Proceedings of 2013 IAHR Congress

Analysis of the Scale Ratio in Nappe Flow Case by Means of CFD Numerical Simulation

Luis Gerardo Castillo,
Professor, Dept. of Civil Engineering ,Universidad Politécnia de Cartagena, Spain. E-mail: luis.castillo@upct.es
José María Carrillo,
PhD Student, Dept. of Civil Engineering ,Universidad Politécnia de Cartagena, Spain. E-mail: jose.carrillo@upct.es

ABSTRACT: Rectangular jet or nappe flow constitutes one of the energy dissipation methods in arch dams. The high turbulence and aeration phenomena that appear in fall jets and dissipation basins make difficult to carry out their characterization only based in classical methodologies: reduced models and empirical formulae.
The Hydraulics Laboratory of the Universidad Politécnica de Cartagena (Spain) has an infrastructure designed specifically for the study of turbulent jets and energy dissipation in plunge pools. The device allows us to study air-water two-phase phenomena (aeration, spray, spread and impact). The mobile mechanism lets us to vary the discharge heights between 1.70 and 4.00 m and flows between 10 and 150 l/s. To improve the knowledge of the phenomenon of turbulent jets, we are measuring aeration rates by means of fiber optical equipment, velocities in different sections of the stilling basin with Doppler instrumentations and pressures on the bottom of the plunge pool with piezoresistive transducers.
Computational Fluid Dynamics programs (CFD) are based on numerical solution of the Reynolds Averaged Navier-Stokes (RANS) equations, together with turbulence models of different degrees of complexity. The programs simulates the interaction between different fluids, such as the air-water two-phase flows, and constitute a new and powerful tool that could let contrast and complement the lab measurement.
There are many studies modeling spillways with CFD methodology using different eddy viscosity turbulence models with accurate results. However, the study of overflow nappe impingement jets has not been sufficiently studied.
Scale effects are important effects that need to be taking account. In this study, numerical simulations has been carry out using different scales ratio of the lab device according to Froude similarity (1:1; 1:10; 1:20 and 1:40).
This paper compares the Parametric Methodology proposed by Castillo (2006, 2007) for the evaluation of hydrodynamic action in plunge pools, revised by Castillo and Carrillo (2011, 2012), with more and new laboratory measurements and the simulation results obtained with CFD program.

KEY WORDS: Nappe flow, Plunge pools, Energy dissipation, Froude similarity, CFD.

1 INTRODUCTION

In recent years, the increase magnitude of design floods has made necessary to carry out a reevaluation of spillways capacity. In some cases, dam owners can accept overtopping of a dam during extreme events (Wahl, et al, 2008).

Rectangular jet or nappe flow constitutes one of the most usual overtopping flow in concrete dams.

This flows often raises issues of potential erosion and scour downstream from the dam, where the rectangular jet impacts with the cushion water in the downstream river channel. To evaluate the need for protection of these areas, a comparison of the potential hydraulic attack and erosion resistance of these materials is needed (Annandale, 2006).

The selection of the plunge pool depth is usually a technical and economic decision between a deep pool which need not lining, or a shallow pool which needs a lining. Therefore, a designer needs to know the magnitude, frequency and extent of the dynamic pressure on the pool floor as a function of the jet characteristics.

The characterization of pressures in plunge pools has been obtained using different scale models: Moore (1943), Lencastre (1961), Ervine and Falvey (1987), Withers (1991), Ervine et al. (1997), Bollaert (2002), Bollaert and Schleiss (2003), Manso et al. (2005) and Federspiel (2011).

In Spain these line of research has been undertaken at Universidad Politécnica de Cataluña by Castillo (1989, 1998), Armengou (1991), Puertas (1994), Castillo et al. (1991, 1999) and at Universidad Politécnica de Cartagena by Castillo (2002, 2006, 2007, 2009, 2010, 2011, 2013).

The principal mechanism of energy dissipation are the spreading of the plunging jet (aeration and atomization in the air), air entrainment by the entering jet and diffusion in the pool and, finally, the impact with the pool base. For design considerations, we define both the issuance conditions and the impingement conditions (Figure 1).

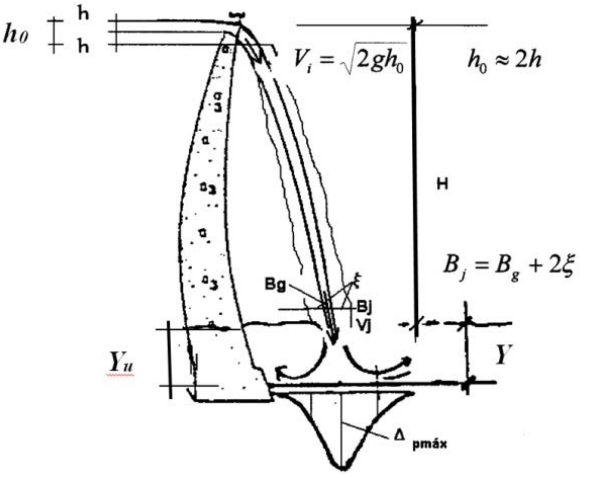


Figure 1 Issuance and impingement conditions in a falling jet

The issuance conditions, located at the exit of the spillway structure, are defined by the mean velocity $V_i = (2gh_0)^{1/2}$, where h_0 is approximately equal twice times the energy head, h.

The principal impingement conditions situated at entrance to the pool are the mean velocity, V_j, and the impingement jet thickness, $B_j = B_g + \xi$, in where B_g is the thickness by gravity conditions and ξ is the jet lateral spread distance by turbulence effect and is approximately equal to the square root of the fall distance (Davies, 1972), and on the other hand, the jet thickness decreases due to gravity effect.

Another important parameter is the jet break-up length, L_b. Beyond this distance the jet is completely developed (disintegrated). It no longer contains a core but consists essentially of blobs of water that disintegrate into finer and finer drops. For flows smaller than 0.25 m^2/s (laboratory tests values), the Horeni´s formulae $L_b \sim 6q^{0.32}$ (cited by Ervine et al., 1997) seems to be correct (Castillo, 2006).

For the nappe flow case, Castillo (2006, 2007) proposed some estimators of the turbulence intensity at issuance conditions (T_u^*), jet break-up length (L_b), lateral spread distance (ξ), impingement thickness (B_j) and the mean dynamic pressure coefficient (C_p). Table 1 resumes these formulations together with the jet trajectory ($y=f(x)$) and the pool depth under nappe in the basin (Y_u).

Table 1 Parametric formulations in nappe flow case

<table>
<tr><th>AUTOR</th><th>FORMULAE</th><th>OBSERVATIONS</th></tr>
<tr>
<td>Castillo (2006) and new adjustment proposed in this paper.</td>
<td>
Issuance conditions:

$$T_u^* = q^{0.43} / IC$$

$$IC = 14.95 g^{0.50} / (K^{1.22} C_d^{0.19})$$

$$\frac{L_b}{B_i F_i^2} = \frac{K}{(k_\varphi T_u^* F_i^2)^{0.82}}$$

Impingement conditions:

$$\xi = 2\varphi\sqrt{h}\left[\sqrt{2H} - 2\sqrt{h}\right]$$

$$B_j = \frac{q}{\sqrt{2gH}} + 4\varphi\sqrt{h}\left[\sqrt{2H} - 2\sqrt{h}\right]$$

Mean dynamics pressure (C_p):
No effective cushion, $Y/B_j \le 5.5$:
Undeveloped jet, $H/L_b \le 1.05$

$$C_p = 1 - 0.4089 (H/L_b)^{4.7958}$$

Developed jet, $H/L_b > 1.05$

$$C_p = 0.6088 (H/L_b)^{-3.803}$$

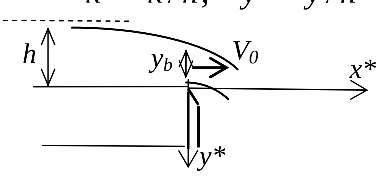
Effective cushion, $Y/B_j > 5.5$:

$$C_p = \frac{H_m - Y}{V_j^2/2g} = a e^{-b(Y/B_j)}$$
</td>
<td>
T_u^*, Turbulent intensity issuance conditions (q<0.25 m^2/s)
q, Specific flow
IC, Initial conditions
$g \approx 9.8$ m/s^2, Gravity acceleration
K, Non-dimensional fit coefficient ($\approx$0.85)
C_d, Discharge coefficient
L_b, Jet break-up length. q<0.25m^2/s, $L_b \approx 6q^{0.32}$
B_i, Issuance jet thickness
Fi, Issuance conditions Froude Number
B_j, Impingement jet thickness
H, Height between upstream water level and downstream water level
$\varphi = k_\varphi T_u = 1.24 T_u^*$, Turbulence parameter q<0.25 m^2/s
$h_0 \approx 2h$, Level of issuance conditions
H_m, Head mean registered at plunge pool bottom
Y, Depth at plunge pool
V_j, Impingement velocity

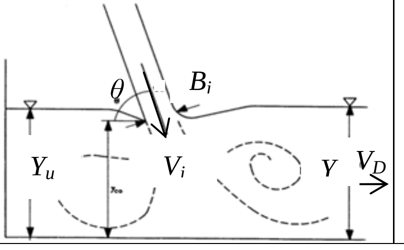
<table>
<tr><th>H/L_b</th><th>a</th><th>b</th><th>C_p (Y/B_j <= 5.5)</th></tr>
<tr><td>< 0.85</td><td>2.50</td><td>-0.20</td><td>0.832</td></tr>
<tr><td>0.85-0.89</td><td>2.35</td><td>-0.20</td><td>0.782</td></tr>
<tr><td>0.90-0.99</td><td>1.94</td><td>-0.20</td><td>0.646</td></tr>
<tr><td>1.00-1.09</td><td>1.54</td><td>-0.20</td><td>0.513</td></tr>
<tr><td>1.10-1.24</td><td>1.15</td><td>-0.20</td><td>0.383</td></tr>
<tr><td>1.25-1.29</td><td>0.85</td><td>-0.20</td><td>0.283</td></tr>
<tr><td>1.30-1.39</td><td>0.55</td><td>-0.20</td><td>0.183</td></tr>
<tr><td>1.40-1.60</td><td>0.35</td><td>-0.20</td><td>0.117</td></tr>
<tr><td>> 1.60</td><td>0.24</td><td>-0.20</td><td>0.080</td></tr>
</table>
</td>
</tr>
<tr>
<td>Scimeni (1930)</td>
<td>
Trajectory of jet central nappe:

$$x^* = [2.155 (y^* + 1)^{1/2.33} - 1]$$

$$x^* = x/h; \quad y^* = y/h$$



</td>
<td>
x, y, Coordinates axis
h, Total head
$V_0 = \alpha V = \alpha (q/y_b)$, Jet initial velocity on weir crest
α, Coriolis coefficient
$y_b \approx 0.85h$ Jet depth on weir crest
</td>
</tr>
</table>

AUTOR	FORMULAE	OBSERVATIONS
Cui Guang Tao et al. (1985), Castillo (1989)	Pool depth under nappe $Y_u/Y=\sqrt{1-2F_D[(V_j/V_D)\beta\cos\theta-1]}$ 	Y_u, Pool depth under nappe Y, Water cushion $F_D=V_D^2/(gY)$, Square Froude number V_j, Impingement velocity V_D, Downstream velocity $\beta_0=1-\beta$, Head loss coefficient $\beta_0\approx0.40$ Bidimensional jet $\beta_0\approx0.45$ Tridimensional jet θ, Impingement jet angle

2 PHYSICAL DEVICE

The device of turbulent jets and energy dissipation in the nappe flow case (Figure 2) allows us to study air-water two-phase phenomena (aeration, spray, spread and impact).

The mobile mechanism lets us to vary the discharge heights between 1.70 and 4.00 m and flows between 10 and 150 l/s. The plunge pool is a methacrylate´s box (1.60 m height and 1.05 m wide) in which different water cushions can be regulated. Instantaneous pressure measurements are registered with piezoresistive transducers located on plunge pool bottom, instantaneous velocities with ADV equipment and mean velocities and aeration rates with fiber optical instrumentation.

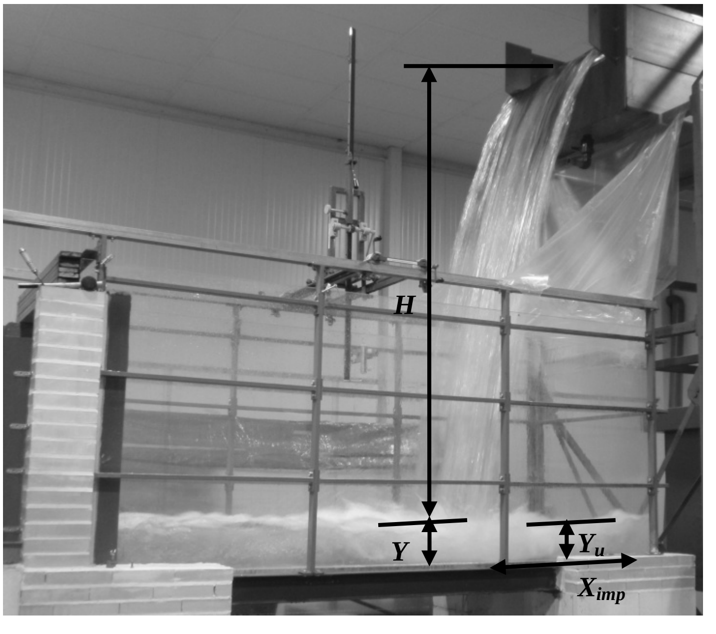


Figure 2 Device of turbulent jets

3 SCALE MODELS

Lab measurements have been carried out in a physic structure of relatively small size. To extrapolate the results to prototype dimension, a similarity law is necessary.

The similarity law have to conserve the relative values of the different physical characteristics that have influence in the fluid movement (gravity, pressure, density, viscosity, elasticity, superficial stress). The choice of the most interest variables will determine not only the shape of the structure, but also his utilization limits and his reliability and credibility.

The Π Theorem of Buckingham consider different adimensional numbers that related the physical characteristics with the inertial forces. In this way, we have:

- Newton Number (*Ne*): ratio between pressure forces and inertial forces.
- Froude Number (*Fr*): ratio between inertial forces and gravitational forces.
- Reynolds Number (*Re*): ratio between inertial forces and viscosity forces.
- Weber Number (*We*): ratio between inertial forces and superficial stress forces.
- Cauchy or Mach Number (*Ca*): ratio between inertial forces and elastic forces.

In order to physical model results can be considered as representatives, they have to satisfy similarity laws. There are three different similarities:

- Geometric (k_l). The ratio between model and prototype must be constant for any of their geometric characteristics.
- Cinematic (k_t). Adding to the geometric rate a time scale, allows to relate velocities and accelerations between model and prototype.
- Dynamic (k_F). Enlarge the previous ones with force scales.

If there is geometric, cinematic and dynamic similarities, we have mechanical similarity. The forces over them, acting in homogeneous points will be proportional:

$$k_{Ne} = k_{Fr} = k_{Re} = k_{We} = k_{Ca} = 1 \tag{1}$$

It is not possible to satisfy all the similarities at the same time. However, is not necessary to comply all the equalities if we consider that not all actuating forces have the same importance.

In free falling jets, authors consider that some forces have minor effects:

- The water compressibility have not importance.
- Viscosity tends to be neglected due to the turbulence is enough to not consider viscous stress.

As far as aeration is concerned, there is an important scale effect. This is a problem difficult to solve. It is necessary to accept that model pressures will be bigger than prototype ones. In this way, an experiment that not consider aeration (viscosity and superficial stress are neglected) will be result on the safety side.

Accepting that the aeration is an scale effect that we cannot solve, a Froude similarity law have been used (eulerian models similarity law). In this conditions, the pressures due to the impact of a jet whose velocity have been acquired by gravitational effects maintain a proportional ratio. Scale ratios for different variables have been compiled in Table 2.

Table 2 Froude scale ratios for different variables

		Scale ratio			
Variable	Scale factor	1:1	1:10	1:20	1:40
Length	k_l	1	10	20	40
Time	$\sqrt{k_l}$	1	3.16	4.47	6.32
Velocity	$\sqrt{k_l}$	1	3.16	4.47	6.32
Acceleration	1	1	1	1	1
Flow	$k_l^{5/2}$	1	316.23	1788.85	10119.29
Force	k_l^{3}	1	1000	8000	64000
Pressure	k_l	1	10	20	40

4 NUMERICAL MODELING

Computational Fluid Dynamics (CFD) programs offer the possibility to investigate physical fluid systems, providing lots of data, increased profitability, flexibility and speed than that obtained with experimental procedures. However, to a correct use, it is necessary to contrast and to calibrate with data obtained in prototypes and/or in physical models.

In this paper, CFD methodology is applied to the investigation of flows highly aerated and turbulent, using ANSYS CFX v12.1 program (2009). This program, based on finite volume scheme, solves the differential equations of the phenomenon in control volumes defined by the meshing of the fluid domain, retaining the reference quantity (mass, momentum, energy) in the three directions for each control volume identified.

To complement the numerical solution of Reynolds equations and average Navier-Stokes (RANS), turbulence models has been used. There are many turbulent models of diverse complexity, from the isotropic models of two-equation like the classic k-ε to the second moment closure models (SMC) like the Reynolds Stress Model.

Two equations models have been widely applied in the solution of many flows of engineering interest. The k-ε (k-epsilon) model, has been implemented in most general purpose CFD codes and is considered the industry standard model, but may not be suitable to solve flows with boundary layer separation. The k-ω based models try to give a more accurate predictions of these types flows. For this reason, we have used the k-ω based Shear-Stress-Transport (SST) model.

All scenarios have been modeled by a transient calculation time. To obtain the results, we have used a 20 Hz frequency, the same as used in the lab pressure measurements. In Figure 3 we can observe that permanent conditions are reached after 20 seconds of simulation when 1:1 scale model is considered.

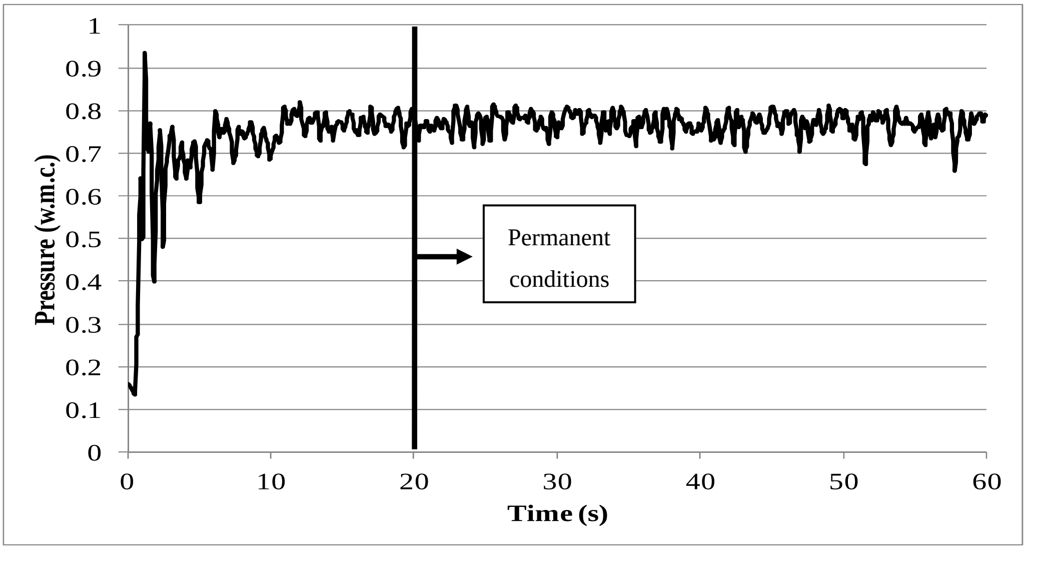


Figure 3 Transient of total pressure at the stagnation point of the plunge pool

To solve the two-phase air-water we used the homogeneous model. It can be considered as a limit case of the inhomogeneous model, in which the transfer rate at the interface is very great. A common flow field is shared by all fluids, remained valid in flows domain by gravity when the phases are completely stratified (case of a free surface flow in which the interface is well defined).

In the study of turbulent jets, the flow separation and high turbulence exist. They need high quality mesh elements in order to solve the problem with the highest accuracy. The mean difference among the different mesh elements is the number of nodes and their distribution. In this way, more node number drove to obtain better results.

We have used hexahedral mesh elements, with length scale of 0.01 m in the falling jet boundary and at stagnation point for scale model 1:1. For the other scales models, the mesh length scale has been changed according to the correspondent k_l scale factor.

The model boundary conditions correspond to the flow and turbulence inlet, upstream and downstream levels and their hydrostatic pressures distributions, taking into account the scale factors.

Due to the high air entrainment in the jet, a ratio of 0.8 of Air Volume Fraction have been used in order to obtain the free surface in the jet and in the stilling basin. Figure 4 shows the mesh size and the free surface obtained with the CFD program when permanent conditions are reached. We can see that the jet profile is very similar to the lab jet shows in the description of the lab device.

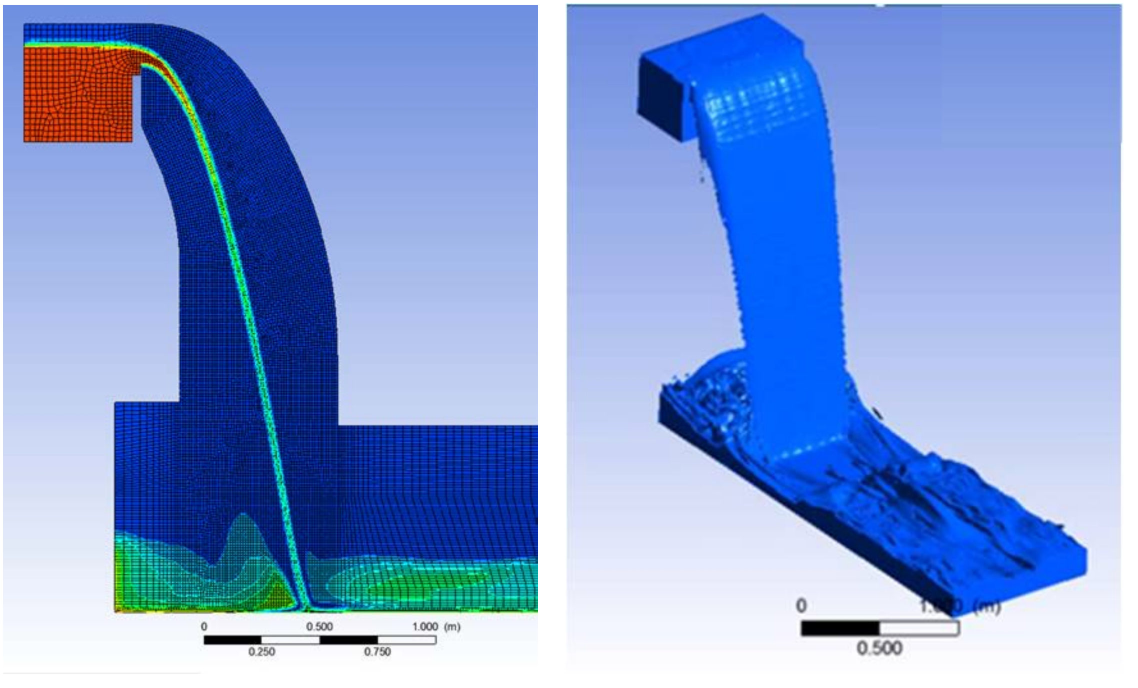


Figure 4 Mesh size and free surface of turbulent jet, for 1:1 scale model (q = 0.058m^2/s, H = 2.28m, h = 0.091m, Y = 0.17m)

5 RESULTS AND DISCUSSION

Tables 3 to 6 compare the results obtained with the three methodologies (CFD simulation, scaled lab results and Parametric Methodology), considering different water cushions. The contour conditions are q, H, h and Y. This parameters have been showed in order to know their value when the different scales are considered.

Table 3 Comparison of the principal measurement and calculated variables (lab scale q = 0.058m^2/s, Y = 0.03 m)

	Scale Factor											
	1:1			1:10			1:20			1:40		
Variable	CFD	Lab	Param	CFD	Lab*	Param	CFD	Lab*	Param	CFD	Lab*	Param
q (m^2/s)	0.058	0.058	0.058	1.83	1.83	1.83	5.19	5.19	5.19	14.63	14.63	14.63
H (m)	2.41	2.41	2.41	24.11	24.11	24.11	48.22	48.22	48.22	96.44	96.44	96.44
h (m)	0.091	0.091	0.091	0.91	0.91	0.91	1.82	1.82	1.82	3.64	3.64	3.64
Y (m)	0.03	0.03	0.03	0.30	0.30	0.30	0.60	0.60	0.60	1.20	1.20	1.20
L_b (m)	$> H$	-	2.32	$> H$	-	23.24	$> H$	-	46.49	$> H$	-	92.98
H_m (w.c.m.)	1.13	1.26	1.27	11.14	12.60	12.66	21.62	25.20	25.32	43.67	50.40	50.65
C_p	0.46	0.51	0.51	0.45	0.51	0.51	0.44	0.51	0.51	0.44	0.51	0.51

* Using scale ratios

Table 4 Comparison of the principal measurement and calculated variables (lab scale q = 0.058m^2/s, Y = 0.165 m)

	Scale Factor											
	1:1			1:10			1:20			1:40		
Variable	CFD	Lab	Param	CFD	Lab*	Param	CFD	Lab*	Param	CFD	Lab*	Param
q (m^2/s)	0.058	0.058	0.058	1.83	1.83	1.83	5.19	5.19	5.19	14.63	14.63	14.63
H (m)	2.28	2.28	2.28	22.76	22.76	22.76	45.52	45.52	45.52	91.04	91.04	91.04
h (m)	0.091	0.091	0.091	0.91	0.91	0.91	1.82	1.82	1.82	3.64	3.64	3.64
Y (m)	0.165	0.165	0.165	1.65	1.65	1.65	3.30	3.30	3.30	6.60	6.60	6.60
L_b (m)	> H	-	2.32	> H	-	23.24	> H	-	46.49	> H	-	92.98
H_m (w.c.m.)	1.07	1.15	1.20	10.89	11.50	11.38	21.86	23.00	22.77	42.12	46.00	45.53
C_p	0.40	0.43	0.46	0.41	0.43	0.43	0.41	0.43	0.43	0.39	0.43	0.43

* Using scale ratios

Table 5 Comparison of the principal measurement and calculated variables (lab scale q = 0.058m^2/s, Y = 0.25 m)

	Scale Factor											
	1:1			1:10			1:20			1:40		
Variable	CFD	Lab	Param	CFD	Lab*	Param	CFD	Lab*	Param	CFD	Lab*	Param
q (m^2/s)	0.058	0.058	0.058	1.83	1.83	1.83	5.19	5.19	5.19	14.63	14.63	14.63
H (m)	2.19	2.19	2.19	22.11	22.11	22.11	44.22	44.22	44.22	88.44	88.44	88.44
h (m)	0.091	0.091	0.091	0.91	0.91	0.91	1.82	1.82	1.82	3.64	3.64	3.64
Y (m)	0.25	0.25	0.25	2.50	2.50	2.50	5.00	5.00	5.00	10.00	10.00	10.00
L_b (m)	> H	-	2.32	> H	-	23.24	> H	-	46.49	> H	-	92.98
H_m (w.c.m.)	0.88	0.73	0.72	9.51	7.30	6.72	18.90	14.60	13.44	37.94	29.20	26.87
C_p	0.29	0.22	0.21	0.32	0.22	0.19	0.32	0.22	0.19	0.32	0.22	0.19

* Using scale ratios

Table 6 Comparison of the principal measurement and calculated variables (lab scale q = 0.058m^2/s, Y = 0.35 m)

	Scale Factor											
	1:1			1:10			1:20			1:40		
Variable	CFD	Lab	Param	CFD	Lab*	Param	CFD	Lab*	Param	CFD	Lab*	Param
q (m^2/s)	0.058	0.058	0.058	1.83	1.83	1.83	5.19	5.19	5.19	14.63	14.63	14.63
H (m)	2.11	2.11	2.11	21.11	21.11	21.11	42.22	42.22	42.22	84.44	84.44	84.44
h (m)	0.091	0.091	0.091	0.91	0.91	0.91	1.82	1.82	1.82	3.64	3.64	3.64
Y (m)	0.35	0.35	0.35	3.50	3.50	3.50	7.00	7.00	7.00	14.00	14.00	14.00
L_b (m)	> H	-	2.32	> H	-	23.24	> H	-	46.49	> H	-	92.98
H_m (w.c.m.)	0.56	0.55	0.56	5.70	5.50	5.37	11.13	11.00	10.75	22.51	22.00	21.50
C_p	0.10	0.10	0.10	0.11	0.10	0.09	0.10	0.10	0.09	0.10	0.10	0.09

* Using scale ratios

The jet break-up length, L_b, is not well definite with CFD simulations. The Eulerian-Eulerian approach used cannot follow the disintegration of the jet into drops. This results mach with the studies of Ho and Riddette (2010), whose consider limited progress to date due to L_b requires a very fine mesh resolution.

C_P is a non-dimensional parameter. For this reason, the different lab scales does not do any effect on it. ANSYS CFX, due to the mesh size have been scaled with the same Froude similitude, tends to scale all the parameters following the similitude law. The Parametric Methodology shows similar values on the head mean registered at the stagnation point, H_m, and on the mean dynamic pressure coefficient, C_P.

Figures 5 to 8 show the impingement jet velocity V_j, mean dynamic pressure coefficient C_P, horizontal distance to the stagnation point X_{imp} and pool depth under nappe Y_u, when different scales are considered. These values are classified like simulated results (CFD), lab measurements (LAB), Parametric Methodology (Param), and the contour conditions and geometry model scaled (LAB scaled).

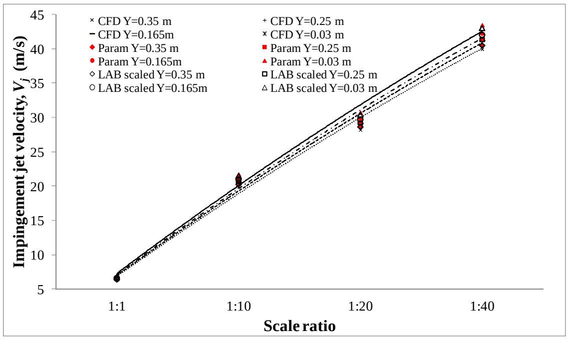


Figure 5 Impingement jet velocity, V_j

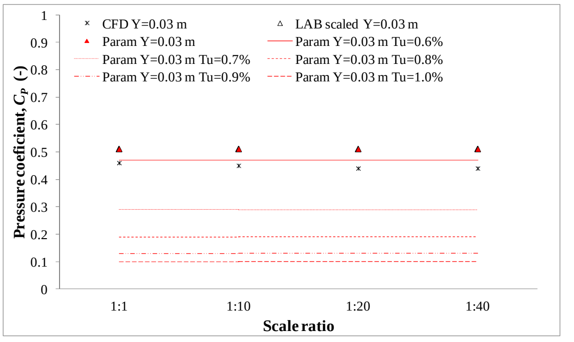


Figure 6 Pressure coefficient C_p. Case Y/B_j=1.30

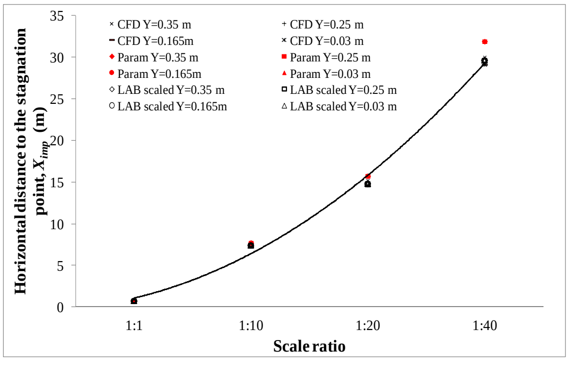


Figure 7 Horizontal distance to the stagnation point, X_{imp}

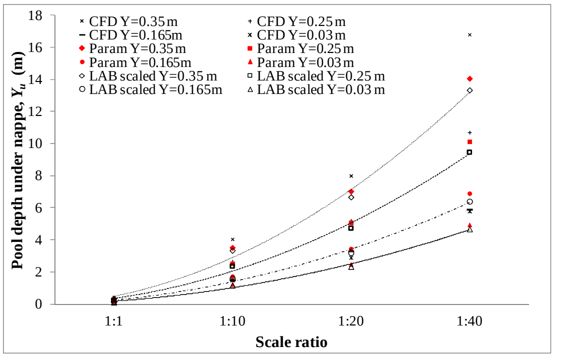


Figure 8 Pool depth under nappe, Y_u

For the impingement velocity, the Parametric Methodology considerer the free falling velocity without air deceleration. The lab measurements have been obtained with optical fiber, so the velocity is a bit smaller. The velocities with ANSYS are a little smaller than the lab measurements.

Figure 6 shows the C_p values obtained in the case of direct impact (Y/B_j=1.30). The CFD values are smaller than LAB scaled and Parametric Methodology. By the way, we can observe that issuance turbulence conditions have great effect in the C_p results. According to their real value in the prototype, the differences can be very important: if T_u=0.6 % then C_p=0.47, but if T_u=1.0 % then C_p=0.10.

We can see in Figure 7 that the Parametric methodology obtains bigger distances to the stagnation point X_{imp}, while ANSYS obtains values close to the lab scales. This results are according to the V_j values.

Finally, Figure 8 shows the pool depth under nappe. It is really difficult to estimate Y_u with numerical simulations due to the high mix air-water that appear in this region. For this reason, the values of CFD are more dispersed.

6 CONCLUSIONS

This study compares the results obtained in free rectangular jets using three methodologies (LAB, Parametric Methodology and CFD simulations) when different ratio scale are considered.

The Parametric Methodology used in this paper is based only on the results of measurements of instantaneous pressures at the bottom of the stilling basin. The C_p formulation has been improved in this paper with more experimental studies of pressures and velocities.

The laboratory results allow us to validate the CFD programs. This studies need a really fine mesh in order to characterize the jet break-up. A compromise between computational time and mesh size has been used. Good results in the majority of the parameters have been obtained. However, the simulation of the air-water interaction is not completely satisfactory.

We appoint to that the estimation of issuance conditions of the jet (turbulence intensity, T_u) is essential in the characterization of the hydrodynamic actions of the energy dissipators. We suggest to calculate the turbulence intensity in model scale (for example 1:40) with the Parametric Methodology and to maintain this value in prototype.

ACKNOWLEDGEMENT

The study is supported by the Ministerio de Economía y Competitividad and the FEDER through the project “Natural Aeration of Dam Overtopping Free Jet Flows and its Difussion On Dissipation Energy Basins“ (BIA2011-28756-C03-02).

References

Annandale, G.W., 2006. Scour Technology. McGraw-Hill, N.Y, USA.

ANSYS, Inc., 2010. ANSYS CFX. Reference Guide. Release 13.0.

Armengou, J., 1991. Vertido libre por coronación presas bóveda. Análisis del campo de presiones en cuenco amortiguador. PhD Thesis. Universidad Politécnica de Cataluña, España (in Spanish).

Bollaert, E. and Schleiss, A., 2003. Scour of rock due to the impact of plunging high velocity jets Part I: A state-ofthe-art review. Journal of Hydraulic Research, 41 (5), 451-464.

Castillo, L., 1989. Metodología experimental y numérica para la caracterización del campo de presiones en los disipadores de energía hidráulica. Aplicación al vertido libre en presas bóveda. PhD Thesis. Universidad Politécnica de Cataluña, España (in Spanish).

Castillo,L., Puertas,J. and Dolz,J., 1999. Discussion about pressure fluctuations on plunge pool floors. Journal of Hydraulic Research, .37 (2), 272-788.

Castillo, L., 2002. Parametrical analysis of the ultimate scour and mean dynamic pressures at plunge pools. Proceedings École Polytechnique Fédérale de Lausanne, Switzerland. Schleiss & Bollaert (eds). A.A. Balkema.

Castillo, L., 2006. Areated jets and pressure fluctuation in plunge pools. Proceeding The 7th International Conference on Hydroscience and Engineering (ICHE-2006), IAHR, ASCE, Drexel University. College of Engineering. DSpace Digital Lybrary. DU Haggerty Library. Philadelphia, USA.

Castillo, L., 2007. Pressure characterization of undeveloped and developed jets in shallow and deep pool. Proceeding 32nd Congress of IAHR, the International Association of Hydraulic Engineering & Research, Venice, Italy, 2, 645-655

Castillo, L., Puertas, J. and Dolz, J., 2007. Discussion about Scour of Rock due to the impact of plunging high velocity jets. Journal of Hydraulic Research, 45 (6), 715-723.

Castillo, L. and Carrillo, J.M., 2011. Numerical simulation and validation of hydrodynamics actions in energy dissipation devices. Proceeding 34th IAHR World Congress. International Association of Hydro-Environment Engineering and Research, Brisbane, Australia.

Castillo, L. and Carrillo, J.M., 2012. Hydrodynamics characterization in plunge pools. Simulation with CFD methodology and validation with experimental measurements. Proceeding 2nd IAHR European Congress. International Association of Hydro-Environment Engineering and Research, Munich, Germany.

Cui Guang Tao, Lin Ji Yong and Liang Xing Rong, 1985. Study on the force and effect of the Arch dam overflow water tongue on the river bed, Journal of Hydraulic Engineering, Beijing, 8, 53-68 (in Chinese).

Davies, J.T., 1972. Turbulence phenomena. Academic Press, New York.

Ervine, D.A. and Falvey, H.R., 1987. Behavior of turbulent jets in the atmosphere and plunge pools. Proceedings of the Institutions of Civil Engineers, Part. 2, 83, 295-314.

Ervine, D.A., Falvey, H.T. and Withers, W.A., 1997. Pressure fluctuations on plunge pool floors. Journal of Hydraulic Research, 35 (2), 257-279.

Federspiel, M.P., 2011. Response of an Embedded Block Impacted by High-Velocity Jets. PhD Thesis. École Politechnique Fédérale de Lausanne, Suisse.

Ho, D.K.H. and Riddette, K.M., 2010. Application of computational fluid dynamics to evaluate hydraulic performance of spillways in Australia. Australian Journal of Civil Engineering, 6 (1), 81-104.

Lencastre, A, 1961. Descarregadores de lâmina livre. Lisboa: LNEC (in Portuguese).

Manso, P.A., Bollaert, E.F.R. and Schleiss, A.J., 2005. Dynamic pressures generated by plunging jets in confined pools under extreme flood discharges. Proceedings of the XXXI IAHR Congress, Seoul, 2848-2860.

Moore, W.L., 1941. Energy loss at the base of a free overfall. American Society of Civil Engineers. Proceedings ASCE, (108), 1343-1360.

Puertas, J., 1994. Criterios hidráulicos para el diseño de cuencos de disipación de energía en presas bóveda con vertido libre por coronación. PhD Thesis. Universidad Politécnica de Cataluña, España (in Spanish).

Scimeni, E., 1930. Sulla forma delle vene tracimanti. L´Energia Elettrica, Aprile, 293-305 (in Italian).

Wahl, T.L., Frizell, K.H. and Cohen, E.A., 2008.Computing the Trayectory of Free Jets. Journal of Hydraulic Engineering, 134(2), 256-260.

Withers, W., 1991. Pressure fluctuation in plunge pool of an impinging jet spillway. PhD Thesis, University of Glasgow, United Kingdom.