

37th IAHR WORLD CONGRESS 13-18 August, 2017 Kuala Lumpur, Malaysia

HYDRAULIC MACHINERY

EXPERIMENT STUDY OF GRAVELS STABILITY WITH BARELY SUBMERGED AND SUBMERGED FOLIAGE PLANTS

MUXI BAO⁽¹⁾, & JIANFENG TAO⁽²⁾

^(1,2) State Key Laboratory of Hydrology-Water Resources and Hydraulic Engineering, Hohai University,Nanjing,China, baomuxi1990@163.com
^(1,2) College of Harbor, Coastal and Offshore Engineering, Hohai University,Nanjing,China,

ABSTRACT

A better understanding of the flow characteristics through emulation plants would facilitate more effective river restoration and wetlands engineering. In this experiment, we investigate the flow structure around foliaged plants deployed in a channel with gravels on the bed under 3 conditions. Velocity data was collected with Acoustic Doppler Velocimetry (ADV) Vectrino (NORTEK). For the barely submerged flow and small bed slope case, the velocity speed up at the near-bed region. The associated Reynolds stress and TKE at the bed are larger than those for the submerged flow and small bed slope case. Compared to the barely submerged flow and large bed slope case, the difference in velocity, Reynolds stress and TKE are small. The results indicate that the presence of barely submerged foliage plants can enhance the channel bed instability and the water depth is a more important factor than bed slope that affect the gravels stability.

Keywords: Open-channel flow; vegetation patches; foliaged plants; channel stability.

1 INTRODUCTION

The vegetation as a kind of economical and effective, biological river environment management method has attracted extensive attention (Chao et al., 2015). As we know, although vegetation obstruct flow by increasing flow resistance and water depth, it reduces river bed erosion and water turbidity, increases bank stability, provides habitat for aquatic and terrestrial wildlife, and filters pollutants (Shields, 1991). It is beneficial to the healthy development of the river. Therefore, better understanding of the interaction between flow and vegetation is important.

Traditionally, study always focuses on the effects of long, uniform meadows on the bulk flow properties. For example, in early research, researchers like to use Manning roughness coefficient (n) or friction factor (f) to describe vegetation roughness. Kouwen and Fathi-Moghadam (2000) gave a table to estimate Manning's n value for flow through vegetation based on the mathematical model. LóPez and GarcíA (2001) find that Manning's coefficient is almost uniform for some critical plant density. Järvelä (2002) showed large variations in the friction factor (f) with depth of flow, velocity, Reynolds number, and vegetative density. Obviously, it is inaccurate, but it is easy and practical in concept model. To improve the resistant relationship, researchers focus more on drag coefficient (C_d). Hygelund and Manga (2003) find several relationships between drag coefficient and different factors. Lee et al (2004) find that the stem spacing and the Reynolds number are important parameters for the determination of vegetation drag coefficient. Wu (2008) gave the formulas of drag coefficient, C_d and equivalent Manning's roughness coefficient, nd which were derived by analyzing the force of the flow of non-submerged rigid vegetation in open channel. Panigrahi and Khatua (2015) study on the effect of emergent rigid vegetation on the prediction of the effective vegetal drag coefficient, C_d for various flow depth combinations has been explored. They use different empirical formulas with drag coefficient to describe resistant force due to vegetation.

However, vegetation is often found in patch or single rigid vegetation, not continuous meadows. The interaction between the patches and flow is actually an extremely complex problem of flow around a circular cylinder (Sumner, 2010). The high frontal width of vegetation patches not only increase flow resistant, but also has group trees effect (Pasche and Rouvé, 1985). It makes the interaction more complex. The existence of vortex also obstructs the flow (Cui and Neary, 2008). Therefore, it is important to understand the detailed characteristics of flow structure through vegetation patch. The use of a single parameter to describe the vegetation effect is not enough.

With the deepening of the research and advanced measuring instruments, the study of internal structure of flow has further develop. Liu et al. (2008) used discrete measurements at multiple locations within the canopy to develop velocity and turbulence intensity profiles and observe the changes in the flow characteristics as water travels through a vegetation array simulated by rigid dowels. Zeng and Li (2014) used physical experiment and numerical simulation to investigate flow through semi-rigid vegetation patches and quantified empirical equations of the adjustment lengths for mean velocity and Reynolds stress. Meire et al (2014) showed a clear flow characteristic around two porous patches, and by considering the interaction between neighboring patches, they identified a new positive feedback for lateral growth.

The researches above treat vegetation patch as a uniform patch from bottom to top, but it does not accord to fact. They ignore the influence of vegetation shape on the flow. The existence of foliage on some vegetation affects the drag and flow characteristics as compared to those of a group of cylinders. Within a vegetation domain, foliage will affect the velocity field. Around the portion of a plant with foliage the velocity will be retarded and deflected. At regions adjacent to the foliage portion the velocity may be increased. The significant local variation of the velocity can cause morphological change of the channel by sedimentation and erosion and affect the channel bed stability. Little works have been done on the study of the flow structure within a vegetation domain with foliage.

In the present work, experiments were carried out to measure the mean flow and turbulence structure of a vegetation domain consisting of model plants with foliage. The data were used to investigate the effect of water depth and slope to the mean flow and turbulence structure. Some gravels were placed around the plants for the investigation of their potential instability.

2 LABORATORY EXPERIMENTS

2.1 Laboratory experiments setup

Experiments were conducted at the Hohai University in a tilting flume. In order to allow a continuous recirculation of stable discharges the outlet and the inlet structures of the flume were connected to a hydraulic circuit. There is a tailgate at the downstream end of flume to control the water depth. The flume is 20m long by 0.31m wide. The velocity of flow was measured by Acoustic Doppler Velocimetry (ADV) Vectrino (NORTEK). Measurements at each point were taken for 75s at 75Hz, yielding 5000 samples to give plants reliable average value. The experimental region is 2m in the longitudinal direction and the initial section is located at 7.5m from the entrance section. We used semi-rigid artificial plants to model vegetation patches. The artificial are 0.165m tall, the crown of plants are 0.105m tall and 0.1m in diameter. The gravels average thickness is 0.025m; the details of experimental setup are shown as follows. The measurement locations are shown in Figure 1. A sketch of foliaged plant is shown in Figure 2.Table 1 summarizes the key parameters of three sets of experiment. The discharge Q was set to 60m3/h. The flume bed slope was set to 0 or 1/120. The value of water depth was set to 0.18m or 0.24m at the upstream.







Figure 2. Sketch of foliaged plant.

Table 1. Summary of experimental conditions.								
Case	Q(m ³ /h)	h(m)	S ₀	U _{av} (m/s)	R _e	Fr	W/h	h/hv
1	60	0.18	0	0.3	5376	0.05	1.72	1.09
2	60	0.18	1/120	0.3	5376	0.05	1.72	1.09
3	60	0.24	0	0.22	5376	0.02	1.29	1.45

 U_{av} is the cross-sectional averaged velocity at leading edge, R_e is the Reynolds number ($U_{av}h/ U$), F_r is the Froude number (U_{av}^2/gh) , W/h is the aspect ratio (width/depth), and h/hv is the water depth/ artificial plant height.

2.2 General observations

Before measuring the characteristics of flow, we did a visual inspection of the flows. Test was divided into two parts: with barely submerged vegetation and without vegetation. Due to the limit of flow velocity, gravels cannot move when flume bed is paved with gravels, so the gravels were placed within circular areas of radius of 5cm around the plant stem. Water discharge Q=70m3/h, the flume bed slope was fixed at 1/300, water depth was set to vegetation height, other details of experiment was the same as above.

By the contrast test of with vegetation and without vegetation, we found that for the scenario of no vegetation, the gravels remain stable and exhibit no motion. The shear stress at the channel bed was thus below the critical shear stress for the gravels. In the presence of vegetation, some gravels became unstable and moved downstream. Through the observation of condition with vegetation, we found that for the whole region gravels at the downstream move first, and for each patch the gravels at the leeward side of the plant moved first. Vegetation is often found to enhance sediment accretion and reduce sediment erosion. This phenomenon shows that the vegetation patch with foliage has different effect on flow and turbulence structure than those of a group of cylinders.

EXPERIMENTAL RESULTS 3

This presentation of the experimental results is divided into two parts: Mean flow characteristics of 3 cases, and Turbulence intensity characteristics of 3 cases.

3.1 Mean flow characteristics

Figure 3 depicts the vertical profiles of the longitudinal velocity which are located on the longitudinal centerline section of the channel behind the plants of 3 cases. The location of each vertical profile is shown in figure 1. Figure 3a is the measured vertical profiles of longitudinal velocity for case 1, figure 3b is for case 2 and figure 3c is for case 3.

Since the plants have irregular cross sections, the results have a lot of scattered points. There is a general trend of this profiles, the lowest velocity occurs immediately behind the VP. The velocity behind the plant is reduced due to the shielding effect of the plant. A wake region is formed there with low velocity and pressure and high energy loss with the presence of vortices. Further downstream, the velocity is higher. The peak velocity occurs at a level just beneath the foliage part of the plant as the shielding effect is absent. The velocity decreases towards the bed due to the frictional effect offered by the gravels and the bed. The velocity measurements were not up to the water surface because of the limitation of the Vectrino profiler. In case1 and case2 near the water surface there should be a high velocity region due to the absence of shielding effect.



Figure 3. Measured vertical profiles of longitudinal velocity for 3 cases.

Figure 4 is the velocity profiles behind plant (x=1.44m) of different cases. There is a little displacement of case 2, because when the flume bed has a slope the water depth change along the section, and we treat water surface as the baseline, so there is a vertical displacement of two velocity profiles. In addition to the vertical displacement, the velocity profiles for case 1 and case 2 are almost the same. In case 3, velocity is much smaller than case 1 and 2 at bottom. This is mainly because in case 1 and case 2, very little flow can pass above the top of the plant, thus the flow in the upper layer tends to move downward, case 3 are submerged conditions, the flow blocked by the plant can pass above the plant crown, so the flow velocity accelerate gradually above the plant, less flow pass below the plant.



Figure 4. Longitudinal velocity profile behind plant (x=1.44m) of different cases.

Figure 5 shows the contour plots of velocity components w, as well as vector plot of velocity u-w (m/s) in centerline x-z plane for 3 cases. The measured results of 3 cases show the same trend. At the upstream region(x=0.4), there is a clear downward speed. In the gap region (x=0.66m) between two adjacent plants, a clear upward speed is observed near the bottom. In the wake region of a plant (x=0.9m-1.1m), the blocking effect of the plant causes a significant reduction in the longitudinal velocity. The reduced flow there tends to move towards the upper layer, resulting in a relatively large vertical velocity (w>0.03cm). In the second gap region (x=1.14m) downstream of the wake region, the flow characteristics are similar to those in the first gap region (x=0.66m). The extent of increase in longitudinal velocity, and the magnitude of upward flow towards the upper layer are however smaller due to the presence of the wake region immediately upstream. This is mainly because in the wake region the flow profile is largely different from that at the approaching region.

Comparing case1 and case 2, the trend of velocity components w and velocity u-w are the same, and when the flume bed slope become large, the value of velocity component w decreases. The trend of velocity components w and velocity u-w are still the same in case2 and case 3, when the water depth become large, the value of velocity component w decreases.

3.2 Turbulence intensity characteristics

Figure 6 is the Reynolds stress and TKE (Turbulent Kinetic Energy) profiles behind plant (x=1.44m) of different cases. As mentioned above, there is a little displacement of the two Reynolds stress profiles for case 1 and case 2. In case 1 and case 2, the maximum Reynolds stress happens near the bed, and as for case 3, it happens both near the bed and at the top of the plant crown. The value of maximum Reynolds stress near the bed of case 2 is a little larger than case 1. In case 3, Reynolds stress is much smaller than case 1 and 2 at bottom. It is because the velocity gradient near flume bed of case 3 is smaller than case 1 and case 2, as shown in figure 5. In case 1 and case 2, the maximum TKE happens from the bottom to the plant crown. In case 3, the maximum TKE happens at the top of the plant crown, and the average TKE at bottom of case 1 and case 2 is almost 3 times bigger than that of case 3. This phenomenon can prove that the gravels are unstable in barely submerged condition. When it is barely submerged condition, upper layer flow is blocked by the plant, most flow is forced downward to bottom layer. A strong downward velocity present near the bed immediately will generate large vortex near bed. Therefore, both the turbulence kinetic energy and Reynolds stress become larger, and the flow drag force to gravels become bigger, and the gravels become easy to move.



Figure 5. Contour plots of velocity w (m/s) and flow field in x-z direction.



Figure 6. Reynolds stress and TKE (Turbulent Kinetic Energy) profiles behind plant (x=1.44m) of different cases.

Figure 7 shows the turbulence kinetic energy (TKE) field on the central longitudinal plane. In case 1, the maximum TKE occurs within the wake region, at x=0.97m, z = 0.035m, at the level of the bottom of vegetation crown. In the wake region, the mean velocity is significantly reduced and small scale vortices are formed, leading to the increase in turbulence. Another peak region of TKE exists at the upper part of the gap region just downstream of the approaching flow. This is the region where the flow is diverted towards the upper unobstructed layer and the turbulence is significant. In the second gap region (x=1.44m), similar flow characteristics have been observed. A peak TKE also occurs at the bottom part of the gap, but the strength is much smaller. This may be due to, beyond the wake region, the flow is more steady and the turbulence

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

becomes weaker. In case 2, the phenomenon of maximum TKE is a little different. The maximum TKE happens both at the bottom of plants crown and at the top of plants crown. It is because when the flume bed has a slope, the water depth getting larger along the section, the water depth at x=0.97 of case 2 is larger than case 1, so the ADV can measure part of the high velocity region near the water surface. In case 3, the maximum TKE just happens at the top of plants crown. It is due to the little velocity increase at the bottom.



Figure 7. Contour plots of measured TKE (m2/s2) and vector plot of velocity u-w (m/s) in centerline x-z plane of 3 cases.

Figure 8 shows the Reynolds stress component $-\langle u'w' \rangle$. The spatial distribution of the Reynolds stress is similar to that of the TKE. The high velocity causes pronounced energy exchange, and leads to high Reynold stress. At the wake region (x~0.97-1.1m), the Reynolds stress is negative. This is mainly due to the blockage of the foliaged plant which generates a velocity profile with the maximum velocity occurred below the bottom level of the foliaged portion of the plant, and a negative velocity gradient is developed there.



Figure 8. Contour plot of measured Reynolds stress -<u'w'> (m2/s2) and vector plot of velocity u-w (m/s) in centerline x-z plane of 3 cases.

4 DISCUSSION

Previous works generally consider that the presence of vegetation will enhance sediment accretion and reduce sediment erosion (e.g. Shields, 1991). For vegetation patches, sediment accretion will occur within the patches and sediment erosion will occur at the edges of the patches (Bouma et al., 2009). In the present work, 3 scenarios with different water depth and bed slope were investigated. For the barely submerged flow and small bed slope case, the velocity speed up at the near-bed region, and the associated Reynolds stress and TKE at the bed are larger than those for the submerged flow and small bed slope case. Compared to the barely submerged flow and large bed slope case, the difference in velocity, Reynolds stress and TKE are small. The results indicate that the presence of barely submerged foliage plants can enhance the channel bed instability, and the water depth is a more important factor than bed slope that affect the gravels stability.

The possible occurrence of this scenario includes steep streams, rivers or gabion channels with vegetation. For a channel of steep slope, the water depth will be small and the flow velocity will be high. It is more likely that the vegetation will be emergent.

5 CONCLUSIONS

The flow structure within a vegetation domain with foliaged plants in a channel with gravels on the bed was investigated experimentally. The experimental results show that there is a large local variation of the flow field. Within the gap region between two plants the velocity and TKE are increased. At the wake behind a plant, the velocity and TKE is reduced. For the barely submerged case, a velocity speed up exists at the near-bed region. The associated Reynolds stress and TKE closed to the bed are large. For submerged case, the velocity speed up less than barely submerged case at the near-bed region. The associated Reynolds stress and TKE closed to the bed are smaller than those for the barely submerged cases, and the maximum value happens at the top of plants crown. The implication is that water depth is an important factor that affect the gravels stability in a vegetation domain with foliaged plants in a channel.

REFERENCES

- Chao, W., Zheng, S. S., Wang, P. F., & Hou, J. (2015). Interactions between Vegetation, Water Flow and Sediment Transport: a Review. *Journal of Hydrodynamics, Ser. B*, 27(1), 24-37.
- Kouwen, N., & Fathi-Moghadam, M. (2000). Friction Factors for Coniferous Trees Along Rivers. *Journal of Hydraulic Engineering*, 126(10), 732-740.
- Järvelä, J. (2002). Flow Resistance of Flexible and Stiff Vegetation: a Flume Study with Natural Plants. *Journal of Hydrology*, 269(1-2), 44-54.
- Hygelund, B., & Manga, M. (2003). Field Measurements of Drag Coefficients for Model Large Woody Debris. *Geomorphology*, 51(1), 175-185.
- Lee, J. K., Roig, L. C., Jenter, H. L., & Visser, H. M. (2004). Drag Coefficients for Modeling Flow Through Emergent Vegetation in the Florida Everglades. *Ecological Engineering*, 22(4-5), 237-248.
- Panigrahi, K., & Khatua, K. K. (2015). Prediction of Velocity Distribution in Straight Channel with Rigid Vegetation . *Aquatic Procedia*, 4, 819-825.
- Sumner, D. (2010). Two Circular Cylinders in Cross-Flow: a Review. *Journal of Fluids & Structures*, 26(26), 849–899.
- Pasche, E., & Rouvé, G. (1985). Overbank Flow with Vegetatively Roughened Flood Plains. *Journal of Hydraulic Engineering*, 111(9), 1262-1278.
- Jie Cui, & Vincent S. Neary. (2008). Les Study of Turbulent Flows with Submerged Vegetation. *Journal of Hydraulic Research*, 46(3), 307-316.
- Liu, D., Diplas, P., Fairbanks, J. D., & Hodges, C. C. (2008). An Experimental Study of Flow Through Rigid Vegetation. *Journal of Geophysical Research Atmospheres*, 113(F4), 66-83.
- Zeng, C., & Li, C. W. (2014). Measurements and Modeling of Open-Channel Flows with Finite Semi-Rigid Vegetation Patches. *Environmental Fluid Mechanics*, 14(1), 113-134.
- Meire, D. W. S. A., Kondziolka, J. M., & Nepf, H. M. (2014). Interaction between Neighboring Vegetation Patches: Impact on Flow and Deposition. *Water Resources Research*, 50(5), 3809–3825.
- Shields Jr F D. (1991). Woody Vegetation and Riprap Stability along the Sacramento River Mile 84. 5-119. *Water Resources Bulletin*, 27(3): 527-536.
- Bouma T J, Friedrichs M, Wesenbeeck B K V, Temmerman, S., Graf, G., & Herman, P. M. J. (2009). Density-Dependent Linkage of Scale-Dependent Feedbacks: a Flume Study on the Intertidal Macrophyte Spartina Anglica. *Oikos*, 118(2):260–268.

CAVITATION EROSION WITH CONSIDERING VISCOSITY OF AERATED FLOW

YU WANG, JIANHUA WU, FEI MA & KUNPENG SU

College of Water Conservancy and Hydropower Engineering, Hohai University, Nanjing, China, jnwangyu@hhu.edu.cn

ABSTRACT

Air entrainment has been a simple and economic technology of protecting hydraulic facilities from cavitation erosion. However, the mechanisms of air entrainment against cavitation erosion are yet to be fully elucidated. In the present work, the experiments about the effect of air pressure, air bubble size and temperature of airwater mixture on cavitation erosion were conducted by means of a vibratory apparatus. Moreover, changes in viscosity of air-water mixture were investigated. The findings suggested that cavitation erosion decreases with high air pressure, small entrained air bubbles and low temperature. The increase of viscosity in air-water mixture upon air entrainment is the dominant mechanism of the observed suppression of cavitation damage.

Keywords: Air entrainment; air bubble size; viscosity; cavitation erosion.

1 INTRODUCTION

Cavitation erosion is a common phenomenon for the working of hydraulics machinery. Air entrainment is a kind of effective and economic means to protect the overflow surface of spillways against cavitation erosion. In earlier years, air entrainment was successfully used in the Grand Coulee Dam in America and this technology is applied in almost all the release works of high dam for the decrease of cavitation erosion (Wu et al., 2006). Laboratory tests have been conducted to confirm that relatively small quantities of entrained air in water can significantly eliminate cavitation erosion (Peterka, 1953; Wu et al., 2011).

However, opinions differ regarding the mechanisms that entrained air could reduce cavitation erosion. It was proposed that entrained air bubbles would suppress pressure transient and oscillation to relieve cavitation (Martin, 1974; Lee et al., 2002). Alternatively, some ideas suggested entrained air bubbles act as large nuclei, and increase of volume and air content of cavitation bubbles would buffer the collapse (Reisman et al., 1997). With the aid of high-speed photography, the investigators contributed the positive role of air bubbles to damping of shock wave (Russell et al., 1974; Hammitt, 1980), and the deflection of micro-jets (Xu et al., 2010; Goh et al., 2014). Meanwhile, it was discovered that air entrainment would change water properties, especially tensile strength, relative density and compressibility of air-water mixture (Zhang et al., 2013).

Numerous researches have proved air concentration is a key concern for reducing cavitation erosion, and entrained air bubble size and liquid temperature may influence cavitation erosion (Chen et al., 2003; Hattori et al., 2006). As far as we are aware of, detailed studies about effect of different air bubbles, sizes and liquid temperature on erosion in aerated water are rarely reported. Furthermore, the mechanisms of cavitation erosion reduction still present major problems in different aerated conditions.

In the present work, a new method for air entrainment was developed and air bubbles were distributed in test water. Cavitation tests were conducted when the test conditions varied with different air pressure in the supply line, different orifice diameter of holey sheet and temperature of air-water mixture. The correlation between liquid viscosity and air concentration was shown and discussed.

2 EXPERIMENTAL SETUP AND METHODOLOGY

2.1 Experimental setup

The experiments were carried out in the High-speed Flow Laboratory of Hohai University, Nanjing, China. Figure 1 shows the schematic diagram of the special vibratory cavitation test facility.

The basic equipment is a vibratory apparatus with the power of 1100W, the vibration frequency of 19.6 \pm 0.5 kHz, and a peak-to-peak amplitude of 50µm according to GB/T 6383-2009 test method. The specimen was fixed in the tip of a magnetostrictive ultrasonic horn, and its surface was immersed in water with a depth of approximate 2mm. The cooling bath system keeps the test liquid at stable temperature.

In order to introduce air into water, a special device with micro-scale orifice diameters was designed to form bubbles in water. The 0.3-mm-thick nickel alloy sheet, where orifices were drilled using laser beams, was fixed between the vessel and an air chamber. The air would send out through the sheet when air pressure is high enough, so that the entrained air bubbles would distribute in water. The air pressure in the supply line was regulated with a relief valve, stabilized by an air tank and read on a precision gauge.



Figure 1. Schematic diagram of the special vibratory cavitation test facility

_

Table 1 Chemical composition (mass %) of specimen								
Chemical composition	С	Cr	Mn	Ni	Р	S	Si	
1045 carbon steel	0.42 - 0.50	≤0.25	0.50 - 0.80	≤0.25	≤0.035	≤0.035	0.17 – 0.37	

The test specimens, made of carbon steel specified in the American Society for Testing and Materials (ASTM) standard 1045, were 16mm in diameter, the density was 7.85 g/cm³, and the chemical composition of specimen is shown on the Table 1, furthermore, the surfaces were polished mechanically before tests.

The testing was continued for 240min in order to achieve a constant weight loss process. The specimens were weighed at 20-min and 40-min intervals for 0-120min and 120-240min, respectively, which aimed to gain a more accurate shape of erosion-time curve. It was washed by an ultrasonic cleaner and dried before and after each test, then weighed using a lab electronic balance within the precision of 10^{4} g. Viscosity measurement of the entrained air water was made in a rotating cylinder viscometer with rotational speed of 12 r/min, the inner and outer cylinders have a diameter of 2.5cm and 3cm, respectively. The air concentration in vicinity of the specimens was recorded using the CQ6-2005 aeration apparatus (made by the China Institute of Water Resources and Hydropower Research (Beijing)), which is a resistance-type of instrument that collects and processes the aeration data on walls by sensors and a microcomputer. Its sampling rate and period are 1020Hz and 10s, respectively, and the error is $\pm 0.3\%$.

2.2 Experimental methodology

As listed in Table 2, eight experimental cases were conducted. Case No.0, as a reference, was a test without air entrainment, while another seven referred to the effects of different conditions of air entrainment on cavitation erosion. For Case No. 1—3, the three sheets with different orifice diameters (*d*) had the same total area of orifices (A=0.79mm²), therefore, the numbers of orifices were 10000, 2500 and 400, corresponding to d = 10, 20 and 50µm, respectively. Case No. 3—5 discussed the effect of air pressure (*p*) from 20kPa to 40kPa with the same orifice diameter and temperature. Case No. 4, 6 and 7 investigated the temperature under the same orifice diameter entrained air concentration.

Case No.	<i>d</i> (µm)	р (kPa)	T (°C)	Remarks
MO	0	0	25	
M1	10	30	25	No.0: for reference
M2	20	30	25	orifice diameter
M3	50	30	25	same orifice diameter
M4	50	20	25	different air pressure
M5	50	40	25	same orifice diameter
M6	50	20	10	and air pressure but different temperature
M7	50	20	40	

 Table 2 Chemical composition (mass %) of specimen

3 EXPERIMENTAL RESULTS AND DISCUSSION

3.1 Effect of air pressure on cavitation erosion

Figure 2 shows the effect of air pressure on weight loss (*WL*) against *t*. From the figure, it is evident that at this temperature (T=25°C), the cases of entrainment resulted in less cavitation erosion, which is in agreement with the traditional knowledge that entraining air into water could effectively reduce the intensity of cavitation erosion. The cavitation erosion reductions were 19.3%, 23.2% and 33.7% comparing with case No.0, corresponding to *p* = 20, 30 and 40kPa, respectively.

The average air concentration in aerated water is defined as $C_a = Q_a / (Q_a + Q_w)$, i.e., the ratio of the air discharge entrained through the aeration holes and the total discharge of both air and water. We know, for the conditions of same temperature and the orifice diameter, the larger the air pressure, the higher the air concentration is. Therefore, Figure 2 indicates cavitation at higher air concentration leads to less cumulative weight losses.



Figure 2. Variation of *WL* against *t* for different air pressure

3.2 Effect of orifice diameter on cavitation erosion

As is shown in Figure 3, when air is introduced into the water, cavitation damage becomes more alleviated as orifice diameters become smaller. For the sheets of case No.1—3, the total areas of micro orifices were all identical, now supposing the sheets of different orifice diameters bring about the same number of air bubbles at the same air pressure, therefore, the sheets with larger orifice diameter will result in a higher air concentration. However, an interesting phenomenon occurred. Cavitation erosion reduction of case No.1 was better comparing with case No.2 and No.3 even when case No.1 produced lower air concentration. It seems to imply there is another factor which really dominates the cavitation. Figure 3 shows the effects of air bubble size which is a much more dominant factor for cavitation erosion. This might be a significant discovery in exploring the mechanism of air entrainment against cavitation erosion. However, the mechanism of the better effects of smaller air bubbles on less erosion should be further investigated.



Figure 3. Variation of WL against t for different orifice diameter

3.3 Effect of temperature on cavitation erosion

Figure 4 shows the effect of temperature on cavitation erosion in aerated water. It can be seen that increasing temperature significantly aggravates cavitation damage within the temperature range in this investigation, which tallies with previous pertinent results of Hattori et al. (2002) who proposed that for aerated water, erosion increases as temperature increases from 10°C to 45°C.

Previous researchers (Plesset, 1972; Preece, 1979) attributed the effect of temperature on cavitation erosion to several mechanisms, including (i) changes in liquid properties such as viscosity, surface tension, vapor pressure and density, (ii) thermodynamic effects on bubble growth and collapse, (iii) changes in the dissolved gas content of liquid, and (iv) changes in the material properties. However, the detailed mechanism of temperature on cavitation erosion for aerated water is not yet clear.



Figure 4. Variation of WL against t for different temperature

3.4 Changes of viscosity in aerated water

To further investigate changes of liquid properties in aerated water, we measured air concentration and viscosity for 27 cases (p=20, 30 and 40kPa, d=10, 20 and 50µm, T=10, 25 and 40°C, respectively). Figure 5 (a), (b) and (c) show the variety of viscosity in aerated water for different air pressure, orifice diameter and temperature, respectively. Figure 5 (a) shows liquid viscosity increases within the limitations of 1.2 to 1.7 mPa \cdot s, 1.3 to 2.1 mPa \cdot s and 1.5 to 2.2 mPa \cdot s, corresponding to air concentration in the range of 1.5% to 3.3%, 2.2% to 5.0% and 3.0% to 5.9% for p=20, 30 and 40kPa, respectively. As a whole, viscosity increases with the increase of air concentration for different air pressure. Moreover, larger air pressure leads to larger air concentration, which brings a decrease in weight loss according to 3.1. Therefore, the increase of viscosity may restrain cavitation erosion.

The general trend of Figure 5 (b) indicates liquid viscosity increases with increase of air concentration, however, different air bubble sizes hardly influence liquid viscosity obviously. Previous researches deemed that the size of air bubble is an important parameter that dominates the direction of micro-jets from the collapse of a cavitation bubble (Xu et al., 2010). Here, we can suggest that smaller air bubbles may induce the deflection of the micro-jet when a cavitation bubble collapses, and then lead to less cavitation erosion.

Figure 5 (c) shows the proportion of liquid viscosity to air concentration for different temperature. From this figure, it could be clearly seen that the lower temperature could produce higher liquid viscosity at different air concentration.

As mentioned above, the increase of liquid viscosity is a plausible explanation of effect of aerated water on cavitation erosion. Figure 6 shows the relation between liquid viscosity and weight loss for eight cases listed in Table 2. It is evident that weight loss is monotonically declining with the increase of viscosity. Therefore, it could be explicitly stated that changes of liquid viscosity may be a mechanism of less erosion in aerated water since entraining air changes in the values of liquid viscosity. Generally speaking, when the liquid viscosity increases, the pressure intensity of shock wave and the velocity of micro-jet due to the collapse of cavitation bubbles will be attenuated, and then, mitigates cavitation erosion.



Figure 5 Relationship between air concentration and viscosity considering (a) air pressure; (b) orifice diameter of alloy sheets; (c) temperature

4 CONCLUSIONS

A device with micro-scale orifice diameter was specially designed to entrain air bubbles into water. The experiments of aerated water demonstrate that higher air pressure, smaller air bubbles and lower temperature have better effects on cavitation erosion reduction. The important knowledge of this present work is that changes of liquid viscosity may be a reasonable mechanism of reducing cavitation erosion in aerated water as entraining air changes the values of liquid viscosity.

ACKNOWLEDGEMENTS

This research has been financially supported by the Fundamental Research Funds for the Central Universities (Grant No. 2016B09914), Graduate Research and Innovation Projects in Jiangsu Province (KYZZ16_0280).

REFERENCES

Chen X.P., Xi R.Z., Shao D.C. & Liang B. (2003). New Concept of Air Entrainment Effect on Mitigating Cavitation Damage (In Chinese). *Journal of Hydraulic Engineering*, 34(8), 70-74.

Goh B.H.T., Ohl S.W., Klaseboer E. & Khoo B.C. (2014). Jet Orientation of a Collapsing Bubble near a Solid Wall with an Attached Air Bubble. *Physics of Fluids*, 26(4), 90-96.

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

Hammitt F.G. (1980). Cavitation and Multiphase Flow Phenomena. Mcgraw-Hill, New York, USA, 250-251.

Hattori S., Goto Y. & Fukuyama T. (2006). Influence of Temperature on Erosion by a Cavitating Liquid Jet. *Wear*, 260, 1217-1223.

- Hattori S. & Tanaka Y. (2002). Influence of Air Content and Vapor Pressure of Liquids on Cavitation Erosion. *Transaction of JSME*, 68(675), 3080-3086. (In Japanese).
- Lee T.S. & Ngoh K.L. (2002). Air Entrainment Effects on the Pressure Transients of Pumping Systems with Weir Discharge Chamber. *Journal of Fluid Engineering*, 124(4), 1034-1043.

Peterka A.J. (1953). The Effect of Entrained Air on Cavitation Pitting. *Proceedings of Minnesota International Hydraulics Convention*, Minnesota, Minneapolis, USA, 507-518.

Plesset M.S. (1972). Temperature Effects in Cavitation Damage. *Journal of Basic Engineering*, 94(3), 559-566.

Preece C.M. (1979). Treatise on Materials Science and Technology. Erosion Academic, 16, 1-450.

Martin C.S. & Padmanabhan M. (1974). *The Effect of Entrained Air on Pressure Transients*. Georgia Institute of Technology, Atlanta, USA, 1-209.

Reisman G.E., Duttweiler M.E. & Brennen C.E. (1997). Effect of Air Injection on the Cloud Cavitation of a Hydrofoil. *Proceedings of ASME Fluids Engineering Division Summer Meeting*, London, England, UK, 1-10.

Russell S.O. & Sheehan G.J. (1974). Effect of Entrained Air on Cavitation Damage, *Canadian Journal of Civil Engineering*, 1(1), 97-107.

Wu J.H. & Luo C. (2011). Effects of Entrained Air Manner on Cavitation Damage. *Journal of Hydrodynamics*, 23(3), 333-338.

- Wu J.H., Wu W.W. & Ruan S.P. (2006). On Necessity Of Placing An Aerator In The Bottom Discharge Tunnel At The Longtan Hydropower Station. *Journal of Hydrodynamics*, 18(6), 698-701.
- Xu W.L., Bai L.X. & Zhang F.X. (2010). Interaction of a Cavitation Bubble and an Air Bubble with a Rigid Boundary. *Journal of Hydrodynamics*, 22(4), 503-512.
- Zhang H.W., Liu Z.P., Zhang D. & Wu Y.H. (2013). Study on the Sound Velocity in an Aerated Flow. *Journal of Hydraulic Engineering*, 44(9), 1015-1022. (In Chinese).

IMPROVEMENT OF FLOW PATTERN IN MULTI-UNIT PUMPING STATION WITH SIDE-INTAKE BASED ON CFD NUMERICAL SIMULATION

LUO CAN⁽¹⁾ & LIU CHAO⁽²⁾

(1,2) College of Hydraulic & Power Engineering, Yangzhou University, Yangzhou Jiangsu, China. acan19880221@126.com, liuchao@yzu.edu.cn.

ABSTRACT

The pumping station which has 16 intake sumps for pumps was set combined with the ship lock together. The ship lock was used as the forebay of pumping station. 14 axial flow pump units were installed in middle of the pumping station. The single discharge of each unit is $1.5m^3/s$. The designed water level is 1.76m and the total designed discharge is $20m^3/s$ for irrigation. The vortices and recirculation flow were often found in the forebay and sump of the pumping station which resulted in vibration and decrease efficiencies of the pump system. The goal of the study is to improve the flow pattern in the intake sump by means of some measures. The multi-segment non-continuous diversion pier, the back baffle and the column were used to eliminate the vortices. To predict the flow fields in the forebay and sump before and after improvement with different measures, the numerical simulation was conducted using the software Fluent. The results show that the recirculation zones get smaller or disappear. The velocity circulation near the back wall is significantly reduced. The flow velocities at the inlet of the pumps are uniform and symmetrical. The velocity distribution on each section is more uniform. The flow pattern is well improved for the operation of pump.

Keywords: Pumping station; sump; side intake; recirculation; numerical simulation.

1 INTRODUCTION

The pumping station which has 16 intake sumps for pumps was set combined with the ship lock together. The ship lock was used as the forebay of pumping station (see Figure 1). 14 axial flow pump units were installed in middle of the pumping station. The single discharge of each unit is $1.5m^3/s$. The designed water level is 1.76m and the total designed discharge is $20m^3/s$ for irrigation. The vortices and recirculation flow were often found in the forebay and sump of the pumping station, which resulted in vibration and decrease efficiencies of the pump system (Rajendran, et al., 1999; Rajendran, et al., 2000; Ansar, et al., 2001; Choi, et al., 2010; Feng, et al., 2012; Liu, et al., 2016). The objective of the study is to improve the flow pattern in the intake sump by means of some measures. To predict the flow fields in the forebay and sump before and after improvement with different measures, the numerical simulation was conducted using the software Fluent. Three different measures were added in the sumps to improve the flow pattern:

- (1) Scheme 1, sills, which is commonly used in the forward forebay (1-3).
- (2) Scheme 2, piers, diversion measure such as diversion pier, guide wall, baffle and diversion grid etc. (4-6).
- (3) Scheme 3, columns, the column and pressurized water plate (7-8).

2 NUMERICAL SIMULATION

2.1 Governing Equation

For the complex flow in the pumping station and large Reynolds number, the continuity and momentum equations can be written in Cartesian coordinate system (x, y, and z) for incompressible turbulent flow such as:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0$$

$$\frac{\partial}{\partial x_i} (\rho \overline{u_i u_j}) = -\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_i} [\mu(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}) - \rho \overline{u_i u_j}] + F_i$$
[2]

where, x_i , x_j (i, j=1, 2, 3) represent coordinate x, y and z, respectively; u_i and u_j represent the average velocity u, v, w, respectively; p is the average pressure; μ is the dynamic viscosity coefficient; F_i is the body force; ρ is the density.

2.2 Computational domain and mesh

The layout and dimensions of the pumping station are shown in Figure 1. The 14 pumps of which each pump has an individual sump are marked with numbers 1-14 along the flow direction. The distance between the center lines of two pumps is 3.5 times of the diameter of suction pipe. The diameter of the suction pipe is indicated by *D*.



Figure 1. 3-Dimension view of the pump intake sumps

The computational domain includes the forebay, the intake sump and the suction and discharge bent pipe which is shown in Figure. 3. The scale of computational model and prototype station is 1:1. The bottom level of the pump station is -2.00m and the intake water level is 1.76m, which gives the water depth of 3.76m. The bottom clearance between bell mouth of the suction pipe and the bottom was set as 0.8*D*.

Considering the small size of several zones and complicated structure, the unstructured grids were adopted to discretize the computational domain. Part of the mesh is shown in Figure 2.

The independence analysis of grid quantity to the hydraulic losses of the intake flow was performed. The results showed no significant change in the losses with grid quantity greater than 1.6×10⁶, which was selected for the meshing of the calculation domain.



Figure 2. Mesh for calculation of original

2.3 Boundary conditions

As shown in Figure1, the entrance of the upper lock head was chosen as the inlet of computational domain and the boundary condition of inlet was set as velocity-inlet of which the magnitude is 0.4m/s calculated with the given designed discharge. The velocity distribution of the inlet was assumed uniform and the hydraulic diameter of the inlet was calculated as 4.9m. The exit of the discharge bent pipe was chosen as the outlet of computational domain, which was set as outflow condition.

No slip boundary conditions and wall functions were used for the solid walls. The velocity magnitude of *x* component, *y* component and *z* component was 0. Considering that the Reynolds number near the wall was small, the law of the wall was applied. The free water surface was set as symmetry ignoring the heat exchange and shear stress between the water surface and air. The Reynolds averaged Navier-Stokes equations (RANS) and the Realizable *k*- ε turbulent model SIMPLE algorithm were selected for numerical simulation. The second-order upwind differencing scheme was used for the momentum, turbulent kinetic energy and turbulent dissipation rate equations. In the process of calculation, the residual absolute of the monitoring parameters (continuity, *v_x*, *v_y*, *k* and ε) is 1.0×10⁻³.

3 THE CALCULATION RESULTS OF ORIGINAL SUMP

3.1 The pump sump of original

Original design of the pump sump is shown in Figure 1. The length and the width of the sump are 10*D* and 3*D*, the maximum water depth is 4*D* respectively. The back wall is arc shaped. The original sump design is shown in Figure 3. The zero point of the coordinates was set at the bottom and middle of the forebay, leading of the division pier for the calculation of flow fields.



Figure 3. Original design of pump sump

3.2 The flow pattern of original design

The contours of velocity component $v_{x at}$ upper section (Z=3.5*D*) near the water surface of the pump sump is shown in Figure 4. The large-scale recirculation zones flowing clockwise were found in the sumps 1~11 of which the flow pattern. There was anticlockwise recirculation around the suction pipe which affects the pump operation directly. The strength of the recirculation decreased gradually from sump 1 to sump 11.



Figure 4. V_x contours and streamlines at lower and upper sections of the pump sumps, original (a) Upper section, Z=3.5D (b) lower section, Z=0.5D (c) cross section, y=5.5D

From Figure 4 (a), it is seen that in upper layer of the sump, the large-scale recirculation was located at the left of the sump caused by the change in flow direction. The main flow was pushed to a narrow zone at the right of the section by higher pressure of recirculation zone. The velocity at the right was much greater than the velocity at the left recirculation zone. The velocity distribution was extremely non-uniform which could cause vibration and decline of pump performance.

From Figure 4 (b), it is seen that in lower layer of the sump, the front recirculation zone was reduced both in size and strength. But the back-recirculation strength increased significantly which tended to rotational direction of pump impeller.

From Figure 4 (c), it is seen that in the cross section (y=5.5D), there was a strong anticlockwise recirculation at the lower-left of the section. The velocity gradient of the sump 1 was the greatest among all the sumps. Such poor flow pattern usually can result in severe hydraulic vibration of pump unit which is often found in the side-intake of the pumping stations. While in the sump 11~14, the large-scale recirculation zones disappeared due to the lower velocity of approaching flow, in which flow pattern looks quite good. However, the small-scale recirculation still exists near the back wall which could result in vortices into the suction pipe and affect the pump operation.

4 THE CALCULATION RESULTS FOR THE IMPROVING DESIGN

4.1 The improving design of sump

To improve flow pattern and eliminate recirculation flow in the pump sump, different measures were applied in the sump accordingly. Considering the navigation of ships, the measures were not allowed to be set inside of the lock which is the forebay of the pump station, otherwise, it could reduce the space of the lock and affect the navigation of the ships.

To improve the flow pattern, two segments diversion piers 1 and 2 (indicated as scheme 1) were installed on the entrance of the intake sump considering the position of gate slots. The computation results showed that the recirculation zones in the intake sump of 1~10 units became smaller obviously. The recirculation zone in the intake sump 11 disappeared. The flow around back baffle was still asymmetrical. The discharges of passing flow at both sides of the diversion piers were different.

Based on the scheme 1, another small diversion pier 3 was added in the middle of the two segments diversion piers as scheme 2 with three segments diversion pier. To avoid the asymmetrical flow, the back baffle was set at the back wall. The computation results showed that the recirculation flow in the sump 7~10 were significantly reduced. However, the large-scale recirculation flow zones still existed in the sump1~6. The flow was asymmetrical around the suction pipe near the back wall.

Based on the scheme 2, two triangular columns (prisms) were set symmetrically on the both sides of the three segments diversion pier to eliminate larger recirculation flow in the sump. It is expected that the wake flow behind the column due to the separation at the prism sides may change the larger recirculation flow in the sump (Feng Xusong, 1998; Cheng, et al. 2002; Liu, et al., 2009).

From the analysis above, the combined measures of three segments diversion pier, back baffle and triangular prism column named scheme 3 were determined as the final scheme for improving flow pattern in the pump sump 1~11. The details of the scheme 3 are shown in Figure 5.



Figure 5. improving design of pump sump

4.2 The flow fields in the sump improved

Figure 6 shows the results of the scheme 3 with combined measures of piers, prisms and back baffles for the sump 1~11. The flow pattern was well improved into desired way. The main flow was divided by the diversion pier. The flow velocity distribution was basically symmetry from left to right sides of the sump, seen

in Figure 6(a) and Figure (b). Figure 6(c) gives the evidence that the large recirculation flow in the crosssection y=5.5D was eliminated. The flow streamlines tended to the centerline of the section which represents the character of conflux flow.

Figure 6(d) shows the detail flow fields in the sump 1 of which the original flow pattern is the worst. The wake flow appeared behind the prism column instead of large recirculation flow. The large recirculation flow zone was symmetrically divided into two parts which were much reduced and limited at the front of the columns. The flow redistributed due to the momentum exchange through the wake behind the column and the flow before pipe inlet was getting uniform.

The circulation around the suction pipe was totally changed due to obstruction of the back baffle. The inlet flow entered in the pipe symmetrically. The velocity distribution in the sump 1 was well improved which can ensure the safety and efficiency of pump operation.

Meanwhile, the flow pattern remained the same as the original in the sump 12~14. It is evident that the combined measures can effectively improve the flow pattern in the side-intake pumping station.



Figure 6. V_y contours and streamlines at lower and upper sections of the pump sumps, improved (a) upper section, Z=3.5D (b) lower section, Z=0.5D (c) cross section, y=5.5D (d) detail flow in sump 1

5 CONCLUSIONS

To improve the intake flow pattern of the side-intake pump sump, the numerical simulations were performed for original design and improved design with combined measures. The computational results of the sumps with combined measures and original sump were analyzed. The summary can be drawn as the following:

- (1) The flow distribution in the most sumps of the side-intake pumping station is obviously non-uniform which cannot meet the requirement of pump inlet condition due to the large recirculation flow and vortices.
- (2) The suitable division piers can reduce the size of recirculation zone and thus change the original non-uniform distribution of velocity. The three-segment division pier is effective to divide the main flow into two parts, the large-scale recirculation is reduced to two small scale recirculation and limited at the front of the columns.
- (3) The prisms column set at the both sides of the diversion pier can destroy the recirculation zone and increase the kinetic energy through the wake flow behind the column. It is conductive to flow redistribution. The flow patterns in the sump are well improved in the sump.
- (4) The back baffles installed can obstruct the circulation flow around the suction pipe and conduct the flow to symmetrically enter the suction pipe. The flow condition can meet the requirement of pump operation and to ensure the safety and efficiency.

ACKNOWLEDGEMENTS

This project was supported by China Nature Science Funds (51279173), "The 12th Five-year" Key Project of National Science and Technology Support Plan(22015BAD20B01) and Jiangsu provincial open funds for Key-Lab (K100017). Support for construction/assembly of the facility was also provided by JSS Key Laboratory of Hydrodynamic Engineering.

REFERENCES

Ansar, M. & Nakato T. (2001). Experimental Study of 3D Pump-Intake Flows with and Without Cross Flow. *Journal of Hydraulic Engineering*, 127(10), 825.

- Constantinescu, G.S. & Patel V.C. (1998). Numerical Model for Simulation of Pump-Intake Flow and Vortices. *Journal of Hydraulic Engineering*, 124(2), 123.
- Cheng, L., Liu, C., Zhou, J. & Yan, B. (2002). Numerical Simulation of Optimal Sump for United Pumping Station. *Irrigation and Drainage*, 21(3), 68-70. (In Chinese).
- Feng, X. (1998). Bottom Sill improving the Flow Pattern in Pumping Station Forebay and Analysis of The Downstream of the Sill. *Hydraulic Science and Technology*, 1(1), 31-33. (In Chinese).
- Feng, J. & Li, X. (2012). Research on the Measures of improve the Flow Pattern of the Forebay of City Intake Water Pumping Stations. *Applied Mechanics and Materials*, 170, 1856-1861. (In Chinese).
- Jong, W.C., Young, D.C. & Chang, G.K. (2010). Flow Uniformity in a Multi-Intake Pump Sump Model [J]. *Journal of Mechanical Science and Technology*, 24(7), 1389-1400.
- Liu, C., Zhou, Q., Qian, J., Jin, Y. & Xie, C. (2016). Flow Characteristics of Two-Way Passage Vertical Submersible Pump System. *Transactions of the Chinese Society for Agricultural Machinery*, 47(10), 59-65, (In Chinese)
- Liu, C., Han, X., Zhou, J., Jin, Y. & Cheng, L. (2009). Numerical Simulation of Turbulent Flow in Forebay with Side- Intake of Pumping Station". *Drainage and Irrigation Machinery*, 27(5), 281-286. (In Chinese).
- Rajendran, V.P., Constantinescu, S.G. & Patel, V.C. (1999). Experimental Validation of Numerical Model of Flow in Pump-Intake Bays. *Journal of Hydraulic Engineering*, 125(11),1119.
- Rajendran, V.P. & Patel V.C. (2000). Measurement of Vortices in Model Pump-Intake Bay by PIV [J]. *Journal of Hydraulic Engineering*, 126(5), 322-334.
- Zhou, L. (2010). Numerical Analysis on Improvement of Flow Conditions in Forebay of Pumping Station by Setting Separation Piers. *Journal of Yangtze River Scientific Research Institute*, 27(2), 31-33. (In Chinese).

FLOW IN MORNING GLORY SPILLWAY OF ALBORZ DAM

AMIR KHOSROJERDI⁽¹⁾ & ATEFEHMOSAHEBI MOHAMMADI⁽²⁾

^(1,2)Water Engineering Department, Science and Research Branch, Islamic Azad University, Tehran, Iran Khosrojerdi@srbiau.ac.ir

ABSTRACT

The goals of this research are to simulate the flow field on a spillway with actual dimensions, investigate various factors affecting the efficiency of numerical method in the simulation of flow field, and the possibility of cavitation in the evacuation system of Alborz Reservoir Dam. In this regard, flow hydraulic in the spillway is investigated using ANSYS CFX software, in which governing equations of fluid flow including equations of continuity and Navier-Stokes motion are simulated by finite volume and implicit discretization at each time step. For this purpose, the free-surface two-phase flow by using k- ϵ turbulence model simulated to achieve the appropriate pattern of flow, and also flow parameters in different discharges are evaluated. Results have shown that flow depth and velocity changes are almost constant. Considering pressure in maximum discharge, negative pressure is observed and this has caused cavitation possibility to be included in the current research.

Keywords: Numerical model; shaft spillway; ANSYS CFX; cavitation; dam hydraulic.

1 INTRODUCTION

Morning glory spillway or generally shaft spillway is a separate spillway which can be an alternative to the side spillway. It consists of a circular concrete crest which conducts the flow into a vertical axis connected to a tunnel with low slope and the axis is connected to the tunnel by a curvature with large radius (D. Vischer and W.H. Hager, 1995). Thus, these shaft spillways consist of three main parts, crest, shaft, and a horizontal or nearly horizontal tunnel. In these structures, the flow crosses over the crest, and passes the vertical shaft and exits through downstream of the tunnel. These spillways are settled in the reservoir to transfer water to downstream and are used in dams with narrow valley topography, capable to connect them to a downstream tunnel. The value of hydraulic conductivity in these spillways depends on the geometry of spillway crest, its location in the reservoir toward dam crest, water head, input flow velocity into the crest, existence of energy depreciation systems such as whirlpool breakers, spillway shaft profile, and bottom tunnel slope. This type of spillway is very economical, when the tunnel is used as diversion tunnel. This spillway with its trumpet shape is independent to dam body, and despite all its advantages, in heads higher than design head, some problems such as whirlpool and consequently vibration in structure, and negative pressure may occur. Flow field simulation of shaft spillway with laboratory actual dimensions using numerical modeling in ANSYS CFX, possibility of cavitation in the shaft spillway of Alborz reservoir dam, investigation of various influencing factors, and the efficiency of numerical method in the simulation of flow field are the most important goals of this research.

2 THE GOVERNING EQUATIONS

In this study, by examining the features and capabilities and limitations of available software, ANSYS CFX software was used to simulate the flow field. Governing equations of incompressible viscous fluid motion in a turbulent state are expressed using averaged Nervier-Stokes equations that are so-called Reynolds (J.W. Toso Karnafuli, 1987). Since the flow at the bottom of spillway is always highly turbulent, thus here, the Reynolds equations with two-equation turbulence model were used to solve turbulent flow and to calculate turbulence transfer in the computational domain. It is worth mentioning that for solving equations governing the motion and distribution, second order upwind method was used.

In the Cartesian coordinate system, the governing equations include the continuity equation and momentum equation, which are provided below.

$$\frac{\partial U_i}{\partial t} + \frac{1}{V_F} \left(U_i A_i \frac{\partial U_i}{\partial x_i} \right) = \frac{1}{\rho} \frac{\partial p'_i}{\partial x_i} + g_i + f_i$$
[2]

where, V_F is the ratio of volume fraction of open space to flow, and ρ , R_{DIF} , and R_{SOR} are the fluid density, turbulence permeability term, and mass source, respectively. u, v, and w, are velocity components and A_x , A_y , and A_z are fraction area of perimeter to flow in x, y, and z directions, respectively.

In the above equations, P', g_i and f_i refer to pressure, gravity force in i direction and Reynolds stress, respectively.

In modeling of flow in Navier-Stokes equations in order to solving flow field, turbulent flow pattern was used, and according to the performance of multi-phase models, transfer between phases is free that in the simulation of fluids (water and air), by an explicit separable interface separated from each other, has the best performance.

K- ϵ turbulence model that was used in current study, includes two transfer equations for turbulent kinetic energy (k) and turbulence depreciation rate (ϵ) to obtain the Reynolds stresses and a whirl viscosity (J.C. Amorim, 2004).

In the current study for simulation of turbulence and results of the analysis of K- ϵ and K- ω turbulence models, the numerical simulation was calibrated based on a given discharge and the investigation of 3-Dimension flow pattern over the shaft spillway using K- ϵ turbulence model as the most appropriate option, showed that this model in determining the pressure and thus the water level profile had a good accuracy.

3 RESEARCH METHOD

Current study was based on experimental work conducted at Water Research Institute in the case of Alborz Dam spillway. Intended field including flow inlet from approach channel, then passing on the spillway crest, and also a part of the vertical shaft was investigated. For this purpose, the flow entered from the spillway crest and passed the shaft and exited through the middle of the shaft. In this regard, the effects of roughness on flow field, the comparison of simulation based on turbulence models, the comparison of performance of different multi-phase models, were studied and the optimal meshing was evaluated in the flow simulation. To achieve this aim, the model was validated based on a given discharge and factors affecting the simulation were investigated. After ensuring the accuracy of numerical models, the simulation was conducted based on four different discharges. The effects of discharge on flow field were also studied and for these discharges, hydraulic specifications of flow and occurrence probability of cavitation were investigated.

Given that in this study numerical modeling was done by computational fluid dynamic, ANSYS CFX was used to analyzing flow, and meshing of fluid volume in the numerical model, especially in the models with complex geometry (as the current investigative model), was the biggest limitation and the suitable choice can lead to increasing calculation accuracy. In this study, for the analysis and discretization of flow field in the studied model, unstructured mesh was used. In the analytical model of the structure, in order to achieve proper and optimal mesh from different mesh sizes, sensitivity analysis was used. Figure 1 shows the fluid volume and numerical model meshing of Alborz Dam spillway.



Figure 1. A view of fluid volume and meshing of Alborz Dam spillway

In order to evaluate the accuracy of the numerical model and its sensitivity to the applied mesh, in areas near the bottom of approach channel, different meshing was used for each turbulent model with regards to the aim of the project that was reaching a pressure distribution near the experimental data and finally optimal meshing was chosen in terms of accuracy and required time for solving the model. By investigating the expected results for pressure, it was observed that for anticipated pressures in the numerical modeling with square meshes, the results improved and better compliance with experimental results was obtained, and the sufficiency of mesh size from the view of calibration was completely confirmed.

In the current study, with regards to the essence of numerical modeling, the flow should be simulated as unsteady flow. In the numerical simulation, a couple solver was used to solving the hydrodynamic equations. In the field entrance, specified discharge boundary condition was used. For the purpose that the input velocity profile be completely developed and flow condition at first to be monotonous, the input boundary condition was far from the spillway sufficiently. For the field output boundary that was after the sill, pressure boundary condition was used and for other surfaces, wall boundary condition was considered as a slick and non-slip flow. Since only air was there, the upper face was considered as open boundary to let the air to be exchanged. Desired boundary condition for pressure was also selected equal to atmospheric pressure.



Figure 2. A view of the fluid basin in the numerical model

4 WATER SURFACE PROFILE

In this step, after calibration and validation of the model, the most suitable condition in terms of better and more accurate simulation in the view of the prediction of free surface flow pattern based on optimal meshing and initial and boundary condition in accordance with the hydraulic model, was selected and had been used as the basis of other hydraulic analysis. After ensuring the accuracy of numerical models, the simulation was conducted based on four different discharges and the effects of discharge on flow field were also studied. For these discharges, hydraulic specifications of flow such as pressure, water and air concentration, and velocity that indicated energy dissipation were investigated in different sections, and cavitation possibility was evaluated.



Figure 3. Comparative curve of flow depth over the spillway crest in the experimental and numerical model (maximum and minimum amount of discharge).

As it can be seen from the results, for a given discharge, the flow depth on the spillway crest was almost uniform, in the low levels of discharge the spillway function was free and for maximum amount of discharge, the spillway was in the state of complete throttle and water depth changes increase indicated that velocity changes reduced and spillway function was in the state of full.





For the minimum discharge, the flow struck the walls as a blade and was discontinuously in the beginning of vertical shaft. With increase of discharge, the flow was coupled with air and water in the beginning of the shaft, in the state of partially submerged, the shaft was full and rotational flow with the air layer moved, and also in the state of maximum discharge, the spillway was throttled and with air voids in the area of central axis

to the beginning of vertical shaft. It should be noted that the design discharge was $800 \frac{m^3}{sec}$, and that the obtained results indicated a desirable performance of the spillway.

5 DISTRIBUTION OF VELOCITY AND PRESSURE VECTOR

It was observed from various model analyses that in the free function of spillway, the velocity and pressure distribution over the spillway is uniform for a defined discharge. Moreover, with the increase of discharge, the pressure reduces and all pressures are negative. In the full and throttle function of spillway, the speed up over the spillway indicates that hydraulic conductivity of the spillway will be performed easily and flow rejection does not exist and the spillway does not reach full throttling. And like the previous, with the increase of discharge, the pressure reduces and all pressures are negative until an area close to the beginning of the shaft.







Figure 6. Comparison of pressure distribution curve in the vertical shaft of spillway crest (maximum discharge)

6 CAVITATION PHENOMENON ANALYSIS

Due to the negative pressure that had been seen in the spillway body, cavitation phenomenon was investigated by two numerical and analytical methods. To obtain cavitation, information such as average velocity and applied pressure on the bottom of various parts of the structure, were needed. However, the velocity in this part was not computed in the laboratory, hence, the cavitation index had been calculated by software, and the results indicated that cavitation had occurred. The investigation was conducted according to the spillway structure of numerical model, and it was specified that pressure in the output part flow was considered zero by software and pressure became zero from its negative value. This pressure difference which caused accurate results was not achieved by the software, hence the cavitation index was computed through the analytical method and flow velocity was obtained from software and measured pressure from experiments. Obtained results showed that cavitation had not occurred in this part of shaft spillway.

D'	Distant	TI	Static pressure (m) Axe B Q=1050 cms				
Plezometers	Distance	Elavation					
Num.	From Spillway Crest(m)	(m)	Numerical	Error%	Experimenta		
1	11.70	300.94	2.50	29.20	1.77		
2	11.32	301.00	1.20	-65.83	1.99		
3	10.68	300.87	2.15	4.19	2.06		
4	9.90	300.50	2.30	-44.78	3.33		
5	8.93	299.78	3.11	-13.50	3.53		
6	7.71	298.46	3.61	-24.38	4.49		
7	6.31	296.20	2.60	-76.54	4.59		
8	5.14	293.18	2.57	-33.46	3.43		
9	4.40	289.41	-1.27	-62.20	-2.06		
10	4.12	286.23	-3.00	-57.00	-4.71		
11	3.85	281.86	-5.00	-46.40	-7.32		
12	3.77	277.87	-6.00	-19.83	-7.19		



Figure 7. Specifications of static pressure for maximum discharge

Figure 8. Curve of cavitation index using analytical method

7 CONCLUSIONS

According to the obtained results, the changes of flow depth in different discharges are almost uniform and close to experimental data, indicating that the model is well calibrated and flow simulation is well performed and the model accuracy is high. Obtained results from the changes of flow velocity in desired discharges indicate that the flow velocity in minimum and maximum amount of the discharge is almost identical. Also in maximum discharge, the flow average velocity is higher indicating that the flow passes easily and flow rejection does not occur in the spillway, so the spillway will not be in a throttle state. By investigating the hydrostatic pressure, it is observed that the pressure in minimum discharge is uniform and positive, although in the maximum discharge, negative pressures have been seen in the beginning part of vertical shaft. These negative pressures which have caused cavitation are also evaluated. When examining the phenomenon of cavitation, two numerical and analytical methods are used. In the numerical method, ANSYS CFX software is used so that the software calculates pressure and velocity based on the developed numerical model and calculates the cavitation index to indicate the cavitation occurrence. In the analytical study, measured pressures from experiment and calculated velocity in the software have been placed in the cavitation index formula. The obtained cavitation index which is higher than critical index shows that cavitation does not occur in this part of the spillway.

REFERENCES

Amorim, J.C. (2004). A Numerical and Experimental Study of Hydraulic Jump Stilling Basin, Advances in Hydro-Science and Engineering, 4.

Bowers, C.E., TosoKarnafuli, J.W. (1987). Project Model Study of Spillway Damage, ASCE, 114 (5).

Chanson, H. (2004). The Hydraulics of Open Channel Flow, Butterworth-Heinemann.

Fiorotto, V. & Rinaldo, A. (1992). Fluctuating Uplift and Linings Design in Spillway Stilling Basins, *Journal of hydraulic engineering*, 118(4), 578-596.

Khatsuria, R.M. (2005). Hydraulics of Spillways and Energy Dissipators. CRC Press.

Peterka, A.J. (1984). *Hydraulic Design of Stilling Basins and Energy Dissipators-Engineering Monograph 25*, Department of the Interior, Bureau of Reclamation.

Vischer, D. & Hager, W.H. (1995). Energy Dissipators, IAHR Hydraulic Structures Design Manual 9. CRC Press.

LA COCHE PELTON ENHANCEMENT PROJECT SCALE MODEL

GREGORY GUYOT ⁽¹⁾ & MATHIEU RODRIGUEZ ⁽²⁾

⁽¹⁾ EDF-Hydro Engineering Center, Le Bourget du Lac, France gregory.guyot@edf.fr
⁽²⁾ Centre d'Etude et de Recherche de Grenoble, Le pont de Claix, France, mathieu.rodriguez@cerg-fluide.com

ABSTRACT

EDF has designed the La Coche Hydraulic power unit upgrade. The construction is currently in progress. A new 240 MW Pelton power plant will be erected on the existing hydropower plant area. One of the biggest challenges was to design the power plant stilling basin that faces all the contradictory constraints imposed by the neighborhood, the water quality and the particular site. Thus, the experimental design must be able to answer to hydraulic, air exhausting and sediment transportation topics. As such matters cannot be solved by analytical calculations, a three-step method was performed to provide a relevant design. The process was composed by a main scale model (1/8.33), a wide experimental measurement to determine the bubble distribution to test on the main scale model and a global head losses model (1/16.41) that gives the head losses in the new hydraulic network and the maximum level of water under the Pelton wheel. The issues of this study are firstly, the way of determining the bubble distribution, secondly, the method used to test the air exhausting on the main scale model and the sediment deposit management. The use of a hydraulic wall ensures an acceptable level of sediment deposit, a quieter flow and an equal level of air exhausting in the stilling basin. Finally, the air exhausting system improvements bring about an acceptable air exhausting for the project.

Keywords: Experimental set up; scale model; air exhausting; sediments transportation.

1 INTRODUCTION

The la Coche hydro power plant was one of the first pumped storage facilities in the world. This kind of prototype needs heavy maintenance effort. To simplify the maintenance and consequently reduce the operation costs, EDF is currently building the La Coche Hydraulic power unit upgrade. A new 240 MW Pelton power plant will be erected in the actual used area.

This article will give an overview of La Coche Pelton power plant unit project. In particular, it will focus on this unique scale model's achievement to optimize the stilling basin. Firstly, it will explain the objectives of the scale model. Then, it will tackle the method used. Finally, it will turn to the main scale model main observations and improvements achieved.

2 CONTEXTS

2.1 La Coche facility area

The new Pelton power plant is located on the La Coche site near Moutier in the Savoie Mont Blanc area. As described in the following figure, the new project is situated between the two existing penstocks (La Coche Penstock and the Randens Penstock) and between the existing underground power plant and the high voltage transformer.

Clearly, the operation water levels into the new power plant stilling basin are directly linked with the level of the Aigueblanche Basin (Figure 1). The downstream levels are strictly imposed by the Randens penstock (Figure 1). The available area is substantially limited by the existing facilities and by the mountain shape.

The hydraulic challenge was to design the downstream power plant stilling basin which must be able to face all these sharp constraints due to the restricted available area and to the hydraulic constraints imposed by the existing hydraulic network and the water quality. That means that the classical Pelton turbine stilling basin was fully irrelevant to this project. The stilling basin to be designed must actually allow a large amount of sediment to be evacuated and to ensure that the air induced by the Pelton wheel is properly removed before the junction with the downstream hydraulic network.



Figure 1. Overview of the la Coche Pelton Hydro facility area.

2.2 Analytical design

An analytical study was performed to create a first design.

The minimum velocity value was given by the basic sediment transportation approaches, mainly based on the shield parameter. This enables the team to design the shape and the bottom slope of the free flow basin, the slope and the diameter of the downstream pipe (Figure 2).

Then, the outlet of the free surface part of the stilling basin was designed to avoid vortices that may induce secondary flows into the downstream pipe. The approach was based on the available submergence criteria. The different propositions from Hager and Schleiss (2009) were studied to design the anti-vortex device and the outlet shape.

Lastly, the air exhausting system was designed with reference to Wickenhauser (2008) and Falvey (1980) publications. A special, simplistic venting pipe composed of a rectangular chamber and a pipe was designed (Figure 2).

2.3 Scale model objectives

In a nutshell, the scale model must optimize the stilling basin design made by the EDF project team. Besides, the scale model is the perfect tool to estimate how the different turbine technologies or the civil engineering modifications impact the stilling basin operation.

Three steps were carried out for all the tested configurations in order to determine the best stilling basin design.

The four objectives of the scale model are: studying hydraulic behavior, sediment transportation, air exhausting and head losses.

The first objective is to ensure that the hydraulic flows are smooth enough to allow the air exhausting and strong enough to generate the sediment transportation.

Secondly, the air exhausting system must be assessed and if needed, modified to enable the best efficiency. It is essential for the existing power plants that an extremely low amount of air is trapped in the hydraulic network.

Then, the sediment transportation must be verified. The sediments must be evacuated even if a large amount of materials are stocked in the stilling basin.

In addition to the previous steps, the head losses of the new hydraulic network must be accurately estimated for all the operation modes. The tests must also provide the maximal level in the stilling basin for steady and transient operation modes.

2.4 Tested operation modes

A lot of parameters were considered for the study.

For the first three steps, only steady flows were required. Two levels of water were imposed in the stilling basin. It represents the maximal level of water available when the upstream reservoir is at the maximum level and the lower level of water available when the upstream reservoir level is at the minimum. For each level of water, four discharges were tested 15, 20, 25, 28m3s-1. It enabled the understanding of the evolution of the phenomenon with discharge variations.

The sediment diameter to test was fixed at 3 cm. It is the maximal sediment diameter that can flow into the hydraulic network.

For the last experiment concerning the head losses and the maximal level that can occur in the stilling basin, the flow rate in the Randens penstock was taken into consideration. As a result, 50 configurations were tested in steady state and 6 in unsteady flow.



Figure 2. Analytical design of the stilling basin (plan view up and elevation view down).

To answer to the need, it was decided to split the experimentation in three different scale models.

3 GLOBAL HEAD LOSSES MEASUREMENT SCALE MODEL

3.1 Experimental apparatus

The scale of this mode was 1/16.41. The model was based on the Froude law of similarity with a controlled Reynolds distortion. It simulated the hydraulic network from the Aigueblanche reservoir down to the Randens-Pelton power plant junction as shown in Figure 3.

	Min scale model
Stilling basin Reynolds number	6 10 ⁴
Randens penstock Reynolds number	2.10 ⁴

The pressure levels were captured by high frequency camera. The films were analyzed to provide the temporal pressure values.

This model gave the maximal level into the stilling basin for steady and unsteady running operations. It allows the engineers to figure out the lower possible level for the Pelton turbine and to determine the head losses in the new hydraulic network for the various operation modes.



Figure 3. View of the head losses scale model (changed to an English version).

This model gave the maximal level into the stilling basin during steady and unsteady operation modes. The results allowed the engineers to determine the turbine level that warrant the energetic efficiency of the power plant.

4 PROTOTYPE BUBBLE SIZE AND AIR FLOW RATE ESTIMATIONS

This step a special feature of this scale model. From a survey of the bibliography [GUYOT et al], the air entrainment consequences of large plunging jets was scarcely investigated. The case of large scale jets was still not well understood. The air entrainment in such conditions still escaped prediction.

Moreover, there is no law of similitude to determine directly the air exhausting process with the main scale model.

To obtain a first estimated representation of the prototype bubble size distribution, a dedicated experimentation was clearly needed. It also aimed to give an idea of the air flow rate entrained by the jets.

The experimental set up was erected on the Pont de Claix channel (EDF France). It (Figure 1 and Figure 2) aimed at analyzing the behavior of the air entrained by the jet under the free surface in a downstream channel.

4.1 Channel experimental apparatus

The experimental apparatus was located 300 m downstream the inlet weir of the straight channel and 500 m upstream the channel end. The channel is 8 meters wide (I) and 5 meters deep. The water level in the channel was controlled by an inlet weir. The channel flow rate (Qc) was imposed by a hydro power plant downstream. The available range of channel flow rates was 35-80 m3s-1. For practical reasons, the channel flow rate (Qc) was set to 45 m3s-1. Thus, the average channel velocity (Vc) under the jet was 1.35 ms-1 (Figure 4). The stability of the Channel velocity and flow rate was checked by a SonTek Acoustic Doppler current profiler (ADCP) upstream the apparatus.



Figure 4. View of the channel upstream (a), of the experimental platform (b), of the channel downstream (c).



Figure 5. Stream wise section sketch (a) and transverse section sketch (b) of the channel experimental apparatus.

The jet flow rate (Q) was pumped out of the channel by two pumps which fed the PVC 164.3 mm internal diameter circular pipe network. The outlet of this network was the injector itself, which comprised of a divergent (I.D. 164 mm to 320 mm) linked to a 600 mm of 320 mm I.D. pipe. A 90° Elbow linked the inlet network to the injector. Two calming flow straighteners were located in the straight section to decrease the turbulence level before the nozzle inlet.

As half of the turbine outflow rate was assumed as round shaped jets plunging vertically, two circular jets were selected to describe the two operation modes.

It was assumed that the jets were in free fall under the turbine, the two impact velocities were 10.35 m/s for the lower level operation mode and 8.35 ms-1 for the highest level operation mode. The experimental set up was designed to reproduce impact velocities.

The two nozzles selected are conical with the same convergent 0.32 (H/V) slope. The nozzle outlet internal diameters (D0) are 164 mm and 135 mm with a respective length of 500mm and 592mm. Consequently, the jet falls down in an atmospheric surrounding.

The flow depth (h) of the channel was set to ensure a 2.57 m falling height (Lc) between the jet nozzle and the free water surface (Figure 5).

A Krohne Optisonic electromagnetic flow meter located upstream the injector measured the flow rate. All the results regarding air entrainment were obtained with an RBI optical probe. This probe was attached to a mast which moved in three directions: X upstream-downstream (from 1 m downstream the jet to the end of the bubble cloud), Y left bank- right bank (between -2.5 m to 2.5m centered on the jet) and Z flow depth (up 2 m penetration depth H), as shown in Figure 5. For each jet configuration, the probe was used to provide the void fraction, the penetration depth and the bubble plume shape along four channel sections downstream the jet. All the locations of the measurement sections are mentioned in table 1.

Table Z. Location of the measurement section	0115 101	the two	o opera		Jues.
Distance between jet impact and section (mm)	1750	2900	5400	6450	7900
Impact velocity 10.35 ms-1		Х	Х	Х	Х
Impact velocity 8.45 ms-1	Х	Х	Х	Х	

Table 9. Logation of the measurement exciting for the two exercises medes

RBI has sold the double optical probe and the data acquisition system sold RBI to measure the void fraction and the bubble diameter up to 0.1 mm.

If the void fraction measured by the optical probe was below 0.02, it was assumed that the probe was outside the bubble plume. Hence, this criterion enables the determination of the penetration depth Z max, and the lateral extent X max of the bubble plume.

4.2 Results

A minimum of 1000 bubbles is required to validate a measurement point. The measurements were analyzed with the RBI software to obtain a bubble distribution for each measurement point as described in the following figure.



Figure 6. Bubble distribution in function of the bubble diameter for the central point of the 2900 mm section for the impact velocity of 10.35 ms-1. The three dash lines represent the first d10, d50 and d90.

All the distributions have been analyzed to provide a representation of the global bubble distribution, including all the available points sorted by configuration. It appears that the global average bubble size is around 2.5 mm. The crucial point is that more than 90 % of the air quantity is conveyed by bubbles bigger than 3 mm.

A local gas flux (FI, ms-1) can be calculated as the measured void rate multiplied by the bubble velocity taken equal to the channel velocity. The entrained air flow rate (Qg) (m3s-1) was then derived by integrating the local flux over the channel cross-section. A consistent magnitude order was the air entrainment flow rate calculated at 2900 mm downstream the jet impact. The values were 17% of the jet water flow rate for the 8.45 ms-1 impact velocity and 32% of jet water flow rate for the 10.35 ms-1.

Unfortunately, it was impossible to implement a reliable method to extrapolate the air flow rate under the jet. No air entrainment transfer function from a downstream section to jet is currently available. The key variables and the physical phenomenon which are responsible for the air entrainment process for this kind of jet are still being investigated.

However for the current purpose, this innovative approach gave a guess of the main populations of bubble to be tested on the main scale model. It is essential to test the over 3 mm bubbles. It may be also valuable to test the smaller bubble size to ensure that the small bubbles may be exhausted in the power plant design even if their terminal velocity is lower than the others and that the air quantity conveyed by the small bubble is negligible. For the previous reasons but also for practical reasons, two different classes were chosen to be tested on the main scale model. The first category composed of bubbles sized at the prototype scale from 0.65 to 0.9 mm and the second group composed of bubbles sized from 1 to 30 mm.

5 MAIN SCALE MODELS

5.1 Main features

The main scale model is a 1/8.33 scale model. It is operating with the Froude law of similitude.

Table 3: Key numbers for the main scale model				
	Min scale			
	model			
Stilling basin Reynolds number (Re)	8 10 ^₄			
Stilling basin Weber number (We)	240			

 $\text{Re} = \rho_l V_0 D_0 / \mu$ [1]: Reynolds number; $Fr = V_0 / \sqrt{gD_0}$ [2]: Froude number, $We = \rho_l V_0^2 D_0 / \sigma$ [3]: Weber number

It is possible to test the sediment transportation with this scale model.

The air exhausting efficiency was estimated with the method described in the following section.



Figure 7. Global view of the main scale model on the left and stilling basin view during PIV measurements.



Figure 8. Terminal velocity of a bubble in function of the bubble diameter, in blue, the first bubbles category and in red, the second category of bubbles. View of the particles used to simulate the two categories of bubbles (0.65 mm to 0.9 mm on the left and 1 to 30 mm on the right)

5.2 Air exhausting assessment method

The size and the density of the chosen material enabled the simulation of the terminal bubble velocity on the scale model.

To assess the air exhausting in the downstream power plan basin is to introduce a large volume of a particle category and to count the particles by exhausting location. Regarding the method used, the effect of the bubble coalescence cannot be simulated. However, the particles are not deformable so that they are more difficult to be trapped in the air exhausting system especially the venting pipes. Consequently, it is clear that the exhausting efficiency measured on the scale model is lower than the real one.

The diameter and the density of the sediments used in the model were fitted with respect to the shield parameter similarity linked with the scale of this model.

The Pelton turbine was simulated thanks to round shaped pipes. If the number of injectors were 6, there are 6 pipes that deliver the half of the flow rate on the power plant wall and 6 pipes that provide the other half of the flow rate vertically. If the number of injector differs, pipes distribution stays similar.

The discharge was measured by an ultrasonic flow meter. The hydraulic behavior was quantified by PIV measurements at the beginning of the process. Then, it was assessed through visualization with dye injections.

5.3 Main observations and design modifications

5.3.1 Anti-vortex design and hydraulic features

For the highest level of operation, two vortices were observed at the downstream corner of the free surface. A anti vortex device composed of two horizontal beams and a horizontal trash racks was added in the basin as described in the following figure.

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)



Figure 9. View of the vortex appearance in the first design (on the left) and of the free surface basin with the new anti-vortex device and without the central pear for the highest operation model (the right).

This device was designed to stop the vortex and to reduce the amount of bubble which flow downward. It consequently improved submergence at lower water level. The wide central pier into the basin, which was originally required for building reasons, was reduced to increase the efficient hydraulic section in the free surface basin. In fact, the best hydraulic behavior was observed without central pier.

In the free surface basin, a large recirculation occurred. The two levels of operation mode were impacted by the important phenomenon.



Figure 10. View of the large recirculation location in the stilling basin for the highest operation level (on the left) and for the lowest operation level (on the right).

This large recirculation is attributed to a hydraulic jump located nearby the downstream wall of the free surface basin conjugated with a massive air lift. As a consequence, the water mainly flows just under the surface before deeply plunging at the extreme end of the stilling basin into the downstream pipe. This phenomenon has to be carefully studied to avoid sediment deposition and air exhausting problem especially for the highest water level in the stilling basin but this also occurred in the lower level operation modes.

5.3.2 Sediment transportation optimization

The first result was that the design of the downstream pipe was appropriate because the pipe is able to evacuate the sediment for the strongest flow rates in both of the operation levels.

The main problem was caused by the large recirculation in the basin. It induced a sedimentation deposit observed in all operation cases (Figure 11).


Figure 11. View of the steady shape of the sediment deposit for the highest operation level (on the left) and for the lowest operation level (on the right).

Before trying to avoid the sediment deposition, it was checked that the sediment deposit shapes were steady for more than 5 hours of operation. For that purpose, a large amount of sediment was introduced in the basin. The areas of the sediment deposit are shown in the Figure 11. Nevertheless, the central area of sediment deposit may increase by the addition of sediment but this test precisely locates the deposition areas.

Research to find a way to decrease drastically the deposit was achieved. A hydraulic was separated into three parts with three different void rates located at the end of the central pier. This was the only way found that enabled a sediment deposit shrinking and a stagnation of the air exhausting efficiency. This wall acts to reduce the flow for the highest level of operation as Figure 11 and Figure 13 show. The vertical distribution of the void rate along the wall allows a progressive relocation of the flow rate. While initially mainly flowing under the surface, the flow was, attributed to the wall, vertically distributed. It results in a sufficient speed along the bottom slope which drastically reduces the sediment deposit.



Figure 12. View of the hydraulic wall at the end of the central pier (Red part void rate= 0, Yellow part void rate = 0.23, blue part void rate = 0.46).



Figure 13. View of the steady shape of the sediment deposit with the hydraulic wall for the highest flow rate and for the highest operation level (on the left) and for the lowest operation level (on the right).

5.3.3 Air exhausting efficiency

The air exhausting system was improved to reach the best efficiency. Firstly, a second venting pipe was added just downstream the outlet. The outlet design remained rough as shown in Figure 14. These two options enabled the model to stimulate great air exhausting immediately at the outlet.

Secondly, the existing venting pipe was pushed downwards at the second location shown on the left picture of Figure 7. This leaves a longer distance to the bubble rise, leading better venting pipe efficiency.



Figure 14. View of the flow at the outlet with the added venting pipe (lower operation mode and highest flow rate.

After analyzing the injected bubble exhausting in the main scale model, the air exhausting efficiency was estimated. As an example, the most unfavorable case result (highest flow rate) is given in the following table. Regarding the assumptions made, this value is clearly an overestimation of the air flow rate to water flow ratio.

Table 4. Unexhausted air flow rate to water flow rate for the highest flow rate.

Operation level	Lowest	Highest		
28 m3/s	6.1 %	2.2%		

These levels were considered as acceptable by the project team.

6 CONCLUSIONS

The special method used to transform an analytical design into an optimized design has been described. Even though the air exhausting and sediment transportation are contradictory phenomena, a reasonable design in accordance with the small area available was found. Attributed to the different devices added, the Pelton power plant stilling basin is able to exhaust a large part of the air induced by the Pelton wheel. It is also able to evacuate a large amount of the incoming sediments.

However, the lack of knowledge regarding the air entrainment induced by a large plunging jet required a special experimentation to provide a first estimation of the influent bubble distribution. This first estimation allowed innovative assessment of the air exhausting efficiency. Nevertheless, the difficulties and the questions associated to that experimentation clearly show that the research must be pushed forward to obtain more accurate results concerning the large plunging jets air entrainment. Finally, the optimization of the existing hydraulic power plant leads to precise understanding of phenomena which were never investigated at the hydraulic network scale.

REFERENCES

Edel'CIK, I. (1969). Memento Des Pertes de Charges.

Falvey, H.T. (1980). Air-Water Flow in Hydraulic Structures. NASA STI/Recon Technical Report N, 81.

- Garnier, J. (1997). Measurement of Local Flow Pattern in Boiling R12 Simulating PWR Conditions with Multiple Optical Probes, OECD/CSNI Specialist Meeting on Advanced Instrumentation and Measurements Techniques, Santa Barbara, CA.
- Guyot, G., Rodriguez, M., Pfister, M., Matas, J. & Cartellier, A. (2016). Experimental Study of Large Plunging Jets. Hydraulic Structures and Water System Management. 6th IAHR International Symposium on Hydraulic Structures, Portland, OR, 27-30 June.
- Hager, W. & Schleiss, A. (2009). Ecoulements Stationnaires, Constructions Hydrauliques (TGC volume 15), Traité de génie civil – 2e édition, PPUR - Collection

Violet, P. Chabard, J., Eposito, P. & Laurence, D (2002). Mécanique des fluides appliquée.

- Wallis, G. (1969). One Dimensional Two Phase Flow. Mac Graw Hill.
- Wickenhäuser, M. (2008). Zweiphasenströmung in Entlüftungssystemen von Druckstollen, Eidgenössische Technische Hochschule Zürich, Book.

STUDY ON CAVITATION AND VIBRATION CHARACTERISTICS AND ANTI-CAVITATION MEASURES OF HIGH HEAD INDUSTRIAL VALVES

XIN WANG⁽¹⁾, YA-AN HU⁽²⁾, XIU-JUN YAN⁽³⁾, WEN-XUN QIAN⁽⁴⁾, ZI-YANG LI⁽⁵⁾ & CHENG CHEN⁽⁶⁾

^(1,3,4,5,6) State Key Laboratory of Hydrology-Water Resources and Hydraulic Engineering, Nanjing Hydraulic Research Institute, Nanjing, China,

xwang@nhri.cn ⁽²⁾Key Laboratory of Navigation Structures, Nanjing Hydraulic Research Institute, Nanjing, China, yaahu@nhri.cn

ABSTRACT

In view of cavitation induced vibration problem of the large high-head industrial valves used in one hydraulic ship lift, prototype observation on the cavitation and vibration characteristics of the valve and pressure pipes was carried out. In order to find vibration reasons and study on damping measures, Atmospheric and decompression model tests have been conducted, and for the different working conditions of the water filling valve and the emptying valve, different measures were put forward. Combined measures of sudden enlarged body behind the valve and aeration before the valve were adopted for the water filling valve, while the aeration measure was also taken before the water emptying valve. After reconstruction of the anti-cavitation measures of the project, prototype observation was carried out again. It is shown that the valve cavitation and vibration were properly resolved by the engineering measures. Valves can run smoothly at large opening and the cavitation phenomenon disappears. Vibration responses of the valves and pressure pipes were reduced by 80%~90%.

Keywords: Industrial valve; high head; cavitation; vibration; aeration.

1 INTRODUCTION

The phenomenon of flow-induced vibration is very common for hydraulic structures. Especially when the hydraulic structures are impacted by bad flow pattern or high speed flow, severe vibration may happen. Flow-induced vibration often appears in sluice gates (Wang and Luo, 2012), guide walls, overflow dam, valves of ship lock (Wang and Yan, 2013), hydro-machines (Wang et al., 2009) and so on. A lot of researches have been carried out for the problem and relative results are also very rich. As a result of the complicated vibration resources formed by the flow, vibration characteristics of the structures are different. Generally, flow-induced vibration can be divided as three kinds, which are, random vibration, periodic vibration and impact vibration (Wang et al., 2005). The stable random vibration is often induced by the normal pulsation pressure of the flow. The periodic vibration is induced by some period loads of the special flow pattern, such as flow around the water seal. Structure blocking in the process of operation or cavitation of the special flow pattern may induce impact vibration (Long et al., 2008). As the severe flow-induced vibration may cause great harm for the structure and its ancillary facilities (Wang et al., 2014), severe vibration induced by bad flow pattern should be avoided. In this paper, in view of vibration of high water head industrial valves, prototype observation and model test were conducted to study the flow-induced vibration characteristics and damping measures. Then, the practical effects of the anti-cavitation and damping measures of the project were evaluated.

2 VIBRATION OF THE INDUSTRIAL VALVE

Hydraulic drive ship lift created in China is a new kind of ship lift. Its running is driven by filling or emptying water through the water delivery system. The running speed of the hydraulic drive ship lift is determined by the delivery flow. And the delivery flow is determined by the water head and the operation way of the valve. As a result, the valve is the critical factor for the water delivery system of the hydraulic drive ship lift. The first hydraulic drive ship lift has been built in Yunnan Province of China. The water head at first. Besides, it should be running quickly and smartly. Large flow is needed for the high-speed running and small flow is needed for the slow docking of the chamber. After technical comparison, the kind of piston valve is selected to be the filling and emptying valves of the hydraulic drive ship lift. In order to meet the requirement of the fast and slow running of the ship lift, the distribution way of "main valve and auxiliary valve" is adopted. For filling or emptying valves, three piston valves, one main valve and two auxiliary valves, are set in parallel to control flow. The layouts of the valves are shown in Figure 1. The main valve is set at the middle and at each side of the main valve is an auxiliary valve. Pressure pipes behind the three valves are connected into one main pipe. The flow capacity of the main valve is large, while the flow capacity of the auxiliary valve is relatively small.

When the ship lift wants to run fast, the main valve needs to be operated at a large opening. When the ship lift wants to run slow, only auxiliary valves need to be operated.



(a) Filling valves

(b) Emptying valves

Figure 1. Layouts of filling and emptying valves

The working water head of the valves is about 60 m and the initial pressure behind the valves is almost zero. Some problems are found in the process of adjustment for the water delivery system of the ship lift. When the opening of filling or emptying valves is increasing, the pressure pipes behind the valves indicate strong vibration, terrible noise, long time of filling or emptying and others. In the normal operation way, when the opening of the main valve is going up to 50% from 45%, cavitation takes place in the main valve. Cavitation noise exists in the pressure pipe behind the main valve and strong vibration happens. The operation staffs feel scared and the main valve stops running. Vibration of the pressure pipe behind the main valve has been observed. When the opening of the valve becomes 50%, the pipe vibration increases immediately. The acceleration amplitude of the pressure pipe behind the main valve increases to more than 20g from about 5g and the displacement amplitude increases to about 2000µm from about 800µm, as Figure 2 shows. At the same time, the vibration characteristics turns to unstable impulse vibration from stable random vibration. Obviously, the phenomenon is induced by the cavitation of the main valve. The vibration characteristics is coincident with the cavitation noise characteristics. They have a same changing law. As a result, the main valve cannot operate as the design way, that is, cannot run at a large opening. The running speed of the ship lift is affected directly. So, the problem of the main valve cavitation must be resolved.



Figure 2. Steel pipe vibration behind the main valve

3 STUDY ON ANTI-CAVITATION AND DAMPING MEASURES

Atmospheric and decompression model tests study of the filling and emptying valves have been conducted for the problems of the high head industrial valves. The model scale is 1:10. The model valves shown in Figure 3 were produced according to the scale by the factory of the prototype valves and the model valve structure is completely the same as the prototype valve. The model test results indicated the flow passed the valve is obviously influenced by the winding pipes behind the valves. Spiral pattern shown in Figure 4 was taking place in the winding pipes and swung left and right in the join region of the three pipes. The flow pattern was not stable. So, the spiral flow is one of the main reasons of the pressure pipe severe vibration.

Flow pulsation pressure is the vibration resource of the structure and can reflect the vibration strength. Under the design working way of the valves, flow pulsation pressure is relatively large when the main valve opens to 70% and the auxiliary valves open to 100%. The maximum root mean square value of the pulsation pressure is 4.51 m water head. And valve cavitation bubbles collapse near the pipe wall under the initial

branch pipe body, which raises the terrible vibration and the cavitation erosion damage of the pipes and valves.



Figure 3. The physical model of emptying valves



Figure 4. Spiral pattern of the winding pipe

Two kinds of measures were proposed against the disadvantageous flow pattern and cavitation impact through the model test. At first, sudden enlarged body behind the valve was taken to replace the branch pipes to solve the spiral flow problem. The body variation is shown in Figure 5. After using the new body, the flow pattern behind the valve was improved evidently and the pulsation pressure behind the valve decreased obviously. The maximum root mean square value of pulsation pressure behind the main valve and auxiliary valves was 0.98 m water head and appeared in the condition of 70% opening of the main valve and 100% opening of auxiliary valves. The pulsation pressure under the sudden enlarged body was reduced by about 80% than that of the branch pipe body. The critical cavitation numbers of the main valve under the two different bodies were studied through decompression model test. The critical cavitation numbers of the main valve are different in large opening. The critical cavitation number under the sudden enlarged body is relatively smaller than that under the old body. So, the anti-cavitation performance of the new body is better. Secondly, the forced aeration measure before the valves was proposed to solve the valve cavitation problem. The distribution of aeration pipes is shown in Figure 6. Decompression model test shows the valve cavitation is avoided when the aeration air flow reaches 0.025m³/s which is about 0.3% of the main flow.



(a) The oringinal branch pipe body (b) Sudden enlarged body **Figure 5**. Body variation behind the filling valves



Figure 6. Distribution of the aeration pipe

4 PRACTICAL EFFECT OF ANTI-CAVITATION AND DAMPING MEASURES

According to the physical model test result and the project real condition, different technical measures were employed for filling valves and emptying valves finally. The combined measure of a little forced aeration before valves and sudden enlarged body behind valves was adopted for filling valves, because the aeration air flow should be controlled to make sure the running of the ship lift is not affected by the aerated air. The measure of large flow forced aeration before valves is adopted for emptying valves as the aerated flow will go to the downstream river.

Then, the ship lift water delivery system was remodeled follow the project measures. After it was finished, a series of prototype observations were carried out to evaluate the effect of the anti-cavitation and damping measures.

4.1 The filling valve

4.1.1 Effect of the sudden enlarged body

It is approved through prototype observation, that the flow pattern has been improved using the sudden enlarged body instead of the branch pipe body. The root mean square of pulsation pressure of the same measuring point on the new body was reduced by 50%~95% in the same condition.

The anti-cavitation performance of the main valve is enhanced by using the sudden enlarged body. Running in the same design way, the cavitation appeared when the main valve opened to 60%, that is, the critical cavitation opening of the main valve increased. The water head that the filling valve could bear increased, and that of the auxiliary valves increased by about 17.0 m and that of the main valve increased by about 5.5 m. The minimum relative cavitation number of the main valve increased by 20%~40%.

Damping effect of the sudden enlarged body is outstanding. There was a section of short pipe behind the main valve which was not changed. So, this short pipe vibration was taken for example. Compared with the original branch pipe body, maximum vertical acceleration amplitude of the short pipe under the sudden enlarged body was about 2g reduced from about 10g under the branch pipe body in the same condition. The two horizontal accelerations decreased to about 1g from about 3g. Vibration of other more measuring points decreased by more than 50%.

4.1.2 Effect of forced aeration

Based on the sudden enlarged body, anti-cavitation and damping effect of forced aeration measure is also approved. The environment noise is evidently reduced with aeration before the valves. The process of water delivery was very stable. The ventilation flow was about 0.15 m³/s and the anti-cavitation effect of aeration is very outstanding. In the same running way, the observed average flow noise strength of auxiliary valves without aeration was 50 Pa and maximum value was about 100 Pa, while the average flow noise strength with aeration was 20 Pa. The average flow noise strength of the main valve without aeration was 80 Pa and maximum value aeration collapse "crackling" noise can be heard in the valve room at the 60% opening of the main valve. After aeration, flow noise strength decreased obviously and the maximum flow noise strength was about 80 Pa and the cavitation noise was not found by the observation device. Meanwhile, the vibration of the structure decreased evidently with aeration. Acceleration root mean square (short for RMS) comparison between model with aeration and without aeration is shown in Figure 7. Acceleration of most of the structures decreases by 80%~90%, especially at the top of the sudden enlarged body.



Figure 7. Acceleration RMS comparison between model with aeration and without aeration

4.2 The emptying valve

Similar to the filling valve, the working condition of the emptying valve is remarkably improved with aeration. The opening and born water head of the main valve and auxiliary valves and the water delivery efficiency increase obviously. The opening of the main valve can reach up to 70% from the maximum opening and 45% without aeration. This operation way can meet the requirement of running time and speed of the ship lift. In the same operation way, cavitation happens in the 45% opening of the main valve without aeration and the average strength and maximum strength of the observed flow noise were 100 Pa and 200 Pa respectively. When the forced aeration measure was employed, the water head in the same opening increased about 10~15 m. The main valve cavitation was completely suppressed and the average flow noise strength was just 15 Pa. Acceleration of the pressure pipe behind the main valve was up to 24g when the main valve opened to 45% without aeration and cavitation took place, while acceleration of the same position became just 1.4g after the aeration measure was adopted. Acceleration root mean square (short for RMS) comparison whether aeration is shown in Figure 8. The vibration root mean square value decreased more than 80% after aeration. The cavitation and impulse vibration were well resolved by taking the forced aeration measure before the main valve.



Figure 8. Acceleration RMS comparison whether aeration

5 CONCLUSIONS

Model test and prototype observation on the cavitation and vibration characteristics of high-head industrial valves and pressure pipes have been carried out. Some conclusions are obtained.

- (1) Vibration induced by cavitation shows impulse characteristics and vibration induced by normal pulsation pressure shows stable random characteristics.
- (2) Based on model test, spiral pattern of bend pipe behind the valves and cavitation of main valves are the two main reasons of severe vibration of the structures.

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

- (3) Different measures were put forward according to the different working conditions of the water filling valve and the emptying valve. The combined measures of sudden enlarged body behind the valve and forced aeration before the valves were proposed for the water filling valve, while the measure of forced aeration before the valves was also proposed for the water emptying valve.
- (4) After reconstruction of the anti-cavitation measures of the project, the valve cavitation and vibration were properly resolved. Valves could run smoothly at large opening and the cavitation phenomenon disappeared. The vibration responses of the valves and pressure pipes were reduced by about 80%~90%.

ACKNOWLEDGMENTS

The research reported herein was funded by the National Natural Science Foundation of China (Grant No. 51479124, 51479125) and the National Key Research and Development Program of China (Grant No. 2016YFC0402002).

REFERENCES

Wang, X. & Luo, S.Z. (2012). Flow-Induced Vibration Study of Tunnel Spillway Working Gate on One Reservoir. *Applied Mechanics and Materials*, 16(13), 226-228.

Wang, X. & Yan, X.J. (2013). Dynamic Optimization and Flow-Induced Vibration Study on Plate Valve of Ship Lock. *Port & Waterway Engineering*, 12, 151-154.

Wang, X., Li, T.C. & Zhao, L.H. (2009). Vibration Analysis of Large Bulb Tubular Pump House Under Pressure Pulsations. *Water Science and Engineering*, 2(1), 86-94.

Wang, G.Y., Tao, L., Liu, S.Y. & Qian, J.J. (2005). Characteristics of Vibration Induced by Cavitation. *Journal* of *Beijing Institute of Technology*, 4(4), 411-415.

Long, X.P., Wang, F.J. & Guan, Y.S. (2008). Experiment Study on Reduction of Cavitation Induced Vibration and Noise of Jet Pump by Air Supply Method. *Journal of Vibration and Shock*, 27(10), 32-35.

Wang, X., Luo, S.Z., Liu, G.S., Zhang, L.C. & Wang, Y. (2014). Abrasion Test of Flexible Protective Materials on Hydraulic Structures. *Water Science and Engineering*, 7(1), 106-116.

ANALYSIS OF THE HYDRAULIC PERFORMANCES OF PERFORATED PLATES BASED ON EXPERIMENTAL DATA

ALESSANDRO PAGANO⁽¹⁾ & UMBERTO FRATINO⁽²⁾

^(1,2) DICATECh – Politecnico di Bari, Bari, Italy alessandro.pagano@poliba.it

ABSTRACT

Perforated plates are widely adopted in hydraulic pressurized systems mainly as flow conditioner or, when coupled with control devices, to originate a fixed pressure loss for limiting the onset and the intensity of cavitation phenomena. Several studies were performed through numerical models and laboratory activities to describe key aspects of the hydraulics of single and multi-hole orifices. Particularly, most of the studies aimed to identify the hydraulic conditions affecting their behavior in terms of either dissipative performances or cavitation development. The hydraulic behavior of such devices mainly depends on geometrical features, such as the equivalent diameter ratio β , the dimensionless plate thickness t/d_h , the number n_h and disposition of the holes. Due to the multiplicity of influential parameters and the complexity deriving from the joint analysis of their role, multiple tests are required in order to cover a sufficiently wide range of configurations. Pressure scale effects may also affect the performances of these devices. New experimental data, collected through tests which were carried out at Politecnico di Bari, were integrated to the results on recent studies in order to provide additional information. The present paper focuses on single-hole orifices which were tested in variable flow conditions. Some findings for orifices where β ranges between 0.3 and 0.63 and t/d_b ranges between 0.1 and 0.36 were discussed. The remarkable dependency of the dissipation characteristics of the devices, expressed in terms of pressure loss coefficient E_u , discharge coefficient C_d , and cavitation intensity upon the above mentioned geometrical parameters were discussed.

Keywords: Perforated plates; experimental setup; dissipative performances; cavitation detection.

1 INTRODUCTION

Single-hole and multi-hole plates are widely used in pressurized systems. They are mainly adopted as flowmeters, but their operation may support the efficiency of pressurized systems. Particularly, they can be adopted as flow conditioners to establish a fixed head loss in pressurized pipes, guaranteeing an alignment of flow streamlines and reducing cavitation risk when coupled with a control valve (e.g. Tullis, 1989). One of the applications is to dissipate disturbances due to obstructions and pipe bends. Another application is to diminish the length of straight pipe upstream of a flow measurement device (Bayazit et al., 2014). They are also widely used in order to delay the onset of turbulence and to attenuate cavitation (Ozahi, 2015).

The dissipation performances of these devices have been largely studied, mainly focusing on the pressure drop estimation and on the description of the flow field characteristics across the device. Many studies are also available regarding cavitation intensity analysis. Both experimental and numerical approaches were used. Milestone equations (Idelchik, 1986; Miller, 1990) were defined for estimating the dissipation efficiency of perforated plates, valid in cases of single-hole devices and fully developed flow conditions. Gan and Riffat (1997) and Erdal (1997) used CFD codes for defining the relationship between induced pressure drop and equivalent diameter ratio β , dimensionless plate thickness t/d_h as well as the number and disposition of the holes. Weber et al. (2000) summarized the literature data on the pressure losses through perforated plates by integrating existing experimental data and new experimental tests. Zhang and Cai (1999) proposed the use of the C_{drop} coefficient for estimating the role played by the abruptness of the transition. Fratino (2000) showed that for fixed β , the increase of the number of holes and/or of the dimensionless plate thickness produced a significant decrease in the dissipating efficiency but better cavitation performances. Moreover, the paper confirmed the absence of pressure scale effects on incipient and critical cavitation limits, as suggested by Govindarajan (1972) and Tullis and Govindarajan (1973).

Recently, Malavasi et al. (2008; 2010) investigated the behavior of sharp-edged perforated plates, numerically confirming the experimental findings. Zhao et al. (2011) studied the dissipation characteristics of multi-hole orifices and introduced an empirical formula for estimating the pressure drop. Holt et al. (2011) analyzed the dissipation and cavitation efficiency of baffle plates in circular pipes, while Wu et al. (2011) and Ozahi (2015) introduced new equations which were derived by experimental data for calculating the pressure loss coefficient. Many experimental studies were carried out mainly focusing on cavitation (e.g. Fratino, 2000; Testud et al., 2011; Maynes et al., 2013; Malavasi et al., 2015).

Both dissipation and cavitation characteristics are mainly affected by the geometrical characteristics of the device, namely: the equivalent diameter ratio (β), the dimensionless plate thickness (t/d_h), the number (n_h), disposition and the shape of the holes. The operating conditions, expressed through the Reynolds number Re, are also relevant. The role of operating pressure is crucial as well, specifically for cavitation inception and is described using a scale effect coefficient. Since a significant number of geometrical and flow parameters concurrently affect the hydraulic performances of these devices, several tests are required to hydraulically characterize them.

Several experimental tests have been recently carried out in the hydraulic laboratory of DICATECh. at Politecnico di Bari, with the aim to provide innovative design criteria for a correct assessment of the performances of perforated plates in terms of dissipating efficiency (Fratino and Pagano, 2011; Malavasi et al., 2012) and of onset (and intensity) of cavitation phenomena (Malavasi et al., 2015).

2 LABORATORY SETUP

The experimental tests were conducted at Politecnico di Bari in the hydraulic laboratory of DICATECh. The laboratory setup, sketched in Fig. 1, was composed of 100 mm and 200 mm steel pipes, with a pumping system working with a maximum pressure of approximately 9.0 bar and flow rates up to 100 l/s. For this campaign, the 100 mm pipe was used. Pressures were measured in multiple locations in order to evaluate the gross head loss across the device and downstream the plate. Pressure taps were located 1D upstream and 10D downstream the device and were used for measuring the gross pressure drop. Additional measurements were performed at 0.5D, 1D, 2D, 3D, 4D, 5D and 7D downstream the device for estimating the pressure profile downstream the plate. The pressure drop was measured by a mercury differential manometer and by Burdon tube pressure gauges. The flow rate was evaluated using an electromagnetic flow meter and checked using a volumetric tank. Cavitation onset and intensity were detected using acceleration measurements, according to ISA standard procedure. For this purpose, a Brüel & Kjær 4397 accelerometer with sensitivity of 1.02 mV/(m/s²) and resolution of 0.025 m/s² was used and placed on the pipe approximately 1D downstream the device. Additional measurements on cavitation were taken in order to further improve the reliability of the results by using a Brüel & Kjær 2513 portable integrating vibration meter. At the same time, Sound Pressure Level (SPL) was recorded by a Brüel & Kjær4191 Falcon Range microphone.

The tests for a single plate were replicated in a wide range of testing conditions identified by the Reynolds numbers values. The incipient cavitation number σ_i was estimated using ISA standard procedure.

The main aim of the present experimental campaign is to integrate the available dataset discussed in some other papers by the same authors (Fratino and Pagano, 2011; Malavasi et al., 2012; Malavasi et al., 2015) with additional information specifically related to single-hole devices, whose geometrical characteristics are summarized in Table 1.



Figure 1. The sketch of the testing facilities at the hydraulic laboratory at DICATECh (Politecnico di Bari).

Plate ID	Nh	<i>d_h</i> [mm]	t/d _h	β	d _h ∕D
1	1	28.20	0.36	0.30	0.28
2	1	28.20	0.21	0.30	0.28
3	1	48.00	0.13	0.51	0.48
4	1	48.00	0.21	0.51	0.48
5	1	59.00	0.17	0.63	0.59
6	1	59.00	0.10	0.63	0.59

 Table 1. Summary of the main characteristics of the tested devices.

3 RESULTS AND DISCUSSIONS

3.1 Dissipative performances

The dissipation performances of perforated plates are generally estimated using the dimensionless pressure loss coefficient K (equivalently identified as Euler number Eu), that is defined as follows:

$$K = \frac{\Delta P}{\frac{1}{2}\rho v^2}$$
[1]

where ΔP is the net pressure drop across the device, ρ is the fluid density and *V* is the pipe bulk-mean velocity. Reference can also be made to the discharge coefficient C_d , defined as the ratio between the actual flow rate through the orifice and the theoretical volume flow rate, as computed according to Bernoulli equation. It ranges between 0 and 1 and presents the following equation:

$$C_{d} = \frac{v}{\sqrt{\frac{2\Delta\Delta}{\rho} + v^{2}}} = \frac{1}{\sqrt{1 + K}}$$
[2]

If cavitation were not present, both *K* and C_d are influenced only by the geometry of the device, defined by the following groups according to the available literature: 1) the equivalent diameter ratio β , that is the square root of the total open area ratio, 2) the dimensionless thickness t/d_h , i.e. the ratio between the plate thickness *t* and the hole diameter d_h , 3) the number of holes n_h , 4) the distribution of the holes (in case of multi-hole devices), 5) the shape of the holes.

The Reynolds number has an influence as well, but a range of self-similarity with respect to it can be detected. Particularly, in this paper, both K and C_d were calculated as the mean values in the self-similarity region. The following Fig. 2 summarizes the results in the case of plate n. 3.



Figure 2. Plate n. 3: *K* and *C_d* vs. Re. ©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print) It is widely acknowledged that the main dependence of the pressure loss coefficient *K* is related to β . Starting from the work by Malavasi et al. (2012), Figure 3 integrates most of the literature evidences (both experimental data and equations) with the new experimental data (in red). It is worth to mention that, in the cited paper, no single-hole plates were taken into account, but this limitation is partially overcome with the present work.



Figure 3. Values of *K* versus β for the tested plates and other experimental data.

Referring to the equations, it is worth to emphasize that all the equations take into account the dependence upon the relative thickness, but for the sake of simplicity, the equations were plotted assuming $t/d_h=0.5$. The results confirm that the equivalent diameter ratio is the key geometrical aspect, although a significant data dispersion can be found for the same β . Nevertheless, considering the large scale of Figure 3, the behavior of most points seems homogeneous and the introduction of the new experimental data provides a remarkable agreement, specifically with Idelchik's equation.

 C_d values were also plotted and compared with the analytical ones obtained by Idelchik and Miller equations. The results are summarized in Fig. 4.

The orifices were also tested for estimating the pressure recovery length, which is a crucial design parameter. Referring to the plate n. 3, Figure 5 shows the ratio between the pressure drop ΔP and the downstream pressure value P_D (1.2 bar) versus the dimensionless distance from the plate section x/D. The following Figure 5 represents the curves corresponding to a subset of different values of the Reynolds number, showing that the pressure recovery length is limitedly dependent upon the *Re*, except for high values of *Re* which also correspond to developed cavitation conditions. The dimensionless reattaching length is approximately 2.5-3 D.



 Figure 4. Values of C_d versus β for the tested plates and comparison with Idelchik and Miller equations.

 3098
 ©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)



Figure 5. Plate n. 3: dimensionless pressure profiles at different Re values.[$P_D = 1.2$ bar].

3.2 Cavitation intensity

The onset and development of cavitation phenomena are always undesired since they highly influence the flow dynamics and determines potential damages on devices and pipes. The detection of cavitation limits usually involves both vibration and noise measurements, which are strictly connected with the implosion of bubbles. In fact, the noise is originated by bubble implosion, with a change of the frequency of the elastic waves, whereas vibrations arise since the pressure generated by the bubble collapses.

Different cavitation regimes can be identified by measuring its indirect effects on hydraulic systems: 1) REGIME I: absence of cavitation; 2) REGIME II: incipient cavitation with the development and the collapse of small vapor bubbles in the flow stream; 3) REGIME III: constant cavitation, involving enough vapor to produce a constant level of cavitation; 4) REGIME IV: maximum vibration, that is, the level of cavitation associated with occurrence of choking condition. Since the determination of the cavitation intensity for hydraulic devices is not a straightforward task, the correct operation of hydraulic devices requires typically the identification of the incipient cavitation level (REGIME II) as the acceptable limit in the design phase. Cavitation is usually studied by means of the cavitation number, defined as follows in ISA standard:

$$\sigma = \frac{P_u - P_w}{P_u - P_d}$$
[3]

where P_u and P_d are the pressures measured sufficiently far upstream and downstream of the device and P_w is the vapor pressure. Similar to the dissipative performances, cavitation inception is also affected by the geometry of the devices (Malavasi et al., 2015).

Cavitation regimes were detected according to ISA procedure. Particularly, in a log-log plot, acceleration measures are reported versus σ (see Fig. 6) and three linear trends of data are generally identified. The incipient cavitation number is defined as the intersection of the first two trends (see Fig. 6). Tests were also replicated by considering several fixed downstream pressure values, in order to make the results independent from possible uncertainties due to the pressure scale effects.



Figure 6. Detection of incipient and critical cavitation limits for the plate n.3.

As mentioned before, the detection of incipient cavitation limit is particularly relevant, in order to ensure cavitation free operation for hydraulic devices. Based on the work by Malavasi et al. (2015), a formula relating σ_i /SSE (the value of incipient cavitation limit modified to take into account the size scale effect coefficient proposed by Rahmeyer, 1980) to C_d can be used to estimate σ_i . The proposed equation, obtained by fitting a wide dataset, is reported in the following:

$$\frac{\sigma_i}{SSE} = 2.10 + 6.75 \cdot C_d - 1.99 \cdot C_d^2 + 4.55 \cdot C_d^3$$
 [4]

It was applied to estimate the incipient cavitation limit using the current experimental dataset. Measured and calculated values, represented in Fig. 7, show a quite good agreement (error is lower than 20%).



Figure 7. Comparison between the measured and computed values of incipient cavitation limits according to the equation by Malavasi et al. (2015).

4 CONCLUSIONS

The behavior of perforated plates is not a straightforward task, due to the multiple dependencies on the geometrical features and on the hydraulic conditions at which the hydraulic system works. Several tests were thus required in order to have a comprehensive idea of their functioning in a wide range of hydraulic conditions.

From the evidences of studies recently carried out, the present paper aims at including new data, checking for their consistency. Particularly, differently shaped single-hole plates were tested in the hydraulic laboratory at Politecnico di Bari. The dissipative performances were assessed and hence provided a comprehensive comparison with literature data, while cavitation inception was analyzed by defining the dependency of σ_i on C_d using the equation proposed by Malavasi et al. (2015). These showed that the collected experimental data are coherent with the existing dataset.

REFERENCES

Bayazit, Y., Sparrow, E.M. & Joseph, D.D. (2014). Perforated Plates for Fluid Management: Plate Geometry Effects and Flow Regimes, *International Journal of Thermal Sciences*, 85, 104-111.

Erdal, A. (1997). A Numerical Investigation of Different Parameters that Affect the Performance of A Flow Conditioner. *Flow Measurement and Instrumentation*, 8(2), 93–102.

Fratino, U. (2000). Hydraulic and Cavitation Characteristics of Multihole Orifices, *Machinery and System - 20th IAHR Symposium*. Charlotte (NC).

Fratino, U., & Pagano, A. (2011). Head Loss Coefficient of Orifice Plate Energy Dissipator-Discussion. *Journal* of Hydraulic Research, 49(6), 830-831.

Gan, G. & Riffat, S.B. (1997). Pressure Loss Characteristics of Orifice and Perforated Plates. *Experimental Thermal and Fluid Science*, 14(6), 160–165.

Govindarajan, R. (1972). Cavitation Size Scale Effects, *PhD Thesis*. Colorado State University, Fort Collins (CO).

Holt, G.J., Maynes, D. & Blotter, J. (2011). Cavitation at Sharp Edge Multi-Hole Baffle Plates. *Proceedings of the ASME 2011 international mechanical engineering congress & exposition IMECE2011*.

Idelchik, I.E. (1986). Handbook of Hydraulic Resistance. Washington (DC), USA: Hemisphere.

Malavasi, S., Macchi, S. & Mereghetti, E. (2008). Cavitation and Dissipation Efficiency of Multihole Orifices. 9th International Conference on Flow-Induced Vibrations FIV2008, Prague (CZ).

- Malavasi, S., Messa, G. & Macchi, S. (2010). The Pressure Drop Coefficient through Sharp-Edged Perforated Plates, *XXXII Convegno Nazionale di Idraulica e Costruzioni Idrauliche*, Palermo (IT).
- Malavasi S., Messa G., Fratino, U. & Pagano, A. (2015). On Cavitation Occurrence in Perforated Plates. *Flow Measurement and Instrumentation*, 41, 129-139.
- Malavasi S., Messa G., Fratino U. & Pagano A. (2012). On the Pressure Losses through Perforated Plates. *Flow Measurement and Instrumentation*, 28, 57-66.

Maynes D., Holt GJ. & Blotter, J. (2013). Cavitation Inception and Head Loss Due to Liquid Flow through Perforated Plates of Varying Thickness. *ASME Journal of Fluid Engineering*, 135(3), 131302.

Miller, D.S. (1990) Internal flow system. Bedford, UK: Cranfield.

- Ozahi, E. (2015). An analysis on the Pressure Loss through Perforated Plates at Moderate Reynolds Numbers in Turbulent Flow Regime, *Flow Measurement and Instrumentation*, 43, 6-13.
- Testud, P., Massou, P., Hirschberg, A. & Auregan, Y. (2007). Noise Generated by Cavitating Single-Hole and Multi-Hole Orifices in a Water Pipe. *J. Fluid Struct*, 23(2), 163–89.
- Tullis, J.P. (1989). *Hydraulics of Pipelines. Pumps, valves, Cavitation, Transients*. New York, USA: John Wiley & Sons.
- Tullis, J.P. & Govindarajan, R. (1973). Cavitation and Scale Effects for Orifices, *Journal of the Hydraulic Division*, 99(HY3), 417-430.
- Wu, J., Ai, W. & Zhou, Q. (2010). Head Loss Coefficient of Orifice Plate Energy Dissipater. Journal of Hydraulic Research, 48(4), 526-530.
- Weber, L.J., Cherian, M.P., Allen, M.E. & Muste, M. (2000). *Headloss Characteristics for Perforated Plates and Flat Bar Screens*, Technical report. Iowa City (IA), USA:Iowa Institute of Hydraulic Engineering, College of Engineering, University of Iowa. Mar. Report no. 411; 2000.
- Zhang, Z. & Cai, J. (1999). Compromise Orifice Geometry to Minimize Pressure Drop, *Journal of Hydraulic Engineering*, 125(11), 1150-1153.
- Zhao, T., Zhang, J. & Ma, L. (2011). A General Structural Design Methodology for Multi-Hole Orifices and its Experimental Application. *Journal of Mechanical Science and Technology*; 25(9), 2237–2246.

EXPERIMENTAL INVESTIGATION OF THE EFFECTS OF INTAKE DESIGNS ON COMPACT TURBINES

DANIEL INNERHOFER⁽¹⁾ & MARKUS AUFLEGER⁽²⁾

^(1,2) Unit of Hydraulic Engineering, University of Innsbruck, Austria, e-mail: Daniel.Innerhofer@uibk.ac.at, markus.aufleger@uibk.ac.at

ABSTRACT

New types of compact turbines like the Stream Diver[™] from Voith allow new designs for the intake structures of hydro power plants. By reducing the dimension and complexity of these structures, the construction costs can be reduced and hence power plants with low heads can also achieve high cost effectiveness. However, along comes the need to better understand the influence of the inflow conditions on the energy yield of such unregulated turbines. Since it is known that a distorted flow pattern in front of the turbine can cause negative effects like hydraulic losses, vibrations and a poorer performance of the turbine, a careful design of the inlet structure is needed. For this purpose, the University of Innsbruck's Hydraulic Engineering Unit, together with the energy supplier Grenzkraftwerke Ltd and the turbine manufacturer Kössler Ltd, created a new test rig. In this large-scale model tests, a real turbine in an artificial channel is used. This set up allows the measurement not only of the turbine efficiency but of the overall efficiency in energy production of a run-of-river plant for different intake designs. In this paper, the focus lies on low head power plants, showing various possible designs for these power plants which also meet the criteria for an environmentally sustainable use of hydro power. Based on the findings of this research project, the effect of the various parts of the inlet structure on the efficiency can be evaluated and the use of a complex geometry can therefore be discussed. The test set up and first results are presented within this work.

Keywords: Hydropower; Scale model tests; Compact-turbines; Renewable Energy; Run-of-river power plant.

1 INTRODUCTION

One of the greatest challenges for hydro power plants with very low heads, from two up to five meter, is to be cost effective. This may be achieved by a high efficiency in energy production and reducing the construction costs. To lower the costs, new designs for the power plants must be found apart from the traditional designs to simplify and reduce the size of the structures needed and take account of the ecological issues as well. Some of these plant designs have been already approved like the river flow power plant (Brinkmeier, 2012).

In order to get a high efficiency, it is possible to access new specialized turbine technologies, but it is also necessary to better understand the influence of distorted inflow conditions on the energy conversion of the turbines. Especially for low head power plants, the flow pattern near the inlet structure is of great importance (Ruprecht and Göde, 2002) due to the low heads even small losses in hydraulic head mean a relatively high loss in energy production.

In the recent years, a lot of research has been done on this topic, summarized by Liu et al. (2015) and it is commonly known that flow disturbances in front of turbines can influence their operational behavior and performance. According to Vischer and Hager (1999) and Lichtneger (2011), an inhomogeneous flow or velocity distribution in the section of the guide vanes may cause losses in efficiency or vibrations, leading to damages of the machinery. There exist many criteria to evaluate these phenomena like the one proposed by Fisher and Franke (1987). These criteria, e.g. summarized by Godde (1994), are supposed to check if the inflow fulfills all necessary requirements for a smooth operation of the turbine, but they are all mainly an empirical based qualitative analysis and not standardized yet. Hence this is a field of ongoing research as there are still uncertainties regarding how big the effective influence of unfavourable flow conditions on the energy production really is. For example, Fernando and Rival (2014) investigated different intake structures for very low head conditions and Ferro et al. (2010) focused on the influence of inlet guide vanes for mini hydro turbines, which should allow using hydro power with low investment costs.

The objectives of this study (Innerhofer et al., 2015) are to compare and benchmark different designs of power plants and inlet structures, for sites with only low heads available, regarding their energy and cost effectiveness in order to investigate if new power plant concepts can compete with the traditional designs. The main problem of simplified inlet structures, like the ones presented in this work, is that they may not be able to provide the high quality flow pattern that is requested by the turbine manufacturers: an undistorted flow with a homogenous velocity distribution in front of the turbine. In a worst case scenario, they may even create additional turbulences and flow distortions. However, Pugh (1983) states that simplifying the design and

construction of turbine intakes could lead to savings in both material and labor. Therefore, it has to be investigated if these advantages can compensate the lower energy production over the life span of the power plant.

For this purpose, it was chosen to construct a physical scale model test with a sectional model of a runof-river power plant to be tested with different intake designs, using the same turbine at the same hydraulic boundary conditions for each.

2 COMPACT TURBINES

The Stream Diver[™] turbine from Voith is a relatively new type of compact turbine. Its advanced yet simple design offers a lot of attributes that makes it even useable for sites where conventional hydro power plants are not viable, e.g. sites with very low heads, unconventional arrangement situations (e.g. below the gate of a weir) or where only a flat foundation can be realized (Aufleger and Brinkmeier, 2015). Its comparable small runner diameters make it suitable for shallow waters and low heads. The turbine is characterized by a very compact design, where the turbine and generator form a single unit and are installed directly in the water. As it is equipped with fixed guide vanes and fixed rotor blades, it runs completely oil-free and dispenses the need of additional infrastructure like hydraulic aggregates, therefore greatly reducing the installation costs and space needed for the power plant. Another key feature is the use of multiple turbine units instead of one conventional big turbine, this makes it possible to recover single units for maintenance without the need of shutting down the whole plant and power production.



Figure 1. Left side: full scale prototype of the Stream Diver. Right side: typical application for the turbine. (Photo credit: Voith).

3 METHODS

3.1 Experimental setup

For the experimental investigations, a test rig, for large scale model tests, was built in the hydraulic laboratory of the University of Innsbruck. It consists of an artificial channel in which is installed a fully functional turbine in a scale of 1:4. The flume has a length of 16 meter and a depth of 2.5 meter. This helps to greatly minimize scaling effects and allows a comprehensible investigation of the flow conditions and turbulences near the inlet structures and the turbine. The flume allows the observation of the flow conditions as one side of the channel is made of Perspex. This experimental setup distinguishes itself from the regular tests for turbine design by putting the turbine not into a closed piping, with ideal flow conditions, but in an freesurface channel and thereby simulating the inflow of a real hydro power plant, where there is a direct interaction between flow distortions and power generation of the turbine. Therefore, it is possible to benchmark different designs for the power plant and to see how well the turbine performs in combination with different inlet structures. For the model test, a sectional model of the power plant was used, therefor one turbine field consisting of upstream floor, inlet structure with the turbine and draft tube were cut out. This allows a large scale factor of the model but limits it to a planar investigation of the problem, as it postulates symmetry and no direct interference between multiple turbines in a row. For some of the presented models in Figure 3 where the turbine is placed in a duct, this may be true but for Model 1 and 2, this has to be investigated further. 3D numerical simulations via CFD seem therefor an appropriate way to analyze if an interaction consist between multiple turbines and will subsequently be done to complement the results shown in this work.



Figure 2. The scale model of the turbine used for the experiments and the test rig during a test run.

3.2 Efficiency measurement

The flume was equipped with sensors to measure and log the water levels and discharge. Moreover, it is possible to determine the theoretical power available. The turbine itself was equipped with a torque transducer to measure the actual generated power on the rotor shaft and thus to determine the efficiency.

The theoretical power available to the turbine is the power of the fluid system given by Eq. [1] where Q is the volumetric flow rate and Δp the pressure drop, which can be expressed as the product of density of the fluid, gravitational acceleration and gross head.

$$P_{ideal} = \Delta p \cdot Q = \rho \cdot g \cdot Q \cdot H$$
^[1]

The actual power is given as the product of torque, τ and angular velocity, ω of the rotor shaft:

$$P_{actual} = \tau \cdot \omega = \tau \cdot 2\pi \cdot n$$
^[2]

Hereby, n is the rotational speed of the turbine. The overall efficiency can then be expressed as the ratio of actual power to ideal power:

$$\eta = \frac{P_{actual}}{P_{ideal}}$$
^[3]

This measured efficiency factor includes not only the efficiency of the turbine but also the effects caused from the power plant e.g. head losses from wall friction and turbulence. To ensure that the various designs can be properly compared, the hydraulic boundary conditions and the operation point of the turbine are kept the same for all test runs. For the model tests, the turbine was equipped with an asynchronous generator coupled with a frequency converter to enable speed control of the runner. The basic parameters of the turbine are listed in Table 1. They derive from an actual project site and are scaled 1:4 according to the law of similarity by Froude to ensure the flow conditions in the model scale represent nature as best as possible.

Head Nominal Flow max. Flow Power Runner Diameter [kW] [m] [m³/s] [m³/s] [m] 2.80 - 3.40 10 17.60 250 1.30 Actual 0.70 - 0.85 0.31 0.55 2 0.35 Model

Table 1. Parameters of the turbine for actual and model scale.

3.3 Variation study

In Figure 3, four different inlet designs for a run of river power plant, which are characterized by a different complexity of their geometry and thus the effort for their realization are shown. These variations cover a wide range from a very simple design to a rather complex inlet structure. The idea behind was to start with the simplest geometry possible and then the parts of the inlet structure were added one by one until arriving at the traditional intake design for run of river power plants. They all have in common that they can be overflown and the headwater can be regulated by a flap gate placed on top of the power plant.



Figure 3. Overview of the tested inlet geometries.

Model 1 and Model 2 are rather uncommon designs where the turbine stands free in the headwater and the water flows in an uncontrolled way from all sides to the guide vanes. This arrangement of the turbine grants easy access for mounting by a crane and later revision of the machinery, these models are also typically coming along with the smallest necessary concrete volume. While Model 1 represents a more theoretical design that could be used for special applications like the use in tail water or cooling water channels, the other models are designed for natural rivers. In particular Model 2 seems to be the simplest design for realization in a river, as normally an inlet sill would be necessary to prevent sand and stones from passing through the turbine.

Model 3 forms a rectangular chamber in which the turbine is situated. This layout shows a technically inconsistent gap between the intake collar of the turbine and the contour of the intake. Anyhow, the inflow is already channeled and oriented towards the turbine. In practice, a service tunnel may be integrated in the building part above the turbine from where single units can be accessed. Model 4 is an improvement of the third one and consists of a duct transition from the rectangular cross section to the circular wicket gate. This layout has to be considered as the more complex and cost intensive shape to build but on the other hand it creates a straight flow pattern and continuous acceleration of the flow towards the turbine and is therefore typically used at run of river power plants with Kaplan turbines.

4 RESULTS AND DISCUSSION

For each of the four models, many single test runs were carried out with varying head levels and discharges in order to cover a wide range of operating conditions. The results of the single test runs were then merged and the effectivity was evaluated to get a characteristic diagram for each power plant design. The maximum efficiency for the given data set is offset to a value of 100% and is presented here versus the total head. The curves obtained can be compared and indicated as to which configuration works better for the given boundary conditions. In Figure 4, these curves are shown for three models, since Model 1 has to be considered a preliminary test. The results presented within this work come from tests where the flap gate on top of the power plant was raised in order to prevent a flow over the power plant. This makes it easier to compare the results as the effects of an overflow, e.g. a loss of energy because part of the discharge bypasses the turbine and additional turbulences behind the draft tube, can be factored out.



Figure 4. Efficiency factor versus head in model scale for the different inlet designs. ©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

It is notable that there is quite a difference in the results of the tests with higher heads, where all the models performed similar to a certain point and the test with lower heads where there is a divergence of the efficiency up to 3.5 %. An explanation may be that the local losses caused from the inlet have a higher relative percentage for low heads. This shows the importance of an accurate investigation of the inlet structure used, since poor shaping can cause more negative effects on energy yield than a complete renouncement of any structure.

A bit surprisingly is that Model 2 achieved the highest efficiency, for it was expected that Model 4 would act the best. This may be due to the fact that for Model 3 and Model 4, the flow deviation and higher flow velocities caused by the duct induced more local losses and turbulence. Especially for Model 3, which shows the lowest efficiency throughout, the formation of eddies in the area of the gap in front of the turbine could be observed.

As already mentioned, the presented efficiency factor is very comprehensive as it contains not only all direct and indirect loss-causing effects, such as head losses due to turbulences but also flow pattern that influence the efficiency of the turbine itself. Hence, it is the goal to differ which effect causes the change in efficiency. The turbine is completely unregulated. Thus, the flow rate through the turbine depends on the pressure gradient and subsequently the head. Therefore, it was checked to see if the correlation between flow rate and gross head changes for the different models. This is because a change in the flow rate through the turbine would result in a change in the efficiency because of its characteristic curve. Thus, a head loss can also cause a different efficiency due to a changed Q to H ratio. However, if for the same gross head for the different geometries result almost the same flow rate, the turbine runs at a similar operating point. If there is nevertheless a significant difference in efficiency is not caused only by a head loss. In Figure 5, the corresponding values for the three models are shown and it can be seen that the difference is always less than 1 % of the total discharge, whilst the efficiency factor changes for serval percentage points. Therefore, the discrepancy in energy conversion does not comes only from a change of the hydraulic loss but from the flow distortions induced by the inlet structure.



Figure 5. Interpolation of flow rate versus head.

Observations during the tests also show that a problem due to low head water levels is a high tendency for vortices to occur. Particularly when the water level falls below critical height these vortices can be quasi-stable and cause air entrainment. Such phenomena can cause efficiency to decline and even be harmful to the machinery, therefore, the influence on the long term behavior of the turbine has to be evaluated and taken into account as well. This was mainly observed for Model 2, while for Model 3 and Model 4 the duct could limit the vortices. Tests also show that a possible way to prevent vortices and air entrainment is the overflow over the power plant. Even if this means that part of the discharge is lost for energy production, an overflow may be wanted as it can be used to practically hide the structure in the water and for downstream fish migration.

5 CONCLUSIONS

The results of the model tests confirm that there is an influence of the design of the inlet structure on the efficiency but they also show that this influence turns out to be less than expected. Though it should always be a goal to avoid flow distortions in front of the turbine, the turbine performed well with all of the tested geometries for the inlet structure. Therefore, it can be stated that, according to the experimental investigations

shown above, simplified inlet structures at typical low head power plants, equipped with compact turbines, can compete with the traditional designs regarding the energy efficiency. However, there are also other important points to consider when designing the inlet structure like the operational behavior of the turbine and construction and maintenance costs.

Further tests and evaluation of the obtainable data are planned and supposed to provide a better understanding on how the inlet structures influence the flow conditions and subsequently how these flow distortions affect the efficiency of the turbine.

ACKNOWLEDGEMENTS

The authors would like to thank the Austrian Research Promotion Agency (FFG) for the funding of the project (839351).

REFERENCES

Aufleger, M. & Brinkmeier, B. (2015): Wasserkraftanlagen mit niedrigen Fallhöhen – Verschiedene Konzepte im kritischen Vergleich. Österreichische Wasser- und Abfallwirtschaft, 67(7), 281-291, [In German].

- Brinkmeier, B. (2012). Wasserkraftnutzung an ökologisch sensiblen und erosionsbedingt sanierungsbedürftigen Standorten: das Konzept des Fließgewässerkraftwerkes, Innsbruck university press (IUP).Series. *Forum Umwelttechnik und Wasserbau*, 16, [In German].
- Fernando, J. & Riva, I.D. (2014). Characterizing the Influence of Upstream Obstacles on Very Low Head Water-Turbine Performance. *Journal of Hydraulic Research*, 52(5), 644-652.
- Ferro, L.M.C., Gato, L.M.C. & Falcao, A.F.O. (2010). Design and Experimental Validation of the Inlet Guide Vane System of a Mini Hydraulic Bulb-Turbine. *Renewable Energy*, 35(9), 1920-1928.

Fisher, R.K. & Franke, G.F. (1987): The Impact of Inlet Flow Characteristics on Low Head Hydro Projects. *Proceeding International Conference on Hydropower*, Portland, 1673-1680.

Godde, D. (1994): Experimentelle Untersuchungen zur Anströmung von Rohrturbinen. Berichte der Versuchsanstalt Obernach und des Lehrstuhls für Wasserbau und Wasserwirtschaft der Technischen Universität München NR. 75, [In German].

Innerhofer, D., Lochschmidt, J., Lampl, J., Wührer, R., Brinkmeier, B. & Aufleger, M. (2015): Anströmung von Kompaktturbinen / Inflow at Compact Turbines. *Österreichische Wasser- und Abfallwirtschaft*, 67(7), 307-314, [In German].

Lichtneger, P. (2011). Untersuchungen zur Optimierung der Einlaufströmung an Niederdruckwasserkraftanlagen, 34. Dresdner Wasserbaukolloquium 2011: Wasserkraft (in Dresdner wasserbauliche Mitteilungen; Heft 45) [In German].

Liu X., Luo Y., Karney, B. & Wang, W. (2015). A Selected Literature Review of Efficiency Improvements in Hydraulic Turbines. *Renewable and Sustainable Energy Reviews*, 51, 18-28.

Ruprecht, A. & Göde, E. (2002): Einlaufgestaltung an Kleinwasserkraftwerken, *5. Internationales Anwenderforum Kleinwasserkraftwerke* [In German].

Pugh, C.A. (1983). *Hydraulic Model Studies on Bulb Turbine Intakes*, REC-ERC-82-14, United States Department of the Interior, Bureau of Reclamation.

Vischer D.L. & Hager W.H. (1999) Dam hydraulics. Wiley, Chichester, 1-328.

DIRECT EROSION PREDICTION USING SPH-FEM COUPLING

WIEBKE BODEN⁽¹⁾, JORGE NUNEZ⁽²⁾, GUILLAUME COUDOUEL⁽³⁾, STEPHANE AUBERT⁽⁴⁾, JEAN-CHRISTOPHE MARONGIU⁽⁵⁾, ALAIN COMBESCURE⁽⁶⁾ & RICHARD PERKINS⁽⁷⁾

> ⁽¹⁾ Andritz HYDRO, Vevey, Switzerland, wiebke.boden@andritz.com
> ^(4,7) Ecole Centrale de Lyon (LMFA), Lyon, France, stephane.aubert@ec-lyon.fr, richard.perkins@ec-lyon.fr
> ⁽⁵⁾ Andritz HYDRO SAS, Villeurbanne, France jean-christophe.marongiu@andritz.com
> ^(2,3,6) Institut National des Sciences Appliquées de Lyon (LaMCos), Villeurbanne, France, Jorge.nunez-ramirez@insa-lyon.fr, alain.combescure@insa-lyon.fr

ABSTRACT

A model for solid sediment transport in the framework of a Smoothed Particle Hydrodynamics (SPH) code in Arbitrary Lagrangian Eulerian (ALE) formulation has been developed to investigate the impact parameters of sediments on a surface. Particle trajectories were calculated by the Lagrangian integration of the particle equation of motion, wall effects and near-wall turbulence were taken into account. The simulations covered the particle size range from 10 µm to 500 µm. Next, the sediments' erosion potential was evaluated based on their impact parameters. The normal component of the sediment impact velocity was identified as the key parameter to estimate erosion on coated Pelton turbine buckets by comparing an oblique to a normal impact. where the normal velocity component was taken identical. These simulations were performed as a solid-solid impact without a fluid phase using a Finite Element (FE) code. Surface damage is supposed to be due to fatigue and evaluated firstly, by the range of stresses occurring in the material and secondly, by the number of impacts until damage. This number arises from a generally valid fatigue criterion for low and high cycle numbers. Subsequent fluid structure interaction (FSI) simulations with a coupling between the SPH and the FE code showed a slightly different deterioration pattern compared to the dry simulations due to the presence of the water jet. Erosion estimation was inferior to the results that were found by the uncoupled approach. Then, a comparison between 2D and 3D dry structure-structure simulations of the sediment impact allows error estimation of the damage evaluation made by the more efficient 2D simulations. Finally, based on all presented results, recommendations for future developments of the direct erosion simulations are made.

Keywords: Smoothed particle hydrodynamics (SPH); solid particle erosion (SPE); fluid structure interaction (FSI), pelton turbines.

1 INTRODUCTION

Solid Particle Erosion (SPE), the deterioration of surfaces by solid particles entrained in liquid, represents a major cause of damages in hydraulic turbines. River water carries a significant load of these solid particles, which is particularly important in periods of heavy rainfall, pronounced glacier melting or severe draughts. Sediment impacts on turbine buckets lead to material loss, secondary damage, decreasing efficiency and in the worst case to severe material failure. Consequently, the hydropower unit has to be shut down for maintenance or replacement, which means an interruption of energy supply and thus a loss of income for the supplier. Pelton turbines are particularly concerned of this damage mechanism, because of high flow velocities and the strong curvature of the buckets, which consequently lead to high sediment impact velocities. Using erosion models as a predicting tool in numerical simulations would fail when applied to Pelton turbines, as they are usually calibrated for much defined flow parameters. The range of flow velocities, flow patterns and sediment sizes found in Pelton turbines, by contrast, require a more general way of erosion estimation. That is the reason that we adopt a different, multiscale approach to estimate erosion based on first principle. Firstly, the behavior of the sediments in the fluid phase was investigated at the macroscale in order to ensure right impact velocities and angles on the surface. Then, the resulting damage of the bucket material was obtained via FE simulations on the microscale.

The topic of blade erosion in turbomachinery was first addressed by Montgomery and Clark (1962) and then by Sage and Tilly (1969), both for gas turbine applications. Finnie (1960) and Bitter (1963a,b) developed general erosion models for ductile and brittle materials, whereas Grant and Tabakoff (1975) were the first to develop a general erosion model specified for gas turbine blades of ductile material. Erosion estimation by these models depends mainly on the impact velocity and impact angle of the solid particles as well as empirically or experimentally calibrated constants, and small variations in those parameters can produce large variations in the predicted erosion. Bergeron et al. (2002) obtained specific erosion rates for a Francis turbine

by simulating the particle movement and the transformation of their kinetic energy upon impact. They found that predicted regions of erosion corresponded to observed ones in real machines. Ghenaiet (2015) studied erosion on the same turbine type by applying a Grant-Tabakoff model derivation. Erosion rates were calculated from impact velocities and directions. Neither of these two studies includes quantitative comparisons with measured erosion rates. Recently, the number of erosion studies devoted especially to Pelton turbines increased. Felix et al. (2016) calibrated the semi-empirical equations proposed by the International Electrical Commission for predicting the erosion depth in Pelton turbines using results from numerical simulations. Rai et al. (2016) published a detailed study of blade erosion in Pelton turbines, focusing on the flow in the turbine bucket and on the forces acting on the particles. Indeed the complexity in the computation of the high-energy, turbulent, unsteady flow in a Pelton bucket in order to develop a physically based erosion model is a major issue. Therefore, studies using the SPH and other meshless methods to simulate the flow in Pelton turbines were only published recently. Those meshless methods are especially advantageous for high energy, free surface flows such as those found in Pelton turbines. Jahanbakhsh (2014) developed a Finite Volume Particle Method (FVPM) model which accounts for multiphase solid-fluid flow as well as solid-solid interactions. The lack of turbulence representation prevented a physically correct representation of the fluid flow and thus sediment impact velocity, but his work accounts for direct erosion simulation where fluid phase and eroded surface were represented by the FVPM. Beck and Eberhard (2016) used the SPH method to simulate the solid-fluid flow in a Pelton bucket and estimated wear by the application of different erosion models. Besides, neither of these two studies provides a quantitative analysis of the eroded material amount.

This work yields two novelties: Firstly, it is a step towards a quantitatively correct numerical erosion prediction in a complicated flow field where calibration is impossible. Secondly, this prediction is made without the use of an erosion model. To achieve these aims, we provide correct sediment impact velocities via precise and validated fluid phase calculations (Boden et al., 2017) with the SPH-ALE code ASPHODEL, developed by the company Andritz HYDRO and the laboratory LMFA of the Ecole Centrale de Lyon. On the other hand, a coupling between the SPH-ALE code ASPHODEL and the FEM code EUROPLEXUS (Nunez et al., 2016) allows us to calculate the material damage directly. In order to obtain a realistic material damage in an efficient way, general applicable assumptions based on material properties were made and validated.

2 INVESTIGATED GEOMETRY

The long-term aim of this work is to obtain a quantitative and qualitative valid erosion rate. In order to have the possibility to compare numerical results to experimental ones, all following investigations were based on a simple geometry, which can easily be tested and measured in experiments. According to conducted experiments in the Andritz Wear Lab, the geometry consisted of an oblique round jet, impacting on a flat plate with an angle of 45° (see Figure 1).



Figure 1. Geometry of jet and velocity reference system on the upper right.

The jet diameter, *D* at the inlet was measured to be 10 mm, the inlet jet velocity, v_{jet} was 60 m/s and the nozzle was situated 40 mm above the plate surface. Sediments were released at the jet inlet with the same velocity as the surrounding fluid, and moved downwards towards the plate where they impacted. Impact parameters such as velocity and angle were saved as input to the erosion estimation. In the following sections, the velocity notation is referred to the reference system on the upper right side of Figure 1.

3 ASSUMPTIONS FOR DAMAGE PREDICTION

3.1 Damage mechanism

In order to estimate the erosion damage produced by the impact of spherical quartz sediments on a target with a tungsten carbide coating by FEM simulations, we need to identify the damage mechanism. Depending on the projectile and target properties such as fracture toughness or hardness and the normal impact velocity, Hutchings (1992) proposed a methodology for that purpose. He distinguished between three different target states after a normal impact of a spherical object on a flat plate: the elastic reversible deformation, the plastic irreversible deformation and the Hertzian fracture, which is characterized by radial ©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print) 3109

cone cracks departing from the indention interface. The latter two conditions cause an immediate, visible damage on the target material, whereas the elastic deformation causes a visible damage after several, usually a high number of impacts. Figure 2 illustrates the different possible states after one normal impact of a spherical quartz particle on a tungsten carbide coating, where the relevant region to this erosion problem is highlighted by the blue rectangle. Material properties leading to this map were identified as the following:

- Fracture toughness of the target: K_{ct} = 3.86 MPa.m^{1/2}
- Fracture toughness of the sediment: $K_{cp} = 1.2 \text{ MPa.m}^{1/2}$
- Young modulus of the target: $E_t = 511$ GPa
- Young modulus of the sediment: $E_p = 70$ GPa
- Hardness of the target: HV_{t:0,3} = 1100 HV
- Hardness of the sediment: HV_{p;0,3} = 1500 HV
- Density of the sediment: $\rho_p = 2650 \text{ kg/m}^3$

For sediment sizes in the relevant range and normal impact velocities up to 25 m/s, we observed that elastic deformation was the damage mechanism principally. For the largest sediments with the highest impact velocities, Hertzian fracture is also possible, but the necessary conditions are only present on the splitter edge of a Pelton bucket. The bucket surfaces are prone to fatigue damage by successive elastic deformations. We focused on the latter mechanism in this study.



Figure 2. Damage mechanisms after Hutchings (1992) for the normal impact of spherical quartz sediments on a brittle tungsten carbide coating.

3.2 Important impact parameters

In order to predict the erosion damage on the plate by the sediment impacts in an efficient way, we need to reduce input variables as much as possible, without introducing an important error to the erosion estimation. Existing erosion models give a good overview over parameters that influence the damage of the material. In literature, several hundreds of such parameters can be found. Meng and Ludema (1995) examined 28 of the most popular erosion models, where 90% of these depended on the impact velocity, around 60% on the impact angle and 50% on the sediment size. Thus, we state that impact velocity and angle as well as the sediment size are the most important impact parameters from the fluid simulations. From the material aspect, one has to account for material specific parameters. Even if those were numerous, there is no need to reduce their number, as they remain constant throughout the simulations and are treated within the FEM simulations. In contrast, from the fluid aspect of the problem, erosion damage changes for each combination of sediment size, impact angle and velocity, creating a three dimensional matrix of the problem. As an example, if we divide our problem into 10 classes of sediments to cover the range of 10 µm to 500 µm and calculate the erosion damage for 3 velocities per sediment class and 3 angles per class-velocity combination, we have to conduct 90 individual structure-structure FEM simulations to properly estimate the erosion damage of the whole problem. However, the nature of our problem allows us to reduce this matrix by one dimension and make it independent of the impact angle. Pelton turbines are covered with brittle tungsten carbide coatings of the type WC-Co-Cr, applied by High Velocity Oxygen Fuel (HVOF) or High Velocity Air Fuel (HVAF) techniques. Traditionally brittle materials are experimentally classified as most sensitive to erosion when jets impact at normal angle and increasingly resistant for jets impacting with a lower impact angle. Sand erosion resistance tests of Wood (1999) or Li et al. (2015) showed the kinds of coatings used for Pelton turbines varying curves, with tendency to a more constant resistance over all jet impact angles. Hence, these tests did not allow independency of the impact angle. But all those experiments indicated that the erosion is sensitive with respect to the global jet geometry, not to an individual sediment impact. Locally on the microstructural level, the mechanism that is responsible for fatigue damage of the brittle coating is the formation of subsurface lateral cracks. Those will eventually propagate, connect and lead to material removal (Li et al., 2015). Furthermore, novel impact fatigue tests (Agrianidis et al., 2006) of a sphere impacting on a surface are only dependent on the normal load. All observations on the microscale indicated that the fatigue damage of brittle material was due to the normal component of the loading. The correlation between the normal impact velocities of sediments in different jet geometries found by SPH simulations and the erosion profiles of brittle materials in corresponding experiments supports this assumption. Hence, the matrix of impact parameters that serves as input into the FEM simulations has two dimensions, the sizes of the sediments and their normal impact velocities.

4 FLUID SIMULATIONS

4.1 SPH-ALE method

In Pelton turbines, flow is characterized by unsteadiness, a high energy content and rapidly changing free surfaces. Their geometry is rather complicated, hence domain meshing with traditional mesh based methods, such as the Finite Volume Method (FVM) becomes cumbersome. Additionally, the mesh has regularly to be adapted to the changing free surface, which is computationally very costly, especially when simulating a whole turbine. A meshless method like SPH offers many advantages for this kind of flows. The fluid is discretized by a set of points, which move according to the fluid motion. Therefore, no computational mesh is needed and a free surface is naturally captured. The discretization points are called *particles* and the properties of each particle (density, velocity, mass etc.) are defined as the weighted average of the properties of all particles in the neighborhood. The weighting function is represented by a Gaussian compact smoothing function called *kernel*. SPH terminology is illustrated in Figure 3a. For the fluid modeled as a barotropic, isothermal, inviscid, weakly compressible fluid with constant internal energy, the Navier-Stokes equations are reduced to the Euler equations,

$$\frac{\partial \rho_f}{\partial t} + \left(\overrightarrow{u_f} \cdot \nabla \right) \rho_f = -\rho_f \nabla \cdot \overrightarrow{u_f}$$
[1]

$$\frac{\partial \overrightarrow{u_f}}{\partial t} + (\overrightarrow{u_f} \cdot \nabla) \overrightarrow{u_f} = -\frac{1}{\rho_f} \nabla p_f + \overrightarrow{f}$$
[2]

where the pressure is denoted as p, the fluid velocities as u_f , the density as ρ and the body force as f.

The equations are closed using a barotropic equation of state, Tait's equation. Presently, no turbulence model is available for the SPH-ALE formulation, but to account for turbulent fluctuations in the fluid and their influence on the sediment movement, we developed an ad-hoc turbulence model that is valid in the near-wall region of the plate (Boden et al., 2016). All simulations of the oblique jet presented in Figure 1 were performed in a two-dimensional space, with a three-step *Runge-Kutta* scheme (RK3) for time and a first order *Moving Least Square Scheme* (MLS1) for spatial discretization (Renaut et al., 2015). The SPH discretization size was chosen as r_0 =D/50=0.0002 m and the smoothing length as h/r_0 =2.0. A CFL number of 0.8 ensured a smooth pressure and velocity field, with respect to the sediment transport limiting time step size $\Delta t < min(\tau_p/10, 10^{-6})$, where τ_p refers to the sediment relaxation time. Further information about the SPH-ALE formalism was provided in Vila (1999). The jet geometry of Figure 1 discretized by SPH particles and the corresponding smooth pressure field is illustrated in Figure 3b.



Figure 3. a) SPH principle and b) oblique jet discretized by SPH particles with its pressure field.

4.2 Sediment transport

Sediments are supposed to consist of pure quartz and to be spherical. Their mass and volume concentration in the river water flowing through a turbine under standard conditions are low so that they are passively influenced by the fluid movement. Sediment-sediment contact as well as particle break up are not considered. The sizes of diluted sediments usually range between 10 μ m and 1000 μ m, where sizes larger than 500 μ m can be filtered out upstream of the turbine. Consequently, this work studies hydro-abrasive erosion for very small sediments from 10 μ m to 500 μ m, classified after ISO 14688-1:2002 as fine silt up to medium sand. Those sediments are numerically treated as point-wise particles and their movement through the fluid can be modeled by the particle equation of motion, the Basset-Boussinesq-Oseen equation (Crowe et al., 2011). Under consideration of the small sediment size and the density ratio between quartz and water of the order of 1 ($\rho_p/\rho_f \cong 1$), the force balance between drag and buoyancy force can be written as

$$\frac{d\vec{u}_p}{dt} = \frac{1}{\tau_p} \left(\overrightarrow{u_f} - \overrightarrow{u_p} \right) + \frac{\rho_p - \rho_f}{\rho_p} \vec{g}$$
[3]

where the first term accounts for the drag and the second for the buoyancy force. Subscript *p* refer to sediment characteristics, subscript *f* refer to fluid characteristics, *u* denotes the velocity, ρ denotes the density and *g* denotes the acceleration due to gravity.

As proposed by Chen and McLaughlin (1995), drag force correction factors $C_{c,i}$ in normal and tangential direction were introduced via a modified particle response time $\tau_p/C_{c,i}$ close to the wall to account for wall effects. The sediments collide on the wall when their center of inertia reaches a distance of $d_p/2+k_r$, where k_r corresponds to the surface roughness of an undamaged Pelton turbine coating. A turbulent dispersion model based on the Langevin equation introduces the influence of turbulence on the sediments. Usually these kinds of models necessitate a turbulence representation in the fluid phase. As stated earlier, this is not available for SPH-ALE formulations. Therefore, the normalized Langevin model proposed by Dehbi (2008) provided a possibility to take turbulence into account within a distance of $y^* < 100$ to the wall on the basis of general valid Eulerian turbulence statistics in this region. It has been shown (Boden et al., 2017) that the application of this model captured the turbulence well for the smaller particles, but not for the larger ones, as those impacts occur before their center reaches the turbulent zone. Further investigations showed acceptable results with respect to erosion prediction of impact velocities of larger sediments ($d_p=500 \mu$ m), when applying the model of Dehbi, and also for the outer zone $y^* > 100$ with constant values from $y^* = 100$. Our ad-hoc turbulence model thus allowed the calculation of physically meaningful impact velocities as input for the erosion simulations.

4.3 Sediment impact parameters

Sediment impact parameters at the wall serve as input for the erosion simulations. In the following, erosion values were estimated by using two different techniques. The first technique consisted of a two-step procedure, where firstly, a pure fluid simulation with the SPH method was conducted. This gave the normal impact velocity component at the corresponding position to each sediment. A subsequent FEM simulation with the known impact parameter gave the damage potential of one sediment of a defined size at a defined position. The second technique estimated the erosion directly via a coupled simulation between SPH-ALE and FEM and will be discussed more in Section 5.2. Figure 4 illustrates the normal impact velocities for the case of the oblique jet for five different sediment sizes. sediments per size were released at the inlet to assure a statistical independent velocity-position distribution.



Figure 4. Normal impact velocities for the oblique jet obtained by SPH-ALE simulation.

The representation is restricted to the data taken on the right side of the stagnation point, where the highest normal impact velocities occurred. The data corresponds to the first impacts of the sediments, as further impacts after one or more rebounds had negligible normal impact velocity components. In the simulations, sediments still existed after their first impacts, but no more impact data was recorded. Data points in Figure 4 indicate the averaged normal impact velocities in an equally distributed, symmetric zone around their positions. The error bars indicate the maximum and minimum normal impact velocity in the corresponding zone. Distributions were built from simulations where a minimum of 10⁵

5 DAMAGE PREDICTION

In the previous sections, we identified the relevant damage mechanism, fatigue and presented the normal sediment impact velocities at different positions on the plate for the oblique jet (Figure 4). With these two components, we are able to estimate the erosion damage caused by a defined sediment size class with one defined normal impact velocity. Therefore, we used two techniques, firstly, a dry structure-structure FEM simulation and then a coupled SPH-FEM simulation. All results are shown for the sediment size $d_p=500 \mu m$. The material deterioration are evaluated by two different variables, the maximal stress range in the impact zone and the number of cycles, which are necessary to erode a certain element from the mesh. For the latter one, a general valid fatigue criterion has been implemented into the FEM code Europlexus (Coudouel et al., 2017). Therefore, principal stresses were calculated from the evolution of the Cauchy stresses during one cycle. Then, the maximal stress range was given by the difference between the stresses at the state of highest traction and strongest compression. Data from experimental fatigue tests related those stress range to a critical number of cycles, which caused material damage. Two assumptions were made in order to estimate the number of cycles leading to erosion damage: Firstly, the surface shape does not evolve with time and secondly, the sediment always collides at exactly the same point.

5.1 Structure-structure simulations

From Figure 4, we obtained the normal impact velocity $|v_{norm}|$ of the sediments, which was for the sediments of size d_{ρ} =500 μ m around 20 m/s. We used this velocity as an example to estimate the erosion damage. The geometry consisted of a circle colliding on a rectangle in the two-dimensional case and a quarter sphere impacting on a block with two symmetry planes in the three-dimensional case. Contact was simulated without friction. The target was built up of two layers, the upper layer representing the tungsten carbide coating with a thickness of 300 µm and the lower layer representing the stainless steel corresponding to the base material of Pelton turbines. Dimensions of the target were chosen such that waves due to the impact do not re-influence the impact zone. The circumference of the sediment was discretized by 150 triangular elements, the triangular elements in the contact zone of the target were of the same size to ensure a proper contact detection. Contact was simulated by a sliding interface without friction. Time step was chosen as ⊿t=5*10⁻¹⁰ s. Figure 5 illustrates the transient simulation of one normal impact on the plate for the twodimensional case, showing the mesh and the appearance of a surface acoustic wave, also known as Rayleigh wave, represented by a compression zone in blue and a dilatation zone in red, both travelling outwards with time. Usually only a few percent of the impact kinetic energy is transmitted into those waves, but in combination with material flaws they can lead to propagation of cracks. During the impact, a part of the kinetic energy was transmitted into the kinetic energy of the rebounding sediment and another one was absorbed in the compression zone near the solid-solid interface. The stress range in this zone was the representative for the final damage due to fatigue and was correlated to this final damage in terms of load cycles via the characteristic Wöhler line for the coating material.



Figure 5. Mesh and evolution of hydrostatic pressure for two-dimensional normal impact at a) t=0 s, b) $t=3*10^{-7}$ s, c) $t=4.6*10^{-7}$ s and d) $t=7.6*10^{-7}$ s

As traditional bending fatigue tests to construct a Wöhler line do not exist for the relevant tungsten carbide alloy, we constructed a suitable Wöhler line based on the impact fatigue results of Agrianidis et al. ©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print) 3113

(2006), which were expressed in terms of load force. With the correspondent impact surface area, one obtained the relevant stresses. This Wöhler line is shown in Figure 6 and it resembles the results of classical fatigue test for other tungsten carbide alloys published by Brandt (1995). We stated that the constructed Wöhler line may be even more appropriate as one based on a classical bending fatigue test would be, as the load applications were the same, namely the impact of a sphere.



Figure 6. Wöhler line as the basis for the damage results in terms of load cycles.

Figure 7a) illustrates the stress range in the coating around the contact zone after one impact for a twodimensional simulation with a normal impact angle and Figure 7b) the corresponding fatigue damage in terms of load cycles. This means, for the latter figure, that the element color corresponds to the number of impacts after which an element will be eroded. The maximum stress range caused by the impact occurs at the contact point and amounted to 2.42 GPa, which means an amplitude of 1.21 GPa corresponding to 1210 N/mm². Those elements were eroded after 700,000 or less impacts. Light blue elements, where a stress range of 500 MPa was applied, were eroded after approximately 875,000 impacts. These relations correspond to the data of the constructed Wöhler line and validate the implementation of the fatigue routine. Regarding the erosion shape and progress, one can observe the fastest but also narrowest material deterioration occurred at the contact interface. Beneath the surface, erosion progressed slower but material over a larger width was affected. The erosion progress seems to correspond to what is known from experiments with progressing material removal from the surface, whereas the erosion pattern differed a lot. One would expect plaques forming due to lateral subsurface cracks, therefore producing a much shallower and larger erosion pattern. Although the microstructure of the material plays a major role, we do not account for it in our studies, assuming a continuous material with homogeneous properties. We rather aim at globally predicting the total amount of material removal over time after the impact of a certain number of sediments. The obtained erosion form is therefore not relevant, whereas the total amount of material removed after a certain number of impacts is important.

In the following steps, we want to validate our approach of only taking the normal impact velocity component for erosion damage estimation. Therefore, we compared the results of the normal impact illustrated in Figure 7 with the corresponding results of an oblique impact with an angle of 45°, with the same normal velocity component as before and an additional tangential velocity component of the same absolute value.



Figure 7. a) Stress range per element, b) number of cycles necessary to erode an element for a 2D simulation of a normal impact of a sphere (d_p =500 µm, | v_{norm} |=20 m/s).

These results are represented in Figure 8. The form of the maximum appearing stresses equals the results of the normal impact with a slightly higher maximum value of 2.5 GPa. Then, the overall picture of the cycle dependent eroded material also equals the results of the normal impact with a very small difference at the surface. By comparing Figure 7b) and 9b), no global difference for the amount of eroded material after a certain number of impacts was visible. This fact validates our assumption, of accounting only for the normal velocity component, when investigating erosion damage of brittle materials.



Figure 8. a) Stress range per element, b) number of cycles necessary to erode an element for a 2D simulation of an oblique impact (45°) of a sphere (d_p =500 µm, | v_{norm} |=20 m/s, v_{tan} = 20 m/s).

Two-dimensional simulations of impacting spheres tended to underestimate stresses in the material compared to three-dimensional ones, due to a simulated line contact instead of a point contact. Therefore, Figure 9 shows the same simulation represented in Figure 7 in three dimensions instead of two dimensions. One observed that the maximal stress range was 4.31 GPa, thus larger than that in the two-dimensional case by a factor of 2. Likewise, the form of the occurring stresses changed to a more circular form and a larger width at the surface. A different form for the cycle dependent erosion was also visible. Relatively, more elements were eroded after a small number of impacts with more elements located at the surface, but the overall expansion of the eroded area was smaller, with only 1/2 of the coating height. Mesh convergence was assured in both cases.



Figure 9. a) Stress range per element, b) number of cycles necessary to erode an element for a 3D axisymmetric simulation of a normal impact of a sphere (d_p =500µm, v_{norm} =20m/s).

5.2 Coupled fluid-structure-structure simulations

As an alternative two-dimensional solution to the two-step procedure described in Section 5.1, we calculated the sediment impact and the resulting damage in one step via the SPH-FEM coupling developed by Nunez et al. (2016). Here, the sediment movement prior to impact was not evaluated by a point-wise particle movement, but a discretized simulation of one sediment in a water column impacting on a plate. The sediment and the plate were simulated by FE method with the same parameters as before, the water column by the SPH-ALE method, where r_0 =0.00005 m and h/r_0 =1.2. The time step was equal to the one in the FE domain. At t=0, the water column was released with a normal velocity of $|v_{norm}|$ =42.2 m/s at a distance of 5*10⁻⁶ m of the plate, corresponding to the vertical velocity component of the oblique jet. The space between the end of the water column and the sediment face was discretized by 5 SPH particles, which was found to be an adequate number. The dimensions of the plate had to be much larger than before in order to prevent the re-influence of the sound waves in the material created by the water column, as it collides approximately 2 ns prior to the sediment. Figure 10 illustrates the results obtained by the coupled simulation: the different velocities in the system, with the normal velocity component of the fluid on the upper right side and the normal velocity of the sediment just prior to contact on the upper left side. Some regions of the fluid accelerated due to gravity, the bottom part showed a slightly upward movement due to rebound. The normal velocity of the sediment accounted to |v_{norm}|=15 m/s. This value differed from the value range found in the separate SPH-ALE simulations and was slightly influenced by the width of the simulated water column and the number of SPH

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

particles between the sediment and the plate at *t*=0. On the bottom left side of the plate, a local maximal stress range of 2.8 GPa was visible at the sediment impact point, slightly higher than the value found for the corresponding 2D simulations. This can be related to the discretization of the plate. The bottom right side of the plate depicted the number of cycles until erosion and varied in two points compared to the structure-structure simulations. Firstly, the erosion form was more circular and corresponded to the form observed in the 3D simulations and secondly, the total eroded area was smaller and affected only the upper part of the coating. The second observation can be related to the smaller impact velocity than the value found for the dry simulations, which was caused by the directly simulated compression effect of the water film between the sediment and the plate. This effect might be overestimated compared to the influence of the near-wall correction factors in the fluid simulations. Regarding the erosion potential of the water jet, it had no influence on the plate, which corresponds to the observations in reality, where the mere impact of the water on the turbine bucket does not cause erosion.



Figure 10. Coupled simulation with the normal velocity of the sediment (top left) and the fluid (top right), the stress range (bottom left) and number of cycles necessary to erode an element (bottom right).

6 CONCLUSIONS

We have presented an approach to estimate hydro-abrasive erosion in Pelton turbines from first principle, either by performing a two-phase water-sediment SPH-ALE simulation for the fluid phase separately and then using these result as input into FEM simulations, or by using directly a fluid-structure SPH-FEM coupling. We identified fatigue as the relevant damage mechanism for our problem and posed the hypothesis that only the normal impact velocity component can be taken into account to estimate the amount of erosion. Results have been generated for an oblique jet impact (45°) on a flat plate. We calculated the normal impact velocities of the sediments via SPH-ALE simulations. Dry structure-structure FE simulations have been executed with the normal impact velocity obtained by the SPH simulations for one sediment size for impact angles of 90° and 45° in 2D and for an impact of 90° in 3D. One wet coupled structure-fluid-structure simulation has been performed for a normal impact of 90° in 2D. Comparisons of the different calculations in terms of stress ranges and cycles of impacts until erosion lead to the following conclusions:

- i. The hypothesis of only taking into account the normal impact velocity to estimate the erosion amount has been validated.
- ii. Erosion amount cannot be estimated based on 2D simulations as they predict the false stress range occurring in the material quantitatively and qualitatively. Therefore, 3D simulations have to be performed.
- iii. The coupled fluid-structure simulation estimates slightly less erosion damage than the two-step procedure. This can be due to the smaller sediment impact velocity because of the simulated compression of the water film on the plate.

Based on the obtained results in this work, we will produce correlations between the two relevant impact parameters normal velocity/sediment size and the estimated erosion amount based on 3D structure-structure simulations. Together with impact statistics from SPH-ALE simulations of the oblique jet in 2D, we will be able to estimate the erosion amount of the plate on the jet axis and compare the obtained value to experiments conducted in our Wear Lab. Furthermore, we will investigate the differences between the results from the two-step procedure and the coupled simulations. It is possible that at small scale of this investigation, supplement physical effects have to be added in order to obtain the same sediment impact velocities as for the fluid simulations.

ACKNOWLEDGEMENTS

This work has been supported by the commission of European Communities through the PREDHYMA project (Marie-Curie actions, grant PITN-2013-608393).

REFERENCES

- Agrianidis, P., Anthymidis, K.G., David, C. & Tsipas D.N. (2006). Impact Testing a Capable Method to Investigate Fatigue Resistance. *Proceeding* 16th *European Conference of Fracture*, 219-220.
- Beck F. & Eberhard P. (2016). Study of Abrasive Wear with a Lagrangian SPH Approach. 11th International SPHERIC (SPHERIC 2016) Workshop, SPHERIC11, 35-46.
- Bergeron S.Y., Vu, T.C. & Vincent, A.P. (2002), Silt Erosion in Hydraulic Turbines: The Need for Real-Time Numerical Simulations. *Simulation*, 74(2), 71-74.
- Bitter J.G.A. (1963a), A Study of Erosion Phenomena: Part I. Wear, 6(1), 5-21.
- Boden, W., Aubert, S., Marongui, J.C. & Perkins R. (2016). *Turbulence in Solid Particle Erosion:* A Meshless *Method Approach*, ETMM11.
- Boden, W., Aubert, S., Marongiu, J.C. & Perkins R. (2017). Solid Particle Erosion in Hydraulic Turbines: Towards a Generic Approach, ETC12.
- Brandt, OC. (1995), Mechanical Properties of Hvof Coatings, *Journal of Thermal Spray Technology*, 4(2), 147-152.
- Chen, M. & McLaughlin, J.B. (1995). A New Correlation for the Aerosol Deposition Rate in Vertical Ducts. *Journal of Collision and Interface Science*, 169(2), 437-455.
- Coudouel, G., Combescure, A. & Marongiu, J.C. (2017), Prédiction et Simulation Numérique Du Dommage Résultant De l'Impact De Gouttes d'Eau à Grande Vitesse Sur Les Augets De Turbine Pelton, *13ème Colloque National en Calcul des Structures, Giens, France.*
- Crowe, C.T., Schwarzkopf, J.D., Sommerfeld, M. & Tsuji, Y. (2011), *Multiphase Flow with Droplets and Particles*. 2nd Edition, CRC Press.
- Dehbi, A. (2008), Turbulent Particle Dispersion in Arbitrary Wall-Bounded Geometries: A Coupled Cfd-Langevin-Equation Based Approach, *International Journal of Multiphase Flow*, 34(9), 819-828.
- Felix, D., Abgottsponn A., Albayrak I. & Boes R.M. (2016), Hydro-Abrasive Erosion on Coated Pelton Runners: Partial Calibration of The lec Model Based on Measurements in Hpp Fieschertal, 28th IAHR Symposium on Hydraulic Machinery and Systems, 1191-1200.
- Finnie, I. (1960), Erosion of Surfaces by Solid Particles, Wear, 3(2), 87-103.
- Ghenaiet, A. (2015), Prediction of Erosion by Solid Particles in A Water Turbine, ETC11.
- Grant, G. & Tabakoff, W.C. (1975), Erosion Prediction in Turbomachinery Resulting from Environmental Solid Particles, *Journal of Aircraft*, 12(5), 471-478.
- Hutchings, I.M. (1992), Ductile-Brittle Transitions and Wear Maps for the Erosion and Abrasion of Brittle Materials, *Journal of Physics*, 25(1A), A212.
- Jahanbakhsh E. (2014), Simulation of Silt Erosion Using Particle-Based Methods, PhD EPFL.
- Li Y., Lian Y., Cao J. & Li L. (2015), Solid Particle Erosion Behavior of Hvof/Hvaf Sprayed Wc-Co-Cr Coatings, Proc. IMech Part J: Journal of Engineering and Tribology, 230(6), 634-643.
- Meng HC. & Ludema KC. (1995), Wear Models and Predictive Equations: Their Form and Content, *Wear*, 181-183, 443-457.
- Montgomery J. & Clark J. (1962). *Dust Erosion Parameters for A Gas Turbine*, Society of Automatif Engineer, Technical Paper 620225, 1-12pp.
- Nunez-Ramirez J., Marongiu JC., Brun M. & Combescure A. (2016), A Partitioned Approach for The Coupling of Sph and Fe Methods for Transient Nonlinear Fsi Problems With Incompatible Time-Steps. *International Journal for Numerical Methods in Engineering*, 109, 1391-1417.
- Rai A.K., Kumar A. & Staubli T. (2016), Forces Acting on Particles in A Pelton Bucket and Similarity Considerations for Erosion, 28th IAHR Symposium on Hydraulic Machines and Systems, 917-926.
- Renaut GA., Marongiu JC. & Aubert S. (2015), *High Order Sph-Ale Method for Hydraulic Turbine Simulations*, ETC11.
- Sage, W. & Tilly, G.P. (1969), The Significance of Particle Size in Sand Erosion of Small Gas Turbines, *The Aeronautical Journal*, 73(701), 427-428.
- Vila, J.P. (1999), On Particle Weighted Methods and Smooth Particle Hydrodynamics, Mathematical Models and Methods in Applied Science, 9(02), 161-209pp.
- Wood RJK. (1999), The Sand Erosion Performance of Coatings, Materials and Design, 20(4), 179-191pp.

ANALYSIS OF THE CAVITATING FLOWS IN A LOW SPECIFIC SPEED AXIAL PUMP

ZHE WANG⁽¹⁾, RENFANG HUANG⁽²⁾, XIANWU LUO⁽³⁾, JIAJIAN ZHOU⁽⁴⁾ & LIN WANG⁽⁵⁾

^(1,2,3) Beijing Key Laboratory of CO₂ Utilization and Reduction Technology, Tsinghua University, Beijing 100084, China,
 zhe-wang16@ mail.tsinghua.edu.cn; hrenfang@yeah.net; luoxw@mail.tsinghua.edu.cn
 ^(1,4) Marine Design and Research Institute of China Shanghai 200011, China 13916608004@139.com
 ^(3,5) State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing 100084, China luoxw@mail.tsinghua.edu.cn

ABSTRACT

The present paper mainly treats the cavitating turbulent flows in a low specific speed axial pump, which is designed to be operated under the following conditions: flow discharge of 0.56m³/s and head rise of 11m at the rotational speed of 1400r/min. In order to include the effects of vapor-water mixture on turbulent flow development, a Filter-Based Density Corrected Model, i.e. FBDCM is applied to correctly evaluate the turbulent eddy viscosity. Three-dimensional cavitating turbulent flows in the full flow passage of the pump are conducted based on a homogeneous cavitation model. For the purpose of improving the numerical accuracy, the structural grids are generated for the entire computation domain, and the clearance zone between the impeller tip and pump casing is refined as the mesh treatment near the flow walls. It is noted that hydraulic performance and cavitation in the pump rotor. As the interaction of sheet cavitation and tip leakage flow, there is perpendicular cavitation in the main flow passage of the blade-to-blade channel. The better understanding of cavitating turbulent flow features in the axial pump will be helpful for further optimization to improve the hydraulic performance as well as the cavitation performance for the pump in the future.

Keywords: Axial Pump; Cavitating Flow; Filter-Based Density Corrected Model (FBDCM).

1 INTRODUCTION

Batchelor, (1967) discussed that in liquid flows cavitation occurs if the local pressure drops below the saturated vapor pressure, which leads to the formation of vaporous bubbles to relieve the negative pressure. It is still an important issue for researchers to understand and analyze the cavitation characteristics. Specifically, cavitation in an axial flow pump is worth studying, because the phenomena are commonly much more complicated than other hydraulic machines. Ducoin et al. (2012) proved cavitation took place when an axial flow pump was operated at high rotational speeds or under off-design conditions. The occurrence of cavitation in blades, tip clearance, etc. of the axial flow pump can result in serious problems such as vibrations, noise, erosion, etc. Although some process of cavitation by model test can be observed, the experiment may be time-consuming and expensive. For some engineering applications, experimental tests cannot completely reveal the physics of cavitation because cavitation involves complex turbulent flow and phase change, and some process of the phenomenon is invisible by present technique. For this reason, it is vital to accurately predict the development and evolution of cavitation in the pump.

In the past decades, many researches including experiments and simulations have been extensively conducted to understand the mechanisms of unsteady cavitation in an axial flow pump. Brennen, (1994) discussed the cavitation effect on pump performance and summarized that the blade size, flow velocity and temperature could change the critical cavitation number when the breakdown occurred. Hu and Zangeneh, (1999; 2001) applied different codes to predict the flow field in the water-jet pump. The results were consistent with measured values for torque, and indicated that the rotor did not have essential effect on the torque.

The motivation of this paper is to investigate the cavitating turbulent flow in a low specific speed axial flow pump at different operation conditions. The pump model was experimentally investigated. Numerical simulations have been conducted by Yu et al. (2010) using the Filter-Based Density Corrected Model and cavitation mode proposed by Zwart et al. (2004). The numerical results were compared with the experimental data corresponding to hydraulic performance and cavitation performance. Furthermore, the internal flows are discussed in detail in this study so as to provide a better understanding of cavitation physics inside the axial pump.

2 GOVERNING EQUATIONS

The cavitation flow in simulation is assumed as a homogeneous mixture. The mixture density is given as

$$\rho_{\rm m} = \alpha_{\rm v} \rho_{\rm v} + (1 - \alpha_{\rm v}) \rho_{\rm L}$$
^[1]

where α_V is the vapor fraction, ρ is the density, and the subscripts of m, V and L represent the mixture, vapor and liquid.

The governing equations and momentum conservation equations about the cavitating flow are described as follows:

$$\frac{\partial \rho_{\rm m}}{\partial t} + \nabla \cdot (\rho_{\rm m} U) = 0$$
^[2]

$$\frac{\partial(\rho_{\rm m}U)}{\partial t} + \nabla \cdot \left[\rho_{\rm m}U \times U - \mu_{\rm m}(\nabla U + (\nabla U)^{\rm T})\right] = -\nabla p$$
[3]

where *U* and *p* represent velocity and pressure, respectively.

The cavitation model is based on the Rayleigh-Plesset equation with liquid-vapor conversion associated with cavitation process which is modeled through the source terms m^+ and m^- . m^+ and m^- are for the process of evaporation and condensation, respectively:

$$m^{+} = C_{\rm e} \frac{3r_{\rm g}(1 - \alpha_{\rm V})\rho_{\rm V}}{R_{\rm b}} \sqrt{\frac{2}{3} \frac{\max(p_{\rm V} - p, 0)}{\rho_{\rm L}}}$$
[4]

$$m^{-} = C_{\rm c} \frac{3\alpha_{\rm v}\rho_{\rm v}}{R_{\rm b}} \sqrt{\frac{2}{3} \frac{\max(p_{\rm v} - p, 0)}{\rho_{\rm L}}}$$
[5]

k- ε turbulence model is widely used in the unsteady cavitating flow, but it over estimates the turbulent viscosity. Therefore, some modifications have been performed, such as DCM and FBM by Huang et al. (2014), etc. The present model is called FBDCM, which combines the advantages of FBM and DCM. In this model, turbulent eddy viscosity, μ_T is set:

$$\mu_{\rm T} = \frac{c_{\mu} \rho_{\rm m} k^2}{\varepsilon} f_{\rm hybrid}$$
 [6]

where C_{μ} =0.009, *k* and ε are turbulent kinetic energy and the dissipation rate. The hybrid coefficient, i.e. f_{hybrid} is defined as

$$f_{\text{hybrid}} = \varphi f_{\text{FBM}} + (1 - \varphi) f_{\text{DCM}}$$
^[7]

where φ represents a function blending FBM and DCM.

$$\varphi = 0.5 + \tanh\left[\frac{C_1(0.6\rho_m / \rho_1 - C_2)}{0.2(1 - 2C_2) + C_2}\right] / 2\tanh(C_1)$$
[8]

$$f_{\rm FBM} = \min(1, \frac{\lambda \times \varepsilon}{k^{3/2}})$$
[9]

$$f_{\rm DCM} = \frac{\rho_{\rm V} + (1 - \alpha_{\rm V})^{\rm n} (r_{\rm I} - r_{\rm V})}{\rho_{\rm V} + (1 - \alpha_{\rm V})(\rho_{\rm I} - r_{\rm V})}$$
[10]

 $C_1=4, C_2=0.2, C_{\mu}=0.009.$

3 GEOMETRY MESH AND BOUNDARY CONDITIONS

The axial pump is shown in Figure 1. The axial pump was designed by Michael et al. (2008). It consists of six rotor blades and eight stator blades. NACA 16 chord wise thickness distribution was used for the rotor

blades. At the design operation point, flow coefficient, ϕ is 0.85 (Q=0.56 m³/s), head rise is 11 m under the rotation speed of 1400 r/min. Marquardt, (2011) tested the axial flow pump at 1400 r/min under no cavitation, and at 2000 r/min under cavitation condition. Diameters of the rotor and stator are 303.78 mm and 304.8 mm. The gap between rotor blade tip and the pump casing is 0.51 mm.



Figure 1. The axial flow pump (a) 3D view of the pump (b) generated pump mesh.

For the purpose of improving the numerical accuracy, the structural grids were generated for the entire computation domain as shown in Figure 2. Based on the mesh independence test, the final grid number for the computation domain is about 2.6 million to achieve the almost unchanged head rise of the axial pump. The tip clearance zone is regarded as a ring. We define the rotor zone in a rotating reference coordinate and others in a stationary coordinate. Thus, the interfaces are assigned between the rotor zone and other stationary parts. The solid wall surface of the axial flow pump was set as no-slip wall. The static pressure was set at the domain inlet, and the mass flow rate was assigned for outlet boundary condition.



Figure 2. Mesh of computation domain.

4 RESULTS AND DISCUSSION

4.1 Cavitation performance

The flow coefficient ϕ is defined:

$$\phi = \frac{Q}{nD^3}$$
[11]

where Q is flow rate, *n* is the rotational speed, and D is the rotor diameter. Head coefficient, ψ is defined as:

$$\psi = \frac{gH}{n^2 D^2}$$
[12]

where H is head rise, and g is the gravitational acceleration.

Figure 3 shows the characteristic curve near the designed operation condition. It is noted that the numerical simulation predicted head coefficient fairly well compared with the tested data. The prediction error between the simulation and experiment becomes larger at small flow rate. For the present study, the maximum discrepancy is about 2.4%.



Figure 3. Characteristic curve.

Figure 4 shows the tendency that the pump head drops with the deceasing cavitation number is well captured by the numerical simulation. Near σ =0.23, the head coefficient increases a little, and drops rapidly when cavitation number further decreases.



Figure 4. Cavitation Performance.

Figure 5 shows cavitation performance at several flow coefficients. For comparison, the critical cavitation number numerically predicted at ϕ =0.85, 2000 r/min is also plotted in the figure. Cavitation number, σ is defined as:

$$\sigma = \frac{P_{\text{inlet}} - P_{\text{v}}}{\frac{1}{2}\rho U_{\text{tip}}^2}$$
[13]



Figure 5. Critical cavitation number for several flow coefficients.

It should be noted that the cavitation number equation is different between this paper and Chesnakas et al. (2009). The critical point for pump performance breakdown, where the pump head drops 3% compared to the head without cavitation can be defined to investigate the cavitation development in a pump. It is noted that σ_c is 0.92 for the simulation, and near 0.85 for experiment at the design point ϕ =0.85. The results indicate that the cavitation effect on pump hydraulic performance is a little over-estimated.

4.2 Cavitating flows in the pump rotor

Figure 6 shows the cavity distributions at four operation conditions. The experimental images were proposed by Marquardt, (2011). Figure 6(a)-(d) and the numerical results i.e. Figure 6(e)-(h) are both plotted for comparison. Note that the simulated images are presented using the iso-surface for vapor volume fraction of 10% and in blue color. It is obvious that the cavity is severe, and expands at each rotor blade tip area. These tip cavities are related with the leakage flow in the tip clearance from the pressure side to the suction side of the rotor blade. Thus, the tip cavity is usually known as tip leakage cavitation i.e. TLC. TLC occurs along entire profile from the beginning of the chord. There is also sheet cavity originated from the blade leading edge and attached on the suction side for four cases. For the cases with lower cavitation numbers, e.g. σ =0.2 at ϕ =0.83 and σ =0.17 at ϕ =0.75, the sheet cavities are large in size and connected with the tip cavities as shown in Figure 6(b) and (f), and Figure 6(d) and (h) respectively. The sheet cavity seems to detach from the suction side and migrate to main blade-to-blade flow passage. The developed cavities have enough volume to result in the blockage of rotor passage, and cause the breakdown of the pump performance. It is noted that the scale of sheet cavity is somehow over-estimated by the numerical simulation for each case, as mentioned above.



Figure 6. Cavity in pump rotor at different operation condition.

Figure 7 shows the experimentally observed cavitation pattern in the pump rotor by Tan et al. (2015). Besides tip leakage cavitation and sheet cavitation, there are tip leakage vortex (TLV) cavitation and perpendicular cavitation (PC) vortex. Herein, the perpendicular cavitation is defined by Tan et al.(2015) The present simulation fails to capture the TLV, but well predict the perpendicular cavitation as shown in Figure 8, where the detail PC development is represented within one evolution cycle for a part flow condition ϕ =0.75. The perpendicular cavitation is believed to be the cloud cavitation shedding from the aft part of sheet cavitation at rotor blade suction side by TLV influence. The same phenomenon was also observed by Zhang et al. (2017).


Figure 7. Cavitation pattern in pump rotor.



Figure 8. Perpendicular cavitation evolution (ϕ =0.75, σ = 0.17).

In order to study the internal flow under the perpendicular cavitation condition, we create a cylindrical surface at the diameter of 0.29m under the operation condition of ϕ =0.75 and σ = 0.17. Since this cylinder is near the rotor tip (i.e. the relative radius is 0.955), it is convenient to investigate the interaction between the tip leakage cavitation and sheet cavitation using the numerical data.

Figure 9 shows four snapshots of sheet cavity in one evolution cycle based on the numerical results. It is noted that the sheet cavity locates on the rotor suction side surface and the concentration of cavity increases along chord wise. As the cavitation layer is thin, sheet cavity has no effect on the flow through passage zone. Under the influence of TLV, sheet cavitation becomes unsteady and its shape begins to change. The changes of sheet cavity can block flow passage. At the aft part of cloud cavity, re-entrant jet may accelerate, separating velocity of sheet cavity. In Figure 10, as the variation of cavity volume, the velocity in the flow passage also changes. With occurrence of cavity cloud, the low velocity domain increases at the aft part of the shedding cavity cloud and becomes larger. During the cloud cavity shedding process, the re-entrant jet moves to the upstream. Sheet cavity on the suction side of rotor blade shrinks according to the shedding cavity cloud and a new sheet cavity begins to form for the next cycle.



Figure 9. Distribution of cavity volume fraction at a cylindrical surface (ϕ =0.75, σ = 0.17).



Figure 10. Velocity distribution at a cylindrical surface (ϕ =0.75, σ = 0.17).

Figures 11 and 12 show the pressure distribution at three rotor blade spans for two flow coefficients. For each flow coefficient, the images for the hub span, the middle span and a span near the blade tip were plotted from the left to right. Basically, there is low pressure zone along the suction side due to sheet cavity attached on the suction side of the rotor blade, and the relative length (the ratio of low pressure zone length and the chord length) decreases from hub to tip. At the flow coefficient of ϕ =0.85, the sheet cavity grows along the suction side, and the cavity seems to split at the rear part of the cavity near the blade tip. At the flow coefficient of ϕ =0.83, because the operation has a lower cavitation number than the critical value i.e. σ_c , the low pressure zone develops, and its lengths cover the chord at the hub and middle span. Near the tip area, the low pressure zone extends towards the center of flow passage. In this case, the cavity would separate from the suction side, and cloud cavity occurs.



Figure 11. Pressure distributions at ϕ =0.85 and σ =0.25.



Figure 12. Pressure distributions at ϕ =0.83 and σ =0.2.

5 CONCLUSION

Based on the present work, the following conclusions can be drawn:

- (1) The present numerical method, including the Filter-Based Density Corrected Model, the homogeneous cavitation model and the mesh treatment is suitable for the prediction of cavitation evolution in the axial flow pump. The results indicate that the hydraulic performance and cavitation performance near the design operation condition are predicted with acceptable accuracy compared with the experimental data.
- (2) The internal flows show that there are strong tip leakage cavitation and sheet cavitation in the pump rotor. As the interaction of sheet cavitation and tip leakage flow, there is perpendicular cavitation in the main flow passage of the blade-to-blade channel.

Though the head drop tendency can be captured numerically, the results indicate that the effect of cavitation development on pump hydraulic performance is somehow over-estimated. Thus, the turbulence model used in the paper should be further improved in the future.

ACKNOWLEGEMENTS

This work was financially supported by the National Natural Science Foundation of China (Project No. 51376100), Science and Technology on Water Jet Propulsion Laboratory (Project No. 61422230103162223004) and State Key Laboratory for Hydroscience and Engineering (Project No. sklhse-2017-E-02).

REFERENCE

Batchelor G.K. (1967). An Introduction to Fluid Mechanics. Cambridge University Press, 1-615.

Ducoin A, Huang B. & Yin L Y. (2012). Numerical Modeling of Unsteady Cavitating Flows around a Stationary Hydrofoil. *International Journal of Rotating Machinery*, 2012, 1-17.

Brennen, C. E. (1994). Hydrodynamics of Pumps. Oxford University Press, Oxford, Chap. 7, 1-293.

Hu, P. & Zangeneh, M. (1999). Investigations of 3D Turbulent Flow Inside and Around a Waterjet Intake Duct under Different Operating Conditions. *Journal of Fluids Engineering*, Transactions of the ASME 121, 396– 404.

- Hu, P., Zangeneh, M. (2001). CFD Calculation of the Flow Through a Water-Jet Pump. In: *Proceedings of the International Conference on Waterjet Propulsion III, RINA*, Gothenburg, Sweden, Paper No. 14.
- Yu An , Luo Xian-wu, Ji Bin., Ren-Fang, H., Victor, H., & Hak, K. S. (2010) Cavitation Simulation with Consideration of Viscous Effect at Large Liquid Temperature Variation. *Chinese Physics Letters*, 31(8): 086401.
- Zwart, P. J., Gerber, A. G., & Belamri, T. (2004). A Two-Phase Flow Model for Predicting Cavitation Dynamics, *Fifth International Conference on Multiphase Flow*, Yokohama, Japan, 152.
- Huang B, Wang G Y. & Zhao Y. (2014) Numerical Simulation Unsteady Cloud Cavitating Flow with a Filter-Based Density Correction Model. *Journal of Hydrodynamics*, Ser B, 26: 26–36
- Michael, T. J., Schroeder, S. D., & Becnel, A. J. (2008). *Design of the ONR AxWJ-2 Axial Flow Water Jet Pump*, NSWCCD-50-TR-2008/066, Naval Surface Warfare Center Carderock Div Bethesda Md, 63.
- Chesnakas C J, Donnelly M J, Pfitsch D W, Becnel, A. J., & Schroeder, S. D..(2009). *Performance Evaluation of the ONR Axial Waterjet 2 (AxWJ-2)*, NSWCCD-50-TR-2009/089, Naval Surface Warfare Center Carderock Div Bethesda Md Total Ship Systems Directorate, 89.
- Marquardt M W. (2011) Summary of Two Independent Performance Measurements of the ONR Axial Waterjet 2 (AxWJ-2), NSWCCD-50-TR-2011/016, Naval Surface Warfare Center Carderock Div Bethesda Md, 44.
- Tan D, Li Y, Wilkes I, Vagnoni, E., Miorini, R. L., & Katz, J. (2015). Experimental Investigation of the Role of Large Scale Cavitating Vortical Structures in Performance Breakdown of an Axial Waterjet Pump. *Journal* of *Fluids Engineering*, 137(11), 317–320.
- Zhang D, Shi L, Zhao R, Shi, W., Pan, Q., & van Esch, B. B. (2017). Study on Unsteady Tip Leakage Vortex Cavitation in an Axial-flow Pump Using an Improved Filter-based Model. *Journal of Mechanical Science & Technology*, 31(2), 659-667.

NUMERICAL STUDY OF TURBULENT OSCILLATIONS AROUND A CYLINDER: RANS CAPABILITIES AND SENSITIVITY ANALYSIS

DANIEL VALERO⁽¹⁾, DANIEL B. BUNG⁽²⁾, SEBASTIEN ERPICUM⁽³⁾ & BENJAMIN DEWALS⁽⁴⁾

⁽¹⁾ Hydraulic Engineering Section (HES), FH Aachen University of Applied Sciences, Aachen, Germany University of Liège, Liège, Belgium, valero@fh-aachen.de ⁽²⁾ FH Aachen University of Applied Sciences, Aachen, Germany bung@fh-aachen.de ^(3.4) Hydraulics in Environmental and Civil Engineering (HECE), ArGEnCo Department, University of Liège, Liège, Belgium s.erpicum@ulg.ac.be; b.dewals@ulg.ac.be

ABSTRACT

Numerical modelling is commonly used in a large range of environmental fluid mechanics applications. It has become widely accepted and, with increasing computer power, the employed models are increasing also in complexity. Reynolds Averaged Navier-Stokes (RANS) equations are the preferred approach for 3D problems, being especially suited for the computation of the mean flow. However, little is known about RANS performance for the fluctuating quantities; thus, it is reasonable to expect an impairment in the accuracy. In this study, a simple geometry (cylindrical pier) was subjected to different numerical settings in order to assess their effect on the development of the physically based flow instabilities. Mesh refinement has been shown to enhance the perturbation growth rate while maximum CFL value did not produce any effect. RNG $k - \epsilon$ and $k - \omega$ have shown to be more dissipative than $k - \epsilon$. Some advection schemes seem to increase the spurious perturbation converging to the physically based ones. Finally, introducing an experimentally based perturbation at the inlet has proven to speed up the process.

Keywords: Turbulence modelling; circular pier; flow oscillations; instabilities.

1 INTRODUCTION

Turbulent oscillations are commonly found in environmental flows. From simple flows to complex worldscale phenomenon, flow detachment can be observed with different frequencies and amplitudes. Current available techniques allow numerical study of such flows, with RANS modelling being one of the most widely used approaches of the techniques grouped within Computational Fluid Dynamics (CFD). Despite some RANS limitations (Spalart, 2000; Pope, 2000; Bradshaw et al., 1996), it is often observed that RANS models are able to represent these oscillations - in addition to the mean flow structure - as exemplarily observed in Figure 1. However, little is known on how these flow fluctuations appear or their resemblance with reality. Under similar circumstances to those of Fig. 1, velocity fluctuations do not appear in RANS simulations. This is the case shown in Fig. 2, for a simulation of a turbulent jet where the flow oscillations do not spawn, despite the mean flow structure is accurately reproduced (Valero and Bung, 2016).



Figure 1. Exemplary shear region oscillations produced in a groin field. Left: laboratory shear flow and right: RANS model. Flow from left to right (the tracer indicates an outfall jet discharging in cross flow direction upstream of the structure).

Other studies have indirectly addressed flow oscillations with RANS modelling (e.g.: study of Bayon et al., 2016 for a small Reynolds number hydraulic jump), nonetheless, only a few studies have dealt with this topic directly. This is the case of Peltier et al. (2015), where use of the shallow water equations was made,

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

altogether with a depth-averaged $k - \epsilon$ turbulence model (Erpicum et al., 2009), to reproduce large-scale coherent structures in a shallow reservoir flow (Peltier et al., 2014a; 2014b; Dufresne et al., 2010). Dewals et al. (2008) suggested that, to trigger the instabilities, an asymmetry might be introduced to the flow. Dewals et al. (2008) used a linearly uneven inlet velocity profile, with a deviation from uniform flow of just +/- 1 %. However, as in the case of flow shown in Fig. 1, in the study of Bayon et al. (2016), oscillations appeared without explicitly introducing any perturbation to the flow. Usually, the difference between different simulations settings is very small yet the results differ significantly (some show instantaneous fluctuating profiles and some not). Thus, how small perturbations introduced (consciously or not) in the simulation affect the final resulting velocity fields is still an unknown. Moreover, a refined set of simulations might pass a mesh sensitivity analysis without even showing any turbulent flow fluctuation and, after some refinement, show these fluctuations.

In this study, a flow around a cylinder has been studied. This well-known simple flow (Williamson, 1996), leading to Kármán vortex street, admits investigation of an oscillatory event while allowing sensitivity analysis related to parameters such as mesh refinement, advection scheme, Courant–Friedrichs–Lewy (CFL) maximum value and selection of the turbulence model. The study has been conducted at a low Reynolds number which ensured that a 2D instability takes place instead of a complex 3D one (Williamson, 1996). Hence, a 2D numerical study may result in satisfactory. Numerical simulations have been conducted by means of the commercial CFD package FLOW-3D. Also, by preserving the Reynolds number small, simulations can reproduce a larger time which eases the trigger and development of the instabilities.



Figure 2. Exemplary vertical jet. Left: instantaneous High-Speed image detailing the Region of Flow Establishment and the Near Field. Right: Result of the numerical model, not appearing any turbulence oscillation. Note the difference in the x and z scales, numerical model shows also the Far Field region. See Valero and Bung (2016) for further details on this study.

2 DESCRIPTION OF THE NUMERICAL EXPERIMENT

A cylinder of diameter D = 1 cm is located at x = y = 0 (see Fig. 3) surrounded by water; with density $\rho = 1000 \text{ Kg/m}^3$ and dynamic viscosity $\mu = 0.001 \text{ Kg m}^{-1}\text{s}^{-1}$. The inlet water mean velocity in the *x* direction is $\langle U \rangle = 0.01 \text{ m/s}$; with a null velocity fluctuation in *x* direction (u = 0 m/s). The inlet mean velocity in the *y* direction is null ($\langle V \rangle = 0 \text{ m/s}$), however, its fluctuation (v) follows the expression:

$$\begin{cases} v = v_{max} \cos\left(\frac{2\pi t}{T}\right); & t \le 35 s\\ v = 0; & t > 35 s \end{cases}$$
[1]

where the fluctuation period *T* has been arbitrarily chosen as T = 2 s. v_{max} is the maximum value of the *y* direction velocity fluctuation. v_{max} has taken different values in the simulated cases. In any case, the value of v_{max} is two orders of magnitude smaller than $\langle U \rangle$, which ensures that its value remains physically plausible. The aim of this expression is to introduce a perturbation in the flow region and to cut down the fluctuating velocity energy introduced to the flow once the instability is triggered, which happens at $t \approx 35$ s under normal conditions (case A02, later described in Table 1). Thus, if the instability keeps growing, it is only fed with energy from the mean flow. The critical time t = 35 s corresponds to the authors observations, as later described.

Distances to boundaries were chosen based on numerical study of Baykal et al. (2015) and are marked in Fig. 3. The left boundary corresponds to an inlet boundary, the right one to an outflow, the upper and lower one are symmetry conditions.



Figure 3. Geometry of the case study. Coordinate system centered at the cylinder.

The resulting Reynolds number ($Re = U \cdot D/\nu$) is 100. Low Reynolds number ensures that the expected flow remains two dimensional, allowing a good representation of the studied phenomenon with a 2D simulation. The studied vortex shedding corresponds then to Regime A-B (parallel laminar shedding, 49 < Re < 194), according to Williamson (1996). The Strouhal number, related to the wake oscillations, can be defined as:

$$S = \frac{f \cdot D}{U}$$
[2]

with *f* as the frequency of the vortex, *D* as the cylinder diameter and *U* as the approaching flow velocity. Williamson (1996) showed that experimental results in the parallel laminar shedding regime (both for air and water flows) are in good agreement (\sim 1%) with the following relationship:

$$S = -\frac{3.3265}{Re} + 0.1816 + 1.6 \cdot 10^{-4} Re$$
[3]

which yields a value of 0.164 for the studied case and, consequently, to a frequency f = 0.164 Hz.

3 NUMERICAL SETUPS

3.1 General remarks

3128

The herein described numerical experiments were carried out by means of the commercial CFD package. FLOW-3D version 11.1. Reynolds Averaged Navier-Stokes equations (RANS) as described by Pope (2000) have been numerically solved by discretizing them using the Finite Volume Method (Versteeg and Malalasekera, 2007; Hirsch, 2007). A 2D structured mesh with constant size cells has been used. A total time of 100 s has been simulated for all the cases, which was usually enough to observe the triggering of the instabilities.

A thorough mesh sensitivity test was conducted (see Table 1), sensitivity to maximum CFL number was also tested (see Table 2) and sensitivity to advection scheme, turbulence model and perturbation intensity (v_{max}) were studied. Fulfilling the CFL condition is elemental for stability of explicit schemes of wave and convection equations. It states that the distance travelled during the time interval Δt , by the disturbances propagating with velocity U, needs to remain below the minimum distance between two mesh points. For a 1D problem, the CFL stability condition reduces to:

$$CFL = \frac{U\,\Delta t}{\Delta x} < 1 \tag{4}$$

and has to be satisfied at all the cells of the employed mesh. In other words, it expresses that the ratio $\Delta t/\Delta x$ has to be selected so that the domain of dependence of the differential equation to be solved should be entirely contained in the numerical domain of dependence of the discretized equations (i.e., the cell size). Studying the dependence of the resulting oscillations on this value might help to discard the probability that the oscillations are representing any type of numerical instability instead of the physically based ones, which are contained in the nature of the modelled equations.

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

aa //II\	NUMBER	ADVECTION	CFL	TURBULENCE
$v_{max}/\langle 0 \rangle$	OF CELLS	SCHEME	(MAX)	MODEL
0.01	100,000	2 nd , mon. pres. (*)	0.50	$k-\epsilon$
0.01	200,000	2 nd , mon. pres.	0.50	$k-\epsilon$
0.01	350,000	2 nd , mon. pres.	0.50	$k - \epsilon$
0.01	500,000	2 nd , mon. pres.	0.50	$k - \epsilon$
0.01	650,000	2 nd , mon. pres.	0.50	$k - \epsilon$
order monotonicity p	reserving scheme (n	non. pres.), also known as	flux limiters	or slope limi
	<i>v_{max} /⟨U⟩</i> 0.01 0.01 0.01 0.01 0.01 order monotonicity p	v_{max} / $\langle U \rangle$ NUMBER OF CELLS 0.01 100,000 0.01 200,000 0.01 350,000 0.01 500,000 0.01 650,000 order monotonicity preserving scheme (n	v_{max} / $\langle U \rangle$ NUMBER OF CELLS ADVECTION SCHEME 0.01 100,000 2 nd , mon. pres. (*) 0.01 200,000 2 nd , mon. pres. 0.01 350,000 2 nd , mon. pres. 0.01 350,000 2 nd , mon. pres. 0.01 500,000 2 nd , mon. pres. 0.01 650,000 2 nd , mon. pres. order monotonicity preserving scheme (mon. pres.), also known as and times (2007) for further purplemention	v_{max} / $\langle U \rangle$ NUMBER OF CELLS ADVECTION SCHEME CFL (MAX) 0.01 100,000 2^{nd} , mon. pres. (*) 0.50 0.01 200,000 2^{nd} , mon. pres. (*) 0.50 0.01 350,000 2^{nd} , mon. pres. 0.50 0.01 350,000 2^{nd} , mon. pres. 0.50 0.01 500,000 2^{nd} , mon. pres. 0.50 0.01 650,000 2^{nd} , mon. pres. 0.50 order monotonicity preserving scheme (mon. pres.), also known as flux limiters 0.50

 Table 2. Main features of the simulations conducted for the maximum CFL value sensitivity analysis. Note that case B03 corresponds to simulation A02

CASE	$v_{max}/\langle U \rangle$	NUMBER OF CELLS	ADVECTION SCHEME	CFL (MAX)	TURBULENCE MODEL
B01	0.01	200,000	2 nd , mon. pres.	0.10	$k-\epsilon$
B02 B03(A02)	0.01	200,000	2 nd , mon. pres.	0.25	$k - \epsilon$ $k - \epsilon$
B04	0.01	200,000	2 nd , mon. pres.	0.90	$k-\epsilon$

3.2 Advection scheme

The first order upwind scheme is the most basic method which can be used while ensuring numerical stability (Hirsch, 2007). In order to achieve high orders of accuracy, more mesh points have to be involved in the numerical scheme and, consequently, a second order method might be used. However, some problems might appear. A second order with monotonicity preserving (as described by Van Leer, 1977) allows use of a second order method with some of the benefits of the first order methods. Table 3 sums the main settings of the simulations which have been conducted for the advection scheme sensitivity tests.

 Table 3. Main features of the simulations conducted for the advection scheme sensitivity tests. Note that

 case C03 corresponds to simulation A02

case C03 corresponds to simulation A02.					
CASE	22 // I I\	NUMBER	ADVECTION	CFL	TURBULENCE
CASE	$v_{max} / \langle 0 \rangle$	OF CELLS	SCHEME	(MAX)	MODEL
C01	0.01	200,000	1 st order	0.50	$k-\epsilon$
C02	0.01	200,000	2 nd order	0.50	$k - \epsilon$
C03(A02)	0.01	200,000	2 nd , mon. pres.	0.50	$k-\epsilon$

3.3 Turbulence model

As a consequence of the Reynolds averaging procedure, some additional assumptions are necessary to close the system. In this study, different eddy viscosity models have been used. Despite existence of more complex models (i.e.: Reynolds-Stress Transport models or RST), it seems that CFD community predicts continued use of the well-stablished one and two equation turbulence models (Slotnik et al., 2014). Also, RST models can lack of robustness under some circumstances and are occasionally less accurate than standard RANS models (Slotnik et al., 2014).

The turbulence model used by default was $k - \epsilon$, which is the most widely used by the community. It proposes one equation for the turbulent kinetic energy (k) and another for its dissipation (ϵ). Moreover, also RNG $k - \epsilon$ and $k - \omega$ were tested. RNG $k - \epsilon$ as defined by Yakhot and Orszag (1986) and Yakhot et al. (1992), is expected to reproduce flow separation better than the standard $k - \epsilon$ (Speziale and Thangam, 1992; Bradshaw, 1997). The $k - \omega$ model, developed by Wilcox (1998; 2008), changes the equation of the ϵ by another for $\omega = \epsilon/k$. This model is known to perform superiorly for many types of flows when compared to the standard $k - \epsilon$ (Pope, 2000) although it has not been as widely tested as $k - \epsilon$ model (Bradshaw et al., 1996).

Furthermore, it has been previously noticed that $k - \epsilon$ is less sensitive to free stream values. When choosing a second variable in two equations models, freestream sensitivity should be given higher priority (Spalart, 2000). All model's parameters were selected by default for this study. Settings on the simulations conducted for the turbulence model sensitivity test are summed up in Table 4. Accuracy of these models for mean flow properties was widely documented in literature (e.g.: Valero and Bung, 2016; Valero et al., 2016; Bayon et al. 2016), but little is known of their capabilities to reproduce flow fluctuations. Provided that they alter the flow viscosity, changes in the resulting oscillations might appear.

CASE	$v_{max}/\langle U \rangle$	NUMBER OF CELLS		CFL (MAX)	TURBULENCE MODEL
D01(A02)	0.01	200,000	2 nd , mon. pres.	0.50	$k - \epsilon$
D02	0.01	200,000	2 nd , mon. pres.	0.50	RNG $k - \epsilon$
D03	0.01	200.000	2 nd , mon. pres.	0.50	$k - \omega$

 Table 4. Main features of the simulations conducted for the turbulence model sensitivity tests. Note that case D01 corresponds to simulation A02.

3.4 Inlet perturbation

Following advice of Dewals et al. (2008), a perturbation was introduced at the inlet in the form of Eq. [1]. The parameter v_{max} , which represents the maximum intensity of this perturbation, was kept considerably small to avoid creating unrealistic fluctuations. In order to use prudent estimations of v_{max} , Pope (2000) recommendation on turbulence intensities below 5 % was embraced. Similarly, Tachie et al. (2000) found values around 2.4 to 3.5 % for open channel flows. It must be noted that the parameter v_{max} represents a maximum value and so it is larger than the actual mean turbulence intensity. Information on the simulations conducted to analyse the inlet perturbation influence is summarized in Table 5.

Table 5.	Main features of the simulations	conducted for the inlets	perturbation se	ensitivity tests.	Note that
	case E02	corresponds to simulation	n A02		

CASE	$v_{max}/\langle U \rangle$	NUMBER OF CELLS		CFL (MAX)	TURBULENCE MODEL
E01	0.00	200,000	2 nd , mon. pres.	0.50	$egin{array}{c} k-\epsilon\ k-\epsilon\ k-\epsilon\ k-\epsilon\ k-\epsilon \end{array}$
E02(A02)	0.01	200,000	2 nd , mon. pres.	0.50	
E03	0.02	200,000	2 nd , mon. pres.	0.50	
E04	0.05	200,000	2 nd , mon. pres.	0.50	

4 RESULTS

4.1 General remarks

Flow is introduced at the inlet and takes around 7 s to arrive at the cylinder. The solid contour, which is a discontinuity in the fluid region, forces the flow to slow down and to modify its trajectory. Fluid is attached to the cylinder up to the detachment point were the downstream wake starts (see Fig. 4). Landau and Lifshitz (1987) described the laminar wake as follows: "at great distances behind the body, the velocity (fluctuation) $(u \ v)$ is noticeably different from zero only in a relatively narrow region near the *x* axis. This region, called the laminar wake, is reached by fluid particles which move along streamlines passing fairly close to the body". The fluid viscosity smooths out the velocity discontinuity produced close to the solid surface and the rotational produced immediately close to the cylinder body penetrates into the fluid region, from which it is transported by convection into the wake region. In a different manner, the viscosity has a negligible effect at any point where the fluid particles have not interacted with the solid body. Consequently, the flow at a certain distance from the body could be considered as potential flow everywhere except in the wake.

Despite the simplicity of the formulated problem, it is remarkable that the cylinder wake involves the interactions of three shear layers: a boundary layer, a separating free shear layer and a wake (Williamson, 1996). The numerical simulations agree with Landau and Lifshitz (1987) considerations and Williamson (1996) description of the flow. This fact supports a deeper analysis of the represented phenomenon, although the goal of this study is to assess the importance of different variables in the trigger of the observed oscillations.



Figure 4. Case A04 results decomposed in mean and instantaneous turbulent fluctuation. Left: mean flow and Right: turbulence intensity at t = 100 s, defined as the magnitude of the fluctuation divided by the magnitude of the mean velocity.

4.2 Mesh sensitivity

Mesh sensitivity is the first parameter which was assessed. Simulations listed in Table 1 contain the simulations setup for this analysis. For the sake of clarity, only v was herein examined at a single location: x = 2.5 cm ($2 \cdot D$ away from the downstream cylinder surface). Figure 5 shows the temporal series for cases A01, A02, A03 and A04 (with a characteristic cell size of 0.0547 cm, 0.0387 cm, 0.0293 cm and 0.0245 cm respectively).



Figure 5. Time series of v at x = 2.5 cm downstream the cylinder center for cases listed in Table 1. Note that the inlet perturbation was shut down after t = 35 s.

It can be observed that in A02 and A03 simulations, the instability was increasing after $t \approx 40$ s, while for cases A01 and A04 it happened slightly before. However, the most remarkable difference was that when the mesh was refined, the temporal growth rate of the perturbations also increased while keeping the characteristic frequency stable. It must be noted that the coarser mesh reproduced the cylinder diameter with at least 18 cells and the finer with more than 40 cells, which represents more than what is usually used to reproduce geometry details in more complex geometries. Convergence was reached between simulations A04 and A05. However, case study A02 was taken as base case since the goal of this study was just to perform a sensitivity analysis.

4.3 Maximum CFL value sensitivity

Different maximum CFL values were tested, corresponding to cases listed in Table 2. The results show that there was no effect upon this parameter (see Fig. 6) so dismissing the hypothesis that the observed oscillations in the numerical results are from numerical nature and not due to the flow equations (physically based fluctuations).



Figure 6. Time series of v at x = 2.5 cm downstream the cylinder center for cases listed in Table 2. Note that the inlet perturbation was shut down after t = 35 s.



Figure 7. Time series of v at x = 2.5 cm downstream the cylinder center for cases listed in Table 3. Note that the inlet perturbation was shut down after t = 35 s.

4.4 Advection scheme sensitivity

Three explicit advection schemes were tested, as listed in Table 3. First order and second order with monotonicity preserving schemes yielded similar results, despite the first order scheme (case C01) was initially triggered with the first fluid relocation downstream the cylinder (when the simulation was started). Notably, case C02 (with second order scheme) resulted a larger growth rate of the perturbations than the first order scheme. It seems that case C02 converged up to a similar level to the one later observed at section 4.6.

Monotonicity preserving algorithms (also called flux limiters or slope limiter) are usually employed to avoid spurious oscillations that might appear when advecting a flow quantity (Hirsch, 2007). This might be the reason that cases C01 and C02 showed an earlier trigger of the instability (numerically triggered) and C03 developed later (more physically triggered). An interesting point is that all cases seemed to grow as one may physically expect despite the difference in their trigger time.

4.5 Turbulence model sensitivity

Turbulence model tests for the cases listed in Table 4 show an important disparity in the results. For all the tests conducted using $k - \epsilon$ model, a small amplitude perturbation seemed to be fed with energy gradually and it consequently increased in amplitude up to, in some cases, a stable value. However, cases D02 and D03 (RNG $k - \epsilon$ and $k - \omega$ respectively) showed opposing behaviors. After the first fluid flow circled the cylinder, a big amplitude perturbation was formed and was slowly dissipated later during the remaining simulation time, without new fluctuations appearing. It must be noted that the values for the inlet perturbation were the same for all these cases ($v_{max} = 1\% \langle U \rangle$), which might be insufficient to trigger the physical instability with the other two turbulence models. It is remarkable that the $k - \omega$ case damped the oscillation faster than the RNG $k - \epsilon$ case.



Figure 8. Time series of v at x = 2.5 cm downstream the cylinder center for cases listed in Table 4. Note that the inlet perturbation was shut down after t = 35 s.

4.6 Inlets perturbation sensitivity

As listed in Table 5, different v_{max} values (see Eq. [1]) were tested to assess the influence of the inlet perturbation on the resulting flow. It must be noted that Eq. [1] ceased to affect the flow at t = 35 s, so that the energy which can be transported to the wake is the same at the steady scenario for all the cases. Consequently, it could be expected a similar final result for all the cases while the transient could show some differences still.

Figure 9 shows the obtained results. It can be observed that cases E04 and E03 (v_{max} taking a 5 % and 2% of $\langle U \rangle$ at the inlet, respectively) converged to a similar amplitude. The most remarkable result is that only the case with no perturbation (E01) did not clearly develop the physical instability. However, when zooming on the results data, it can be observed that some inconspicuous waves started to appear at around $t \approx 80$ s while the order of magnitude of its amplitude was considerably smaller. All the cases showed a very similar frequency, independently of the intensity of the input perturbation. It must be noted, as well, that the frequency of the input perturbation corresponded to 0.5 Hz, thus faster than the developed instability. No test was conducted to investigate how slower perturbations affect the triggering of the laminar wake instability. However, as the case with null v_{max} also showed the same frequency as the other cases, it could be expected that no big difference should occur.



Figure 9. Time series of v at x = 2.5 cm downstream the cylinder center for cases listed in Table 5. Note that the inlet perturbation was shut down after t = 35 s.

4.7 Discussion on the frequency

All the frequencies found are below the physically based value of 0.164 Hz (at a mean value of 90.3%). Despite it cannot be concluded from the conducted experiments, this could be due to the effect of the increased viscosity resulting from the use of eddy viscosity models. This might cause a damping of the fluctuations. Nonetheless, results have proven to be in good agreement with the physically based value and are not as much affected as the oscillations growth rate.



Figure 10. Frequency accuracy (% of agreement) for all the modelled cases. All the obtained frequencies are below the physically based one.

5 CONCLUSIONS

Sensitivity to numerical setup parameters for a simple 2D flow has been conducted. It has been observed that RANS modelling is capable of reproducing cylinder laminar wake when using the $k - \epsilon$ turbulence model. Further testing is necessary to conclude the same for the RNG $k - \epsilon$ and $k - \omega$ turbulence models. It has been also observed that maximum CFL value did not exert any effect on the simulated instability, which allowed concluding that the instability is not of numerical nature. However, small inconspicuous waves generated by the advection scheme may help triggering the physically based instability. Luckily, these numerically triggered instabilities seemed to converge to the same characteristics as reproduced by the modelled equations. The result from the inlet perturbation sensitivity analysis is also interesting. Following suggestion of Dewals et al. (2008), the inlet boundary condition was slightly perturbed. For the case where no perturbation is considered, the instability appeared several orders of magnitude smaller, and hence almost invisible. Independently of the intensity of this perturbation, it seemed to be converging to a similar amplitude – and so the instability is likewise turbulent or energetic.

REFERENCES

- Baykal, C., Sumer, B.M., Fuhrman, D.R., Jacobsen, N.G. & Fredsøe, J. (2015). Numerical Investigation of Flow and Scour around a Vertical Circular Cylinder. *Philosophical Transactions of the Royal Society of London A: Mathematical, Physical and Engineering Sciences,* 373(2033), 20140104.
- Bayon, A., Valero, D., García-Bartual, R., Vallés-Morán, F.J. & López-Jiménez, P.A. (2016). Performance Assessment of OpenFOAM and FLOW-3D in the Numerical Modeling of a Low Reynolds Number Hydraulic Jump. *Environmental Modelling & Software*, 80, 322-335.
- Bradshaw, P., Launder, B.E. & Lumley, J.L. (1996). Collaborative Testing of Turbulence Models. *Journal of Fluids Engineering*, 118(2), 243-247.
- Bradshaw, P. (1997). Understanding and Prediction of Turbulent Flow—1996. International Journal of Heat and Fluid Flow, 18(1), 45-54.
- Dewals, B. J., Kantoush, S. A., Erpicum, S., Pirotton, M. & Schleiss, A. J. (2008). Experimental and Numerical Analysis of Flow Instabilities in Rectangular Shallow Basins. *Environmental Fluid Mechanics*, 8(1), 31-54.
- Dufresne, M., Dewals, B., Erpicum, S., Achambeau, P. & Pirotton, M. (2010). Classification of Flow Patterns in Rectangular Shallow Reservoirs. *Journal of Hydraulic Research*, 48(2), 197-204
- Erpicum, S., Meile, T., Dewals, B. J., Pirotton, M. & Schleiss, A. J. (2009). 2D Numerical Flow Modeling in a Macro-Rough Channel. *International Journal for Numerical Methods in Fluids*, 61(11), 1227-1246.
- Hirsch, C. (2007). *Numerical computation of internal and external flows: The fundamentals of computational fluid dynamics*. Butterworth-Heinemann.
- Landau, L.D. & Lifshitz, E.M. (1987). Fluid Mechanics. Butterworth Heinemann, Book, Volume 6.
- Peltier, Y., Erpicum, S., Archambeau, P., Pirotton, M. & Dewals, B. (2014a). Experimental Investigation of Meandering Jets in Shallow Reservoir. *Environmental Fluid Mechanics*, 14(3), 399-710.
- Peltier, Y., Erpicum, S., Archambeau, P., Pirotton, M. & Dewals, B. (2014b). Meandering Jets in Shallow Rectangular Reservoirs: POD Analysis and Identification of Coherent Structures. *Experiments in Fluids*, 55(6), 1-16.
- Peltier, Y., Erpicum, S., Archambeau, P., Pirotton, M. & Dewals, B. (2015). Can Meandering Flows in Shallow Rectangular Reservoirs Be Modeled with 2D Shallow Water Equations? *Journal of Hydraulic Engineering*, 141(6), 04015008.
- Pope, S.B. (2000). Turbulent flows. Cambridge University Press.
- Slotnik, J., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E. & Mavriplis, D. (2014). CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences. NASA Technical Report Server (NTRS), NASA/CR-2014-218178.
- Spalart, P.R. (2000). Strategies for turbulence modelling and simulations. International Journal of Heat and Fluid Flow, 21(3), 252–263.
- Speziale, C.G. & Thangam, S. (1992). Analysis of an RNG Based Turbulence Model for Separated Flows. International Journal of Engineering Science, 30(10), 1379-1388.
- Tachie, M.F., Bergstrom, D.J. & Balachandar, R. (2000). Rough Wall Turbulent Boundary Layers in Shallow Open Channel Flow. *Journal of Fluids Engineering*, 122(3), 533-541.
- Van Leer, B. (1977). Towards the Ultimate Conservative Difference Scheme. IV. A New Approach to Numerical Convection. *Journal of Computational Physics*, 23(3), 276-299.
- Valero, D. & Bung, D.B. (2016). Sensitivity of Turbulent Schmidt Number and Turbulence Model to Simulations of Jets in Crossflow. *Environmental Modelling & Software*, 82, 218-228.
- Valero, D., Bung, D. & Oertel, M. (2016). Turbulent Dispersion in Bounded Horizontal Jets: RANS Capabilities and Physical Modeling Comparison. In Sustainable Hydraulics in the Era of Global Change: Proceedings of the 4th IAHR Europe Congress, Liege, Belgium, 27-29 July 2016, 14.
- Versteeg, H.K. & Malalasekera, W. (2007). An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education.
- Wilcox, D.C. (1998). Turbulence Modeling for CFD. La Canada, CA: DCW industries, Vol. 2, 103-217.
- Wilcox, D.C. (2008). Formulation of the K-Omega Turbulence Model Revisited. AIAA Journal, 46(11), 2823-2838.
- Williamson, C.H. (1996). Vortex Dynamics in the Cylinder Wake. Annual review of fluid mechanics, 28(1), 477-539.
- Yakhot, V. & Orszag, S. A. (1986). Renormalization Group Analysis of Turbulence. I. Basic Theory. *Journal of scientific computing*, 1(1), 3-51.
- Yakhot, V., Orszag, S. A., Thangam, S., Gatski, T. B. & Speziale, C. G. (1992). Development of Turbulence Models for Shear Flows by a Double Expansion Technique. *Physics of Fluids A: Fluid Dynamics*, 4(7), 1510-1520.

PHYSICAL MODELLING TESTING FOR RAM PUMP PERFORMANCE

NOOR AZME OMAR⁽¹⁾, ICAHRI CHATTA⁽²⁾, MOHD BAHARUM MUHAMAD DIN⁽³⁾, MOHD FAUZI MOHAMAD⁽⁴⁾, MOHD KAMARUL HUDA SAMION⁽⁵⁾, AHMAD FARHAN HAMZAH⁽⁶⁾, MOHD KHAIRUL NIZAR BIN SHAMSUDDIN⁽⁷⁾ & MOHD RADZI ABD. HAMID⁽⁸⁾

(1.2.3.4.5.6.7.8) National Hydraulic Research Institute of Malaysia (NAHRIM), Seri Kembangan, Selangor, Malaysia. azme@nahrim.gov.my

ABSTRACT

The provision of adequate domestic water supply for the scattered rural population is a major problem in many developing countries including Malaysia. Operation for the common pumping system has become an issue of water supply to rural and remote areas due to a few reasons such as power supply shortage, inaccessible infrastructure and new settlement. In order to minimize the problem, a study was conducted in NAHRIM's laboratory using ram pump technology. The aim of the study is to identify the performance of ram pump for achieving sustained performance at optimum operational condition in laboratory scale. The testing component consisted of the ram pump and inverter controlled pumping system with a series of pipelines. A total of 2 testing conditions were conducted during the study, which were the dam simulation and river simulation with 6 testing scenarios. It was found that the ram pump could transfer 10% of the total volume of water sources from any open channel using only kinetic energy. The result of this study also found that the ram pump has a potential in supplying water for rural and remote settlement area especially near the river.

Keywords: Gravity flow; ram pump; water resources; testing conditions; kinetic energy.

1 INTRODUCTION

A ram pump is a hydraulic pump that uses energy from a falling quantity of water to pump some of it to an elevation much higher than the original level at the source. No other energy is required as long as there is a steady flow, it will work continuously and automatically (De Carvalho et al., 2011; Filipan and Bergant, 2003; Herlambang and Wahjono, 2006). The provision of adequate domestic water supply for the scattered rural population is a major problem in many developing countries. Fuel and maintenance costs to operate conventional pumping system are becoming prohibitive. The ram pump is an alternative pumping device that is a relatively simple technology that uses renewable energy and is durable. Ram pump has been used for over two centuries in many parts of the world (Suarda and Wirawan, 2008; Tessema, 2000; Inthachot et al., 2015). Their simplicity and reliability made them commercially successful, particularly in certain European countries, in the days before automation technology became widely available (Maratos, 2003). As technology advanced and became increasingly reliant on sources of power derived from fossil fuels, the ram pump was neglected. Large-scale ram pumps became attractive and small-scale ram pump technology became less attractive (Twort et al., 2000). In United Kingdom, manually controlled precursor of the ram pump has been invented (Sakenian Dehkordi and Arshad, 2012; Shuaibu Ndache Mohammed, 2007; Atharva Pathak et al., 2016). Although hydraulic ram pumps come in a variety of shapes and sizes, they all have the same basic component states of a ram pump, which are the main vessel, waste valve, delivery valve, snifter valve, air chamber and relief valve (Matthias Inthachot et al., 2015; Wallace and Warren, 1941). Ram pump's cycle had three phases: acceleration, delivery and recoil. A cyclic pumping action will produce characteristics beat during operation. The aim of the study is to identify the performance of ram pump for achieving sustained performance at optimum operational condition in laboratory scale. As to meet the objective above, the scope of works includes: testing the performance of ram pump system with capacity of 40 meters of a total maximum head and 30m³ volume water per day (equal to supplying water to 10 houses model), undertaking full scale test on the ram pump system inside NAHRIM's Hydraulic Instrumentation Laboratory, testing ram pump supply by river flow simulation, testing ram pump supply by reservoir/ lakes/ dam flow simulation and observing breakdown during testing for maintenance input.

2 METHODOLOGY

The setup components in the project were: a set of supply pump with inverter system, a unit of ram pump, water tanks (as water supply and receiving tank), a set of PVC pipeline system, flowmeters, valves and joints, structural component such as pump plinth and tank platform as well as some civil work.

Inverter pump was used to supply water to the ram pump at certain flow rate. The flow rate was measured by two units of factory calibrated flow meter that were located before and after the ram pump. The most common concept of surface water flow is caused by two types of sources, which are river water flow sources and dam

water flow sources from upstream to downstream. Based on the inconsistency flow rates, the ram pump testing was carried out in several scenarios. There were two types of flow, which are (i) river flow simulation (by M1, By Passed using Inverter system); and (ii) dam simulation (by M2, Static Tank Method as illustrated in Figure 1 as below. There were six (6) testing scenarios during the testing period, which were precommissioning testing and pre-dam simulation testing, dam simulation testing, pre-river simulation testing, river simulation testing, the 24hrs simulation testing and the free flow simulation testing.



Figure 1. Schematic Diagram for Ram Pump Performance Test.

2.1 Pre-commissioning testing

The Pre-Commissioning Testing purposes was to make sure that the system operate smoothly. No testing data was recorded during the test run. This test served as a calibration for ram pump. All ram pump system components were assembled accordingly for design in order to have a stable ram pump performance. The valves were controlled to assure optimum water supplied to the ram pump from dam tank.

Water was supplied to the ram pump from0 to 10 m³/hour for about 5-10 minutes. The ram pump optimum operation was observed. The flow rate was increased whenever the ram pump was not operating, in interval of 10 m³/hr from 10 up to 50m³/hr. The valve was closed after the ram pump works continuously. The ram pump was started by forcing the chock up and down for a few minutes; the ram pump was started to squirt the water. At certain forcing, ram pump chock automatically moved up and down. The forcing was stopped when the ram pump was squirting at constant flow. The inverter pump flow rate was increased if the chock did not move up and down automatically. At this stage, the pressure gauge at the top of the ram pump vessel was monitored. The pressure value was increased from 0 to 1.5 bars (30psi). When the pressure reached 1.5 bars, ram pump was started to squirt the water consistently. The valve after the ram pumps was opened slowly to let the water flow from ram pump to the receiving tank. Water sent recorded on flow meter after ram pump was remarked. From this moment onwards, the ram pump was ready for further scenarios testing. After the ram pump had performed its pressure in a stable range for 5 - 10 minutes of each inverter pump flow rate setting, the inverter setting was increased in order to find out which of the settings that make the ram pump shut down (unperformed), consistently performed and strongly performed. Now, the ram pump had performed in consistent inverter pump setting and ready for further testing. The scenario was run for a few hours continuously as to oversee any occurrence of breakdown.

2.2 Pre-dam simulation testing

After the Pre-Commissioning Testing, ram pump was ready to be tested in further scenarios. This testing aim to find out the exact setting of inverter pump flow rate as to determine the minimum workable flow rate for selected size of ram pump. Firstly, the ram pump was operated in steady performance at inverter pump setting begin at 39 Hertz. Then, ram pump performance was recorded every hour for eight (8) hours continuously and the decrease in the water level at Dam Tank was also recorded. The ram pump was restarted after the ram pump stop operating. This time, it was decreased from 39 Hertz to 38 Hertz, and repeated until it reached 30 Hertz. Through this testing, the ramp pump design factor at application site could be determined in terms of annual flow rate of the water body or by the necessity of having small size water dam.

2.3 Dam simulation testing methodology

This test was to evaluate whether the ram pump require a small dam for the effective and continuous operations during on-site application. During this testing, Inverter Pump was set in interval of one (1) between

33 to 39 Hertz for water supply to the Dam Tank. There was a different level of water detention at Dam Tank. Firstly, the ram pump was operated as usual; Inverter Pump setting was started at 39 Hertz. The water level of Dam Tank was observed after the ram pump operated for a few hours continuously. The test was repeated with Inverter Pump setting at 38 Hertz to 37.8 Hertz and the water level of the Dam Tank was observed;

2.4 Pre-river simulation testing methodology

This test was to confirm that ram pump could be operated without the assistant of Dam Tank. There were four (4) adjustable values to control the flow to the ram pump in order to achieve a constant flow rate. Firstly, the Inverter Pump constant flow rate was set at 33 Hertz. Then, Values A, B, C and D were adjusted as follows: Fully Closed D-Value, Fully Open A-Values, ¼ Opened B-Values and ½ Opened C-Values. After that, Inverter Pump was set at more than 33 Hertz, followed by the setting at less than 33 Hertz.

2.5 River simulation testing methodology

This test was to confirm that ram pump could be operated without the assistant of Dam Tank at a certain setting of Inverter Pump. Firstly, Inverter Pump constant flow rate was set at 43 Hertz. Then, Valves A, B, C and D was adjusted to ensure only one way flow to the ram pump was produced. Inverter Pump setting at more than 43 Hertz to 50 Hertz were repeated.

2.6 The 24 hours simulation testing methodology

This was conducted to test whether the designed ram pump could operate continuously exceeding 6 hours, 12 hours and 24 hours. This test was executed after the River Simulation Testing had completed.

2.7 Free flow simulation testing methodology

This test was to determine flow rate of Inverter Pump Setting to support the assembly factors of ram pump at the site. From this testing, correlation between Inverter Pump Setting and flow rate was produced as references to be used for application. The result was recorded only after the flow rate was stable. Based on standard operating procedure of current meter in-situ measurement and its tolerant factors, the test period was selected for 300 second (5 minutes) of each testing.

3 RESULTS AND DISCUSSIONS

A total of 17 test were carried out successfully as shown in Table 2. Some of the constraints during the experiments are: (i) modification of pre-commissioning test due to unstable pressure at the receiver tank and (ii) rectification of inverter pump due to damage to the rubber bush coupling. A total of six (6) test scenarios were carried out during the testing period.

_

Table 2. Successfu	I Test Recorded.
Months	No. of Test
May	9
June	3
July	5
Total	17

The Pre-Commissioning Testing and Pre-Dam Simulation Testing were performed to make sure that the
system operate smoothly and no data was recorded for the tests. Data was recorded for the rest of the testing
scenarios for analysis. Table 3 shows the testing period for different testing scenarios.

No.	Scenarios	Testing Period(Day)	Type of Data
i.	Pre-Commissioning Testing and Pre-Dam Simulation Testing	7	Visual Observation
ii.	Dam Simulation Testing	6	Flow Data
iii.	Pre-River Simulation Testing	4	Visual Observation
iv.	River Simulation Testing	6	Flow Data
٧.	24hrs Simulation Testing	1	Flow Data
vi.	Free Flow Simulation Testing	1	Flow Data
	Totals	25	

Table 3. Testing Period According To Testing Scenario.

3.1 Result of pre-commissioning testing

After the ram pump system had been set up, pre-commissioning test was run to determine ram pump steady operation. No flow data was recorded for this testing, but observations were recorded.

3.2 Pre-dam simulation testing

Pre-commissioning testing was conducted successfully as pre-requisite for further test. After the precommissioning testing, pre-dam simulation testing was run to determine ram pump operation stability. Some of the observations and flow were recorded in Table 4.

Inverter Pump Setting	Impact of Water Level At Dam	Ram Pump Operation Status
(Hertz)	Tank	
30.0	Decreased	Not Operating
30.5	Decreased	Not Operating
31.0	Decreased	Not Operating
31.5	Decreased	Not Operating
32.0	Decreased	Not Operating
32.5	Decreased	Not Operating
33.0	Decreased	Not Operating
33.5	Decreased	Not Operating
34.0	Decreased	Not Operating
34.5	Decreased	Not Operating
35.0	Decreased	Not Operating
35.5	Decreased	Not Operating
36.0	Decreased	Not Operating
36.5	Decreased	Not Operating
37.0	Decreased	Not Operating
37.5	Decreased	Not Operating
37.8	Constant	In Operation
38.00	Constant	In Operation
39.00	Constant	In Operation

 Table 4. Pre-Dam Simulation Testing At Different Setting of Inverter Pump.

3.3 Results of dam simulation testing

From the testing, it was found that certain setting of Inverter Pump caused its water level to decrease and subsequently the ram pump performance was interrupted. However, at certain setting from 36 to 37 Hertz, the ram pump operated intermittently at times. These could be an early warning sign of ram pump system failure and it shows that the ram pump requires a certain amount of detention water above the ram pump to give constant water pressure to the ram pump. Results of dam simulation testing were shown in Table 5. This testing scenario proved that ram pump required a small size of retention water structure or mini dam, with a minimum capacity of 2,271 liters for continuous ram pump operation.

Table 5. Summar	y Result of Dam	Simulation	Testing.
-----------------	-----------------	------------	----------

Inverter Pump Flowrate	Setting (Hertz)	Ram Pump Operation Status	Flowrate (m ³ /hr)
33.0		Not Operating	No Data
33.5		Not Operating	No Data
34.0		Not Operating	No Data
34.5		Not Operating	No Data
35.0		Not Operating	No Data
35.5		Not Operating	No Data
36.0		Intermittent Operation	151.16
36.5		Intermittent Operation	151.16
37.0		Intermittent Operation	161.7
37.8		In Operation	151.9
38.0		In Operation	137.8
39.0		In Operation	137.0

3.4 Results of pre-river simulation testing

Generally, from this testing scenario, it is possible for a ram pump to operate without the assistance of retention water. However, it requires some complicated valves controlling procedures. Pressure gauge at ram pump did not give high reading, meaning that no additional pressure at ram pump during high and low peak of ©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print) 3139

flows. However, by the assistance of two valve's setting, Valves C and Valves D, the ram pump were able to operate continuously.



Figure 3. Water Flows and Valves Setting During the Ram Pump Testing.

3.5 Results of river simulation testing

For the Pre-River Simulation Testing results, some flow rate setting on Inverter Pump were found as stable testing methodologies. In this test, Inverter Pump were set to six (6) setting which generated incoming water to ram pump from 34.4 to 40m³/hr (50 Hertz). The summary of testing result is as shown in Table 6.

		Table 6. Summary of F	River Simulation Testing.	
No.	Inverter Pump	Average Incoming	Average	Percentage
	Setting	Water Supply to Ram	Delivery Water Supply	Differences
	Flowrate	Pump Flowrate	from Ram Pump	between Incoming
	(m ³ /hr)	(liter/second)	Flowrate	and Delivery
			(liter/second)	
1	34.4	2.022	0.079	3.89%
2	36.0	2.007	0.144	7.17%
3	37.6	1.933	0.103	5.30%
4	38.4	1.880	0.065	3.47%
5	39.2	1.852	0.072	3.87%
6	40.0	1.784	0.096	5.40%

At the setting from 36m³/hr till 38.4m³/hr of Inverter Pump, the average of incoming water supply to and fro from ram pump deteriorated. From Table 6, at the highest setting, 40 m³/hr and at the lowest setting, 34.4 m³/hr of Inverter Pump, the ram pump delivery also deteriorated. This led to the conclusion that optimum water flow rate to the ram pump would be a requirement. However, due to the river flows fluctuation, the ram pump was recommended to have water retention structure in order to ensure continuous performance.

3.6 Results of 24 hours simulation testing

The ram pump could be operated for more than 6 hours, 12 hours and 24 hours continuously. It was also found that no breakdown occurred after 24 hours. No heat development that could led to component wear and tear. Ram pump successfully supplied $11.647 (\sim 12m^3)$ per day or 7.03% of incoming water. Conversion of the supplied water was: $12m^3/day$ or $0.5m^3/hours$ or $0.0083m^3/minutes$ or $0.0001388m^3/second$ or 0.1388liter/second ($\sim 0.14liter/second$). As recorded by the pressure vessel of ram pump that the pressure decreased by 20% at the end of the testing from 10psi to 8psi and the ram pump still could be operated. In this test, $165 m^3$ (1.91 liter/second) were supplied to ram pump delivering 0.14 liter/second. From these findings, it was stated that the flow rate ratio at site (laboratory or river) and ram pump were 1.91:0.14 liter/second resulted in the ram pump deliverables at 7% water from the site. This means that, if the ram pump

located near to the river of 1.91 liter/s, the ram pump requires 4.5 hours to fill in 2,270 liter seized of water tank. From these findings, the numbers of selected sized ram pump could be determined by calculation. Table 7 shows the calculation for determining numbers of ram pump for certain residential usage.

 Table 7. Ram Pump Calculation Determining Numbers of Ram Pump for Certain Residential Usage.

 Conditions/ Formula/ Input

Conultions/ Formula/ input	Output
Resident Population/ Water Demands or Water Supplied	1000m ³ for 100 people
Water should be received by ram pump according to ratio formulation (1:10).	10,000m ³
Adjustment of Water Supply In Time of 12 hours or 720	The River/ Dam Flow Rate Requirement:
minutes or 43,200seconds (Proposed 7.00PM to 7.00AM)	= 1000 m ³ / 12 hours or 720 minutes or 43,200seconds = 83 m ³ /hour or 1.39 m3/minutes or
No. of ram pump required based on early finding.	Option 1: Three (3) unit of ram pump at 1.5 Bar or 30 psi
[Early finding: 5 hours at 7.5liter/second supplying 145m ³ for a unit of ram pump at 1.5 Bar or 30psi]	Option 2: One (1) unit ram pump with achievement at 3.0 Bar or 60 psi

3.6 Results of free flow simulation testing

In this testing, the testing range of the Inverter Pump were from 36 Hertz to maximum 50 Hertz. From the testing, due to the distance from Inverter Pump to flow meter (before Ram Pump), the amount of the losses is about 27 - 34%. However, there were only small differences between flow meter reading and Inverter Pump manufacturer calculation. As such, this free flow testing result was used as references for acceptable tolerance between Inverter Pump Setting and real flow meter measurement. The summaries of free flow testing result are shown in Table 8.

Setting of Inverter Pump	Volume (liter)	Test Period (second)	Flowrate Recorded (liter/second)	Flowrate Calculated	Percentage Differences
(Hertz)				(liter/second)	
36	1703	300	5.7	8.0	29%
37	1756	300	5.9	8.2	28%
38	1819	300	6.1	8.4	27%
39	1843	300	6.1	8.7	30%
40	1858	300	6.2	8.9	30%
41	1900	300	6.3	9.1	31%
42	2027	300	6.8	9.3	27%
43	1966	300	6.6	9.5	31%
44	1956	300	6.5	9.8	34%
45	2009	300	6.7	10.0	33%
46	2032	300	6.8	10.2	33%
47	2099	300	7.0	10.4	33%
48	2115	300	7.1	10.7	34%
49	2199	300	7.3	10.9	33%
50	2201	300	7.3	11.1	34%

Table 8. Summary of Free Flow Simulation Testing.



Figure 4. The Finding of Inverter Pump Setting and Flowrate Measurement.

4 CONCLUSIONS

This project has successfully evaluated and determined the ram pump performance. At the laboratory scale, it was confirmed that ram pump had its potential to provide an alternative means of water supply delivery system though not as efficient as conventional supply. For the study, we have found the Ram Pump Delivery Ratio (RPDR).

This study has met performance tested of ram pump. Results showed that ram pump delivered 10% of the incoming water injected to the ram pump to the end user. Water was supplied by a ram pump in the ratio of 1:10. For example, if 10,000m³ go through the ram pump, only 1,000m³ of water will be the supply volume. The objective of this study is also to identify optimum sustained water flow from ram pump through simplified calculation in Ram Pump Design Calculator (RPDC). The RPDC was produced based on 1:10 ratio to determine the ram pump size, quantity and requirement according to water supply demand. Table 9 shows the RPDC.

From the test carried out, it was found as follows: (a) the ram pump could perform consistently with the mini dam concept; (b) factors affecting the ram pump performance are the consistency of water sources to start-up the ram pump. Although the ram pump does not use electricity, some additional accessories in controlling ram pump performance are recommended to be installed, to assure its continued operation. It is recommended to be integrated with surveillance unit, semi or fully automated units for controlling purposes. This should be considered due to the engineering knowledge constraint of rural communities. Standard Operating Procedure documents should be prepared and delivered to the community as basic information to ram pump start up, monitoring and troubleshoots.

ACKNOWLEDGEMENTS

The authors would like to thank the top management of National Hydraulic Research Institute of Malaysia (NAHRIM) for the research project's financing. Special thanks to NAHRIM's Research Assistances, Jegajeevan Naidu Balasundram, Zairi Jalaludin, Hafizul Husni Mohamed Sidin, Suriani Othman, Safarudin Salehuddin, Fardzir Johari involvement in this project.

REFERENCES

- Atharva P. Santosh, K., Sagar M. & Mamta, P. (2016). Design of Hydraulic Ram Pump. *International Journal for Innovative Research in Science & Technology*, 2(10), 290-293.
- De Carvalho, M. O. M., Diniz, A. C. G. C. & Neves, F. J. R. (2011). Numerical Model for a Hydraulic Ram Pump. *International Review of Mechanical Engineering*, 5(4), 733.
- Filipan, V., Virag, Z. & Bergant, A. (2003). Mathematical Modeling of a Hydraulic Ram Pump System, *Journal of Mechanical Engineering*, 49(3), 137-149.
- Herlambang, A. & Wahjono, H. D. (2006). Hydram Pump Design for Communities in Rural Indonesian, *Journal of Aquaculture*, 2(2): 178-186.
- Inthachot, M., Saehaeng, S., Max, J. J., Müller, J. & Spreer, W. (2015). Hydraulic Ram Pumps for Irrigation in Northern Thailand, *Agriculture and Agricultural Science Procedia*, *5*, 107-114.
- Maratos, D. F. (2003). Technical Feasibility of Wavepower for Seawater Desalination Using the Hydro-Ram (Hydram). *Desalination*, 153(1-3), 287-293.
- Matthias, I., Suchard, S., Johannes, F.J., Max, J. & Müller, W.S. (2015). Hydraulic Ram Pumps for Irrigation in Northern Thailand, 1st International Conference on Asian Highland Natural Resources Management, Asia HiLand, Agriculture and Agricultural Science Procedia, 5 (2015),107–114.
- Suarda, M. & Wirawan, I. K. G. (2008). Experimental Study of the Influence of Air Pressure on Hydram Head Pump. Academic *Journal of Mechanical Engineering CAKRAM*, 2(1), 10-14.
- Sakenian Dehkordi, N. & Arshad, S. H. (2012). Design, Construction and Evaluation of A Hydraulic Ram Pump Made of Polyethylene Materials. *Iranian Water Research Journal*, 6(10), 1-10,
- Shuaibu, N. M. (2007). Design and Construction of A Hydraulic Ram Pump, Department of Mechanical Engineering, Federal University of Technology, Minna, Nigeria, Leonardo Electronic. *Journal of Practices* and Technologies, 11, 59-70.
- Tessema A. A. (2000). Hydraulic Ram Pump System Design and Application, *Proceedings Of ESME 5th* Annual Conference on Manufacturing and Process Industry,.
- Twort, A. C., Ratnayaka, D. D., & Brandt, M. J. (2000). *Hydrology and Surface Supplies*. Water Supply (Fifth Edition), Butterworth-Heinemann, Elsevier.
- Wall M.L. & Warren, G.D. (1941). An Analytical and Experimental Study of the Hydraulic Ram. University of Illinois Urbana, University of Illinois Bulletin, Vol. XXXVIII, No.22, Engineering Experiment Station Buletin Series No. 236.

INTERNAL FLOW ANALYSIS AND DESIGN OPTIMIZATION FOR A LOW SPECIFIC SPEED AXIAL PUMP

DANDAN YANG⁽¹⁾, ZHE WANG⁽²⁾, XIANWU LUO⁽³⁾, JIAJIAN ZHOU⁽⁴⁾ & HONGYUAN XU⁽⁵⁾

^(1, 2, 3) Beijing Key Laboratory of CO₂ Utilization and Reduction Technology, Tsinghua University, Beijing 100084, China

yangdd15@mails.tsinghua.edu.cn; zhe-wang16@mails.tsinghua.edu.cn; luoxw@mail.tsinghua.edu.cn ⁽⁵⁾ State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing 100084, China xhy@mail.tsinghua.edu.cn

⁽⁴⁾ Marine Design and Research Institute of China, Shanghai 200011, China 13916608004@139.com

ABSTRACT

The present paper aims to simulate the three-dimensional turbulent flows in an axial pump. The pump has the flow discharge of 0.562 m³/s, and the head rise of 11.24 m at the rotational speed of 1400 r/min. The numerical simulation was conducted based on the Reynolds average Navier-Stokes equations and Shear Stress Transport (SST) model to depict the flow evolutions in the pump. The predicted hydraulic performance on the pump agreed fairly well with experiment data. It is noted that the shaft power is not sensitive to the axial spacing for all operation conditions, and the hydraulic efficiency may improve a little due to the better flow pattern at stator zone for the smaller axial spacing between the rotor and stator. The head rise and shaft power increased with decreasing the tip gap and the efficiency reached the maximum value at the tip gap 0.3 mm. The maximum vorticity occurred near the pressure side of the rotor blade decreased with the increase of the tip gap. Near the hub, the tip gap effect on the pressure coefficient distribution is not remarkable with increasing the tip gap from 0.5 mm to 0.7 mm. Further, the effect of numerical method on pump internal flow was also investigated by two turbulence models, i.e. SST model and modified Partially-Averaged Navier-Stokes (MPANS) model. Though both turbulence models over predicted the velocity wake deficit near the hub, MPANS model achieved a closer axial velocity distribution to the experimental result and can capture the vortex structure with better accuracy compared with SST model. This study would be helpful for the design optimization for the axial pump with low specific speed ranging from 630-1260 m³min⁻¹·min⁻¹·m in the future.

Keywords: Axial pump; internal flow; modified partially-averaged navier-stokes (MPANS) model; design optimization; hydraulic performance.

1 INTRODUCTION

Axial flow pump is a low-head pump which is widely used in drainage and irrigation, power station, water diversion projects, etc. Recently, it has also been applied in the shipping industry as water-jet propulsion. The water-jet axial pump gains several advantages over traditional screw propellers such as high-speed capacity, good maneuverability and improved cavitation performance. With the development of the computational fluid dynamics (CFD), many researchers devoted on the numerical simulations of the axial pump instead of model test, which is time-costing and expensive. Park et al. (2005) analyzed the details of the flow phenomena in the waterjet propulsion system by using sliding multiblock technique to handle the rotor-stator interaction. Kinnas et al. (2007) used the boundary element method to predict the internal flows in the water-jet propulsor and considered the circumferentially averaged effect to evaluate the interaction between rotor and stator. Gao et al. (2008) employed the TASCflow to investigate the performance and flow fields in a water-jet pump and discussed the influence of a rear stator on the overall performance. Lee et al. (2008) optimized the total efficiency of a low-speed axial flow fan blade by modified the blade profile. Zhang et al. (2010) investigated the axial flow pump using unsteady Reynolds-average Navier-Stokes (RANS) equations. They found the periodic pressure fluctuation on the rotor and stator, and the dominant frequency is equal to the frequency of blade passing. Wang et al. (2013) studied the effects of gap between the rotor and stator by using $k-\varepsilon$ model. Their results showed that the performance was improved with increasing the gap in low flow rate conditions and had no obvious effect under design condition and large flow rate conditions. Shi et al. (2015) used numerical techniques by software iSIGHT to optimize the axial flow blade, which is both effective and timesaving. Turbulence model demands careful consideration in every simulation which has significant impact on the predicted accuracy of the flow field. Direct Numerical Simulation (DNS) model is the most accurate and expensive method because it resolved the whole scale of turbulence. The Large eddy Simulation (LES) model attempted to simulate large-scale turbulence eddies and the small scale turbulence eddies on large ones were considered by an approximate model. It still needs adequate computer memory. The Reynolds-Average

Navier-Stokes (RANS) model such as k- ε model costs less computational resource, but tends to over-predict the turbulence viscosity. Thus, some hybrid models were proposed to combine the benefit of these models. Spalart (2006) proposed a detached-eddy simulations (DES) model which was made a combination between RANS and LES. Girimaji (2006) proposed a Partially-Averaged Navier-Stokes method (PANS) which varied from RANS to DNS with two filter-control parameters, i.e. the unresolved-to-total ratios of kinetic energy f_k and dissipation f_{ε} . Huang et al. (2016) adopted a modified PANS (MPANS) model to investigate the flow field around a backward facing step. They validated that MPANS model provided an improved result over k- ε model, especially in turbo-machines which encounter with flow separation and recirculation.

In this paper, three-dimensional steady turbulent flows in a water-jet axial pump were simulated with commercial CFD code ANSYS CFX. The effects of geometrical parameters on pump performance were investigated by applying the different rotor tip gaps and different spacing between the impeller and the guide vane. In addition, Shear Stress Transport (SST) model and MPANS model were used to investigate the effects of turbulence model on the internal flows.

2 GOVERNING EQUATIONS

The continuity and momentum conservation equations for steady state are given by Eqs. [1]- [2].

$$\frac{\partial \left(\rho u_{j}\right)}{\partial x_{j}} = 0$$
[1]

$$\frac{\partial \left(\rho u_{i} u_{j}\right)}{\partial x_{j}} = -\frac{\partial p}{\partial x_{j}} + \frac{\partial}{\partial x_{j}} \left[\left(\mu + \mu_{t}\right) \left(\frac{\partial u_{i}}{\partial x_{j}} + \frac{\partial u_{j}}{\partial x_{i}}\right) \right] + f_{i}$$
[2]

where u_i is the velocity in the *i* direction, p is the pressure, p is the density, μ is the laminar viscosity, μ_t is the turbulence viscosity given by the turbulence model and f_i is a component of the body force.

The formulation of the SST turbulence model is given by Eq. [3]-[4].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_{j}k)}{\partial x_{j}} = \frac{\partial}{\partial x_{j}} \left[(\mu + \sigma_{k}\mu_{t})\frac{\partial k}{\partial x_{j}} \right] + P_{k} - \beta^{*}\rho k\omega$$
[3]

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho u_{j}\omega)}{\partial x_{j}} = \frac{\partial}{\partial x_{j}} \left[\left(\mu + \sigma_{\omega}\mu_{t}\right) \frac{\partial\omega}{\partial x_{j}} \right] + \alpha \frac{1}{\nu_{t}} P_{k} - \beta \rho \omega^{2} + 2(1 - F_{1})\rho \sigma_{w2} \frac{1}{\omega} \frac{\partial k}{\partial x_{j}} \frac{\partial\omega}{\partial x_{j}} \right]$$

$$\tag{4}$$

where *k* is the turbulence kinetic energy and ω is the turbulence frequency. Note that F_1 is blending function which is equal to one in the boundary layer and turns to zero in the wake region. The constant values and detailed explanation of the equations can be found by Menter et al. (2009).

3 NUMERICAL IMPLEMENTATION

3.1 Geometry and computational domain

The present work made use of the axial water-jet pump in the literature (Michael et al., 2008). As illustrated in Figure 1, the pump was comprised of the hub, casing, a six-blade rotor and an eight-blade rear stator. The inlet diameter *D* was 304.8 mm and the rotor blade tip clearance was 0.5mm for practical model test. The design point corresponds to a flow discharge of 0.562 m³/s, and a head rise of 11.24 m at the rotational speed *n* of 1400 r/min. The stator reference line position was moved along the axial line for different spacing between rotor and stator. In order to obtain stable flow simulation, the inlet and outlet plane of the computation domain were extended to avoid the occurrence of backflows. The distance between the inlet plane and the rotor inlet is 2*D*, and that between the outlet plane and the stator outlet is 5*D*.



Figure 1. Geometry of the axial pump.

3.2 Mesh generation

ANSYS ICEM was used to generate the structured grids in the whole computation domain. The local refinement was applied in the regions close to the leading and trailing edges of the vanes. The mesh near the solid walls was also refined to meet the requirement of wall function. The final computational grid number was around 2.8 million. The generated mesh is illustrated in Figure 2. In Figure 2(a), the inlet plane of the computation domain is at the most left side, and the outlet plane is at the most right side. For better understanding, the mesh surface of the rotor and stator is shown in Figure 2(b).



Figure 2. Generated mesh.

3.3 Boundary conditions

The total pressure at the inlet plane was assigned according to the experiment setup (Chesnakas et al., 2009). As for the boundary condition at the outlet plane, the mass flow rate was set. A rotating frame was applied for the rotor zone, and other regions were in a stationary frame. The GGI interfaces were used between the rotor zone and the stationary parts. Non-slip wall condition was set for all solid surfaces.

4 RESULTS AND DISCUSSIONS

In the report of Michael et al. (2008), the stator reference line is located at x/R=1.53 relative to the upstream end of the rotor hub. For the simulations, the stator reference line position was chosen at 1.47R, 1.53R and 1.59R to investigate the effects of axial spacing on pump performance. By moving the reference line downstream, the spacing between the rotor and stator increased. The effect of rotor blade tip gap on the pump was also studied. For discussion about the effect of axial spacing, the tip gap was set to 0.5mm as the design value. Similarly, the reference line position was fixed at 1.53R when studying the effect of tip gap.

4.1 The effects of axial spacing

The numerical results were compared to the experiment data measured by Chesnakas et al. (2009) on pump performances such as head rise, shaft power and efficiency. For convenience, the non-dimensional parameters, i.e. flow rate coefficient, head rise coefficient, power coefficient and pump efficiency, are defined as Eqs. [5]-[8].

The flow rate coefficient is defined as

$$Q^* = \frac{Q}{nD^3}$$
^[5]

The head rise coefficient is defined as

 $H^* = \frac{gH}{n^2 D^2}$ [6]

The power coefficient is defined as

$$P^* = \frac{P}{\rho n^3 D^5}$$
^[7]

Thus, the pump efficiency can be expressed as

$$\eta = \frac{\rho g Q H}{P} = \frac{Q^* H^*}{P^*}$$
[8]

where Q, H and P are the volumetric flow rate, head rise and shaft power, respectively.

The hydraulic performance curves are shown in Figure 3, where the numerical results were compared with the experimental data under different axial spacing between the pump rotor and stator. As a whole, the predicted results agreed fairly well with the experimental data at the 1.53R reference line position. The maximum discrepancies between the experiment and calculation data for head rise, shaft power and efficiency were 2.3%, 1.2% and 2.5%, respectively. The experimental efficiency was 87.9% at the design point, i.e. flow rate coefficient of 0.85, and the corresponding predicted efficiency was 89.4%. There were little larger discrepancies in the head rise, shaft power and efficiency between the experimental and predicted results outside the designed operation condition. Unfortunately, the head rise coefficient at part flow condition was underestimated due to the limited capability of the present numerical method.

It is also noted that the shaft power was not sensitive to the axial spacing for all operation conditions. Though the pump head changed slightly with the axial spacing between the rotor and stator, there were no remarkable effects on the hydraulic performance for the pump. The head rise and efficiency increased at the 1.47R and 1.59R position in the large flow rate condition. The maximum efficiency was about 90% at the 1.47R position. It indicates that the hydraulic efficiency may improve a little due to the reduction of the distance between the rotor and stator.



Figure 3. Experimental and numerical hydraulic performance curves.

Figure 4 shows the streamline and pressure distribution near the stator casing with different axial spacing. It shows that the flow passed over the stator vane smoothly, though there were a pair of vortexes at the leading edge near the pressure side at the axial spacing of 1.47R. On the contrary, there was a minus incidence for the case of 1.59R, where the flow impinged the stator vane pressure side near the leading edge. There was a relatively homogeneous pressure distribution for 1.47R, compared with other cases. This kind of better flow pattern may benefit the hydraulic efficiency for the stator zone at smaller axial spacing.



Figure 4. Streamline and static pressure distributions around the stator vane at the pump casing, Q*=0.85: (a) 1.47R, (b) 1.53R, (c) 1.59R.



Figure 5. Pressure distribution of different stator blade sections, Q*=0.8.

Figure 5 shows the comparisons of the pressure coefficient around three stator blade sections by different reference line position at the design flow coefficient $Q^*=0.85$. The pressure coefficient is defined as $C_p=(p-p_{ref})/(0.5\rho v_{in}^2)$, where *p* is the static pressure, p_{ref} is the reference pressure ($p_{ref}=101325$ Pa) and v_{in} is the average velocity of the inlet plane. As shown in Figure 5, the pressure magnitude at the leading edge of the blade was higher than other regions. The pressure on the pressure side increased near leading edge and decreased near trailing edge from 10% to 90% span. By increasing the axial spacing, the pressure coefficient 3148 ©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

decreased near leading edge and increased near trailing edge at 10% span position. The pressure coefficient increased with the decrease of the axial spacing at 50% span and 90% span, especially near trailing edge of the stator. In addition, the effect of the axial spacing on the pressure coefficient is more sensitive on the suction surface.

4.2 The effects of blade tip gap

Comparisons of hydraulic performance by three blade tip gaps are listed in Table 1. The result shows that the head rise and shaft power increased with decreasing the tip gap. The head rise improved 2.3% by decreasing the tip gap from 0.5 mm to 0.3 mm, and the shaft power decreased 1.5% by increasing the tip gap form 0.5 mm to 0.7 mm. The efficiency reached the maximum value of 0.911 at the tip gap of 0.3 mm compared to other cases.

Table II companeerie e	r nyaraane perien		lade lip gape.
Case	H*	P*	η
gap 0.3 mm	2.24	2.089	0.911
gap 0.5 mm	2.19	2.079	0.894
gap 0.7 mm	2.17	2.048	0.901

|--|

Figure 6 shows the contour plots of vorticity around the rotor gap flow path with different tip gaps at the design flow rate coefficient. The vorticity near the blade tip plane was near zero due to the presence of the rotor blade, and the maximum vorticity was near the pressure side of the rotor vane. The vorticity magnitude near the pressure side of the rotor at the tip gap 0.3 mm was higher than those of the other two cases. Similarly, the vorticity due to the rotor movement became weaker at the largest tip gap, 0.7 mm.



Figure 6. Vorticity around the rotor gap flow path, Q*=0.85: (a) gap 0.3 mm, (b) gap 0.5 mm, (c) gap 0.7 mm.

Figure 7 shows comparisons of the pressure coefficient around three stator sections by different tip gaps at the design flow coefficient $Q^*=0.85$.



Figure 7. Pressure distribution of different stator blade sections, Q*=0.85.

The load on blade increased near the leading edge and decreased near the trailing edge along the radial direction from hub to tip. At the same span, the load changed with the tip gap. It is obvious that the load for the gap of 0.5 mm was larger than that of 0.7 mm at 50% span, and the load for the gap of 0.3 mm was larger than that of 0.7 mm at 90% span. At 10% span, the effect on the pressure coefficient distribution was not remarkable with increasing the tip gap from 0.5 mm to 0.7 mm.

5. FURTHER CONSIDERATIONS

To depict the internal flow clearly, we applied a newly developed dynamics turbulence model, i.e. MPANS model, for the present numerical simulation.

The challenge of the PANS model is to determine the two filter-control parameters, i.e. the unresolved-tototal ratios of kinetic energy f_k and dissipation f_{ϵ} , which are defined as

$$f_{\rm k} = \frac{k_{\rm u}}{k}, f_{\varepsilon} = \frac{\varepsilon_{\rm u}}{\varepsilon}$$
[9]

The turbulence governing equation in the PANS model also treats the standard k- ϵ turbulence model as parent RANS model as listed in Eqs. [10]-[11].

$$\frac{\partial(\rho k_u)}{\partial t} + \frac{\partial(\rho u_j k_u)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_{ku}} \right) \frac{\partial k_u}{\partial x_j} \right] + G_{ku} - \rho \varepsilon_u$$
[10]

$$\frac{\partial(\rho\varepsilon_{u})}{\partial t} + \frac{\partial(\rho u_{j}\varepsilon_{u})}{\partial x_{j}} = \frac{\partial}{\partial x_{j}} \left[\left(\mu + \frac{\mu_{t}}{\sigma_{\varepsilon u}} \right) \frac{\partial\varepsilon_{u}}{\partial x_{j}} \right] + \frac{C_{1\varepsilon}\varepsilon_{u}}{k_{u}} G_{ku} - C_{2\varepsilon}^{*} \rho \frac{\varepsilon_{u}^{2}}{k_{u}}$$
[11]

where G_{ku} is the unresolved production term, the unresolved kinetic energy σ_{ku} , the dissipation Prandtl numbers $\sigma_{\varepsilon u}$ and the value of $C_{\varepsilon 2}^*$ are defined by

$$\sigma_{\rm ku} = \sigma_{\rm k} \frac{f_{\rm k}^2}{f_{\varepsilon}}, \sigma_{\varepsilon u} = \sigma_{\varepsilon} \frac{f_{\rm k}^2}{f_{\varepsilon}}, C_{z_{\varepsilon}}^* = C_{1\varepsilon} + \frac{f_{\rm k}}{f_{\varepsilon}} (C_{2\varepsilon} - C_{1\varepsilon})$$
[12]

The PANS turbulence viscosity is described as

$$\mu_{t} = \rho C_{\mu} \frac{k_{u}^{2}}{\varepsilon_{u}}$$
[13]

For the MPANS model, the unresolved-to-total ratio of kinetic energy f_k is a variable which is based on the physical grid ($\Delta = (\Delta x^* \Delta y^* \Delta z)^{1/3}$) and local turbulence length scale ($I = k^{1.5/\epsilon}$) (Girimaji et al., 2005).

$$f_{\rm k} = \min(1, 3(\Delta/l)^{2/3})$$
 [14]

Figure 8 shows the comparison of experimental and computational circumferential average axial velocity under different turbulence models at the nozzle exit plane. The predicted axial velocity matched well with the measured data away from the hub wake by MPANS models. However, the axial velocity magnitude was over predicted at the stator wake by SST model. Though both of the numerical cases over predicted the velocity wake deficit near the hub, the axial velocity calculated by MPANS model was closer to the experiment result.



Figure 8. Circumferential average axial velocity at the nozzle exit plane.



Figure 9.Streamline around the rear stator at the casing position, 1.53R.

Comparisons of streamline and pressure distributions around the rear stator casing with three operating conditions by SST turbulence model and MPANS turbulence model are shown in Figure 9. The left column is the numerical result predicted by SST model and the right is the numerical result predicted by MPANS model. The stator reference position was at 1.53R and the rotor blade tip gap was 0.5 mm. For SST model, the result shows that there was a large vortex on the leading edge of the pressure side and a small vortex on the trailing edge of the suction side at the lower flow rate coefficient. With the increasing of the flow rate, the vortex was smaller and tended to disappear. For MPANS model, it shows that there were two vortexes on the leading edge of the pressure side and a small vortex on the trailing edge of the suction side at the lower flow rate coefficient, and the vortex structure was clearer compared with the vortex captured by SST model. At the design flow rate condition, there was a distinct vortex on the leading edge of the pressure side, which was not captured by SST turbulence model. It can be inferred that the MPANS model captures the vortex structure more accurate compared with the SST model.

6. CONCLUSIONS

In this paper, the CFD method was employed to estimate the internal flow in an axial water-jet pump. Conclusions can be drawn as follows:

(1) It was confirmed that the present numerical method can successfully predict the hydraulic performance for the axial pump compared with the experimental data.

(2) It is also noted that the shaft power is not sensitive to the axial spacing for all operation conditions, and the hydraulic efficiency may improve a little due to the better flow pattern at stator zone for the smaller axial spacing between the rotor and stator. In addition, the effect of the axial spacing on the pressure coefficient is more sensitive on the suction surface.

(3) The head rise and shaft power increased with decreasing the tip gap and the efficiency reached the maximum value at the tip gap 0.3 mm. The maximum vorticity occurred near the pressure side of the rotor blade decreased with the increase of the tip gap. Near the hub, the tip gap effect on the pressure coefficient distribution was not remarkable with increasing the tip gap from 0.5 mm to 0.7 mm. However, the load for the gap of 0.5 mm was larger than that of 0.7 mm at 50% span, and the load for the gap of 0.3 mm was larger than that of 0.7 mm at 90% span.

(4) Though both turbulence models over predicted the velocity wake deficit near the hub, MPANS model achieves a closer axial velocity distribution to the experimental result and can capture the vortex structure with better accuracy compared with SST model.

ACKNOWLEDGEMENTS

This work was financially supported by the National Natural Science Foundation of China (Project No. 51376100), Science and Technology on Water Jet Propulsion Laboratory and State Key Laboratory for Hydroscience and Engineering (Project No. sklhse-2017-E-02).

REFERENCES

- Chesnakas, C.J., Donnelly, M.J., Pfitsch, D.W., Becnel, A.J. & Schroeder, S.D. (2009). *Performance Evaluation of the Onr Axial Water jet 2 (AxWJ-2)*, Naval Surface Warfare Center Carderock Div Bethesda Md Total Ship Systems Directorate.
- Gao, H., Lin, W. & Du, Z. (2008). Numerical Flow and Performance Analysis of a Water-Jet Axial Flow Pump. *Ocean Eng.*, 35(16), 1604-1614.
- Girimaji, S. & Abdolhamid, K. (2005). Partially-Averaged Navier Stokes Model for Turbulence: Implementation and Validation. 43rd AIAA Aerospace Sciences Meeting and Exhibit.
- Girimaji, S.S. (2006). Partially-Averaged Navier-Stokes Model for Turbulence: A Reynolds-Averaged Navier-Stokes to Direct Numerical Simulation Bridging Method. *Journal of Applied Mechanics*, 73(3), 413.
- Huang, R.F., Luo, X.W., Ji, B. & Ji, Q.F. (2016). Turbulent Flows Over A Backward Facing Step Simulated Using a Modified Partially-Averaged Navier-Stokes Model. *Journal of Fluids Engineering*, 139(4), 1-7.
- Kinnas, S.A., Lee, H., Michael, T.J. & Sun, H. (2007). Prediction of Cavitating Waterjet Propulsor Performance Using a Boundary Element Method. 9th International Conference on Numerical Ship Hydrodynamics Ann Arbor, Michigan, USA.

Lee, K.S., Kim, K.Y. & Samad, A. (2008). Design Optimization of Low-Speed Axial Flow Fan Blade with Three-Dimensional Rans Analysis. *Journal of Mechanical Science and Technology*, 22(10), 1864-1869.

- Menter, F.R. (2009). Review of the Shear-Stress Transport Turbulence Model Experience from an Industrial Perspective. *International Journal of Computational Fluid Dynamics*, 23(4): 305-316.
- Michael, T.J., Schroeder, S.D. & Becnel, A.J. (2008). *Design of the Onr Axwj-2 Axial Flow Water Jet Pum*p, Naval Surface Warfare Center Carderock Div Bethesda Md.
- Park, W.G., Jin, H.J., Chun, H.H. & Kim, M.C. (2005). Numerical Flow and Performance Analysis of Water jet Propulsion System. *Journal of the Society of Naval Architects of Korea*, 32(14), 1740-1761.
- Shi, L., Tang, F., Xie, R., Qi, L. & Yang, Z. (2015). Design Optimization of Axial-Flow Pump Blades Based on Isight. ASME/JSME/KSME 2015 Joint Fluids Engineering Conference, American Society of Mechanical Engineers.
- Spalart, P. (2006). Trends in Turbulence Treatments. Fluids 2000 Conference and Exhibit.
- Wang, W.J., Liang, Q.H., Wang, Y., Yang, Y., Yin, G. & Shi, X.X. (2013). Performance Analysis of Axial Flow Pump on Gap Changing Between Impeller and Guide Vane. *IOP Conference Series Materials Science and Engineering*, 52, 66-71.
- Zhang, D.S., Shi, W.D., Chen, B. & Guan, X.F. (2010). Unsteady Flow Analysis and Experimental Investigation of Axial-Flow Pump. *Journal of Hydrodynamics*, 22(1), 35-43.

A PRELIMINARY STUDY FOR A FLUID STRUCTURE INTERACTION MODEL BY SMOOTHED PARTICLE HYDRODYNAMICS AND CONTACT MECHANICS

ABDULLAH DEMIR⁽¹⁾, ALI ERSIN DINÇER⁽²⁾ & CÜNEYT YAVUZ⁽³⁾

⁽¹⁻²⁻³⁾Middle East Technical University, Civil Engineering Department, Ankara, Turkey ⁾abdemir@metu.edu.tr;aliersin@metu.edu.tr;cyavuz@metu.edu.tr

ABSTRACT

Various fluid structure interaction (FSI) models have been proposed in literature. FSI systems have been solved by simulating the fluid and structure either with solely mesh free or mesh-based methods. However, there are few researches that couple mesh free and mesh-based methods to interact fluid and structure. In the present study, a fluid structure interaction model is proposed in which fluid is simulated with Smoothed Particle Hydrodynamics (SPH) as a mesh free method and structure is simulated with a well-known mesh-based method (FEM). A well-known contact mechanic model is modified to simulate the interaction. In the model, water particles invading the solid domain are taken out of the solid structure by using the modified contact mechanics algorithm. The validations show that the proposed FSI model is promising. It should be noted that since the displacement of the structure should be very small in the experiments, structural displacements are not included in the present study.

Keywords: fluid-structure-interaction, FSI, smoothed particle hydrodynamics, SPH, contact mechanics

1 INTRODUCTION

Fluid-Structure Interaction (FSI) is an important subject in engineering and many more areas. Therefore, researchers have studied on this subject lately. Although simulating FSI problems is complex and requires much computational effort, some progress has been made in FSI technology (Bathe et al., 1999; Dowell et al., 2001; Ohayon and Felippa, 2001; Tezduyar and Bazilevs, 2008). In the remainder of the introduction, a very brief review of Smoothed Particle Hydrodynamics (SPH) and contact mechanics which create the current FSI model will be given. After that, the studies about the proposed FSI method will be pointed out.

Smoothed Particle Hydrodynamics (SPH), used to obtain mathematical solutions of equations by replacing the material with a set of particles, was invented in 1977 by Gingold and Monaghan (1977) and Lucy (1977) to simulate astrophysical problems. Then, Monaghan (1994) applied SPH to fluid dynamics problems. The equations in SPH are in Lagrangian form in which conservation of momentum and energy is guaranteed. In SPH, in order to define interactions between particles, which reproduce the equations of fluid dynamics, Gingold and Monaghan (1977) used a kernel estimation technique. Over the past decades, SPH was applied to multi-phase flows (Monaghan and Kocharyan, 1995), gravity currents (Monaghan, 1995), heat conduction (Chen et al., 1999), solid mechanics problems (Swegle et al., 1995; Benz and Asphaug, 1995) and others.

Contact mechanics is a method to prevent the penetration between each defined domain. There are many formulations of it (Bathe 1985, Deuflhard 2008, Chawla 1998, Khenous 2006). These formulations were made for solution of different solid bodies. This is because the behavior of solid bodies (Feng 2006) are important for solution as changing elastic, geometric and surface properties. Recent researches were made for dynamic contact problems (Deuflhard 2008, Chawla 1998). A solution method named discrete-finite element method (Munjiza 2004) is used to solve those types of problems. This method is a solution for solid structures being in motion in space and in contact with other defined solid structures. Proposed method is different from those researches composing meshless and mesh-based methods with contact mechanics; SPH is used as a meshless method to model the fluid and FE method is used as a mesh-based method to model the solid.

There are limited studies with coupled SPH-FE method. Firstly, study was conducted to deal with structure-structure impact model and proposed by Attaway (1994). Then, Vuyst et al. (2005) applied SPH-FE method to fluid-structure impact problems. SPH code was coupled with commercially available FE software, DYNA3D. In their method, FE nodes are regarded as SPH particles and a particle to particle contact approach is performed. Violent FSI interactions in the presence of free-surface and elastic structures was studied by Fourey et al (2010) and Groenenboom and Cartwright (2010). Lately, Hu et al. (2014) developed a searching algorithm method to improve the computational efficiency of SPH-FE method.

In most of the studies in the literature, the coupling was satisfied by calculating forces from one domain and applying them to the other domain. In the present study, as a main difference, contact mechanics satisfy the self-interaction between solid and fluid domains by solving water particles invading the solid domain and whole structure together. Since the structure is taken as fixed in the experiments, the displacement of the structure is not mentioned in this study.

The paper is organized as follows: Firstly, the governing equations and necessary parameters in SPH are explained. Then, governing equations of contact mechanics are defined. After that, validation of the method is done by comparing the results of the proposed method and the experimental data. Finally, conclusions are drawn.

2 NUMERICAL MODEL

2.1 Smoothed Particle Hydrodynamics Equations SPH interpolation for a function f can be expressed as:

$$f(r) = \int f(r') W(r - r', h) dr'$$
[1]

For density, ρ , velocity vector, \dot{U} , position vector, \vec{U} and pressure, p, the Euler equations in discrete SPH form are given as:

$$\frac{d\rho_i}{dt} = \sum_{j=1}^N m_j \left(\vec{U}_{ij}\right) \cdot \nabla \vec{W}_{ij}$$
[2]

$$\frac{d\dot{U}_i}{dt} = -\sum_{j=1}^N m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \pi_{ij}\right) \cdot \nabla \vec{W}_{ij}$$
[3]

$$\frac{d\vec{U}_i}{dt} = \vec{U}_i + \varepsilon \sum m_j \left(\frac{\vec{U}_{ij}}{\vec{\rho}_i}\right) W_{ij}$$
[4]

$$p_i = c_0^2 (\rho_i - \rho_0)$$
 [5]

where subscript i denotes ith particle and subscript j denotes the neighbors, j, of particle i, $\vec{U}_{ij} = \vec{U}_i - \vec{U}_j$, $\vec{W}_{ij} = \vec{W}(r_{ij}, h)$ is the cubic spline kernel and a detailed information about the Kernels can be found in the studies of Liu and Liu (2003). $\nabla \vec{W}_{ij}$ is the gradient of the kernel, and h is the size scale of the kernel support. The artificial viscosity, π_{ij} , mainly used to stabilize the numerical algorithm, can be calculated from:

$$\pi_{ij} = -\frac{\alpha h c_0}{\overline{\rho}_{ij} (r_{ij} + 0.01h^2)} \min(\vec{U}_{ij} \cdot r_{ij}, 0)$$
[6]

where α is an empirical coefficient and taken as 0.01 as proposed by Monaghan (1994) and c is the wave speed, $r_{ij} = \left| \vec{r_{ij}} \right|$ and $\vec{\rho}_{ij} = \left(\frac{\rho_i + \rho_j}{2} \right)_2$.

2.2 Contact mechanics

Contact mechanics is basically the application of penalty method which applies a displacement constraint to a structure by finding corresponding forces. The following derivations are taken from the research made by Bathe.

Contact mechanics takes place in the existence of a target body and contact body. Target body is the solid structure and contact bodies are water particles of SPH. Thus, a point to line contact is defined. Defining the overlap as penetration of the particle to the defined line $\vec{\Delta}_p^{(i-1)}$; in Eq. 7.

$$\vec{\Delta}_{p}^{(i-1)} = {}^{t+\Delta t}\vec{x}_{p}^{(i-1)} + {}^{t+\Delta t}\vec{x}_{C}^{(i-1)}$$
[7]

where ${}^{t+\Delta t}\vec{x}_p^{(i-1)}$ is the position of particle, ${}^{t+\Delta t}\vec{x}_C^{(i-1)}$ is position of contact point which is the closest point on defined target surface, t is time increment and i is iteration increment. Corresponding potential is in Eq. 8.

$$\Psi_{p} = \left({}^{t+\Delta t}\vec{\lambda}_{p}^{(i-1)} + \Delta\vec{\lambda}_{p}^{(i)}\right)^{T} \left[\Delta\vec{u}_{p}^{(i)} + \vec{\Delta}_{p}^{(i-1)} - \Delta\vec{u}_{C}^{(i)}\right]$$
[8]

where ${}^{t+\Delta t}\vec{\lambda}_p^{(i-1)}$ is force applied to water particle by solid structure due to penetration, $\Delta \vec{\lambda}_p^{(i)}$ is change in force, $\Delta \vec{u}_p^{(i)}$ is change in position of water particle, and $\Delta \vec{u}_C^{(i)}$ is change in position of point C.Governing finite element equations for solid structure is achieved by the addition of potential defined in Eq. 8 to the present potential of solid structure.

$$\begin{bmatrix} t+\Delta t M_p^{(i-1)} & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} \Delta \ddot{U}_p^{(i)} \\ \Delta \lambda^{(i)} \end{bmatrix} + \left\{ \begin{bmatrix} t+\Delta t K_{solid}^{(i-1)} & 0 \\ 0 & 0 \end{bmatrix} + \begin{bmatrix} t+\Delta t K_c^{(i-1)} \\ \Delta \lambda^{(i)} \end{bmatrix} \right\} \begin{bmatrix} \Delta U_{solid}^{(i)} \\ \Delta \lambda^{(i)} \end{bmatrix} = \begin{bmatrix} t+\Delta t R \\ 0 \end{bmatrix} - \begin{bmatrix} t+\Delta t R \\ 0 \end{bmatrix} + \begin{bmatrix} t+\Delta t R_c^{(i-1)} \\ t+\Delta t A_c^{(i-1)} \end{bmatrix}$$
[9]

where ${}^{t+\Delta t}M_p^{(i-1)}$ is the mass matrix of water particles, $\Delta \ddot{U}_p^{(i)}$ is incremental acceleration vector of water particles, ${}^{t+\Delta t}K_{solid}^{(i-1)}$ is tangential stiffness matrix of solid structure, ${}^{t+\Delta t}K_c^{(i-1)}$ is contact stiffness matrix, $\Delta U^{(i)}$ is incremental displacement vector, $\Delta \lambda^{(i)}$ is incremental contact force vector, ${}^{t+\Delta t}R$ is total applied external load vector, ${}^{t+\Delta t}F^{(i-1)}$ is equivalent nodal force vector, ${}^{t+\Delta t}R_c^{(i-1)}$ is contact force vector, and ${}^{t+\Delta t}\Delta_c^{(i-1)}$ is overlap vector.

Applying Newmark method [] taking $\alpha = 0.5$, $\beta = 0.25$ and defining a pseudo time position for the position of water particle invading the defined target surface, the governing finite element equations is:

$$\begin{cases} \begin{bmatrix} t + \Delta t K_{solid}^{(i-1)} & 0 & 0 \\ 0 & \frac{2}{\Delta t^2} \cdot t + \Delta t M_p^{(i-1)} & 0 \\ 0 & 0 & 0 \end{bmatrix} + \begin{bmatrix} t + \Delta t K_c^{(i-1)} \end{bmatrix} \\ \begin{cases} \Delta U_{solid}^{(i)} \\ \Delta U_p^{(i)} \\ \Delta \lambda^{(i)} \end{bmatrix} = \begin{bmatrix} t + \Delta t R \\ 0 \\ 0 \end{bmatrix} - \begin{bmatrix} t + \Delta t R \\ 0 \\ 0 \end{bmatrix} + \begin{bmatrix} t + \Delta t R_c^{(i-1)} \\ t + \Delta t A_c^{(i-1)} \end{bmatrix}$$
[10]

The time interval of motion of fluid particle in contact with solid structure is formulated by using Newmark method as indicated. This time interval can be named as pseudo time interval for SPH solution. Also, this part is assumed timeless for contact mechanics solution. Briefly, the motion of a water particle depends on SPH formulations before it penetrates the solid structure. After the penetration, contact mechanic equations lead the movement of the water particle. In fact, a real penetration will not occur during solution, this action happens at pseudo time interval of SPH method. In timeless solution step of contact mechanics (at pseudo time interval of SPH method), water particle is taken out of the solid structure.

Contact mechanics presents a solution for defined structures being in contact. Therefore, a contact search algorithm is needed to detect the penetrations between structures. A 2D contact search algorithm is needed, which is Node to Line contact. Nodes are contact particles (water particle) and lines are solid structures. Details of contact search algorithm will not be given in this paper, but will be given in further researches.

Solution of contact mechanics depends on the properties of defined solid structure. These are material, geometrical and surface properties. However, defined governing equations do not change with structural conditions. In this article, notation of Bathe (1982) is used for total Lagrangian formulation and governing contact mechanic equations are modified from the studies of Bathe and Chaudhary (1985).

3 VALIDATION

The experiments of Lobovsky et al. (2013) is used to validate the proposed FSI model. They studied on dam break flow over a dry horizontal bed. A side view of the experimental setup can be seen in Figure 1. Left side in the figure shows the initial position of the water. In the experiments, fresh water with the density of 997 kgm⁻³ and the kinematic viscosity of 8.9E-7 Nm⁻² were used. The initial water depth, H, was taken as 300 mm. The right side of the figure shows the wall located at the downstream end of the channel. Five pressure sensors were mounted on the wall to measure the pressure due to the impact of the water wave.



Figure 1. A side view of the experimental setup

In the numerical simulations of FEM part of FSI model, four node plane stress elements were used. Geometry of the container was constructed with 143 number of finite elements as seen in Figure 2. Supports are also shown in this figure. Although nonlinear effects were taken into consideration, resultant displacements were very small and there is no experimental data available, so displacements will not be shown as an output. Polymethly methacrylate (PMMA) was used as the material of the container and young's modulus of this material was taken as 3000 MPa with 0.3 poisson's ratio.





In the numerical simulations of SPH part of FSI model, 1860 water particles, shown with red dots in Figures 3 and 4, were used. The initial spacing between particles for this case was 0.01 m. Kernel length was taken as 1.33 times of the initial particle spacing. The initial configuration of the water particles can be seen in Figure 3. In the figures, green-colored water represents the experimental data of Lobovsky et al. (2013). Time step was calculated from CFL condition and taken as 0.00001 s. The authors are aware that time step is very small and currently they are studying on this subject.



Figure 3. Initial configuration of the water particles (Lobovsky et al., 2013)

Although, in the calculations, the experiments were simulated with FSI approach, the boundaries used in the experiments were fixed. However, the authors attempted to focus on the force on the structure. This is because when the force on the structure due to water is calculated precisely, then the movement of the structure can be simulated correctly. Therefore, this study should be regarded as a preliminary study for a complete coupled FSI problem which can be found in other studies of the authors.

In Figure 4, the measured and calculated free surface profiles at different times can be seen. In the figure, a1, b1 and c1 are the experimental data whereas a2, b2 and c2 are the numerical results. Due to the angle of photography in the experiments (a1, b1, c1), two free-surface profiles can be seen on the front and back wall which also can be seen in a2, b2 and c2. As shown, the measured and simulated free surfaces are in good agreement.



Figure 4. Free surface profiles at t= 0.1599 (a), 0.2766 (b) and 0.3733 (c) seconds.

Although, simulating free surfaces correctly give an insight that the FSI mechanism works, the forces calculated from FSI method should also be investigated. The measured pressures at transducer 2 shown in Figure 1 were used to validate the forces. The calculated and measured pressures can be seen in Figure 5. The calculated and simulated water-front reaches to the wall located at the end of the channel approximately at the same time. The calculated and measured impact pressures are in good agreement. Minor oscillations in the numerical results are observed due to the nature of the numerical calculation. The simulation of remaining experimental results can be seen in the other studies of the authors.



Figure 5. Impact Pressures at 2nd Sensor

CONCLUSIONS 4

In the present study, an FSI model combining meshless and mesh-based methods was proposed. For the meshless model, SPH algorithm, for the mesh-based method, finite element method was used. For the interaction between SPH and FEM, a modified contact mechanic algorithm was used. Although this method is capable of simulating the FSI problems with large geometric and material deformations, a benchmark problem was used to validate the interaction between water and structure (stiff) by verifying the forces or pressures. In the present study, the forces were directly calculated from contact mechanics whereas in some other researches they were calculated from the fluid equations. Proposed FSI method was validated with the experimental study of Lobovsky et al. (2013). It is found that the calculated free surfaces and impact pressures were in good agreement with the experimental data. In the experiments, structure was assumed to be fixed, so the displacement of the structure was not mentioned in the present study although very small displacements at the downstream gate were obtained. In a follow-up study, structural displacements calculated from proposed FSI model will be compared with the measured data of the experiments conducted by the authors recently.

REFERENCES

- Attaway, S., Heinstein, M. & Swegle, J. (1994). Coupling of smooth particle hydrodynamics with the finite element method. Nuclear Engineering and Design, 150(2-3), 199-205.
- Bathe, K.-J. (1982). Finite element procedures in engineering analysis. Prentice-Hall.
- Bathe, K.-J. & Chaudhary, A. (1985). A solution method for planar and axisymmetric contact problems. International Journal for Numerical Methods in Engineering, 21, 65-88.
- Bathe, K., Zhang, H. & Ji, S. (1999). Finite element analysis of fluid flows fully coupled with structural interactions. Computers & Structures, 72(1-3), 1-16.
- Benz, W. & Asphaug, E. (1995). Simulations of brittle solids using smooth particle hydrodynamics. Computer Physics Communications, 87(1-2), 253-265.
- Chawla, V. & Laursen, T. (1998). Energy consistent algoritms for frictional contact problems. International Journal for Numerical Methods in Engineering, 42, 799-827.
- Chen, J. K., Beraun, J.E. & Carney, T.C. (1999). A corrective smoothed particle method for boundary value problems in heat conduction. Journal for Numerical Method in Engineering, 46(2), 231-252.
- De Vuyst T, Vignjevic R. & Campbell, J.C. (2005) Coupling between meshless and finite element methods. Journal for Numerical Method in Engineering, 31(8):1054–1064.
- Deuflhard, P., Krause, R. & Ertel, S. (2008). A contact-stabilized Newmark method for dynamical contact
- Dowell, E.H. & Hall, K.C. (2001). Modeling of fluid-structure interaction. *Annual Review Fluid Mechanics*. 33 (1). 445–490.
- Fourey, G., Oger, G., Touzé, D.L. & Alessandrini, B. (2010). Violent Fluid-Structure Interaction simulations using a coupled SPH/FEM method. IOP Conference Series: *Materials Science and Engineering*, 10, 012041.
- Gingold, R. A. & Monaghan, J.J. (1977). Smoothed particle hydrodynamics: Theory and application to nonspherical stars. *Monthly Notices of the Royal Astronomical Society*, 181(3), 375-389.
- Groenenboom, P. H. (2009). Hydrodynamics and fluid-structure interaction by coupled SPH-FE method. *Journal of Hydraulic Research*, 48(1), 61-73
- Hu, D., Long, T., Xiao, Y., Han, X. & Gu, Y. (2014). Fluid–structure interaction analysis by coupled FE–SPH model based on a novel-searching algorithm. *Computer Methods in Applied Mechanics and Engineering*, 276, 266-286.
- Khenous, H.B., Patrick, L. & Renard, Y. (2006). Comparison of two approaches for the discretization of elastodynamic contact problems. *Mathematical Problems in Mechanics*, 342, 791-796.
- Liu, G.R. & Liu, M.B. (2003). Smoothed particle hydrodynamics: A meshfree particle method. Singapore: World Scientific.
- Lobovský, L., Botia-Vera, E., Castellana, F., Mas-Soler, J. & Souto-Iglesias, A. (2014). Experimental investigation of dynamic pressure loads during dam break. *Journal of Fluids and Structures*, 48, 407-434.
- Monaghan J.J. (1995). *Simulating gravity currents with SPH lock gates,* Applied Mathematics Reports and Preprints, Monash University.
- Monaghan, J. (1994). Simulating Free Surface Flows with SPH. *Journal of Computational Physics*, 110(2), 399-406.
- Monaghan, J. & Kocharyan, A. (1995). SPH simulation of multi-phase flow. Computer Physics Communications, 87(1-2), 225-235.
- Munjiza, A. (2004). The Combined Finite-Discrete Element Method. John Wiley & Sons.
- Ohayon, R. & Felippa, C.E. (2001). Advances in computational methods for fluid–structure interaction and coupled problems. *Computer Methods Application of Mechanical Engineering*, 190, 2977–3292.
- Swegle, J., Hicks, D. & Attaway, S. (1995). Smoothed Particle Hydrodynamics Stability Analysis. *Journal of Computational Physics*, 116(1), 123-134.
- Tezduyar, T. & Bazilevs, Y. (2008) Fluid-structure interaction, Computational Mechanics, 43, 1–189.

STUDY ON INTERNAL FLOW OF REVERSIBLE BULB TUBULAR PUMP WITH SYMMETRIC AEROFOIL

YAN JIN⁽¹⁾, HONGCHENG CHEN⁽²⁾, JUNXIN WU⁽³⁾ & CHAO LIU⁽⁴⁾

(1.2.3.4) School of Hydraulic, Energy and Power Engineering, Yangzhou University, Yangzhou, China, jinyan_yz@163.com; chenhc_yz@163.com; versionwjx@163.com; liuchao@yzu.edu.cn

ABSTRACT

The 3D flow field in reversible bulb tubular pump with "S" shape aerofoil impeller is analyzed. The pressure distribution on blade surface is similar at the two-way operating conditions, which indicates the symmetric aerofoil impeller with the same performance in both conditions. However, after adding the other parts to form pumping system, the performance of the reversible pumping system became different between the forward condition and reverse condition because of the asymmetry structure in the inlet passage and outlet passage. The main research is the influence of bearing support arrangement on the reversible tubular pump and the results show that the arrangement position of bearing support is certain to influence the system performance in the forward and reverse conditions. In the forward condition, the pump efficiency of the scheme of the bearing support and bulb on both sides of the impeller is higher than the scheme of the bearing support and bulb on the same side of the impeller, and in the reverse condition, the efficiency of the same side scheme is slightly higher than the both sides scheme. The forward operation efficiency is higher than the reverse one in two schemes.

Keywords: Reversible bulb tubular pump; symmetric aerofoil; bearing support; internal flow; numerical simulation.

1 INTRODUCTION

Nowadays, more and more urban pumping stations need to have the function of irrigation and drainage. In the plain areas, the head of many pumping stations are very low, even only about one meter. Considering the efficiency and practicability, the reversible tubular pump with low head and large discharge is chosen to realize the function of irrigation and drainage.

Symmetrical aerofoil is often used to implement reversible pumping of tubular pump. The symmetrical aerofoil is used when water pump reverse pumping, import side of the original blade changing into export side, and the pressure surface of the original aerofoil changing into suction surface, which is equivalent to the operation condition that water flow and rotation direction is invariable, the blade rotating around the center of rotation with 180°. To make the impeller with the same performance of forward and reverse, two types of aerofoil can be used: the first type is the aerofoil with straight central line, such as flat profile, the aerofoil flow of this type is not good, and the performance is poor; another type is a reverse symmetrical aerofoil, namely the "S" shape symmetrical aerofoil with good energy and cavitation performance.

"S" shape aerofoil blade is symmetrical, theoretically, the forward and reverse performance of impeller is the same, but the impeller must be combined with a structure of in and out passages and others to apply in practical engineering, so other asymmetry flow components of pump units can lead to the differences in forward and reverse performance. To facilitate the arrangement of the bearing, it is common to set up the guide vane which is used to recycle the impeller outlet vortex energy. The impeller, with the head about 3.0 meter, outlet vortex energy accounts for about 6% of the total energy. The guide vane of reversible act as both a rear diffuser and a front guide vane, the flow outlet angle of one-way axial flow guide vane is 90° conventionally, namely axial water, which does not retain export vortex. This design in turn works as a front guide vane, and the prewhirl flow can make the reverse performance of pump fell sharply. Quite a few scholars studied the guide vane form and arrangement of the reversible tubular pump (Tang et al., 2003; Wang et. al., 2000). Researches on the reversible tubular pumping system and symmetrical aerofoil are limited. Tang et al. (2004; 2002) compared the hydraulic performance of two reversible tubular pump installation. The experiments showed that both the pump installations had better performance characteristics than that from a usual vertical submersible pump installation. Tang also studied the reversible pump design with "S" shape aerofoil blade. Yang et al. (2012) studied on the three-dimensional fluid flow inside a diving tubular pumping system with symmetric aerofoil blade on dual-directional operation using CFD technology. Jiang et al. (2005) studied on the characteristics of tubular pumping devices by using the numerical simulation technique. Yan et al. (2008) carried on the forward and reverse energy and vibration test on reversible tubular submersible pump model. In addition, there are some scholars (Fang and Xie, 2010; Tang et al., 2013; Liu et al., 2010; Li et al., 2007; Lewis, 1996) studied the reversible tubular pump and symmetrical aerofoil. In this paper, based on the commercial CFD software, a reversible tubular pumping system with different positions of

bearing support is calculated and the influence of guide vane on the pumping system is analyzed. The relationship between the internal flow mechanism and the external characteristic of reversible tubular pump, which can provide a beneficial reference for this type of pumping station's construction, is revealed through the numerical simulation.

2 CALCULATION MODEL AND SCHEMES

In this paper, the bidirectional impeller was generated by the symmetrical aerofoil with double circular arc "S" shape (shown in Figure 1) (Tang et al. 2002). In Figure 1, F is the camber, L is the length of the areofoil and R is the arc radius (Eq. [1]), according to the design experience and previous model test study, the best performance corresponding to the camber ratio is about 4%.



Figure 1. The double circular arc "S" shape aerofoil.

$$R = \frac{F}{2} + \frac{L^2}{32F}$$
[1]

In order to simplify the calculation, the guide vane was not set, only a cross bearing supports used for bearing seat was set up, although there are bearing supports, there is no guide vane recovery for the function of circulation, but it can also have the effect of recycling part of the circulation and rectification. Besides, the forward operation condition when the bulb unit is in the outflow side was specified, on the contrary, the bulb unit in the inflow side is reverse operation condition. In order to study how the bearing support layout position affect the hydraulic performance of the reversible tubular pump, respectively, two different schemes need to be calculated: the bearing supports and the bulb unit on both sides (scheme 1) and the bearing supports and bulb unit on the same side (scheme 2) (single line diagram as shown in Figure 2).



©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

In terms of flow passage design, the symmetry in and out of the water conditions, the water flow inlet face of inlet passage and the outlet face of the outlet passage design are ensured to be the same size, and let the length of inlet passage and outlet passage size close as far as possible.

3 NUMERICAL SIMULATION

3.1 Computational Domain and Parameters

The computational domain for the numerical simulation is from the inlet to outlet which includes inlet passage, impeller, guide vane, bulb unit and outlet passage. The structure of bidirectional impeller model for the calculation is shown in Figure 3, and the three dimensional numerical calculation model of reversible bulb tubular pumping system of different schemes are shown in Figure 4. The specific parameters of the model pump are shown in Table 1.



Figure 3. The bidirectional impeller model perspective.



(a) Scheme 1.



(b) Scheme 2. **Figure 4**. The reversible tubular pump of different schemes.

Table 1. Parameters of the model pump.							
300							
1375							
0.36							
220~340							
3							
4							

3.2 Grid Generation and Boundary condition

In this paper, unstructured cells were used to define the inlet metric structure of the entire system which is very complicated, and the local mesh refinements were used in the impeller and guide vane, while the non-refinements were used in the inlet passage and outlet passage, which can avoid the waste of computing resources under the premise of ensuring the accuracy. The total number of cells was about 207360.

Boundary conditions of reversible tubular pump is particular, on forward operating conditions, boundary conditions were set as follows, passage 1 is the inlet passage and the passage 2 is the outlet passage; on reverse operation condition, the boundary condition should be switched to import and export conditions, passage 2 is the inlet passage and the passage 1 is the outlet passage; the impeller rotating direction is on the contrary to the forward condition, together with the rotating blades and hub of relative rotation direction was set to the opposite direction.

The inlet boundary condition adopts mass-flow rate for each computation, and the direction specification method is normal to boundary. Outflow condition was defined at the outlet. The method of multiple reference frames (MRF) was applied for numerical simulation. The flow in the impeller was computed in a moving reference frame, while the flow in the inlet pipe and guide vane was calculated in the stationary frame of reference. Shroud of the impeller was set as absolutely stationary, and the blade and hub of the impeller were relatively stationary. No slip boundary conditions and wall functions were used for the solid walls.

4 CALCULATION RESULTS AND ANALYSIS

Through the whole flow field numerical simulation of reversible tubular pumping system with different bearing support positions, the flow field information within the various flow components was obtained. At the same time, the hydraulic performance of reversible tubular pump system of forward and reverse operation condition were calculated, and the external characteristic curves were compared.

To facilitate the analysis of the reversible tubular pump within the three-dimensional steady turbulent flow calculation results, the best efficiency point (BEP) was chosen at the discharge Q = 280 l/s on forward condition, and the BEP was chosen at the discharge Q = 300 l/s on reverse condition as the typical points. In this paper, the unit in the pressure distribution figures is Pa, the unit in the velocity distribution figures is m/s. Scheme 1 in the figure represents the bearing supports on both sides of the impeller and bulb unit; and scheme 2 is on behalf of the bearing supports and bulb unit on the same side.

4.1 Analysis of the internal flow field of the pumping system

Figure 5 shows the velocity line and static pressure contour of vertical plane at best efficiency point (BEP) in different schemes. The results show that no matter where the bearing supports layout is, the flow pattern in the inlet passage is straight in two schemes, which change to complex when the water flow through the impeller, and the flow pattern became turbulence in outlet passage because of the different diffusion angle of guide vane hub. On the forward condition of scheme 1, due to the diffuser being after the impeller and without guide vane, the circulation of the impeller outlet is very large, but the manhole and the bulb support after the diffuser play a role on adjusting the flow pattern, therefore, after water flows into the outlet passage, although flow pattern is very disorder, the current rotating trend is not obvious; when operating on the diverse direction, water flows out of the impeller directly into the bearing supports, because of the large amount of water ring, bearing supports the existence of the disrupted flow of water, making water flow in the circumferential becomes uneven, the bias current is obvious, and there is a lot of vortex and backflow region in the bottom of outlet passage. Forward operation condition in scheme 2, bearing supports and bulbs on the same side, the flow after the impeller outlet regulated by the bearing supports, the manhole and the bulb unit support, the flow pattern of scheme 1 is better, however, since the bearing supports and bulb are all in the impeller outlet side, water shock loss will increase, so this part of the hydraulic loss will increase. On reverse operation condition, the bulb and the bearing supports are in the inflow side, the impact on the water flow is not great, but in the outflow side, there is no other flow structure, the flow of rotating impeller outlet flows into the outlet passage, so the flow in the outlet passage presents spiral flow due to the circulation, and the flow condition is very poor.

4.2 Flow field of the impeller

The pressure distributions on blade surface at BEP in the forward and reverse directions of scheme 1 and scheme 2 are shown in Figure 6. The trend of pressure distribution on the surface of the blade in different schemes is consistent, and the pressure distribution on the surface of the three blades is basically the same. The pressure of the pressure surface is higher than that of the suction surface. The pressure of the pressure gradually decreases from the inlet side to the outlet side, and that of the suction surface gradually increases. The pressure surface near the inlet side has local high pressure, while the suction side near the inlet side has local low pressure. Since the blades of the "S" aerofoil are symmetrical, the impellers have the same forward and reverse performance. It can be seen from the pressure distribution contour of the blade surface that the pressure distribution is very similar when impellers rotate in the forward and reverse directional impeller exchange under forward and reverse operating conditions. Flow components in the inlet and outlet passages have different structures, so the hydraulic performance of the forward and reverse operating conditions is still different for the entire reversible tubular pumping system.



(a) Scheme 1.



Figure 5. The velocity line and static pressure contour of profile at BEP in different schemes.



(b) Scheme 2. Figure 6. The pressure contour of blade surfaces.

4.3 Flow field of the bearing support

The velocity vectors of the bearing support outlet section in the forward and reverse directions of scheme 1 and scheme 2 are shown in Figure 7. Seeing from the figure, the bearing support is on the inflow side under the forward operating condition of scheme 1 and the outlet section is the inlet section of the impeller. Therefore, the flow velocity of the bearing support is 120° symmetrically distribution by the impact of threeblade impeller. Flow rate is uniform. In reverse operation, the bearing support is located on the outflow side, and the high-speed rotating water flows into the bearing support unit directly, so the velocity distribution in the outlet section is very uneven, bias flow and vortex occur obviously. In scheme 2, when the pumping system is running in the forward direction, the bearing support unit is on the outflow side and the outlet section is the water inlet side of the bulb unit. Therefore, the flow area is small. When water flows through the diffusion pipe with the bearing support, there is still a large amount of circulation, which results in backflow and vortex. When the pumping system operates in reverse direction, the bearing support and bulb are in the outflow side. Water flows into the bearing support more smoothly, and then flows into the impeller ideally. At the outlet of the bearing support, only a periodic flow with a uniform change in flow rate is exhibited.



(b) Scheme 2. Figure 7. The velocity vectors of the bearing support unit.

5 HYDRAULIC LOSS AND EXTERNAL CHARACTERISTICS OF PUMPING SYSTEM

5.1 Calculation of hydraulic loss in each section

Table 2 shows the hydraulic loss of each part of the pumping system at BEP for the forward and reverse running of scheme 1 and scheme 2.

Table 2. Hydraulic loss of each part of the pumping system.Unit: m						
Sche	emes	Passage 1	Passage 2	Bearing support unit	Bulb unit	
Scheme 1	Forward	0.010	0.150	0.099	0.549	
	Reverse	0.385	0.013	0.544	0.129	
Sahama 2	Forward	0.107	0.113	0.463	0.408	
Scheme 2	Reverse	0.736	0.046	0.094	0.019	

It can be seen from the table that in the forward operating condition of scheme 1, since passage 1 is relatively short (excluding the bearing support unit) and gradually shrinking, the hydraulic loss of the section is small, and the hydraulic loss in the bearing support on the inflow side is not large, but larger than passage 1 due to its relatively large local loss. The flow pattern of bulb unit which is adjacent to the impeller outlet is very poor, accompanied by bias flow and vortex phenomenon, and the hydraulic loss suddenly increases the water in passage 2 and gradually spreads, but there is still a large amount of circulation and relatively large hydraulic loss. When the system runs reversely, passage 2 turns into the inlet passage. The bulb section is also on the inflow side, and flow pattern is good. The hydraulic loss in passage 2 and the bulb unit is relatively

small. The hydraulic loss of the bearing support unit and passage 1 on the outflow side increases significantly. In general, the forward hydraulic loss is less than the reverse.

When the system runs forward in scheme 2, hydraulic loss in passage 1 which mainly consists of loss along the passage is not large. The hydraulic loss of the bearing support and bulb unit which on the impeller outlet side increases due to the existence of bias flow, vortex and flow separation, passage 2 is short and the local loss is small, so the hydraulic loss compared to the bulb unit is reduced. In the reverse operating condition, the bulb and the bearing support are located on the inflow side, and the hydraulic loss is small. There is no flow components in passage 1 directly connected to the impeller outlet, water flow is spirally diffused in it, and the flow condition is extremely poor with hydraulic loss reaching the maximum value. When the system runs relatively in scheme 2, the sum of all parts hydraulic loss of the system is lower than the forward operating conditions.

5.2 Performance curves of the pumping system

The performance curves of the two schemes were obtained through the numerical simulation, the performance curves of the same scheme under different operating conditions (shown in Figure 8) and that under the same operating conditions of different schemes were compared. (shown in Figure 9).







As shown in the figures above, hydraulic performance of the reversible tubular pumping system with 'S' aerofoil impeller in forward and reverse directions differs due to its different flow component structures. When the bearing support and the bulb unit are on both sides of the impeller, the head and efficiency of the system in forward operating condition are higher than the reverse condition. The efficiency difference is nearly 10%. While the bearing support and bulb are on the same side of the impeller, the maximum efficiency difference of forward and reverse directions is only 4%. When the system operates forward in two schemes, the head-discharge curves are close to each other, and the maximum efficiency of scheme 1 is about 5% higher than that of scheme 2. In the reverse operating condition, the head of the system of scheme 1 and scheme 2 is relatively close under the large discharge condition, and the efficiency curve is similar. Scheme 2 is only one percentage point higher than that of scheme 1.

6 CONCLUSIONS

The internal flow of the reversible bulb tubular pumping system with symmetrical aerofoil impellers was numerically simulated as steady flow by the Reynolds-averaged equation and the RNG k- ε turbulence model. The internal flow characteristics of the bulb tubular pumping system under different operating conditions were analyzed, and the influence of the position of the bearing support on the performance of the system in the reversible tubular pump was studied.

The static pressure distribution regularities of flow field in "S" aerofoil under design condition are similar to that of unidirectional impeller. The static pressure of the pressure surface decreases uniformly from the inlet side to the outlet side, and the static pressure of the suction surface increases uniformly from the inlet side to the outlet side. The local high pressure area appears on the pressure surface, and the local low pressure area appears on the suction surface near the inlet side of the blade. When the impeller is moving forward and backward, the static pressure distribution on the blade surface is similar, which shows that the two-way impeller has the same forward and reverse performance.

The forward operating performance of the bearing support arrangement of the reversible tubular pumping system is higher than that of the reverse direction. Under the same operating condition, the performance of the bearing support and the bulb on both sides of the impeller operating in the forward direction is higher than that of them on the same side, which is the contrary when it runs reversely. The influence of the bearing support on the performance of the reversible tubular pump is obvious, which should be reasonably arranged in practical applications.

ACKNOWLEDGEMENTS

This project was supported by China Nature Science Foundation of China (Grant No.51206139), the National Twelfth Five-year Science-technology Support Plan of China (Grant No. 2015BAD20B01-02), the Project Funded by Development of Jiangsu Higher Education Institutions, and Science and Technology Innovation Fund of Yangzhou University in 2016.

Support for construction of the facility was also provided by JSS Key Lab of Hydrodynamic Engineering.

REFERENCES

Fang, G.L. & Xie, W.D., (2010). Design of Large Diving Tubular Pump and its Application. *China Water Resources*, 16, 8-10

- Jiang, X.X., Li, L., Wang, L.L. & Hu, D.Y. (2005). Numerical Simulation Of Characteristics of Tubular Pumping Devices. *Journal of Hohai University (Natural Science)*, 33(1), 81-84.
- Lewis R.I. (1996). Turbomachinery Performance Analysis. Butterworth-Heinemann, London: Arnold, 1-328.
- Li, L., Wang, Z. & Hu R.X. (2007). Numerical Simulation of the Influence of Guide Vanes on Tubular Pumping Station Performance in Dual-Direction Operation. *Transactions of the Chinese Society for Agricultural Machinery*. 38(1), 76-79.
- Liu, C., Jin, Y., Zhou, J.R., et al. (2010). American Society of Mechanical Engineering, Power. *Conference Numerical Simulation and Experimental Study of a Two-Floor Structure Pumping System*. 777–784.
- Tang, F.P., Liu C., Xie, W.D., Yuan J.B., Zhou, J.R. & Cheng, L. (2004). Experimental Studies on Hydraulic Models for a Reversible, Tubular and Submersible Axial-Flow Pump Installation. *Transactions of the Chinese Society for Agricultural Machinery*, 35(5), 74-77.
- Tang, F.P., Liu C., Xie, W.D., Zhou, J.R., Yuan, J.B., Cheng, L. & Yan, B.P. (2002). Reversible Axial Flow Pump Design with "S" Type Aerofoil Blade. *Pump Technology*, 5, 11-13.
- Tang, F.P., Wang, G.Q., Liu, C., Xie, W.D., Zhou, J.R. & Cheng, L. (2003). Study on a Reversible Axial-Flow Pump Installation with S-Shaped Conduit. *Transactions of the Chinese Society for Agricultural Machinery*, 34(6), 50-53.
- Tang, X.L., Chen, X.S. & Wang, F.J. (2013). Numerical Simulation of Turbulent Flow in Bulb Tubular Pump and Performance Predictions. *Journal of Drainage and Irrigation Machinery Engineering*. 31(10), 1025-1029.
- Wang, L., Liu, D.K. & Chen, D.X. (2000). Experimental Study on the Hydraulic Characteristics of Bidirectional Tubular Pump. *Journal of North China Institute of Water Conservancy and Hydroelectric Power*. 21(4), 29-33.
- YAN, J.S., Zheng, Y. & Zhang, Z. (2008). Study on Energy Characteristics and Vibration Model Test of Reversible Tubular Submersible Pump. *Yangtze River*. 39(13), 82-84.
- Yang, F., Jin, Y. & Liu, C. (2012). Numerical Analysis and Performance Test on Diving Tubular Pumping System with Symmetric Aerofoil Blade. *Transactions of the Chinese Society of Agricultural Engineering*, 28(16), 60-67.

CHARACTERISTICS OF THE FREE SURFACE JET EMANATING DOWNSTREAM OF VORTEX INTAKE STRUCTURES: ANALYTICAL AND NUMERICAL APPROACHES

SEAN MULLIGAN ⁽¹⁾ & EOGHAN CLIFFORD ^(1, 2)

^(1, 2) College of Engineering and Informatics, National University of Ireland Galway (NUIG), Galway, Ireland seanmulligan23@gmail.com
⁽²⁾ The Ryan Institute, National University of Ireland Galway (NUIG), Galway, Ireland

ABSTRACT

Vortex flow drop structures are commonly employed in sewer systems and large hydropower schemes to convey water to lower levels and dissipate energy safely. The free-surface jet that forms just downstream of the intake plays a critical role in the performance of the device and knowledge of its characteristics are of importance to safe and optimum engineering design. In this study, principles of vortex flow and the critical cross-section of the free-surface vortex are used to evaluate the characteristics of the free-surface jet. A physical model of a subcritical vortex chamber was analysed and the nature of the emanating jet was observed for various approach flow conditions. The reduced scale vortex chamber was also simulated using a three-dimensional numerical model employing two different turbulence models: the shear stress transport with curvature correction and the baseline Reynolds stress model. The results indicated that the jet angle is largely independent of the approach flow depth for the investigated vortex chamber and test cases. The numerical models provide an excellent simulation of the free-surface jet position, shape and initial spray angle in addition to the bulk velocity throughout the jet. However, the analytical model developed was shown to produce errors in the region of 24 to 50 % which was pondered to be as a result of neglecting the gravitational constant at low pressures.

Keywords: Vortex; drop shaft; free-surface; jet; dissipation.

1 INTRODUCTION

Vortex flow drop structures are commonly employed in sewer systems to safely convey water from a higher elevation to a lower one (Drioli, 1947; Laushey and Mavis, 1953). They have also been utilized as spillway structures in high pressure hydropower schemes to safely dissipate excess energy (Jeanpierre and Lachal, 1966). The vortex intake and drop shaft structure is highlighted in Figure 1 (a-d). It comprises of (a) a tangential inlet which conveys the design flow Q to (b) a circular or spiral rotation chamber which imparts rotation or circulation Γ_{∞} resulting in a stable free-surface vortex discharging centrally through a vertical orientated orifice of diameter d into (c) a cylindrical drop shaft of length l_s . When the flow enters the drop shaft, the angular momentum imparted to the flow by the vortex chamber results in the fluid clinging to and flowing down the walls of the drop shaft as highlighted in Figure 1(d). The attenuation of swirl and increase in the axial velocity result in energy dissipation through friction on the drop shaft walls. This, as well as the flow stability for various discharges, renders vortex drop shafts preferable to standard plunge-flow drops (Zhao et al, 2006).

Knowledge of the characteristics of the free-surface rotating jet which emanates from the vortex intake is of importance for the safety design of these devices. In fact, although there has been a number of analytical and experimental studies performed on the drop shaft below the vortex intake (Jeanpierre and Lachal, 1966; Crispino, 2016), there is still a limited amount of research on the characteristics of the jet structure at the transition from the vortex chamber to the drop shaft. In this study, the authors adopt principles from the theory of 'vortex breakdown' (Leibovich, 1978; Lucca-Negro and O'Doherty, 2001) and the critical cross-section (Keller, 1995) coupled with existing analytical methods for modelling free-surface vortex flow behavior (Ackers and Crump, 1960; Mulligan et al., 2016). The article first introduces the basic hydraulic principles surrounding the rotating jet structure. Subsequently, experimental observations of the jet structure in typical subcritical vortex chamber are discussed followed by three-dimensional numerical simulations of the model jets. The study closes with a discussion of the ability to predict the annular jet characteristics (such as the angle of emanation and velocity characteristics) as well as the likely effects that this can have on downstream structures. The study aims to introduce techniques and concepts regarding vortex breakdown in drop shafts to be progressed in future studies.



Figure 1. Example of (a) a vortex intake and (b) vortex intake during operation and (c) example of a downstream vortex drop shaft together with (d) a model of the swirling flow down the walls of the shaft. Images of Milton Keynes vortex drop shaft courtesy of Nic Townsend. Model images courtesy of the École polytechnique fédérale de Lausanne (EPFL) (Schleiss, 2015).

2 HYDRAULICS OF THE ANNULAR JET

An end view cross-section of a typical vortex intake chamber with a subcritical approach flow $Fr_i \le 1$ is presented in Figure 2(a). As the flow is conveyed through the tangential inlet, the approach flow geometric arrangement results in strong conditions of circulation Γ_{∞} which generates a stable, vertical air core extending deep into the intake (Mulligan et al., 2016). In the subcritical approach flow, the tangential velocity field varies according to $\Gamma_{\infty} = 2\pi v_{\theta} r$ where Γ_{∞} can be taken as constant throughout the flow field, assuming irrotational flow conditions. In a recent study undertaken by Mulligan et al. (2016), the authors found that for a subcritical approach flow, the circulation can be approximated by:

$$\Gamma_{\infty} = \frac{Qr_{in}}{bh}$$
[1]

which allows one to easily interpret the tangential velocity field down to the outlet of radius r = d/2. Keller (1995) defines the case of a free-surface, 'hollow core' type vortex, as a B-Type vortex which approaches the critical-state as the cross-sectional area of flow decreases. Therefore, the critical-section resides at the outlet where there is a local minima in the air core radius. The critical-section of the free-surface vortex acts as the controlling section and behaves in an analogous fashion to a traditional flow over a weir, where the flow upstream of the critical-section is subcritical and the flow immediately downstream is supercritical. When the free-surface jet downstream of the critical section is contained, it is prone to the breakdown phenomenon (Benjamin, 1962) where the jet increases again in cross-sectional area resulting in recirculation patterns and turbulence in the jet (Escudier and Keller, 1985). This annular hydraulic jump phenomenon has been studied by Binnie (1964) and is analogous to the traditional hydraulic jump in open channels downstream of a weir or constriction in a channel. The annular hydraulic jump would suggest poor design in a vortex drop shaft structure due to a limited flow capacity and can result in the hydraulic jump travelling upstream and drowning the vortex intake completely Binnie (1964).

Figure 2(b) outlines the case where the free-surface jet freely discharges to the atmosphere (i.e. no vortex breakdown occurs within the control volume). Various studies have shown that the axial velocity across the critical cross-section $d/2 \le r \le a_c/2$ is constant (Quick, 1961; Ackers and Crump, 1960 and Mulligan et al., 2016). Therefore, the axial velocity distribution at the outlet can be approximated by:

$$v_{zo} = \frac{Q}{\frac{\pi}{4}(d^2 - a_c^2)}$$
[2]



Figure 2. (a) Cross section of vortex chamber and (b) analytical control volume for the free-surface jet downstream of vortex chamber.

Keller (1995) applied momentum principles to the rotational conditions in the B-Type vortex of a simplex atomizer fuel injector which behaves in the same way as a free-surface vortex intake. In their study, a theoretical solution for the jet angle α was developed. Using boundary conditions for the free-surface vortex as outlined in Figure 2, the jet angle can be computed by:

$$\alpha = 2Tan^{-1} \left[\frac{\left(1 + \left(\frac{\Gamma_{\infty}}{\pi a_c v_{zo}}\right)^2\right) \left(1 - \left(\frac{a_c}{d}\right)^2\right)^2}{\left[\left(1 - \left(\frac{a_c}{d}\right)^2\right) \left(1 + \frac{1}{2} \left(\frac{\Gamma_{\infty}}{\pi a_c v_{zo}}\right)^2 + \left(\frac{a_c}{d}\right)^2 \left(\frac{\Gamma_{\infty}}{\pi a_c v_{zo}}\right) ln\left(\frac{a_c}{d}\right)\right)\right]^2} - 1 \right]^{\overline{2}}$$
[3]

where a_c is the critical air core diameter, d is the intake diameter and v_{zo} is the axial velocity at the critical cross-section. The solution assumes that the jet angle just downstream of the critical cross-section is independent of gravitational conditions which is deemed valid for large flows and pressures.

3 EXPERIMENTAL OBSERVATIONS

Experiments were conducted in a physical model of the vortex chamber constructed from 6 mm transparent acrylic as shown in Figure 3(a). The model had an intake diameter d = 0.067 m, inlet width of b = 0.067 m and an inlet radius of $r_{in} = 0.174$ m corresponding to an approach flow geometric factor $\alpha = 2.768$ (Muligan et al., 2016). The walls scrolled inwards according to the logarithmic spiral $r_p(\theta) = ae^{b\theta}$. Flow was circulated from a lower reservoir to the chamber using a pump with a flow range of between 0 and 4 l/s which was regulated using an inline flow control valve. The model discharged directly to the lower reservoir under free jet conditions as outlined in Figure 2 and 3. The dischargeQ, circulation Γ_{∞} , approach flow depth h and critical air core diameter a_c were measured accordingly as described by Mulligan et al. (2016) and scale effects were found to be negligible when utilizing Froudian similitude. The jet angle α was measured in the model from medium exposure images of the annular jet (see Figure 3(c)). The jet angle was observed for six approach flow depths: h/d = 1.0, 1.5, 2.0, 2.5, 3.0, and 3.5. The system monitoring delivered an accuracy corresponding to maximum error bars for intake Froude and radial Reynolds numbers of Fd = \pm 0.019 and Rr = \pm 34, respectively which corresponding to values 3 orders of magnitude smaller than the experimental range.



Figure 3. Schematic of (a) plan and (b) end view of subcritical vortex chamber and (c) an example of the freesurface jet emanating at the underside of the vortex chamber.

Test	h/d	Q	Γ_{∞}	Νr	a_c/d
case #		(m³/s) × 10⁻³	(m²/s)		-
A1	1.0	0.00070	0.170	12.77	0.746
A2	1.5	0.00115	0.186	8.51	0.597
A3	2.0	0.00170	0.206	6.38	0.507
A4	2.5	0.00219	0.212	5.09	0.463
A5	3.0	0.00260	0.211	4.28	0.448
A6	3.5	0.00310	0.218	3.73	0.430

Table 1. Test parameters of experimental investigation

NUMERICAL SIMULATION 4

Three-dimensional, multiphase numerical modeling of the vortex flow structure and emanating jet was performed using ANSYS CFX (V14.5) which uses a hybrid FEM/FVM (finite element based finite volume method) approach in discretizing the Navier-Stokes equations. Previous numerical analysis performed on strong free-surface vortex flows specify best practice guidelines as follows which will be employed in this paper:

- Radially structured or quasi-structured mesh arrangement (Suerich-Gulick et al., 2013; Mulligan et al. 2016b).
- Choice of numerical scheme resolution has no major effect on the tangential and axial velocity profile (Škerlavaj et al., 2010; Škerlavaj et al., 2014).
- Transient modeling is necessary to resolve flow instabilities in the vortex core region (Mulligan et al. 2016b).
- Time steps in the region of 0.01 seconds are necessary for model studies (Skerlavaj et al., 2014).
- Shear stress transport (SST) with curvature correction (CC) and Reynolds stress models (RSM) are necessary to properly model regions of strong streamline curvature, rotation and anisotropic turbulence near the vortex core (Skerlavaj et al., 2014; Mulligan et al., 2016b).

The boundary condition configuration assigned a mass flow at the inlet and a static pressure condition at the outlet. For the current case however, a mass flow inlet required that an artificial condition be assigned in order to best simulate the approach flow depth. Improper specification of the inlet depth may introduce unphysically realistic momentum conditions as a result of the 'drop' of fluid at the chamber. To resolve this problem, a 'virtual wall' was placed at the inlet which protrudes a small distance below the approach flow depth h to ensure its constancy during simulation. An opening with a zero pressure boundary condition was imposed at the outlet which guaranteed that recirculation of each phase was permitted. The 'entrainment' option was selected which behaved in a similar manner to that of a pressure-specified opening but did not require a flow direction to be specified. The top of the chamber was simulated as an opening boundary condition with zero relative pressure. This permitted air flow normal to the boundary in both directions (see Figure 4 (a)). A no slip boundary condition was imposed to the walls of the vessel.

The two phase fluid domain is modeled using a homogeneous Eulerian-Eulerian multiphase flow model. This is a limiting case of the full Eulerian-Eulerian model which assumes that the interphase momentum transfer is negligible. This is valid for the current test case where the phases are completely stratified and the interface is well defined and interphase mixing is minimal. In the homogenous approach, both phases are treated as interpenetrating continua parted by a well-defined interface and share a common velocity, pressure and turbulence field. Therefore, cells that are located away from the interfacial zone will be representative of either air or water $\phi_k = 1$ and cells in the vicinity of this zone will contain a mixture of both $0 > \phi_k > 1$.





As a conservative measure, a high resolution scheme (second order accurate) was used to model advection and turbulence numeric. According to the previous best practice guidelines, the SST-CC and the Baseline BSL RSM were employed independently to model the turbulence. The flow domain was discretized using a quasi-structured radial meshing arrangement as shown in Figures 5 (b) and 5 (c). 3.18×10^6 elements were used with a minimum cell size of 1 mm close to the vortex core complying with the sensitivity study performed by Mulligan et al., (2016b). In order to properly resolve the turbulent boundary layer using the SST-CC and BSL RSM, the near wall nodes were sized in order to enforce a y –plus of $1 \ge y \ge 6$. Each test case was modelled using a transient simulation for time steps of 0.01 s and 0.005 s for the SST-CC and RSM respectively. Test cases A1, A3 and A6 were investigated using the boundary conditions of Table 1. A conservative physical simulation time of t = 40, 25 and 20 seconds starting from a 'new' initial condition was chosen for test cases A6, A3 and A1 respectively based on laboratory observations. The target value of the normalised residual for each flow variable was set to 10^{-5} . Simulations are performed at the Fionn supercomputer at the Irish Centre for High End Computing (ICHEC). 48 cores were used for the SST-CC transient simulations and 96 cores were used to compute transient RSM simulations.

5 DISCUSSION OF RESULTS

5.1 Experimental observations

Figure 6 (a-c) provides a visual comparison of the free-surface annular jet emanating from the vortex drop shaft for test cases A1, A2 and A3 respectively. For approach flows $h/d \le 0.5$, it was noted that the jet was poorly formed and the fluid film emanating from the intake quickly separated into filaments and drops. The flow in the chamber for these low approach flows was dominated by a radial flow and therefore there was a limited angular velocity available to form a stable rotating jet under the ambient gravitational conditions. As shown in Figure 6, a well formed jet was observed for each approach flow and there was no apparent visual difference between the formed jets in each case. The jet structures appeared to be lightly asymmetric. The overall jet took on a general 'umbrella' like appearance with the initial part of the jet originating from a vertical surface.

Therefore, the jet angle changes throughout the jet according to a tangent to the free surface i.e. $\alpha_s = \frac{d_{z_j}}{dr_i}$. The initial jet angle α was determined from images obtained of the jet.



Figure 5. Images of the free-surface jet for (a) h/d = 1.0 (b) h/d = 2.0 and (c) h/d = 3.5.

5.2 Numerical Results

For the SST-CC models, the root mean square (RMS) residuals for each variable achieved values in the region of 10^{-4.9}. The RMS residuals for the BSL RSM model were well below 10⁻⁵ as required for each of the variables. Furthermore, the global imbalances were found to be small (< 0.00001 %) in each simulation and therefore conservation is essentially achieved. Figure 6 presents a qualitative comparison between the free-surface prediction in both the transient SST-CC and BSL RSM models compared to that of the experimental free-surface.



Figure 6. (a-c) Iso-surface at a water volume fraction of $\phi_w = 0.5$ outlining the position of the predicted freesurface and (d- i) the simulated jet for each approach flow condition as displayed by the contour of the water volume fraction where Red corresponds to $\phi_w = 1.0$ and Blue corresponds to $\phi_w = 1.0$.

Regarding the free-surface vortex, the centrality and shape of the free-surface were well predicted in each case. The Reynolds stress model indicated a slightly higher, more accurate prediction of the free-surface at the inlet. The percentage error in predicting the far-field (inlet) free-surface was recorded to be 12 %, 11 % and 16 % for A1, A3 and A6 respectively. The annular jet formation in each case compared qualitatively well

with the experimental observations of Figure 5 with no significant differences observed between the SST-CC model and the RSM. However, due to the containment of the jet in the drop shaft n the numerical simulations, it was difficult to observe the change of a convex to concave structure as identified in the experimental observations. The thickness of the jet reduced along the z –axis as the flow travelled down the surface of the drop shaft. This is due to the increase in axial velocity and decrease in the tangential velocity as the fluid spirals down the chamber. Jeanpierre and Lachal (1966) describe that the residual rotational energy is much smaller than the potential energy and its function is to stabilize the flow on the drop shaft walls whereas the axial velocity is responsible for induces energy dissipation.

5.3 Jet angle

Figure 7 highlights the relationship between the jet angle α and the approach flow depth for the experimental, numerical and analytical model. It can be concluded that the jet angle is largely independent of the approach flow depth with a minimal variation in the jet angle of 4 degrees observed for the range of test conditions. The relationship appears to show that the angle decreases very slowly as the approach flow depth increases for the experimental range observed. The decrease in the angle is a result of the axial momentum dominating the rotational flow, hence a reduction in the jet circulation number as the approach flow depth is increased. The numerical data agreed well with the experimental results in both cases with an over estimation of the jet angle in the range of 10 to 18 %. However, the analytical model (Eq. [3]) provided a significant over estimation of the jet angle by approximately 24 to 50 %. It was pondered that this could be due to numerous reasons including uncertainty in the jet angle measurement or the position at which the jet angle is being measured. For example, in this study, the jet angle is measured as it directly emanates from the intake and from section 5.1 and 5.2 it is clear that the jet angle varies as a tangent to the curve of the jet. If the jet angle was measured further down in the jet, a larger angle would be obtained which may compare better to Eq. [3]. However, another reason for the observed discrepancy may owe to the fact that the gravitational force is disregarded in the development of the analytical solution (Keller, 1995). In cases where the approach flow depth (or pressure) in the vortex chamber is low, the ambient gravitational conditions may result in a drawdown or decrease of the angle of the real jet conditions.



Figure 7. Relationship between the jet angle α and the approach flow depth for experimental, numerical and analytical model (Eq. [3]).

5.4 Jet velocity

Figure 8 (a-f) present the contours representing the total velocity vector in the region of the jet for the SST-CC model and the BSL RSM for each approach flow condition (h/d = 1.0, 2.0 and 3.5). It is apparent that there is no obvious variation between each of the models. In each case, the velocity reaches its maximum value at the underside of the orifice prior to the expansion of the cross section. The velocity for each test case is in the range calculated from the resultant velocity vector $v = \sqrt{v_{\theta}^2 + v_z^2}$ at the critical-section by combining $\Gamma_{\infty} = 2\pi v_{\theta} r$ and Eq. [2] and assuming that the radial flow in the critical section is negligible. The large magnitude of the axial velocity in the jet for higher approach flows reinforced the observations observed in the reduction of the spray angle.



Figure 8. Velocity conditions in the annular jet as predicted by (a-c) the SST-CC model and (d-f) the BSL RSM for each test case.

6 CONCLUSIONS

In this study, a brief investigation on the characteristics of the free-surface jet emanating from a vortex intake was presented. Considerations for the hydraulics of the rotating jet, together with the critical cross-section and principles of vortex breakdown were discussed with regards their application in evaluating the jet flow. An equation to predict the jet angle directly below the intake was presented as modified from a previous study on the simplex atomizer. Next, an experimental and numerical evaluation of the vortex chamber and jet structure was performed on a reduced scale model. The experimental results indicated that, for the particular model investigated, the jet angle remains largely independent of the approach flow depth with a slight reduction in the jet angle as the approach flow depth increases. It was pondered that the reduction in jet angle was probably a result of the vortex flow tending towards the critical submergence where the circulation number in the flow field decreased. This reduction might be more obvious in a weaker approach flow geometry.

The results of the numerical SST-CC model and the BSL RSM indicated an excellent qualitative comparison between the position, location and shape of the free-surface jet. The angle was found to be overestimated by approximately 10 to 18 %. The analytical model overestimated the jet angle in the region of 24 to 50 %. The error was significant and was pondered to be as a result of jet angle measurement uncertainty or due to neglecting the gravitational force in the analytical solution. In summary, the study introduces some interesting approaches to evaluating the jet emanating from a vortex intake using the critical cross section and available models for the primary flow field. The methods have implications in assessing or predicting the behavior in vortex drop shaft structures such as the nature of the axial and tangential velocity boundary conditions at the start of the drop shaft, the jet spread angle and its impact force on the shaft wall (following an expansion) as well as to aid in the computing of pressures, shear stresses and frictional head-losses.

ACKNOWLEDGEMENTS

The authors would like to thank the Irish Centre for High End Computing (ICHEC) for providing simulation time on the Fionn Supercomputer. The authors would also like to thank Nic Townsend and Giovanni de Cesare of the EPFL for permission for use of drop shaft images.

REFERENCES

Ackers, P. & Crump, E.S. (1960). The Vortex Drop. *ICE Proceedings: Thomas Telford*, 433-442. Benjamin, T.B. (1962). Theory of the Vortex Breakdown Phenomenon. *Journal of Fluid Mechanics*, 14(04), 593-629.

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

- Binnie, A.M. (1964). Annular Hydraulic Jumps. *Proceedings of the Royal Society of London a: Mathematical, Physical and Engineering Sciences,* the Royal Society, 282(1389), (155-165).
- Crispino, G., Dorthe, D., Fuchsmann, T., Gisonni, C. & Pfister, M. (2016). *Junction Chamber at Vortex Drop Shaft:* Case Study of Cossonay.
- Drioli, C. (1947). Su Un Particolare Tipo Di Imbocco Per Pozzi Di Scarico (scaricatoreidraulico a vortice)', L'Energia Elettrica, 24(10), 447-452.
- Escudier, M.P. & Keller, J. (1985). Recirculation in Swirling Flow-A Manifestation of Vortex Breakdown. AIAA Journal, 23(1), 111-116.
- Jeanpierre, D. & Lachal, A. (1966). 'Dissipation D'énergie Dans Un Puits À Vortex', La Houille Blanche, 7, 823-832.
- Keller, J.J. (1995). The Critical Cross-Section of a Vortex. *Zeitschrift Für Angewandte Mathematik Und Physik ZAMP*, 46(1), 122-148.
- Laushey, L.M. (1953). Studies Show Pittsburgh How To Drop Sewage 90ft Vertically To Tunnel Interceptors. *Engineering News Records*.
- Leibovich, S. (1978). The Structure of Vortex Breakdown. Annual Review of Fluid Mechanics, 10(1), 221-246.
- Lucca-Negro, O. & O'Doherty, T. (2001). Vortex Breakdown: A Review. *Progress in Energy and Combustion Science*, 27(4), 431-481.
- Mulligan, S., Casserly, J. & Sherlock, R. (2016). Effects of Geometry on Strong Free-Surface Vortices in Subcritical Approach Flows. *Journal of Hydraulic Engineering*, 142(11), 1-12.
- Mulligan, S., Casserly, J. & Sherlock, R. (2016b). Experimental and Numerical Modelling Of Free-Surface Turbulent Flows in Full Air-Core Water Vortices. In Advances in Hydroinformatics. Springer Singapore. 549-569.
- Quick, M.C. (1961). A Study of the Free Spiral Vortex. University Of Bristol.
- Schleiss, A. (2015). *Rapport D'activité Activity Report,* Laboratory of Hydraulic Constructions, École Polytechnique Fédérale De Lausanne.
- Škerlavaj, A., Lipej, A., Ravnik, J. & Škerget, L. (2010). Turbulence Model Comparison for a Surface Vortex Simulation. IOP Conference Series: Earth and Environmental Science, 25th IAHR Symposium on Hydraulic Machinery and Systems, 12(1), 1-10.
- Škerlavaj, A., Škerget, L., Ravnik, J. & Lipej, A. (2014). Predicting Free-Surface Vortices with Single-Phase Simulations. Engineering Applications of Computational Fluid Mechanics, 8(2), 193-210.
- Suerich-Gulick, F., Gaskin, S.J., Villeneuve, M. & Parkinson, É. (2013). Characteristics of Free Surface Vortices at Low-Head Hydropower Intakes. *Journal of Hydraulic Engineering*, 140(3), 291-299.
- Zhao, C.H., Zhu, D.Z., Sun, S.K. & Liu, Z.P. (2006). Experimental Study of Flow in a Vortex Drop Shaft. *Journal of Hydraulic Engineering*, 132(1), 61-68.

SIMULATION OF STEADY AND UNSTEADY FLOWS THROUGH A SMALL HORIZONTAL FRANCIS TURBINE

AHMED LAOUARI (1) & ADEL GHENAIET (2)

⁽¹⁾Laboratory of Energetic Mechanics and Engineering (LEMI), Faculty of Engineering, University of Boumerdes, Algeria ahmedlaouari@gmail.com
⁽²⁾Laboratory of Energetics and Conversion Systems, Faculty of Mechanical Engineering, University of Sciences and Technology HouariBoumediene, BP32 EI-Alia, Bab-Ezzouar, 16111 Algiers, Algeria

ABSTRACT

Computational Fluid Dynamics (CFD) analysis is a very useful tool for predicting performances at different operating conditions of hydraulic machinery. All theoretical methods for predicting the performance merely gives a value, and one is unable to determine the root cause for the poor performance. Due to the development of CFD code, one can get the performance value as well as observe the actual behavior of the flow in the domain. Analysis and variation of performances can be found out by using CFD analysis. Pressure fluctuation due to rotor-stator interaction and occurrence of vortex rope in draft tube at partial load operation are obvious phenomena in Francis type reaction hydro turbines. These hydrodynamic effects are important issues and should be addressed during the design of hydraulic machines. A 3D transient state turbulent flow simulation in the entire flow passage of a Francis turbine was conducted to investigate the rotor-stator interaction by adopting k-w base SST turbulence model. The commercial 3D Navier-Stokes CFD solver Ansys-CFX was utilized to study the flow through this horizontal shaft Francis turbine in its stationary and transient passages, at small flow, optimal, and high flow rate. The investigated turbine consists of a spiral casing with 6 guide vanes, a runner with 10 blades and draft tube. A periodical behavior was observed for the pressure distribution in guide vanes, runner blades and torgue in the runner blades. Vortex breakdown occurred in the draft tube at small flow rate operation of the turbine. The rotational frequency of the vortex was predicted for the rotational frequency of the runner.

Keywords: CFD; Francis Turbine; hydraulic performances; pressure oscillation; unsteady flow.

1 INTRODUCTION

A Francis reaction turbine is the most commonly used type at hydropower stations (Tushar et al., 2011). This turbine can be used for micro-medium or large hydro-stations, as the operating range of Francis turbines is between 1 m and 900 m. It has a radial or mixed radial/axial flow runner, which is most commonly mounted in a spiral casing with internal adjustable guide vanes (Voith, 2013). On the other hand, compared to impulse turbines, reaction turbines have a better performance in low head and high flow sites (Paish, 2002). The flow in Francis turbines contains a large variety of length and time scales because of different flow phenomena appearing simultaneously, such as turbulence, secondary flows, positive pressure gradient, adverse pressure gradient, boundary layer separation, vortices, vortex breakdown, shock waves, cavitation, erosion, and fluid structure interaction. The prediction and understanding of the flow behavior in casing, runner and draft tube region hold key importance in redefining the flow and developing better flow techniques to overcome the flow instabilities and the detrimental interaction between the components (Anup et al., 2013).

Sabourin et al. (1996) implemented a strategy to simulate low interactions between rotating and stationary components. The distributor and the runner were calculated in a single calculation through the stage interface, whereas the draft tube was calculated separately and the pressure condition at runner outlet was adjusted. Wu et al. (2007) applied CFD to a Francis turbine to integrate three blade designs in order to provide over 3% increase in peak efficiency and 13% increase in power with an improved cavitation coefficient for less than 0.09. In comparison with the original runner, they demonstrated that the pressure exhibits a much more uniform distribution without a low-pressure zone on the suction side near the leading edge. Kumar and Saini (2010) presented a study for different causes of the declined performance of hydro-turbines and the suitable remedial measures based on a literature survey of various aspects related to cavitation.

The source of the torque variation was found to be either a rotor-stator interaction (RSI) or a vortex breakdown in the draft tube. During steady state operation, small discharge fluctuations may couple to the continuous development, destruction, and rotation of vortices at a sub synchronous frequency inside the runner channels (Widmer et al., 2011). Blade-to-blade pressure differences and asymmetric pressure loadings may also cause output torque fluctuations, particularly when the RSI occurs. The interaction between the guide vanes, wakes and the rotating blades creates a complicated pressure and velocity field in the vaneless space. A wave propagates towards the runner blade, and at the same time, the RSI generates a wave,

creating large oscillations in the entire hydraulic system (Brekke, 2010). Cyclic interactions between the blades and the guide vanes create periodic phenomena that cause variations in the torque.

The development of CFD techniques has saved enormous amount of time and cost of repeated measurements in the laboratory. Currently, CFD is considered as a virtual test rig for the hydraulic turbines (Enomoto et al., 2012; Bucur et al., 2012). The measurements are performed for specific requirements only; for example, experimental data required for validating the numerical model (Sun-Sheng et al., 2012).

This paper contributes to predicting the hydraulic performances in the full passages of a small model of a Francis turbine at different operating conditions for steady state simulation, and to give a better insight into the unsteady flow behaviour. The interaction between the stationary and rotating domains and the influence of the fluid flow in various components are studied in this time-dependent analysis. Also, a fast Fourier transformation (FFT) is done, and the rotor-stator interaction frequencies are predicted at all operating conditions.

2 NUMERICAL SIMULATION PROCEDURE

The commercial software CFD-solver Ansys-CFX v16.0 is used to pre-process the time varying unsteady flow through a horizontal shaft of Francis turbine in the stationary and rotating passages at small, optimal and large flow rate where the steady and transient flow fields in spiral casing, distributor vanes, runner and draft tube are simulated. The model of a Francis turbine was taken from a hydraulic laboratory workbench.

The computation domain is divided into 4 components, a spiral casing, 6 distributor vanes, runner of 10 blades with crown and band and a draft tube. The 3D models of the components were generated in Solid-Works CAD (Computational Aided Design) software. The obtained CAD model is shown in Figure 1 and the parameters of the turbine are summarized in Table 1.



Figure 1. 3D scheme of Francis turbine and section view of region near runner.

Parameter	Value
Falameter	Dimension in mm
Inlet diameter of spiral case	38
Outlet diameter of spiral case	149
Height of distributor vanes	8
Inlet diameter of runner	83
Outlet diameter of runner	38
Outlet diameter of draft tube	80
Number of blades of guide vane	6
Number of blades of runner	10

Table	1.	Geometrical	parameters	of	Francis	turbine	model.

The unstructured tetrahedral mesh with eight prism layers, 1.2 high ratio was generated for the spiral casing domain. The mesh of the structured multi-blocks in the draft tube was done with 45 hexahedral blocks. For the runner and distributor vanes, a total of 170 hexahedral passage blocks were adopted for the generated mesh in Turbo-Grid software (see Figure 2).

The entire computational grid is comprised of 3.2 million grid nodes. The entire fluid domain of the turbine was formed by combining the components with an interface between casing and runner and runner and draft

tube, each, using general grid interface (GGI) method for mesh connection. Spiral casing with distributor vanes, and draft tube are stationary component while runner is the rotating component.

The boundary conditions needed in the present simulations are as follows: mass flow set at the casing inlet and an opening-type boundary with a prescribed steady water pressure at the outlet of draft tube.

For transient analysis, the time step of 1.8° rotation of runner was taken for 3 full rotations of the runner. So, the time step was 0.000158 s at optimal operating condition, corresponding to 1/200 of the runner rotational period and the total computational time was 0,0948 s of the runner. A second order backward Euler was used as a transient scheme with a high-resolution advection scheme. The maximum loop coefficient was taken as 10. The k- ω based shear stress transport (SST) model of Menteris applied for turbulence treatment to study the rotor-stator interaction. Near the walls, nodes were positioned in such a way that the value of $y^{+} = \rho y_{p} u_{t} / \mu_{f}$, $u_{t} = V_{f} \sqrt{f/2}$ (V_f: flow velocity) and the friction factor is given by $f = 0.025 R_{er}^{-0.1428}$.

The solution obtained from 3D steady state flow with SST treated turbulence model and a frozen rotor approach was specified as an initial guess for the transient calculation to give it a start-up.



Figure 2. Grid Discretization of the flow domains, spiral case, runner, inlet pipe and draft tube.

3 NUMERICAL RESULTS

3.1 Steady State Analysis

The flow field in this radial turbine was computed using the code ANSYS-CFX16, considering a complete volute and all-blades of distributor and runner and also full- draft tube.

The results of the global hydrodynamic performances parameters characterizing this Francis turbine were presented for different rotational speeds of the runner (1100, 1500, 1900, and 2360 rpm) and for different distributor vanes openings. Figure 3 presents the comparison of the produced power between the CFD prediction and the tests. It can be readily observed that the accuracy of CFD prediction is satisfactory along the low rate range.

Figure 4 shows the relationship between the ratio of unit speed and the ratio of unit discharge, revealing a continual decrease at all rotational speeds. For a rotational speed of 1100 rpm, the ratio of unit speed sweeps the whole operation range for a value of 0.9, whereas for 2360 rpm it is only equal to 0.2. Furthermore, the unit discharge varies proportionally to the low rate at the inlet of the spiral casing provided that the total head is constant.

The hydrodynamic performances map (Figure 5) based on the stage interface were plotted in terms of hydraulic efficiency, produced power versus flow rate, and ratio of unit speed versus ratio of unit discharge. The efficiency curves reveal a maximum of 79.18% for the rotational speed of 1900 rpm and flow rate of 165 l/min. The evolutions of the produced power at different rotational speeds and low rates were illustrated. For all rotational speeds, the produced power increases with the volume flow rate but at different scales.



Figure 3. Produced power and comparison between prediction and test



Figure 4. Ratio of unit speed versus ratio of unit discharge.

For the operating speed of 2360 rpm, the power sweeps the whole operating range for 470W, whereas at the lowest rotational speed of 1100 rpm the gain in power is 124W.



Figure 5. Performances map, (a) Hydraulic Efficiency, (b) Produced Power versus Flow rate

3.2 Transient Analysis

The results of the analysis were performed for small flow rate (SFR), best efficiency point (BEP), and large flow rate (LFR), characterized by the parameters shown in table 2, and figure 6 that presents the openings of distributor vanes.

Figure 7 illustrates the static pressure distribution in casing (spiral and distributor), and runner. The static pressure distributions in the mid-span of shroud entry and hub entry, drops as the potential energy is transformed into kinetic energy conformed to the free vortex design for all operating points. The equivalent pressure is in coherence with the uniform flow that is distributed evenly around the casing, and decreases rapidly toward the inner radius. In the distributor section, the static pressure distribution is much denser than in the leading edge, indicating that the inflow angle matches well with the leading edge angle of distributor vanes. Static pressure distributions around the rotor blades show difference between suction side and pressure side.

At small flow rate (Fig.7. c), the flow in the radial part of rotor has a pattern with strong recirculation, this may explain the radial velocity that diminishes more than the tangential velocity to cause flow circulation to pressure side where it stagnates. For large flow rate, this difference is located both at inducer and mid section (Fig.12.b). The flow approaches rotor with angles taking an average value smaller than for the other points, and the recirculation zone tends to reduce.

Table 2. Characteristics of the operating conditions.





Figure 7. Static pressure at rotor mid-span and the mid-span of casing for three revolution of runner, (a) BEP, (b) LFR, (c) SFR

In the inter-blade channels, the decrease in flow speed near hub and shroud makes the fluid more susceptible to pressure gradient, with a fluid migration from pressure side to suction side leading to radial movements and a passage vortex. As observed from Figure 8 (a), there is presence of vorticity regions over the pressure side of some blades for the optimal point. But for small flow rate point (Figure 8 (c)), the vortices occupy all passages and become larger. The morphology of the flow through the runner is greatly affected for the high water flow rates. Furthermore, the velocity near the pressure side is low, whereas the flow is highly accelerated near the suction side. Reverse against, a recirculation region is notable at the entrance of runner. This recirculation is described as normal forces to balance a current line, which implies that the addition of the centrifugal forces and viscous effects is equal to the pressure gradient from the difference of pressure between the suction and pressure sides. The origin of these vortices is often found within the boundary layers, zones where the fluid is slowed and is therefore much more sensitive to the influence of radial pressure gradients and azimuthal. The observed phenomenon in our case is the vortex passage shown in Figure 9.

Consequently, in the inter-blade channel, the decrease in speed at the hub and shroud makes the fluid more susceptible to pressure gradient. So, there will be migration of the fluid from the pressure side to the suction side of blade near the regions that close to the walls. This transfer of fluid will cause radial movements along the blades and form the passage vortex.



Figure 8. Streamlines in mid span of runner passage for three revolution of runner, (a) BEP, (b) LFR, (c) SFR.



Figure 9. 3D velocity streamlines in the runner for three revolution, (a) BEP, (b) LFR, (c) SFR.

The incorporated elbow type draft tube with a single outflow channel decelerates the flow leaving the runner and converts the excess of outlet kinetic energy into static pressure.

Figure 10 plots the pressure contours for the different operating points, showing the diffusion process which has qualitatively similar trends as reported in (Hellstrom et al., 2007; Xiao, 2010). The pressure near the inner wall is higher due to the channel bend and accordingly a vent set may be chosen. At the bend, a maximum velocity appears near the inner wall and as the low progresses into the duct and due to inertia force this maximum point rapidly moves towards the outer wall.

Much of the turbine's performance depends on the flow behavior in the draft tube and the flow condition in this component depends on point of operation. As shown in figure 11 and 12, the flow accelerates from the casing to the enters of the runner. After the runner transfers the energy into torque by the pressure difference between its blades, the flow decelerates while moving toward the draft tube outlet. The swirl rate appearing behind the runner for small flow rate greatly affects the flow condition in the draft tube. Swirl appears at runner downstream due to the radial component of velocity at runner exit where flow is supposed to be purely axial.

The flow in draft tube passage is streamlined and uniform but the flow field in inner region is disturbed with a very low velocity creating a dead water zone, resulting in flow recirculation and genesis of vortex shedding. This is can be explained by the local pressure pulsation caused by rotor-stator interaction and draft tube vortex precession that propagate along the whole water conduit.



Figure 10. Static pressure in draft tube at mid plane for three revolution, (a) BEP, (b) LFR, (c) SFR.

This region is developed as a result of the flow deceleration along the axis and its size indicates the vortex rope size and strength (Susan-Resiga et al., 2010).



Figure 11. Streamlines in draft tube at mid span for three revolution of runner. (a) BEP, (b) LFR, (c) SFR.



Figure 12. 3D streamlines in draft tube at optimal point for three revolution of runner

3.3 Spectral analyses

Spectral analysis of the time domain signals was perform to investigate the prevailing frequencies in the turbine. A Fast Fourier Transform (FFT) analysis of all the pressure-time data was performed to determine the prevailing frequencies in the turbine at the best efficiency point and large flow rate.

Consequently, when interpreting time-sequence data from a transient solution, it is often useful to look at the data's spectral (frequency) attributes (ANSYS, 2016), to determine the major vortex-shedding frequency from the time-history of the drag force on a body recorded during a simulation. Or, to compute the spectral distribution of static pressure data recorded at a particular location on the surface. In essence, the Fourier transform enables to take any time dependent data and resolve it into an equivalent summation of sine and cosine waves. A Fourier analysis shows that the unsteady simulations are capable of resolving the most important pressure and torque fluctuations (Lucien Stoessel et al., 2015).

The interactions phenomena produced by the movement of the rotor blades against the distributor vanes and runner against draft tube are defined in the chronic periodicity model stipulating that the signal corresponding to each property emerging from the interaction can be represented by a superposition of infinity of rotating waves, and hence all the matter is to describe these rotating waves. This model originates from the fact that the flow is characterized by a spatial and time (double) periodicity, as explained by (Gerolymos et al., 2002). For example, the static pressure presents a space and a time periodicities as follows:

$$p(x,r,\theta,t) = p(x,r,\theta + \Delta\theta,t) = p(x,r,\theta,t + \Delta t)$$
[1]

For a rotor, the spatial period and time period are related to the number of blades and speed of rotation by:

$$\Delta \theta = \frac{2\pi}{N_R}, \Delta t = \frac{60}{\Omega_R N_R}$$
[2]

The period $\Delta\theta$ is equal to 2π , if the greatest common divider of the numbers of rotating and stationary blades N_R and N_S is equal to 1. A particular case will happen when this greatest common divider N is different from 1, and in this case a passage composed of N_R/N rotating blades and a passage composed of N_S/N stationary blades have the same pitch, and subsequently the spatial period is simply

$$\Delta \theta = \frac{2\pi}{N_R}$$
[3]

The static pressure can be described by the double Fourier decomposition as proposed by Tyler and Sofrin (1962).

$$p(x,r,\theta,t) = \sum_{n=0}^{+\infty} \sum_{m=-\infty}^{+\infty} P_{mn}(x,r) e^{i[m\theta - nN_R\Omega_R t + \varphi_{mn}]}$$
[4]

The sum over 'n' represents the time harmonics, whereas the sum over 'm' represents the space harmonics. The theory of Tyler and Sofrin (1962) provides the following relationship between the time harmonic and the space harmonic:

$$m = nN_R + kN_S$$
) $k = \cdots, -1, 0, 1, ...$ [5]

There is a superposition of infinity of rotating waves, each one is characterised by a phase, amplitude and a rotating velocity corresponding to:



Figure 13. Pressure recording locations in distributor vanes and runner.

For the evaluation of pressure fluctuation in the components, 08 locations are chosen and time varying pressures are recorded in spiral casing, distributor vanes, runner blade and draft tube as shown in Figure 13 and Figure 14.





Figure 14. Pressure recording locations in spiral case and draft tube.





Figures 15, 16, 17, and 18, show a comparison of the amplitude of pressure pulsation at optimal and large flow rate operating conditions for different locations in turbine.



Figure 16. Frequency spectrum of the numerical analysis pressure-time domain signals at the BEP.



Figure 17. Pressure fluctuation for 3 runner rotations at Large flow rate, (a) SC1, (b) NGV2, (c) RUN1, (d) DT3.



Figure 18. Frequency spectrum of the numerical analysis pressure-time domain signals at the BEP.

The maximum of amplitude can be seen at the vaneless space location between distributor vane and runner blade (NGV 2), during LFR operating condition. The amplitude and frequency are shown in figure 17 (b) and 18, consecutively. In spiral casing location (SC1) for both BEP and LFR, the pressure oscillation is smooth and have a high frequency signifying the absence of swirl and vortex shedding in this location.

In runner blade (RUN 1) location where the amplitude of pressure fluctuation is higher, the dominant frequencies are (189 Hz, 242 Hz) for BEP and LFR consecutively.Furthermore, the pressure oscillations on runner blades are found to be related to the precession of vortex rope that caused pressure in the runner blade to fluctuate with a dominant frequency.A high amplitude and frequency found in vaneless space and runner for both operating conditions indicated the effect of the unsteady vortical flow entering the runner blade channels; the same phenomena were observed from the numerical data for the flow field. The incoming flow rotated inside the blade rows around a local orbiting axis in the channel just after the runner inlet. The flow

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

continuously accelerated and decelerated as the runner rotated or passed the guide vane; thus, the unsteady phenomena may not have allowed the flow to stabilize inside the blade channels.

4 CONCLUSIONS

The role of each component of this horizontal small Francis turbine and its effect on the hydrodynamic performance were investigated by simulating the single of turbulent flows considering SST turbulence model over the entire low passages. The predicted performance depicts that the nominal point corresponds to a maximum of hydraulic efficiency of 79.28% and an important drop with increased discharge. The details of flow structures show that the most of losses are located in the runner where there are large vortices affecting the stability of operation. The velocity at inlet of draft tube has a substantial circumferential component that initiates a precession motion of a vortex of helical shape. The unsteady flow phenomenon in turbomachinery invites many flow related challenges. To gain an insight into these phenomena and work outsolutions, a CFD based unsteady flow in a Francis turbine is conducted for optimal operating point, small flow rate and large flow rate, in complete flow passage using a robust turbulence model. Several flow features at different sections of the turbines are analyzed and their mutual effects are also correlated, to predict the outstanding flow parameters like velocity profiles, pressure distribution and interaction between the stationary and rotating components.

REFERENCES

Ansys (2017). Theory Guide, Ansys Release 16.0, 2017.

- Anup, K.C., Thapa, B. & Lee, Y.H. (2013). Transient Numerical Analysis of Rotor-Stator Interaction in a Francis Turbine". *Journal of Renewable Energy*, 65, 227-235.
- Brekke, H., (2010). A Review on Oscillatory Problems in Francis Turbine. *New Trends in Technologies: Devices, Computer, Communication and Industrial Systems*, M. J. Er, Ed., Sciyojanezatrdine, Rijeka, Croatia, 217–232 pp.
- Bucur, D., Dunca, G. & Caalinoiu, C. (2012). Experimental Vibration Level Analysis of a Francis Turbine. *IOP Conference Series: Earth and Environmental Science, 26th IAHR Symposium on Hydraulic Machinery and Systems, Iop Publishing*, Uk, P. 062056.
- Enomoto, Y., Kurosawa, S. & Kawajiri, H. (2012). Design Optimization of a High Specific Speed Francis Turbine Runner. *IOP Conference Series: Earth and Environmental Science, IOP Publishing*, UK, P. 032010.
- Gerolymos, G. A., Michon, G. J. & Neubauer, J. (2002). Analysis and Application of Chronic Periodicity in Turbomachinery Rotor/Stator Interaction Computations. *Journal of Propulsion and Power*, 18, 1139-1152.
- Hellstrom, J.G., Marjavaara, B.D. & Lundstrom, T.S., (2007). Parallel CFD Simulations of an Original and Redesigned Hydraulic Turbine Draft Tube. *Advances in Engineering Software*, 38(5), 338–344.
- Jacob, T. & Prenat, E. (1996). Francis Turbine Surge: Discussion and Data Base, XVIII IAHR Symposium, Kluwer, Dordrecht, 855-64 pp.
- Kumar, P. & Saini, R.P. (2010). Study of Cavitation in Hydro Turbines-A Review. *Renewable and Sustainable Energy Reviews*, 14(1), 374–383.
- Lucien, S. & Hakan, N. (2015). Steady and Unsteady Numerical Simulations of the Flow in the Tokke Francis Turbine Model at Three Operating Conditions. *Conference Series; Journal of Physics*, 579 012011.
- Paish, O. (2001). Micro-Hydropower: Status and Prospects. *Proceedings Institute of Mechanical Engineering Part A. Journal of Power Energy*, 216, 31-40.
- Sabourin, M., Labrecque, Y. & De Henau, D. (1996). From Components to Complete Turbine Numerical Simulation. *In Proceedings of the 18th IAHR Symposium on Hydraulic Machinery and Cavitation*, Valencia, Spain.
- Sun-Sheng, Y., Fan-Yu, K., Jian-Hui, F. & Ling, X. (2012). Numerical Research on Effects of Splitter Blades to the Influence of Pump as Turbine. *International Journal of Rotating Machinery*, 2012, 1-9.
- Susan-Resiga, R., Muntean, S., Hasmatsuchi, V., Anton, I. & Avellan, F. (2010). Analysis and Prevention of Vortex Breakdown in the Simplified Discharge Cone of a Francis Turbine. *Journal of Fluids Engineering*, 132(5), 1-15.
- Tushar, G.K. & Mark, P.A. (2011). Energy Resources and Systems, Renewable Resources. Netherlands, Springer.
- Tyler, J.M. & Sofrin, T.G. (1962). Axia L Flow Compressor Noise Studies, SAE Transaction, 70, 309-332.

Voith Siemens Francis Turbines, (2013). Hydropower Generation, Voith Turbines, Francis.

- Wu, J., Shimmei, K., Tani, K., Niikura, K. & Sato, J. (2007). CFD-Based Design Optimization for Hydro Turbines. *Journal of Fluids Engineering*, 129(2), 159–168.
- Widmer, C., Staubli, T. & Ledergerber, N. (2011). Unstable Characteristics and Rotating Stall in Turbine Brake Operation of Pump-Turbine. *Journal of Fluids Engineering*, 133(4), 101-109.
- Xiao, H. &Yu, B. (2010), 3D-Viscous Low Simulation and Performance Prediction of a Complete Model Francis Turbine, *In Proceedings of the International Conference on Mechanic Automation and Control Engineering (Mace '10)*, China. 3942–3944.

PERFORMANCE CHARACTERISTIC STUDY OF ENERGY-RECOVERY PELTON TURBINE WITH DIFFERENT SIZES OF CASING

YOUNG SOO KIM⁽¹⁾, JOO HOON PARK⁽²⁾, 2, YOUHWAN SHIN⁽³⁾ & JIN TAEK CHUNG⁽⁴⁾

^(1,3) Center for Urban Energy Research, Korea Institute of Science and Technology, Seongbuk-gu, Seoul, Korea cusia2002@kist.re.kr; yhshin@kist.re.kr
 ^(1,2)Graduate School of Korea University, Korea University, Seongbuk-gu, Seoul, Korea, joohoon82@korea.ac.kr
 ⁽⁴⁾ Department of Mechanical Engineering Korea University, Scongbuk, gu, Scoul, Korea

⁽⁴⁾ Department of Mechanical Engineering, Korea University, Seongbuk-gu, Seoul, Korea,

jchung@korea.ac.kr

ABSTRACT

The purpose of this study is to investigate performance characteristic changes of the Pelton turbine according to the modification of casing design. The Pelton turbine is widely used as the energy recovery device at the generating system of Pressure Retarded Osmosis (PRO). The casing design of the Pelton turbine affects the performance of the Pelton turbine by changing the splash water distribution that occurs after the water leaves the bucket. Therefore, this study investigates the performance characteristics of the turbine with different casing designs of the Pelton turbine. In this experimentation, laboratory scale equipment of Pelton turbine is manufactured for the performance test. In order to change the shape of the casing, the circular structures are produced with different widths. Through the experimental performance test, the efficiency of the Pelton turbine according to the speed ratio is calculated at the same operating condition (278 LPM, 30 bar) for each shape. As a result, it has been confirmed that the 40 mm width of casing design shows the least windage losses at design conditions. Afterwards, this study can be utilized in the various casing design of the energy recovery turbine.

Keywords: Pelton turbine; energy recovery turbine; pressure retarded osmosis (PRO); impulse turbine; casing; speed ratio.

1 INTRODUCTION

Currently, the global energy environment is rapidly changing. Coal fuel for power generation is no longer permanent and global warming has attracted serious interest in renewable energy. The energy independence will impact on each national economy in the future, therefore development of clean energy and sustainable energy becomes an important consideration. Seawater power generation is a development system that is well suited to this interest.

Pressure-retarded osmosis (PRO) is a power generation process using osmotic energy. A draw solution with higher pressure was produced by passing the feed solution through the semipermeable membrane. The operating conditions of the PRO system are suitable for using the Pelton turbine as the energy recovery device (ERD) because it has operation range with high pressure and low flow rate.

Figure 1 shows a schematic of PRO system. The energy of draw solution formed by semipermeable membrane converts to the electrical energy through the Pelton turbine used as ERD. The geometric design of the Pelton turbine have to be considered and this includes the diameter of the runner, the nozzle diameter, the bucket shape and so on.

Until now, researches on the casing and the wiper of turbine have been in comparison with researches of bucket design and jet shape. Therefore, in this paper, the performance characteristics of the turbine according to the changes of the wiper position and the casing width were investigated.

2 PERFORMANCE TEST

Figure 2 is a lab scale performance test apparatus to simulate the operating condition of PRO system. The centrifugal pump was used to implement the concentrated water passing through the membranes on the experimental apparatus. The high-pressure fluid generated by this pump passes through the nozzle and inject into the buckets.

An electronic flow meter was installed between the nozzle and the pump to measure the flow rate before the fluid entering the nozzle. A pressure transducer on the bended pipe was installed to measure the pressure of the fluid entering the nozzle. A servo motor was also placed to control the experiment under various rotating speed and torque required in the experiments. Servo motor and a peripheral device are used to control the rotating speed up to 5000 rpm and maximum torque up to 117 Nm.



Figure 1. Schematic of PRO system.



Figure 2. Performance test apparatus of Pelton turbine.

The pitch circle diameter of runner is 182 mm and 25 buckets were used for this study. This runner used to convert the pressure energy of the jet from the nozzle into kinetic energy through the runner bucket. During the runner rotation, the shaft torque occurred at the runner was measured through a torque transducer connected to the shaft. The measured data from the flowmeter and pressure transducer are sent to the PC by using the data acquisition unit 1 (MX100). The data from the torque transducer are sent to in the PC through the data acquisition unit 2 (TM301). Table 1 shows the accuracy of the data acquisition device used in the performance test and the measurement output range.

Figure 4 is the shape of the spear nozzle used in this study. One spear nozzle that consist of 55° nozzle and 80° spear was produced for the experiment. The performance test was prepared to validate the effect of the wiper. Two case of wiper with casing and no wiper with casing were prepared. The wiper design was enlarged by 1 mm difference from the design of the bucket. The width of the casing was changed from 40 mm to 70 mm by using the wiper which showed the best efficiency depending on the position of the wiper and the radius of the casing is 109 mm In order to manufacture various casings, it is possible to make the shape of a desired casing by making it into a plate shape as shown in the figure 5.

Table 1. Measurement devices.								
Device	Pressure Transducer	Flowmeter	Data Acquisition Unit 1	Data Acquisition Unit 2	Torque Transducer			
Range	0~1167 LPM	0~40 bar	±100V	±200Nm 0~12,000RPM	±5VDC			
Accuracy Output range	±0.2% 4~20mA	±0.15% 1~5 VDC	0.05%	0.03% R.O. ±5VDC	0.02% FS			



Figure 3. Runner and bucket of Pelton turbine.



Figure 4. Spear nozzle.



Figure 5. Casing design of Pelton turbine.

3 RESULTS AND DISCUSSIONS



Figure 6. Performance curve of Pelton Turbine.

The corresponding performance curves in Figure 6 shows a correlation between the flow rate and pressure variation at constant nozzle opening. The flow rate and pressure showed a typical linear correlation ©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print) 3189

at constant opening position. The flow rate increase according to the square root of pressure as the theory. As designed, it was confirmed that the maximum efficiency region appeared around operation condition (278 LPM, 30 bar).



Figure 6. Speed ratio vs. Pelton turbine efficiency by the effect of wiper.



Figure 7. Turbine efficiency with casing width (wiper, no wiper) at 3600rpm.



Figure 8. Efficiency difference between wiper and no wiper case.

©2017, IAHR. Used with permission / ISSN 1562-6865 (Online) - ISSN 1063-7710 (Print)

In order to measure the turbine efficiency in relation to the wiper, the relationship between the speed ratio and the turbine efficiency is shown in the diagram under the operation condition of 278 LPM, 30 bar. Figure 6 and Figure 7 above shows the efficiency of the Pelton turbine according to the speed ratio with wiper and no wiper. And the speed ratio can be expressed as a ratio of the jet speed to the tangential speed of the runner.



Figure 9. Speed ratio vs. Pelton turbine efficiency for width change of casing.



Figure 10. Turbine efficiency with casing width (40mm, 70mm) at 3600rpm.

$$x = \frac{v_{runner}}{v_{jet}} = \frac{PCD \cdot \pi \cdot \frac{N}{60}}{C_v \cdot \sqrt{2 \cdot g \cdot h}}$$
[1]

Equation (1). C_v , the velocity coefficient of the spear nozzle is between 0.95 and 0.99. (Thake, J., 2000; Zhang, Z. 2016) The C_v value of 0.97 was used in this study. Theoretically optimal speed ratio is 0.5 but experiment result showed a different value of 0.46 due the friction of windage losses

Figure 8 shows the efficiency difference between full wiper and no wiper case according to the speed ratio. After increasing the efficiency difference from the low speed ratio to the operation range, the efficiency difference decreased as the speed ratio increased. Previous study (Matthias, H. B., 1997) mentioned that the direction of flow from the bucket changes as the speed of the runner increases. The amount of entrained water in the upper casing section is the highest during the operation range. The efficiency difference is the maximum in this section because the wiper can prevent entrained water in the upper casing. Above the optimum range, portion of the flow will deviate from the energy transformation and the amount of flow into the upper casing is reduced. This phenomenon influenced the efficiency difference between the wiper case and the no wiper case. In the figure 8, the wiper case shows a rise in speed ratio of about 2% over no wiper.

Equation (2) can be used to obtain the rated speed of the generator through the number of poles and frequency.

$$N = \frac{120 \cdot f}{n_p}$$
[2]

The efficiency at 3600 rpm was measured since the PRO pilot plant will use a generator with a rated speed of 3600 rpm. The case of full wiper and no wiper showed a difference of efficiency of 1.4% at 3600rpm in figure 8. The wiper is used to prevent the splash water from falling back to the runner. Some water out of the bucket can move up to the upper casing and fall into the tailrace. These splash waters degrades the turbine efficiency. Figure 9 shows the turbine efficiency according to the speed ratio of the circular casing with width of 40 mm and 70 mm with wiper installed. A wider casing showed higher turbine efficiency as the speed ratio increases. The case of 40 mm case and 70 mm showed a difference of efficiency of 1.9% at 3600rpm in Figure 10. As the width increases, effect of the air turbulence in the upper part of the casing were reduced, therefore windage losses in the casing increases. For this reason, as the width of the casing increases, the efficiency of the turbine decreases.

5 CONCLUSIONS

In this study, a performance test was conducted on a Pelton Turbine according to the design of casing. Design of casing for one jet Pelton Turbine is carried out by considering the hydraulic and windage losses. The wiper can prevent the splash water from falling into the runner. The experiment shows about 2% efficiency difference depending on whether wiper is used or not. As the width of casing increases, the windage losses at the upper part of casing also increases. The wider casing result in lower turbine efficiency. In order to find the optimal case to suit the operating conditions, additional experiments on cases with various changes are needed.

ACKNOWLEGDGEMENT

This research was supported by a grant (code 17IFIP-B065893-05) from Industrial Facilities & Infrastructure Research Program funded by Ministry of Land, Infrastructure and Transport of Korean government.

REFERENCES

Matthias, H.B., Prost, J. & Rossegger, C. (1997). Investigation of the Flow in Pelton Turbines and the Influence of the Casing. *International Journal of Rotating Machinery*, 3(4), 239-247

Thake, J. (2000). The Micro-Hydro Pelton Turbine Manual: Design, Manufacture and Installation for Smallscale Hydropower (No. 621.24/T364). ITDG publishing.

Zhang, Z. (2016). *Friction and Windage Losses in Pelton Wheels*. In Pelton Turbines, Springer International Publishing, 195-202.