COMPARATIVE ANALYSIS OF THE MODIFICATION OF TURBULENCE AND ITS EFFECTS ON A TRAPEZOIDAL SECTION STILLING BASIN.

LUNA B. J. CÉSAR(1), SALAS V. CRISTIAN(2) , GRACIA S. JESÚS(3) & ORTIZ M. VICTOR(4)

(1,2 3 y 4) *Instituto de Ingeniería de la Universidad Nacional Autónoma de México, D.F. México, Country,*

(1)JlunaB@iingen.unam.mx
(2)cristiansalasv@hotmail.com
(3)JgraciaS@iingen.unam.mx
(4)VortizM@iingen.unam.mx

ABSTRACT

This present work aims to characterize a turbulent flow that occurs in a trapezoidal section stilling basin, which belongs to the "Tunel Emisor Oriente" (TEO) project, concerning to the deep drainage system from the Mexico City, with a design caudal of 240 m³/s. The physical model was constructed with a 1:30 scale, where it was found that its functionality during the design condition set up would be committed due to overflow. Through modification of the geometrical characteristics of the chute, at the entrance section of the tank, we sought to obtain a higher dissipation of turbulent kinetic energy (TKE) and decrease the levels variations of water depth. To evaluate these new geometries and their effects, three different physical assays were performed on the model, these being: the test on the smooth spillway (smooth spillway), the second one was carried out by increasing its roughness using steps in the chute (stepped spillway) and the third and last one exchanging the steps with two rows of baffles with the form of teeth (spillway with deflectors). The physical model was instrumented with acoustic Doppler Velocimeter (ADV) and a digital water level gage. With which the instantaneous speed and level fluctuations was measured, also with this a grid was drawn to calculate the velocity, turbulent kinetic energy (TKE) and its dissipation. In addition, a CFD modeling was performed using the computational software FLOW 3D with same dimensions as the physical model to compare the modification of turbulence. From the comparison of the physical and numerical models some similarity in the calculated indexes is observed, however it is noted that the numerical model underestimates values, so it is necessary some modification in its parameters in order to obtain better results, closer to reality.

Keywords: spillway, turbulent kinetic energy, FLOW 3D

1. INTRODUCTION

In infrastructures of energy dissipation such as stilling basins, the objective has always sought to work efficiently with less depth and longitude in order to reduce construction costs (Fathi-M et al 2011, Su Pei et al 2009, Chen J. et al. 2014). To achieve this, a design with adequate cross section to the project and a longitude to properly develop the energy dissipation is required (Garcia et al 2012).

It is important to know how the energy dissipation within the stilling basin develops, since this will determine the longitude of the basin. It has been observed by experiments that the dissipation of kinetic energy is independent of fluid viscosity, even though the only parameter of the movement equations that can dissipate energy is precisely the viscous (Jimenez S.J. 2011). Jimenez S.J. (2011) also found a solution assuming that large eddies have high Reynolds numbers where the viscosity cannot influence dissipation, which generate small eddies every time until the viscosity is able to dissipate energy as heat. While the later does not happen, energy is transmitted without loss and flow between different scales is constant, which is used to determine the velocity fluctuations. This cascade of energy acts from the largest scale that is not homogeneous and isotropic, to the viscous scale (Jiménez, S. J. 2011).

To understand how the energy dissipation takes place in a stilling basin, is necessary to characterize the turbulence parameters and for that one way to proceed is to know the speed values in time and used them to calculate statistically the turbulence intensity, turbulence kinetic energy and the dissipation of turbulence kinetic energy.

1.1 Turbulence intensity

There is a universal expression to calculate the turbulence intensity, which is calculated for each of the flow directions. The turbulence intensity components in a flow are *u´* in the flow direction, *v´* in transversal direction and *w´* in the vertical direction of the flow. Nakagawa et al. (1975) were some of the first researchers to measure these parameters in free surface flows, with the advent of a hot wire anemometer. They carried out studies to observe the turbulence behaviour, and changing the roughness in a channel. Thus, it was observed that the dominant eddy size and turbulence intensity in the flow direction decreases when the roughness increases; and therefore the redistribution of turbulence energy over a channel with rough bottom develops faster than on one with a plain bottom.

Nezu and Nakagawa (1993) developed equations for the turbulence energy and turbulence intensities in the three directions of flow; derived from their experiments a better fit was observed in the $0.1 < {}^{y}/_{h} > 0.6$ which is where a greater difference was presented in Nakagawa early results. To obtain the value of the coefficients of the equations, they conducted experiments on free surface channels whereby the following equations standardized and derived from the study of two-dimensional flow were developed:

$$u'/U_* = 2.3\exp(-y/h) \quad [1]$$

$$v'/U_* = 1.27\exp(-y/h) \quad [2]$$

$$w'/U_* = 1.63\exp(-y/h) \quad [3]$$

$$k\ /\ U_*^2 = 4.78\exp(-2y/h) \quad [4]$$

Where $\boldsymbol{u'}, \boldsymbol{v'}, \boldsymbol{w'}$, are mean square (turbulence intensity); y is the distance to the channel bottom; k is the turbulence energy and $\boldsymbol{U_*}$ is the cutting velocity $U_* = \sqrt{ghS}$ being g gravity acceleration, h mean hydraulic depth and S the channel slope.

With the above equations turbulence intensity in each direction is obtained by observing $u' > w' > v$, however the turbulence intensity is modified due to the presence of walls, for this case other parameters need to be applied. When performing further experiments, it was demonstrated that these equations have problems with the presence of obstacles in the bottom but generates good results in smooth or slightly rough bottom (Papanicolaou et al. 2012).

1.2 Turbulent Kinetic Energy

TKE is a function of the velocity mean values in three directions, but unlike this variance the TKE is quantified in three directions, so that if the speed fluctuations in the three directions are known, fluid energy at a specific point in a specific time can be known.

TKE is considered as a transport model of turbulence of an equation where the specific energy is associated with the velocity fluctuations of the turbulence flow velocity and to characterize the magnitude of the fluctuation mean square of each variable is used (White F.M. 1991).

Being the continuous values of the velocities in the flow direction: $(u_1, u_2, u_3 \dots u_n)$

$$u'_{rms} = \sqrt{{}^{1}/_{n}\,(u_1^2 + u_2^2 + u_3^2 \dots + u_n^2)} \quad [5]$$

Then, turbulence kinetic energy is calculated as:

$$TKE = 1/2\,(u'^2_{rms} + v'^2_{rms} + w'^2_{rms}) \quad [6]$$

Where $\boldsymbol{u}', \boldsymbol{v}'\ and\ \boldsymbol{w}'$ are velocity fluctuations in the respective flow directions.

1.3 Rate of dissipation of turbulent kinetic energy

There are several criteria for calculating the dissipation rate of the turbulence energy. Nezu and Nakagawa (1993) developed an equation based on the analysis in a direction of normalized spectral function of velocity fluctuations, this only in the predominant direction of flow, where parameters such as macro scale turbulence are involved. They obtained a practical equation that only depends on a constant and the value of the hydraulic depth, where the ε is required.

$$\varepsilon h/U_*^3 = E_1(y/h)^{-1/2}\exp(-3y/h) \quad [7]$$

Where $\boldsymbol{h}$ is the hydraulic depth, $\boldsymbol{y}$ is the depth $\boldsymbol{E_1}$ is approximated of 9.8 for Reynolds numbers between $10^4 - 10^5$.

The dissipation rate for models of an equation is directly related to the turbulence kinetic energy and can be calculated as

$$\varepsilon = C_\mu \sqrt{\frac{3}{2}}\frac{TKE^{3/2}}{l} \quad [8]$$

Where TKE is the turbulence kinetic energy, $\boldsymbol{C_\mu}$ is an empirical parameter typically of 0.09 (dimensionless) and $\boldsymbol{l}$ is the macro scale turbulence, the value of 7% of the hydraulic diameter which in the case of pipes is restricted by the pipeline diameter and the approximate ratio is 7 % of diameter, is recommended (Shojaee and Boyaghcchi, 2007). It is possible to use this parameter in channels; in this case it would be of 7% of hydraulic depth (Flow Science, Inc. 2012).

It is also possible to obtain the dissipation of kinetic energy if one starts from the average of the TKE per unit mass. The equation proposed by Flow Science in its manual of exercises is:

$$\varepsilon = C_\mu^{3/4} k^{3/2} l^{-1} \quad [9]$$

Where $\boldsymbol{C_\mu}$ is 0.09, $\boldsymbol{k} = 1.5(uI)^2$ being u mean velocity and $\boldsymbol{I} = 0.16Re^{-1/8}$, the Reynolds number for open channels is $\boldsymbol{Re} = Rh\,u/\vartheta$, being $\boldsymbol{Rh}$ the hydraulic radius and $\boldsymbol{\vartheta}$ the kinematic viscosity, which for water to 20°C is 1.003e-6.

If the macro scale turbulence of 7% of the hydraulic depth is considered; the rate of dissipation of the kinetic energy equation (Nezu and Nakagawa, 1993) is represented as:

$$\varepsilon = K\frac{U'^3}{l} \quad [10]$$

Where $K = 0.691 + 3.89/\sqrt{Re}$ and is valid if the Reynolds number is greater than 200.

2. PHYSICAL MODEL

A physical model of 1:30 scale was constructed in the laboratory of the Institute of Engineering, UNAM. The geometry modelled corresponds to the exit door of a very important work for the discharge of waste water and storm water of Mexico City, the main relationships between model and prototype are shown in Table 1.

Table 1. Main relationships between model and prototype

CARACTERISTIC	SCALE
Lenght	1:30
Discharge	1:4929
Velocity	1:5.5
Time	1:5.5
Rugosity	1:1.8

The section of interest for this work corresponds to a trapezoidal stilling basin with a slope of 1: 1.5 and dimensions at the base of 1.80 m and the height of free border of 23.33 cm as shown in (Figure 1)

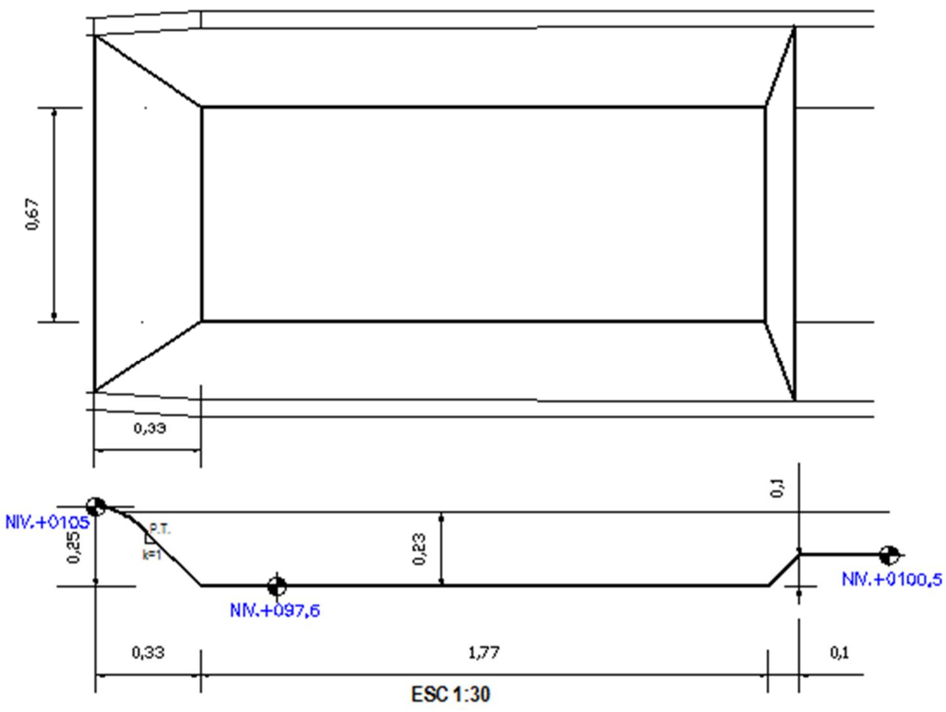

Figure 1. Profile and base of the physical model

In the model, a flow in the prototype 240m³/s of 48.7 l/s was represented. Also, the speed was measured with a Micro ADV Sontek at a frequency of 30 Hz, 45 velocity sampling were performed at different points of the stilling basin, these were distributed in 5 cross sections and on each section 3 verticals, located on the right, center and left banks where measured, and sampling was performed at three different depths 5, 10 and 16cm with respect to the hydraulic reference plane. All these measures give a total of 9 points per section, resulting on a meshing of 45 node points. The level variation of the free surface was also recorded as the basin's operation under normal conditions had an overflow. To register level fluctuations, digital limnimeters transducers were instrumented with the same frequency of 30 Hz.

Figure 2 Instrumentation with Micro ADV and limnimeters

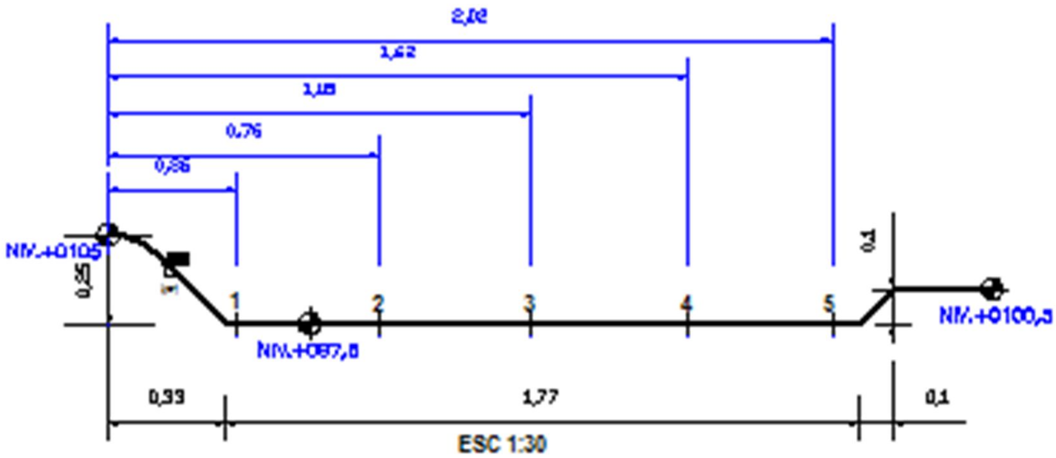

Figure 3 measuring points along the stilling basin

3. NUMERICAL MODELING

In addition, a modelling based on theoretical principles of Computational Fluid Dynamics (CFD) was performed. The numerical model was made in FLOW 3D since this software has given reliable results in the interpretation of similar studies to the work presented in this paper (De Dios et al. 2010, Cervantes et al. 2012, Amorim J.C. et al 2004 Sungyul et al 2002). By using the computational software FLOW 3D, the same dimensions of the physical model are maintained and it is possible to compare the modification of turbulence.

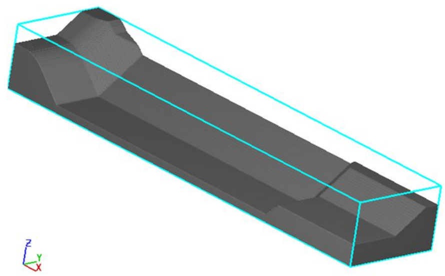

Figure 4. view of the stilling basin in the numerical model

4. METHODOLOGY

As mentioned before is intended that through the modification of the geometrical characteristics of the surface, the energy dissipation inside the basin changes, this with the aim to reduce variations of the hydraulic depth levels. That is why three geometries are proposed in order to characterize the turbulence and determine if there is a structure that modify the dissipation of turbulence within the basin thus is reflected in the reduction of fluctuating levels of the free surface.

The first geometry called "Smooth Spillway" consists basically of operating the model with the original design features (Figure 5).

Figure 5. Trial Smooth Spillway

The second geometry was carried out by increasing the initial rugosity of the smooth spillway, using inclined steps of unconventional dimensions which gradually change in both steepness and footprint length, and given its complexity the steps were modelled in polyethylene through the 3D printing technique. This was designated as the trial "Stepped Spillway" (Figure 6)

Figure 6. Trial Stepped Spillway

Finally, the geometry based on the same principle as the second trial was proposed. The steps were exchanged by two rows of deflectors shaped as serrated edges, built of wood for simplicity and speed; it was called "Spillway with Deflectors". (Figure 7)

Figure 7. Trial Spillway whit deflectors

The numerical model was produced using the FLOW 3D software, the model was developed through a prismatic mesh in which 133.993 cells were included for calculation having each cell a length of 1 cm per side. Borders conditions according to the phenomenon were placed, in the case of upstream a velocity condition to define the flow; downstream was only required an output condition; on the right side the border condition was also output in order to identify overflows; on the left side tests were performed with previous modelling for the purpose of determining if this has a symmetric behaviour and put a symmetry condition. The tests showed that the borders have the same behaviour so it was decided to model only half a basin and computation time was then saved; the border at bottom is of wall and border related to the free surface is of pressure (Figure 8)

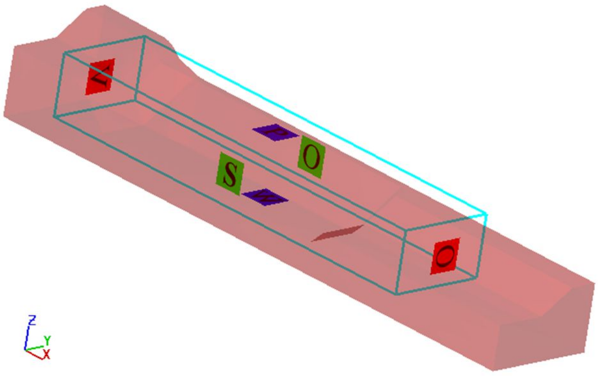

Figure 8. Boundary conditions

Additionally, the same geometries that were tested in the physical model were modelled in order to compare the turbulence rates against the physical model.

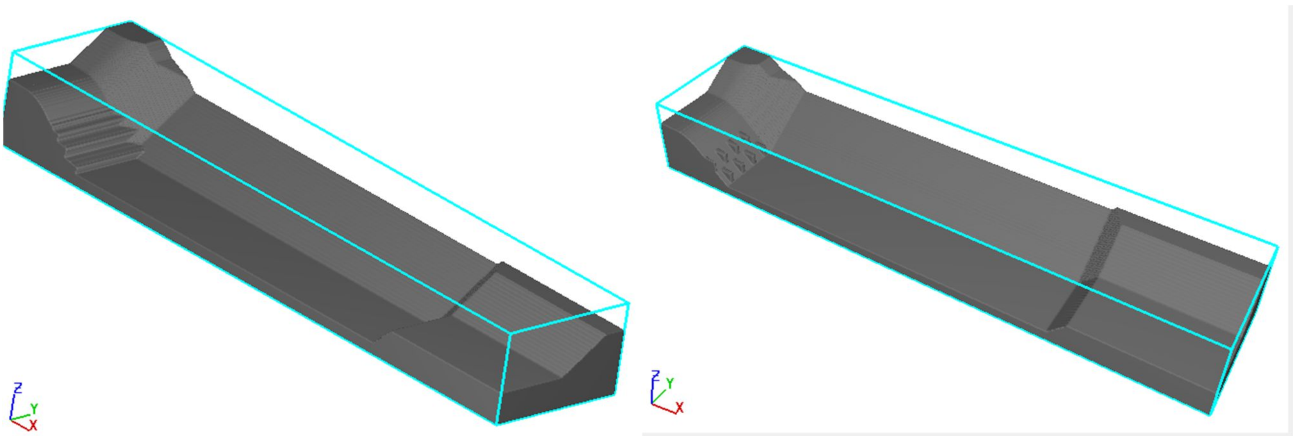

Figure 9. Stepped Spillway and Spillway whit deflectors configurations

5. RESULTS

To analyze the behaviour of the phenomenon graphs of mean velocities, turbulence kinetic energy, dissipation of it, and fluctuation levels were performed, and then compared with those carried out with the numerical model.

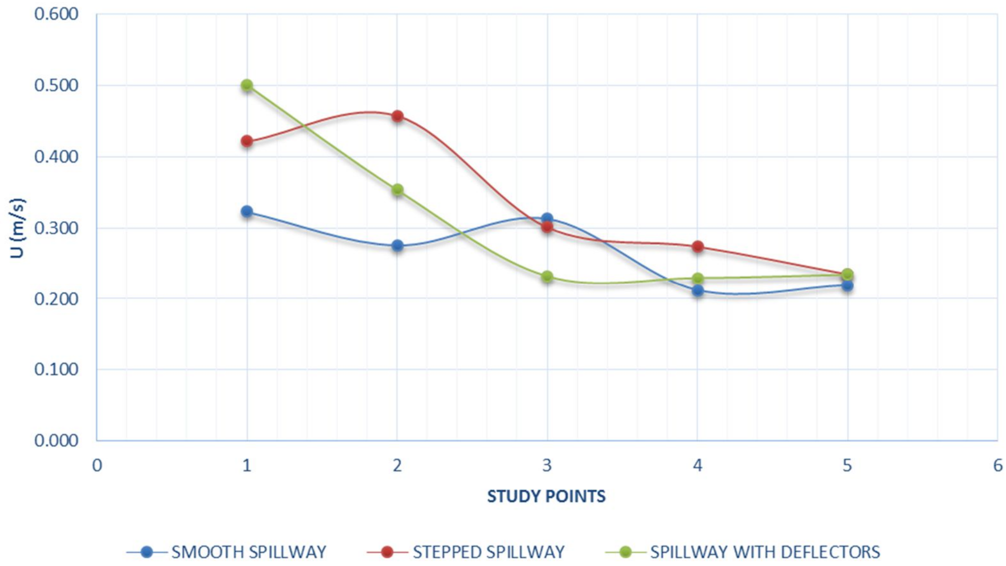

Figure 10. Average Velocity – Experimental data

Figure 10 shows the average velocities obtained at each point of the study and despite what one might think about the trial with the initial configuration, called "Smooth Spillway", the measurements give a lower average velocity of the flow in the initial points. These results are due to these points are where the hydraulic jump submerged occurs and the flow velocity is not characterized yet by a dominant component that governs the flow direction. This means that the flow has both positive and negative velocities affecting the mean values of the velocities in the individual components. As a result, a higher mean velocity of the flow is produced at the point number 1, located 0.76m from the cornice. With respect to the trial with the spillway with deflectors, the flow is accelerated due to the decrease of the sections that generate the strangulation of the geometric configuration, producing jets that create a dominant velocity in the direction of flow. Regarding the results of the stepped spillway trial, one can conclude that lies between the two previous spillways, this means that its velocity reduced relative to the spillway with deflectors and is greater than the smooth spillway, which suggests that the velocity of the flow is more ordered when this enters to the basin but one cannot conclude that this has a dominant direction.

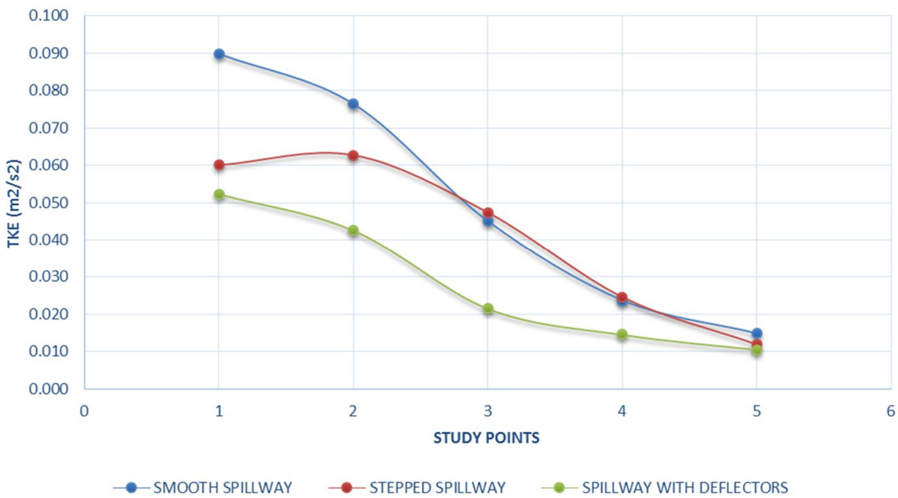

Figure 11. TKE Experimental results

In the above Figure 11, the curve representing the initial configuration (smooth spillway), has its maximum turning point in Section 1, located at 0.36m of the crest and 0.03m of the end of the spillway. From this point, the TKE decreased 83% when it reaches the point 5 which has a TKE close to 0.015m²/s. The curve corresponding to the "stepped spillway" has the maximum point of TKE at the studied point 2, located at 0.76m of the crest of the stepped spillway. Although, a reduction in turbulence kinetic energy of 33% occurs with regards to the original configuration, this has not managed to reduce the oscillations of the water level. Therefore, one can demonstrate that the greater dissipation of turbulence kinetic energy is given by the spillway with deflectors, the curve corresponding to this proposal has a similar behaviour to the curve of the "smooth spillway", however a decrease of TKE from 41% to 28% compared to the "smooth spillway" is presented. Due to the geometry configuration of this trial more recirculation zones are produced that promote greater energy dissipation, which is greater than in the other two, resulting in a lower TKE at all study points.

The oscillation of the maximum levels at each point of study for the different trials with regards to the upper limit of the stilling basin is shown in Fig 12. As mentioned above, the curve representing the trial with the stepped spillway at the inlet section ("Stepped Spillway") decreases the level of maximum hydraulic depth from 2% at study point 1 to 13% in the study point 5, over the original configuration; however this is not sufficient to solve the problem of overflow. No being the case of the third trial ("Spillway with Deflectors"), as shown, this can maintain the levels of the maximum hydraulic depth below the border of the basin of the corresponding study, and as a result a higher energy dissipation is produced in the spillway with deflectors configuration. Therefore, level fluctuations along the entire basin are smaller in comparison with the other two configurations.

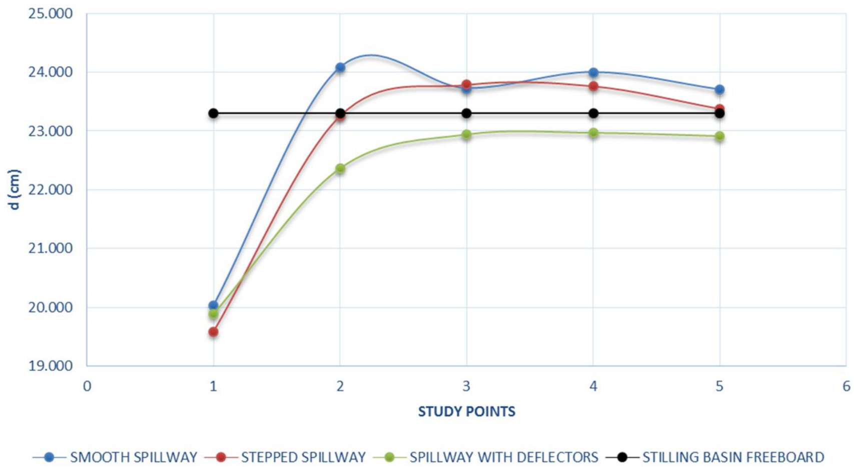

Figure 12. Maximum hydraulic depth

As explained previously, the greater dissipation of turbulence kinetic energy ε is produced in zones of recirculation of trial "Spillway with Deflectors"; these zones are favoured by having greater recirculation and turbulence. It should be noted that this is done on the structure of dissipation and on arrival at the basin it has less TKE. Comparing to the other trials ε is greater in the trial "Stepped Spillway" than the trial "Smooth Spillway"; this is due to the recirculation produced in the steps when forming a pseudo template with the vertices of the steps to the flow at high speed.

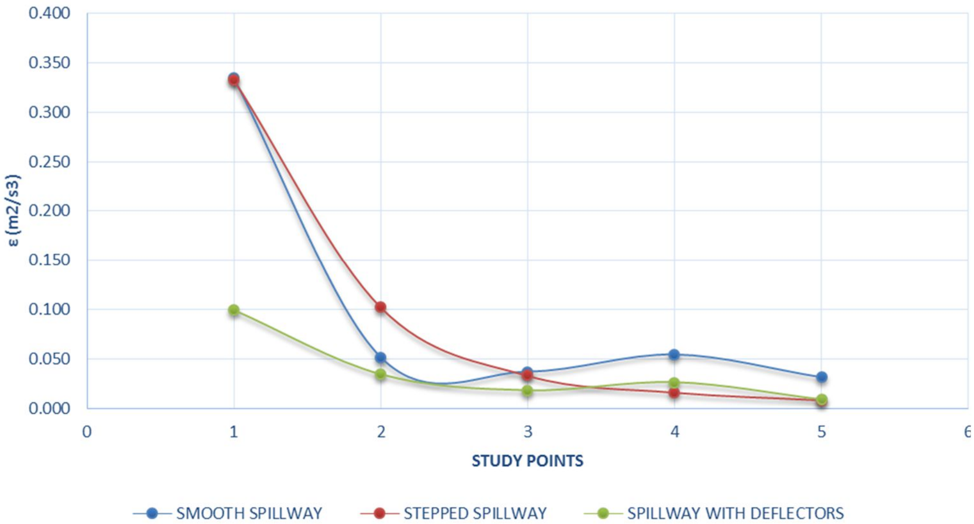

Figure 13. Dissipation of TKE

With regards to the numerical modelling, TKE values, ε, fluctuation levels and mean velocities along the same points that were used for recording data in the physical model, were obtained. It was noted that the numerical model underestimates the values of TKE and ε, but it obtains an acceptable approximation concerning these values and distribution.

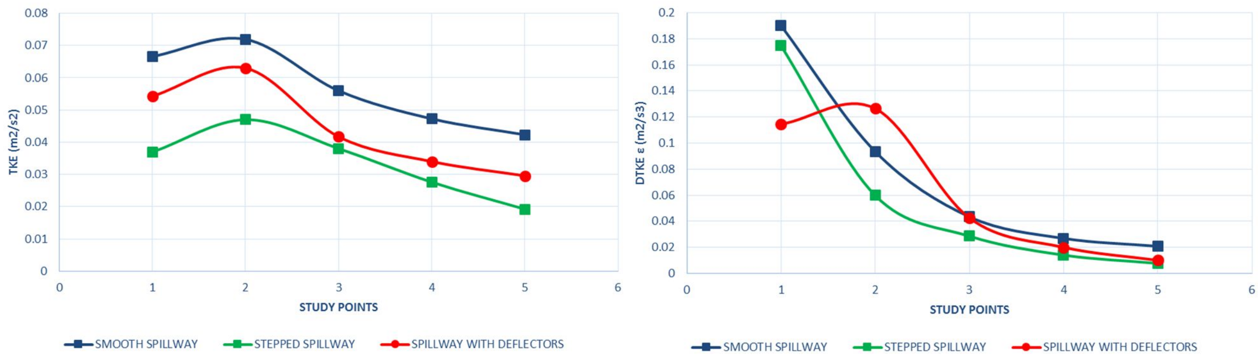

Figure 14 TKE- Mathematical model (left) and ε - Mathematical model (right)

In the figure 14 a different behaviour to the results of the physical model with respect to the TKE can be observed. The behaviour along the basin of the Smooth Spillway configuration is similar to the results from the physical model, being of small magnitude but consistent in shape. Regarding the other two configurations it can be noted that in the first test point the Stepped Spillway has a lower TKE to the Spillway with Deflectors. This is a contrast since in the physical model was observed it develops inverted and both the Spillway with Deflectors and the Stepped Spillway do not present overflow, which is not seen in the physical model where only the Spillway with Deflectors avoided overflow.

The dissipation of the turbulence kinetic energy is developed in a similar manner between measurements and computation, only on the entry significant differences are presented by the Smooth Spillway where ε is 34% higher in the physical model; in the Stepped Spillway configuration, measurements of the numerical model are 19% higher and in the configuration of the Spillway with Deflectors only has a 5% difference. With regards to the general behaviour of dissipation, a greater ratio between the models was observed. Although, there is a small variation, it was also observed that, in general, in the Spillway with Deflectors the dissipation is greater than in the Stepped Spillway which is the inverse in the physical model.

The mean speeds along the basin have considerable variations in the numerical model (Figure 15), this occurs both in the Smooth Spillway and Stepped Spillway, only the Spillway with Deflectors has almost an identical behaviour throughout the entire basin. In the first register point, it is difficult that models behave similarly because it is a complex phenomenon, this zone has increased turbulence and air bubbles that complicate the recording of data, and the fluctuation levels have not been adequately represented in the numerical model, thus at this point some reservations in the interpretation of results must be applied. Regarding the fluctuations level the numerical modelling must be improved to ensure that the fluctuation is the one recorded in the physical model for this, work must be carried out at the interface with the free surface.

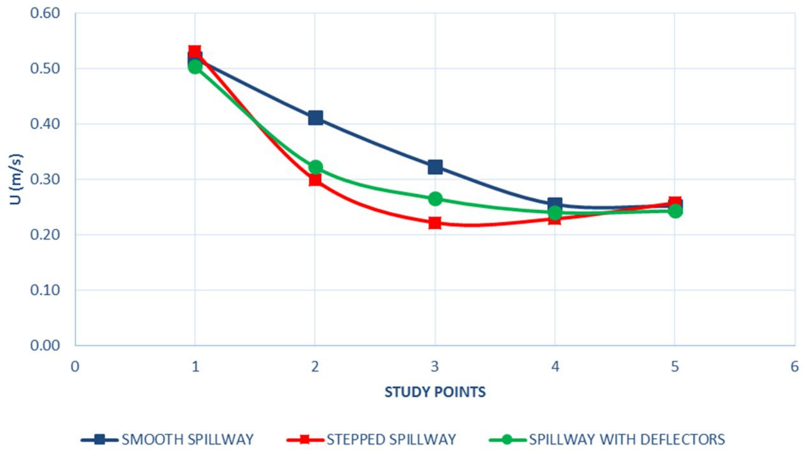

Figure 15. Average velocity mathematical model

6. CONCLUSIONS

In the world of hydraulics one of the most important parts for the validation of a design is the physical modelling. This modelling allows the study and visualization of the hydraulic behaviour of the designed structures. Taking this into account, it can be concluded that the increase in roughness that possess the Smooth Spillway using steps, it produces a greater dissipation of turbulence kinetic energy. However, the problem of overflow, although less, is still appearing along the stilling basin, whereas thanks to the use of deflectors in the form of teeth ("Spillway with Deflectors"), the flow presents greater dissipation of TKE and better performance nearly eliminating the problem of overflow.

Comparing results of both models and observing the graphs of the turbulence kinetic energy and its dissipation, indicates that it is possible to decrease the longitude of the stilling basin by placing structures that increase the dissipation, so the TKE at the entry of the basin is small as possible. All this because using the option called " Spillway with Deflectors" the longitude of the basin under study can be reduced up to 40%.

Also, from the comparison of the physical and numerical models some similarity in the calculated values is observed. However, it is evident that the numerical model underestimates the values and do not represent the behaviour of the free surface realistically. Also, it was not possible achieve that the computational model overflows in any of the three study cases; this is why the latter cannot be entirely validated. Therefore, it is necessary tuning the mathematical model, and modify its parameters in order to obtain better results to describe the phenomenon in question

ACKNOWLEDGMENTS

We appreciate the cooperation of The Instituto de Ingeniería de la Universidad Nacional Autónoma de México and their facilities in order to make possible this work

REFERENCES

Amorim J. C., Rodrigues R. C. and Marques M. G. (2004) A numerical and experimental study of hydraulic jump stilling basin. *Advances in hydro-science and engineering volume VI University of Mississippi*

Cervantes A. R. Berezowsky V. M., Vargas M H, Espinoza A. D. Camargo H. J., Montes L. M. Alejandrina L. (2012) Comparacion de modelo físico y matemático del flujo 3D en un vertedor de demasías. XXII National Congress of Hydraulic Mex.

Chen J. G., Zhang J. m. XU W.L. and Peng Y. (2014) Characteristics of the velocity distribution in hydraulic jump stilling basin with five parallel offset jets in a twin-layer configuration. *Journal of Hydraulic Engineering 2014.140:208-2017*

Fathi-Moghadam M., Haghighipour S, Lashkar-ara B and Aghtouman P (2011). Reduction of stilling basin length with tall end sill *Journal of Hydrodynamics, 2011,23(4):498-502*

Flow Science, Inc (2010). FLOW-3D Hydraulics Exercise Sessions. *Version 9.4 the favor advantage*

García S., Aguirre C, Fonseca M. (2012) Metodología para evaluar la disipación de energía en tanques amortiguadores. *Instituto Tecnológico y de Estudios Superiores de Occidente, Mexico*

Jiménez, S. J. (2011). Las teorías de la turbulencia y la imaginación en la física. Speech read in the act of receipt as academic of grade. Real Academia de Ciencias Exactas, Físicas y Naturales, Madrid Spin.

Nakagawa Hiroji, Nezu Iehisa and Ueda Hiroshi (1975). Turbulence of open channel flow over smooth and rough bed. PROC. OF Japan Soc. Civil Engrs. *No. 24.*

Nezu Iehisa and Nakagawa Hiroji (1993). Turbulence in open-channel flows. *A.A. Balkema, Rotterdam, Brookfield*.

Papanicolaou, A.N., Kramer, C. M., Tsakiris, A.G., Stoesser, T., Bomminayun, S., and Chen, Z. (2012). Effects of a fully submerged boulder within a boulder array on the mean and turbulent flow fields: Implications to bedload transport. *Acta Geophysica. Vol. 60 No. 6 pp 1502-1546.*

Shojaee Fard M.H. and Boyaghchi F.A. (2007). Studies on the Influence of Various Blade Outlet Angles in a Centrifugal Pump when Handling Viscous Fluids. *American Journal of Applied Sciences ISSN 1546 -9239*

Su Pei-lan, Liao Hua-sheng. Qio Yue Y. Li Chen-juan. (2009) Experimental study on a new type of aerator in spillway with low Froude number and mild slope flow. *Journal of Hydrodynamics 2009, 21(3): 415-422*

Sungyul Y. Kiwon H and Manha H (2002) A 3-dimensional numerical study of flow patterns around a multipurpose dam. *Proceedings of the Fifth International Conference on Hydroinformatics, Cardiff, UK ISBN 1 84339 021 3*

White, F. M, (1991). Viscous Fluid Flow. *Second edition McGraw-Hill University of Rhode Island.*