# Numerical simulation and validation of hydrodynamics actions in energy dissipation devices

L.G. Castillo E.[1], J.M. Carrillo S.[1]
[1] Civil Engineering Department. Hidr@m Group
Technical University of Cartagena
Paseo Alfonso XIII, 52, 30203 Cartagena
SPAIN
E-mail: luis.castillo@upct.es, jose.carrillo@upct.es

**Abstract:** *The energy dissipation in plunge pools are produced principally by turbulence generation. In the fall jet and in the dissipation basin appear high turbulence and aeration phenomena, that cannot be correctly studied by the classical methodologies.*

*The methods used in Computational Fluid Dynamics (CFD) are based on numerical solution of the Navier-Stokes and averaged Reynolds equations, together with turbulence models of different complexity grade: vortex´s algebraic models for the viscosity (k-ε, RNG k-ε, k-ω) and Reynolds Stress Models (RSM).*

*Taking into account these problematic and in function of different experimental results, the Hidr@m group is carrying out different studies to characterize velocities (Doppler instrument), pressures (piezoresistive transducers) and aeration (optical fibre). These laboratory results are useful as a base to validate and calibrate some commercial programs CFD.*

***Keywords:*** *dams, spillways, plunge pool, energy dissipation, pressure characterization, CFD.*

## 1. INTRODUCTION

The rectangular jet or nappe flow constitutes one of the types of plunge pools in arch dams. The selection of the plunge pool depth is usually a technical and economic decision between a deep pool which needn't 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) and Manso et al. (2005).

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

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 (see Fig.1). For design considerations we define both the issuance conditions and the impingement conditions.

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 $\xi$ 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.

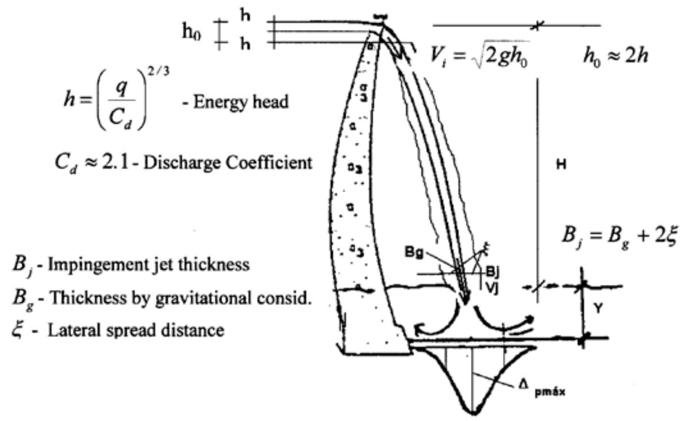



**Figure 1. Plunge pool at dam toe**

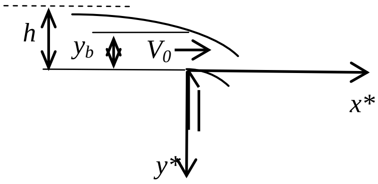
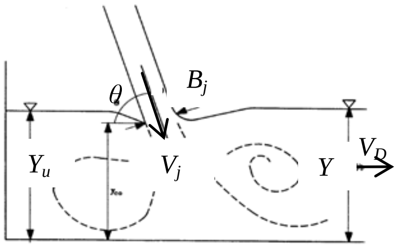
An important parameter constitutes the jet break-up length, $L_b$, beyond this distance the jet is completely developed, it no longer contains a core but consists essentially of blobs of water that disintegrate into finer and finer drops. For flows smaller than 0.25 $m^2/s$ (laboratory tests values), the Horeni´s formula $L_b \sim 6q^{0.32}$ (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 ($\xi$), impingement thickness ($B_j$) and the mean dynamic pressure coefficient ($C_p$). Table 1 resume these formulations together with the jet trajectory ($y=f(x)$) and the pool depth under nappe in basin ($Y_U$).

**Table 1. Parametric formulations in nappe flow case**

| AUTOR | FORMULAE | OBSERVATIONS |
|---|---|---|
| Castillo (2006) | Issuance conditions:<br>$T_u^* = q^{0.43} / IC$<br>$IC = 14.95 g^{0.50} /(K^{1.22} C_d^{0.19})$<br>$\frac{L_b}{B_i F_i^2} = \frac{K}{(1.07 T_u F_i^2)^{0.82}}$<br>Impingement conditions:<br>$\xi = 2\varphi\sqrt{h_0}\left[\sqrt{H} - \sqrt{h_0}\right]$<br>$B_j = \frac{q}{\sqrt{2gH}} + 4\varphi\sqrt{h_0}\left[\sqrt{H} - \sqrt{h_0}\right]$<br>Mean dynamics pressure ($C_p$):<br>$Y \le 4B_j$:<br>$C_p = 0.36(H / L_b)^{-1.04}$<br>$Y > 4B_j$:<br>$C_p = \frac{H_m - Y}{V_j^2 / 2g} = ae^{-b(Y/B_j)}$ | $T^*_u$, turbulent intensity issuance conditions (to calculate $B_i$ and $B_j$)<br>$q$, specific flow<br>$IC$, initial conditions<br>$g \approx 9.8$ $m/s^2$, gravity acceleration<br>$K$, fit coefficient ($\approx 0.95$)<br>$C_d$, discharge coefficient<br>$L_b$, jet break-up length. $q$<0.25 $m^2/s$, $L_b \approx 6q^{0.32}$<br>$B_i$, issuance jet thickness<br>$T_u \approx 0.012$ turbulent intensity issuance conditions (when $q$ >> 0.25 $m^2/s$)<br>$F_i$, issuance conditions Froude Number<br>$B_j$, impingement jet thickness<br>$H$, height between upstream water level and downstream water level<br>$\varphi = 1.07\ T^*_u$, turbulence parameter in nappe flow case<br>$h_0 \approx 2h$, issuance conditions level<br>$H_m$, head mean registered at plunge pool Bottom (stagnation point)<br>$Y$, depth at plunge pool<br>$V_j$, impingement velocity |
| Scimeni (1930) | Trajectory of jet central nappe<br>$x^* = [2.155(y^* + 1)^{1/2.33} - 1]$ | $x$, $y$, coordinates axis<br><br>$h$, total head |

| $H/L_b$ | $a$ | $b$ | $C_p$ ($Y/B_j$<= 4) |
|---|---|---|---|
| < 0.5 | 0.98 | 0.070 | 0.78 |
| 0.5-0.6 | 0.92 | 0.079 | 0.69 |
| 0.6-0.8 | 0.65 | 0.067 | 0.50 |
| 1.0-1.3 | 0.65 | 0.174 | 0.32 |
| 1.5-1.9 | 0.55 | 0.225 | 0.22 |
| 2.0-2.3 | 0.50 | 0.250 | 0.18 |
| > 2.3 | 0.50 | 0.400 | 0.10 |

| AUTOR | FORMULAE | OBSERVATIONS |
|---|---|---|
| | $x^* = x/h; \quad y^* = y/h$   | $V_0 = \alpha V = \alpha(q/y_b)$, jet initial velocity on weir Crest<br><br>$\alpha$, Coriolis coefficient<br><br>$y_b$, jet depth on weir crest |
| Cui Guang Tao (1985), Castillo (1989) | Pool depth under nappe<br>$Y_u / Y = \sqrt{1 - 2F_D[(V_j / V_D)\beta \cos\theta - 1]}$   | $Y_u$, pool depth under nappe<br><br>$Y$, water cushion<br><br>$F_D = V^2_D / (gY)$, Square Froude number<br><br>$V_J$, impingement velocity<br><br>$V_D$, downstream velocity<br><br>$\beta \approx 0.6$, head loss coefficient<br><br>$\theta$, impingement jet angle |

## 2. PHYSICAL MODEL

An infrastructure has been constructed specifically for the study of turbulent jets and energy dissipation in the nappe flow case (see Figure 2), which allows us to study air-water two-phase phenomena (aeration, spray, spread and impact).

It consists of a mobile mechanism that allows to vary the discharge heights between 1.70 and 4.00 m and flows between 10 and 150 l/s. A methacrylate´s energy dissipation basin (1.60 m height and 1.05 m wide) in which can be regulated different water cushion. 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. In addition, there are photographic and high-speed video instrumentation to characterize the phenomenon.

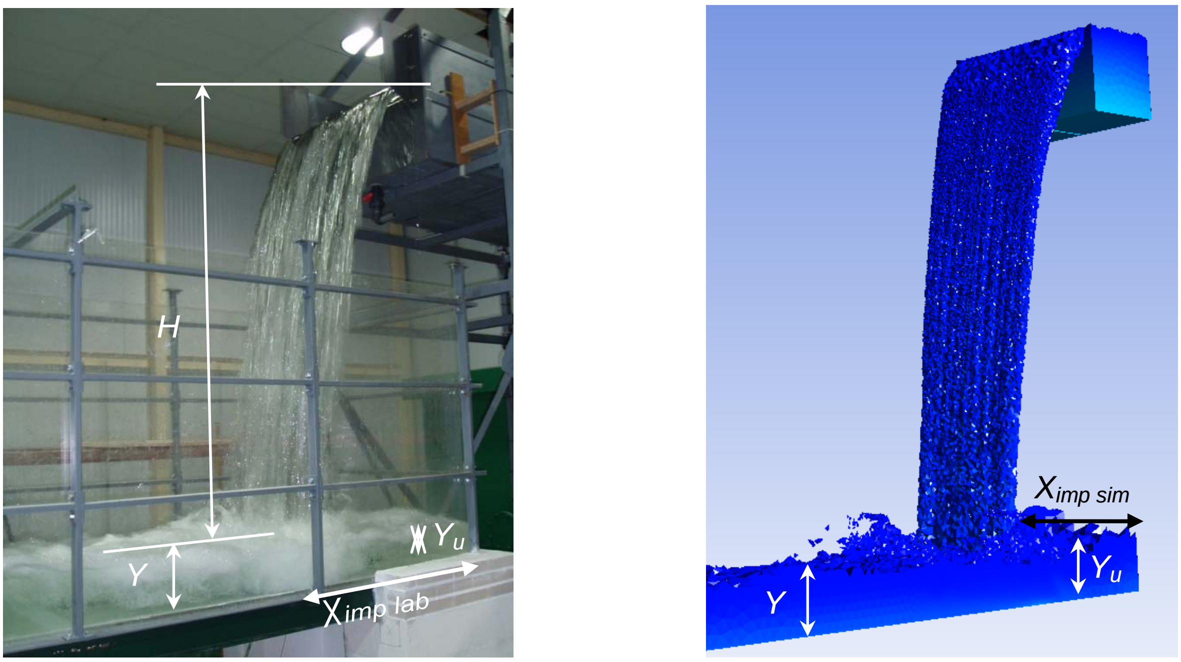



**Figure 2. Laboratory devices versus numerical simulation of plunge pool**

Figure 2 shows the laboratory nappe flow and a numerical simulation visualization (ANSYS CFX 12), for the same flow conditions: $q$=0.056 $m^2/s$, $H$=2.29 m and $Y$=0.16 m. The measured impact point was $X_{imp\ lab}$= 0.74 m and the simulated result $X_{imp\ sim}$ =0.68 m.

## 3. NUMERICAL MODELING

The main advantage of the methodology called *"Computational Fluid Dynamics"* (CFD) is the possibility it offers to investigate physical fluid systems, providing lot 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 prototype or physics model.

In this paper, the CFD methodology is applied to the investigation of flows highly aerated and turbulent, using the code ANSYS CFX (2006). The software 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), has been used *"Shear Stress Transport"* (*SST*) turbulence model. So, the *k-ω* based *SST* model results adequate for the transport of the turbulent shear stress and give highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients.

ANSYS CFX provides two models for simulating multiphase flows, bubbles, drops, solid particles and free surface flow: an Eulerian–Eulerian multiphase model (EE) and a Lagrangian Particle Tracking multiphase model (LP). The EE model has two different sub-models: the homogeneous and the Interfluid transfer or inhomogeneous model.

In the inhomogeneous model, each fluid has its own flow field. The fluids interact through terms of transfer at the interface (interfacial transfer of momentum, heat and mass are directly dependent on the contact surface area between the two phases) and there's a solution field for each phase separately.

The homogeneous model has been used in this study. It can be considered as a limit case of EE 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). The homogeneous model does not need to be applied consistently to all equations. In this case, the velocity field may be modeled as inhomogeneous, but coupled with a homogeneous turbulence model.

The model boundary conditions corresponds to the flow, upstream and downstream levels and their hydrostatic pressures distributions. The total number of elements used in the simulation were 3,852,769 with length scale in the falling jet boundary and at stagnation point of 0.01m. The total wall clock time was 4.099E+05 seconds (CPU with 8 nucleus).

All scenarios have been calculated by a transient calculation time of 60 seconds, using a step interval of 0.05 seconds. Figure 3 shows an example of the pressure register obtained in one of the simulations. We can observe that permanent conditions are reached after 20 seconds of simulation.

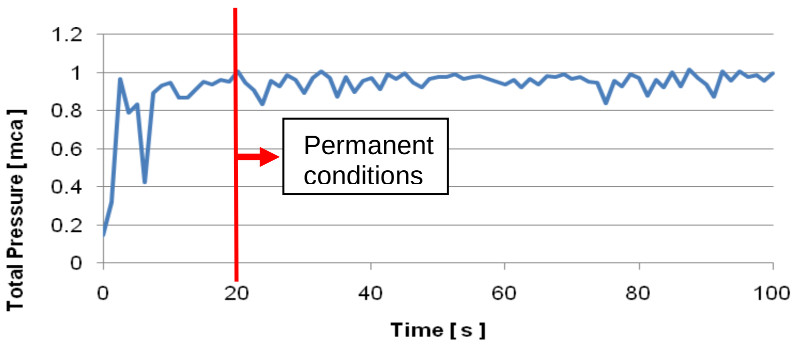



**Figure 3. Transient of total pressure in a point of the bottom of the plunge pool**

## 4. RESULTS AND DISCUSSION

This paper presents a comparison of the most important parameters that appear in the phenomenon of turbulent jets. Data were extracted from the numerical modeling carried out with ANSYS CFX and measurements in the hydraulics laboratory of the Technical University of Cartagena. This data are compared too with the turbulent jets methodology proposed by Castillo (2006, 2007) and Castillo et al. (2007) and defined here like Parametric methodology. Figure 4 shows the definition of variables and relevant sections in the study of turbulent jets.

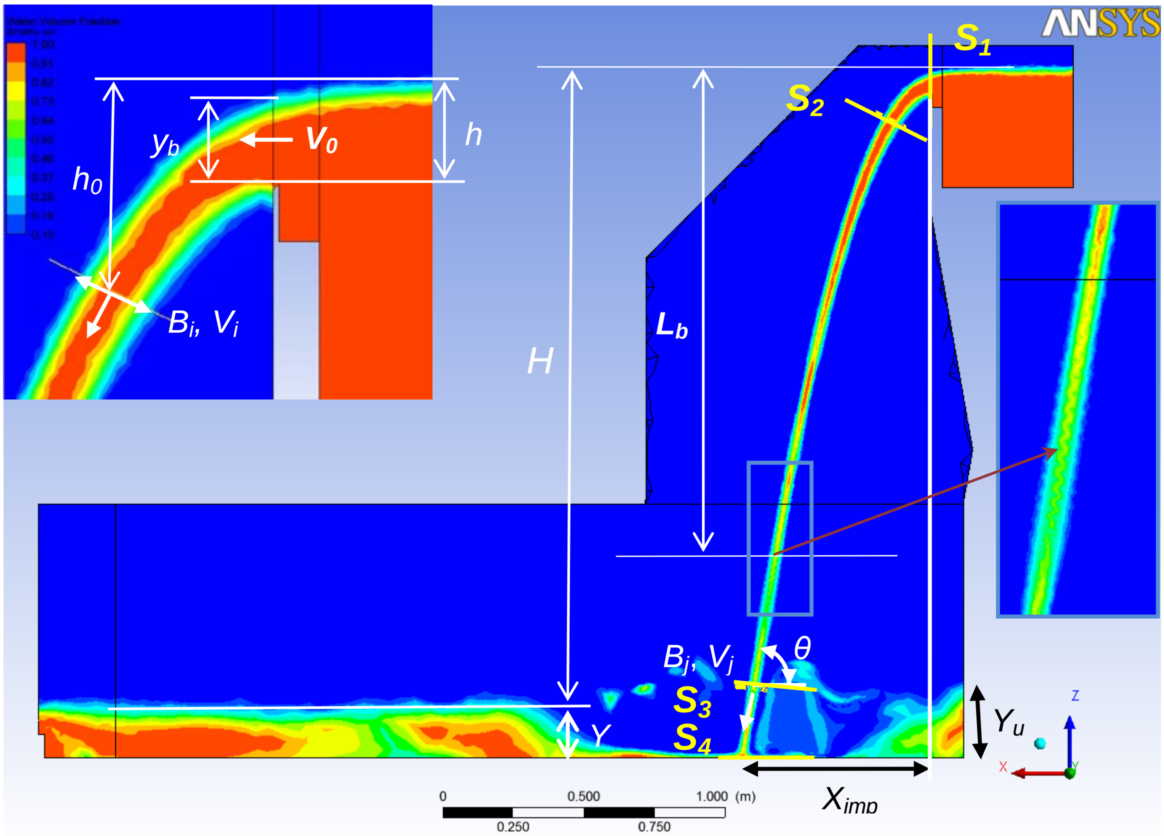



**Figure 4. Variables and relevant sections in the study of turbulent jets**
**Details of weir shape and the jet break-up point**

Table 2 shows the results of the three methodologies for two different scenarios:

- Two specific flows: 0.056 and 0.024 $m^2$/s.
- The weir height is 2.35 m above the bottom of the stilling basin.
- A barrier close to the end of the stilling basin of 0.083 meters.

**Table 2. Comparison of the principal measurement and calculated variables**

**$q$ = 0.056 $m^2$/s, $H$ = 2.29 m, $h$ = 0.098 m, $Y$ =0.17 m, $T^*_u$ = 0.0065, $H/L_b$ = 0.96, $Y/B_j$ = 7.44**

| | $y_b$ (m) | $V_0$ (m/s) | $V_i$ (m/s) | $B_i$ (m) | $L_b$ (m) | $V_j$ (m/s) | $B_j$ (m) | $Y_u$ (m) | $X_{imp}$ (m) | $\theta$ (º) | $C_p$ |
|---|---|---|---|---|---|---|---|---|---|---|---|
| Param. | - | - | 1.71 | 0.046 | 2.38 | 6.63 | 0.022 | 0.15 | 0.76 | 81.51 | 0.39 |
| Lab | 0.075 | 1.01 | 1.48 | - | - | - | - | 0.17 | 0.74 | 84 | 0.34 |
| CFX | 0.070 | 1.08 | 1.82 | 0.059 | 2.24 | 6.40 | 0.024 | 0.17 | 0.68 | 82.48 | 0.42 |

**$q$ = 0.024 $m^2$/s, $H$ = 2.28 m, $h$ = 0.060 m, $Y$ = 0.13 m, $T^*_u$ = 0.0045, $H/L_b$ = 1.26 m, $Y/B_j$ = 11.52**

| | $y_b$ (m) | $V_0$ (m/s) | $V_i$ (m/s) | $B_i$ (m) | $L_b$ (m) | $V_j$ (m/s) | $B_j$ (m) | $Y_u$ (m) | $X_{imp}$ (m) | $\theta$ (º) | $C_p$ |
|---|---|---|---|---|---|---|---|---|---|---|---|
| Param. | - | - | 1.28 | 0.026 | 1.81 | 6.63 | 0.011 | 0.13 | 0.58 | 84.25 | 0.09 |
| Lab | 0.044 | 0.68 | 1.03 | - | - | - | - | 0.13 | 0.54 | 86 | 0.09 |
| CFX | 0.045 | 0.67 | 1.31 | 0.043 | 1.19* | 6.20 | 0.012* | 0.12 | 0.47 | 85.77 | 0.08 |

* Value obtained with water volume fraction of 20%

In general way, the results obtained with the three methods are similar except for some considerations discussed below, so that the results are encouraging to continue to the research.

In the section S1, the depth over the spillway $y_b$ obtained with CFX and the measured value in the laboratory are very similar. However, we have not possibility to compare with the Parametric methodology because we have not any formulation yet.

Figure 5 shows how the velocity distribution at the discharge point (section S1) doesn't match a uniform profile. The mean velocities fits with the following Coriolis coefficients: $\alpha$=*1.35* ($q$=0.056 $m^2/s$) and $\alpha$=1.25 ($q$=0.024 $m^2/s$). This uneven distribution of velocity also affects to the jet thickness ($B_i$ and $B_j$).

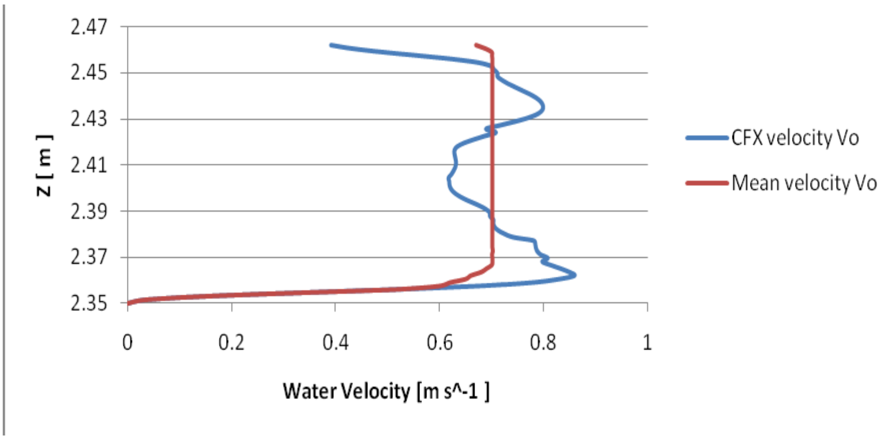



**Figure 5. Velocity distribution over the weir crest**

In sections S2, the turbulence index calculated with Parametrical methodology and CFX are:

$q$ = 0.056 m²/s: Parametric: $T^*_u$=0.0065
CFX: $T_u$=0.0058;

$$(V_i = 1.646\, m/s; \quad u' = 0.0046\, m/s, \quad v' = 0.00216\, m/s, \quad w' = 0\, m/s; \quad \Rightarrow \overline{v_i'} = 0.0051 m/s)$$

$q$ = 0.024 m²/s: Parametric: $T^*_u$=0.0045
CFX: $T_u$=0.0039

$$(V_i = 1.308\, m/s; \quad u' = 0.0046\, m/s, \quad v' = 0.00216\, m/s, \quad w' = 0\, m/s; \quad \Rightarrow \overline{v_i'} = 0.0051 m/s)$$

These results are similar and to confirm one of the hypothesis assumed in Parametric methodology, the perpendicular turbulent velocity to the flow plane is $w' \approx 0$ (Castillo, 2006). However, the horizontal turbulent velocity $u'$ is approximately the double that the vertical turbulent velocity ($u' \approx 2v'$). Then the root-mean-square of the stream wise turbulent component would be $\overline{v_i'} = \sqrt{(2v')^2 + v'^2} \approx 2.24v'$ and the turbulence parameter $\varphi = k(v'_i/V_i) \approx 2.24 T^*_u$. Nevertheless, for confirm this last affirmation, we need determine the indexes with laboratory measurements and carry out more simulations.

For flow $q$=0.056 m²/s, the break-up length calculated with CFX ($L_b$=2.24 m) results very similar to Parametrical result ($L_b$=2.38 m) when the water volume fraction is similar to 50%. However, for low flow ($q$=0.024 m²/s) the break-up length with CFX ($L_b$=0.56 m) is very different to Parametrical result ($L_b$=1.81 m). For that reason we have used in this case the 20% of water volume ($L_b$=1.19 m).

The criterion to determine the depths ($y_b$, $Y_u$) and the jet thickness ($B_i$, $B_j$), was to limit the surface to a value of the 50% in water volume. The differences in the jet thickness in S2 and S3 sections between CFX and the Parametric methodology are due possibly to the arbitrary criterion that we have assumed. With this criterion had not possible to obtain the incident jet thickness for the lower flow and whose value was obtained with the 20% of water volume fraction.

The flow $q$=0.024 m$^2$/s is fully disintegrated in the air (developed jet case) because the ratio break-up length to fall height is greater than one ($H/L_b$ =1.26). By the way, the ratio water cushion to impingement jet thickness is very much greater than four ($Y/B_j$ = 11.52) and so, the jet is totally established. For these reasons, the mean dynamic pressures coefficients ($C_p$) obtained by the three methods are very reduced. On the other hand, the flow $q$=0.056 $m^2/s$ is in the limit of a disintegrated partially jet ($H/L_b$=0.96) and is totally established in the water cushion ($Y/B_j$=7.44). The mean dynamic pressure coefficient ($C_p$) calculated with CFX is slightly greater than the obtained with Parametrical and Lab methods because possibly the homogeneous simulation don't reproduced adequately the energy loss by effect of the jet aeration. Figure 6 shows the simulated pressure distribution in the symmetry line on the bottom of the plunge pool (stagnation point).

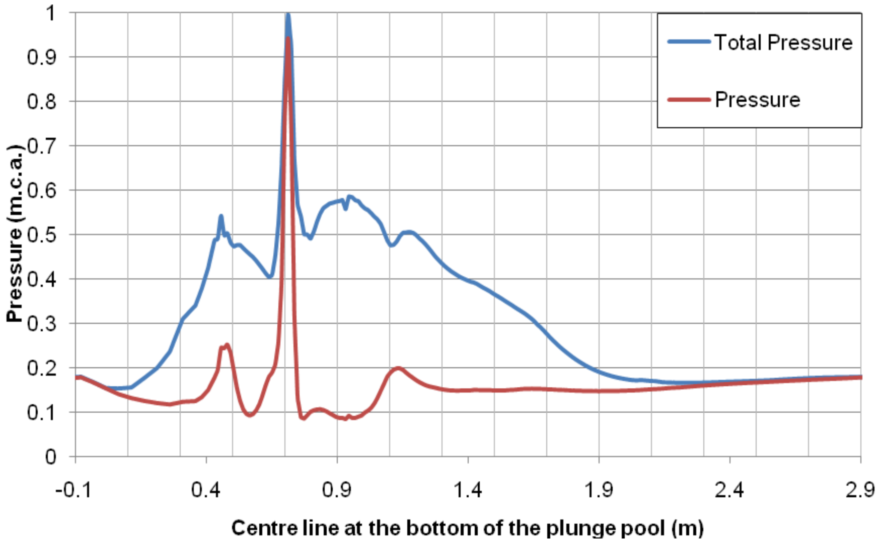



**Figure 6. Pressure distribution at stagnation point**

The jet angles and depths at pool depth under nappe ($Y_U$) are very similar and so we can establish that these variables are adequately simulated.

The CFX´s velocities in the impingement point (section S4) are slight lower than the theoretical velocities obtained with the Parametrical method. These results seems to be correct because would indicate the jet energy loss in the air. However in the section S4 (stagnation point), the simulated impact points $X_{imp}$ are slightly lower with respect to the measurement and calculated values. This is probably due to that the different turbulent parameters in numerical model have not been adequately established yet. By the contrary, the Parametric values are slightly greater that measurements values because the Scimeni´s (1930) formulation not consider the aeration´s loss.

## 5. SUMMARY AND CONCLUSIONS

In order to improve the design of energy dissipation structures: arch dams, overtopping gravity dams, fall structures in channels, it is necessary to advance in the knowledge and characterization of the hydrodynamic actions.

The parametric methodology used in this paper are based only on the results of measurements of instantaneous pressure at the bottom of the stilling basin. To advance knowledge in this area is necessary to make more experimental studies, both physical models and prototypes, simultaneously characterizing the phenomena produced in the jet aeration and measures of pressures, velocities and aeration rates in stilling basin.

The laboratory results allow us to calibrate and validate some commercial software CFD. As can see, progress is being made in the characterization of the phenomenon of turbulent jets with ANSYS-CFX. Later we'll validate the results with FLOW-3D and with some Lagrangian program.

This is the objective of the presented paper, whose results and conclusions, we hope will contribute to advance the understanding of these phenomena.

## 6. ACKNOWLEDGMENTS

The research is part of the project PEPLAN: "Hydrological modelling in Semi-Arid Regions. Subproject 3: Modelling of intakes in ephemeral rivers" (Decreto 420/2008). The authors are grateful for financial support of Consejería de Universidades, Empresa e Investigación of Comunidad Autónoma of Región de Murcia.

## 7. REFERENCES

Annandale, G.W. (2006). *Scour Technology. Mechanics and Engineering Practice*. McGraw-Hill, New York, USA.

ANSYS CFX (2006). *ANSYS CFX. Reference Guide*. Release 11.0.

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

Bollaert, E. (2002). *Transient water pressures in joints and formation of rock scour due to high-velocity jet impact.* Laboratoire de Constructions Hydrauliques. Ed.: A. Schleiss. Ecole Polytechnique Fédérale de Lausanne. Communication 13.

Bollaert, E., 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, Vol. 41, No.5, pp. 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.

Castillo, L. (1998). *Revisión de las formulaciones de presión en los disipadores de energía en presas bóveda y corrección del coeficiente de presión dinámica.* Comunicación personal.

Castillo,L., Puertas,J., Dolz,J. (1999). *Discussion about pressure fluctuations on plunge pool floors*. Journal of Hydraulic Research, Vol.37, No.2, pp. 272-788.

Castillo, L. (2002). *Parametrical analysis of the ultimate scour and mean dynamic pressures at plunge pools.* Poceedings of the International Workshop on Rock Scour due to Falling High-velocity Jets. École Polytechnique Fédérale de Lausanne, Switzerland, 25-28 september 2002. Schleiss & Bollaert (eds). A.A. Balkema. ISBN 90 5809 518 5.

Castillo, L. (2006). *Areated jets and pressure fluctuation in plunge pools*. The 7th International Conference on Hydroscience and Engineering (ICHE-2006), IAHR, ASCE Environment and Water Resources Institute, Drexel University. College of Engineering. DSpace Digital Lybrary. DU Haggerty Library (22 pages). Philadelphia, USA.

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

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, Vol. 45, No. 6, pp. 715-723.

Cui, G. T. (1985). *Gongba yiliu shuishe dui hechuang zuoyonghi ji qi yinxiang de yanjui*. Shuli xuebao (8), pp. 53-68. [*Efeito do impacto, no leito do rio, da lamina descarregada sobre una barragem-abóbada*. I.C.T. TR. 829 LNEC, Lisboa, 1986].

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

Ervine, D.A., Falvey,H.T., Withers,W.A. (1997). *Pressure fluctuations on plunge pool floors*. Journal of Hydraulic Research. Vol. 35, No. 2, pp. 257-279.

Manso, P.A., Bollaert, E.F.R., Schleiss, A.J. (2005). *Dynamic pressures generated by plunging jets in confined pools under extreme flood discharges*. XXXI IAHR Congress, Seoul, CD_Rom, pp: 2848-2860.

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.

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

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