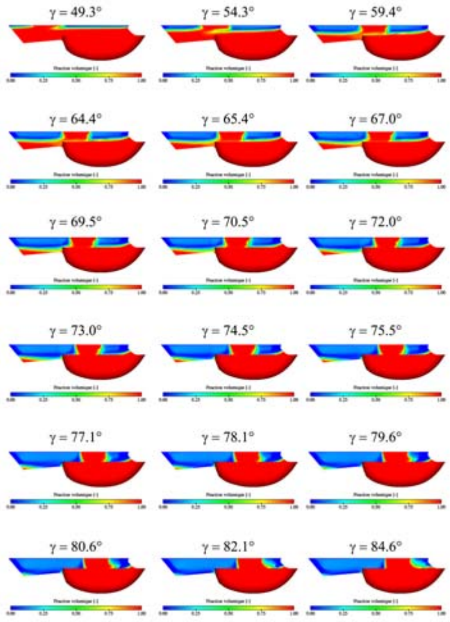

Fig. 9 Numerical jet evolution

ACKNOWLEDGEMENT

The authors would like to thank VA TECH HYDRO SA staff for their help and test rig. The study was possible thanks to the financial support of the Swiss Federal "Commission pour l'Innovation et la Technologie" (CTI Contract n°3808.2) and of VA TECH HYDRO SA. We also would like to thank Dr. Mohamed Farhat for his contribution in obtaining these excellent experimental results.

REFERENCES

Ref. 1 Ishii, M., 1975, "Thermo-Fluid Dynamic Theory of Two-Phase Flow", Collection de la Direction des Etudes et de Recherches d'Electricité de France, 1975, Eyrolles Publications.

Ref. 2 Hirt, C.W., Nichols, B.D., 1981, "Volume Of Fluid (VOF) method for the dynamics of free boundaries", Journal of Computational Physics, January 1981, Vol.39, pp. 201-225.

Ref. 3 Kvicinsky, S., Longatte, F., Avellan, Fr., Kueny, J.-L., 1999, "Free surface flow: experimental validation of the Volume of Fluid (VOF) method in the plane wall case", *Proceedings of 3rd ASME/JSME,* 8-23 July 1999, San Francisco, CA.

Ref. 4 Kvicinsky, S., Kueny, J.-L., Avellan, Fr., 2002, "Numerical and experimental analysis of free surface flow in a 3D non rotating Pelton bucket", *The 9th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery,* 10-14 July 2002, Hawaii, USA.

Ref. 5 Bachman, P., Schärer, C., Staubli, T., Vullioud, G., 1990, "Experimental flow studies on a 1-jet model Pelton turbine", *Proceedings of 15th I.A.H.R. Symposium on Modern technology in hydraulic energy production,* 11-14 September 1990, Belgrade, pp. 1-13.

Ref. 6 Janetzky, B., 2001, "Ein Verfahren zur Berechnung instationärer Stömungen mit freir Oberfläsche", *Universität Stuttgart, Institut für Strömungsmechanik und Hydraulische Strömungmaschinen,* 2001, Thesis work.

Ref. 7 CFX-4: Solver. AEA Technology, UK, 1997.