NUMERICAL MODEL OF THE NETZAHUALCOYOTL DAM EMERGENCY SPILLWAY AND HYDRAULIC VALIDATION OPERATION WITH THE PHYSICAL MODEL

Antonio Nicolás Gómez Arteaga
Independent Consultant

Edwin Fernando Zetina Robleda
National Water Commission

Víctor Hugo Alcocer Yamanaka
National Water Commission

This paper presents the study of the hydraulic operation of the Netzahualcoyotl dam emergency spillway, located in the state of Chiapas, Mexico, by Computational Fluid Dynamics (CFD). The numerical model solves the Reynolds Averaged Navier-Stokes equations (RANS) in three-dimensional cartesian coordinates using the finite volume method on structured meshes and the free surface is determined by the VOF (Volume Of Fluid) method. Respectively results were validated with a physical model (scale 1:65). The versatility of the numerical model allowed to consider scenarios and compare different design proposals.

NUMERICAL MODEL OF THE NETZAHUALCOYOTL DAM EMERGENCY SPILLWAY AND HYDRAULIC VALIDATION OPERATION WITH THE PHYSICAL MODEL

Objective

To develop a numerical model of the Netzahualcoyotl dam emergency spillway and validating the results with measurements in a physical model. This to analyze the hydraulic operation of the structure with the modifications made in the original design. In addition, the results were compared to know the reliability between the numerical model and the measurements of the physical model, with the purpose of validating the use of these techniques of numerical modeling to design, to revise and to improve the design of hydraulic structures.

Introduction

At present, there are several methods to evaluate the hydraulic behavior of large structures, among them, the physical modeling is one of the most used, however, in these cases there are parameters, which are not very representative due to the effect of the scale. In order to analyze the parameters that can not be adequately represented, it is advisable to apply a numerical model.

This work presents the study of the hydraulic operation of the emergency spillway of the Netzahualcoyotl dam (also known as Malpaso) using a numerical model through the application of FLOW-3D® software, which is widely used in hydraulic design due to its capacity of computing fluid behavior in three dimensions.

The Netzahualcoyotl dam was built between 1959 and 1964. It is located in the municipality of Tecpatan, in the state of Chiapas, in Grijalva system (the most important river system for generation of hydropower energy in México), approximately 125 km upstream from Villahermosa, Tab., and the first one to be built on the Grijalva River (CFE, 2011).

The dam's spillway is located on the left bank of the river and is composed by two components, one of service and one of emergency.

The original discharge for the design of the service and emergency spillways were 11,100 and 10,650 m^3/s, respectively. With the construction of the La Angostura dam upstream, the hydrological conditions were modified, leaving the total discharge for both spillways in a maximum of 10,500 m^3/s (to keep the usual risk criteria for dam safety) due to the regulation capacity of the Angostura dam. According to the last hydrological review, the peak runoff corresponding to the return period for 10,000 years is 16,450 m^3/s, this represent a discharge of 7,050 and 9,400 m^3/s in the service and emergency spillways respectively (González, 2012).

XVI World Water Congress
International Water Resources Association (IWRA)
Cancun, Quintana Roo. Mexico. 29 May- 3 June, 2017

With the above, the dam had a capacity for eviction by spillways less than a fifth of the recommended, which made it urgent to modify the spillways to be able to discharge the 10,500 m^3/s with safety for the dam and its structures (González, 2012).

The reasons why the technical intervention was necessary to improve the hydraulic operation of the emergency spillway were the following:

- The flow in the channel was concentrated in the left margin (Figure 1).
- The initial discharge for operation was very high, close to 3,000 m^3/s.
- The discharge of the spout hits the foundation of the structure.
- The energy dissipation occurred by the impact of the jet against the rocky massif, causing erosion.
- The dam had a capacity for eviction by landfills less than a fifth of the recommended.

Figure 1. Operation of the emergency spillway (HRS, 1976)

Derived from the hydraulic performance presented in the emergency spillway, it was necessary to make proposals to optimize the design. The mitigation actions used by CFE (Federal Electricity Commission of Mexico) to improve its hydraulic operation are presented below:

- Design of a central wall that divides the existing discharge channel into two.
- A transversal slope along each channel to reduce the lifting effect and thus obtain satisfactory hydraulic operation of the new discharge channels.
- Placement of an aerator in each channel, in order to minimize the speed in the bottom, and avoid the cavitation phenomena by introducing air to the flow and produce a greater loss of energy.
- Side deflectors and in the central wall in the bucket to achieve energy dissipation at the exit.

Numerical modeling

Nowadays there are several tools that support numerical modeling in one, two and three dimensions. The numerical modeling allows the study of the hydraulic phenomena in original conditions, without the limitations of the scale of the model.

In the design and planning stages of hydraulic structures, technical decisions to evaluate flow behavior are often based on physical models. However, engineers and researchers are increasing the use of Computational Fluid Dynamics (CFD) into the process of design. Nevertheless, the validation of numerical models with experimental or prototype data is very important to ensure correct modeling and to provide a high level of confidence in the results and its application.

The numerical model used is capable of adequately reproduce the analyzed phenomena and the related variables. Usually the problem arises in the way to carry out the solution of the numerical model, so that the model resembles in behavior to the prototype that wants to be represented. Also, the modifications made in a numerical model with respect to the physical model, have a much lower cost and allow the evaluation of different alternatives seeking the cost-performance optimization of the prototype.

With these advantages, CFD is useful for flow simulation, where the Navier-Stokes equations and the continuity equation, are discretized and solved for each element of the calculation mesh.

Pre process

Using computer-aided design (CAD) tools and 3D modeling, the geometry was built (Figure 2). This geometry is composed by two channels, service and emergency spillways, as well as the topography of the terrain where these structures are located.

Figure 2. Components of the geometry

The elaboration of the mesh to discretize the dam is one of the most important stages. To perform the simulation, six different mesh arrangements were tested. The final configuration with its boundary conditions is shown in Figure 3.

The discretization of the mesh is a function of the geometry of the structure: where it presents large curved or sloping elements, the resolution will be greater in comparison where the geometry is flat and straight. The configuration of the final mesh arrangement was discretized in structured blocks every 4, 2 and 1 meter per side.

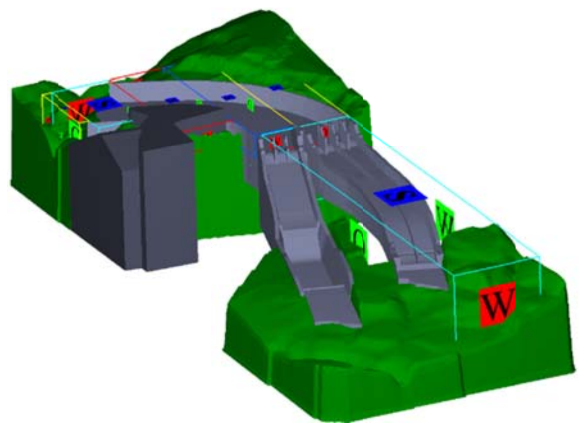

Figure 3. Mesh and boundary conditions

The turbulence model used to solve the RANS was the RNG (Re-Normalization Group) approach, which calculates the turbulent kinetic energy (k) and the kinetic energy dissipation rate (ε). The models based on the RNG equations depend on fewer empirical constants and establish a framework for the derivation of a series of parameters that will be used in different scales of turbulence.

Once all parameters of the pre-processing were stablished, a first simulation is performed to analyze the correct computation of the parameters defined in the pre-processing.

Process

At this stage, the simulation process begins, the convergence and stability of the model must be monitored to know the progress of the results obtained by the program.

For this study case, before the definitive simulation, was necessary five trials with different meshing arrangements and different discretization, with a duration of approximately twenty days each. Finally, the computer used thirty-five days to complete the simulation with the final mesh arrangement.

As a reference of the calculation time invested, the characteristics of the computer used in this modeling are indicated: Lenovo Workstation Model E30, Intel Xeon Processor (8 processors at 3.40 GHz), NVIDIA GeForce GT 520 3Gb video card and 8 Gb Memory RAM, Windows 7.

The review criteria of the simulations was the follow of the evolution of the average kinetic energy of flow during the execution until the variations over time begin to decrease. Once the steady state of this parameter has been reached, it is verified that this stabilization is also reflected in the flow through the domain.

The Figure 4 presents the finished simulation of the emergency spillway.

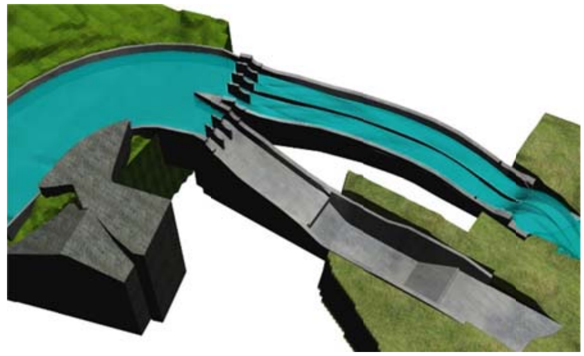

Figure 4. Simulation of the emergency spillway of the Netzahualcoyotl dam

Pos process

In the post process stage, the extraction of data generated by the processing is performed. The results can be obtained graphically, animated or in numerical form, which are subsequently processed using calculation tools and databases to obtain its graphic interpretation. The results are presented in the following sections.

Profiles and Speeds

Figures 5 and 6 show the profiles on the flow axis generated for the right and left channel respectively. In order to analyze the velocity values, the data from the numerical model were processed to obtain the matrices of the velocity quantities for their graphic representation.

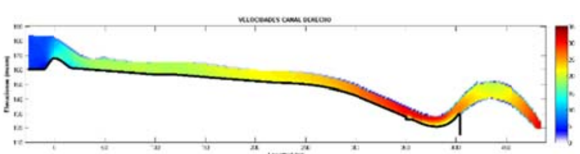

Figure 5. Velocity magnitude in right cannel (m/s)

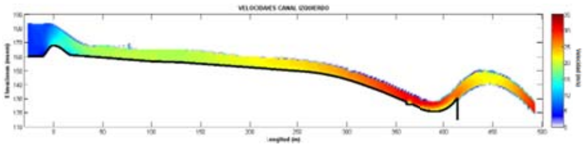

Figure 6. Velocity magnitude in left cannel (m/s)

The maximum velocities are of the order of 31 m/s before the aerator, while the corresponding in the weir are of 11 m/s, and the velocity reduction from 30 m/s to 22 m/s in the bottom after of the aerator.

Pressure head

Using the same procedure that was used to determine velocities, the numerical model data were processed in spreadsheets to generate pressure matrices. The magnitude of the dynamic pressure for the longitudinal profile along the axis of each channel was obtained, where it can be seen that the greatest concentration of pressures in the area of the deflector cuvette is approximately 33 meters in the bottom, Figures 7 and 8.

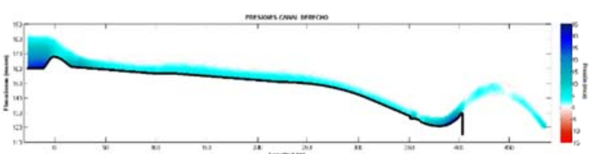

Figure 7. Pressures, right channel (meters)

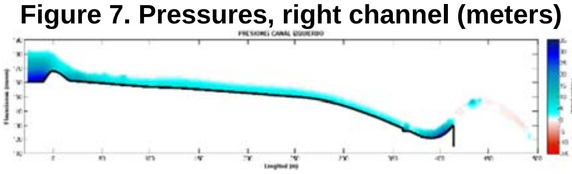

Figure 8. Pressures, left channel (meters)

Numerical-experimental validation

In order to validate the numerical simulation, a comparison of the results obtained by the numerical model with the experimental data of the physical model built by CFE.

The spillways of the Netzahualcoyotl dam was studied in the CFE's Hydraulic Laboratory. In the 1:65 scale hydraulic model, the two spillways were represented with a common channel as well as a part of the dam.

Several alternatives were analyzed, performing experimental tests with different geometries of the central wall, transversal slope, transitions of the wall with slope to vertical in the area of the flip bucket and lateral deflectors and in the central wall.

To verify the hydraulic operation of the emergency spillway, initial discharge, velocities and depths were recorded along the channels of the spillway, variables that would also be useful for the validation of the results obtained from the spillway numerical model.

Figures 9 and 10 show graphically the comparison of the depths for both channels.

This comparison, for the same stations where the depths of the physical model were obtained, the cartesian coordinates were determined and the data of the numerical model were extracted, they were processed to obtain the corresponding depths.

The results allow to visualize the variation of the depths along the channel, the fluctuation of the profile of the free surface of the water when moving away from the crest, the increase at the beginning of the transversal slope, the undulations along the chute by the effect of the slope and the horizontal curvature, as well as the length of the ski jump.

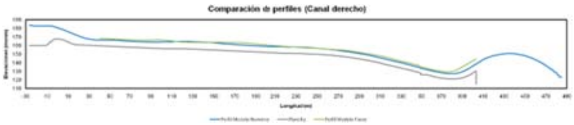

Figure 9. Comparison of depths, numerical and physical model, right channel

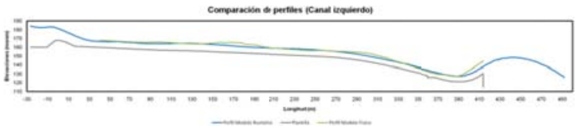

Figure 10. Comparison of tie depths, numerical and physical model, left channel

Figures 11 and 12 illustrate the variation of depths along the spillway. In the numerical and physical model, the similarity of the hydraulic behavior between the two models can be observed.

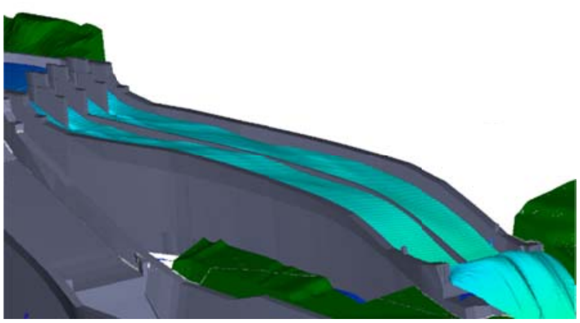

Figure 11. Variation of depths in numerical model

Figure 12. Variation of depths in physical model

To extract the average velocities from the numerical model, it was necessary do perform the same procedure that was used to determine the depths. In tables 1 and 2 the comparison of the velocities of both models is presented, indicating their magnitude and the station where the average velocities were obtained, finding the greatest variation in the flip bucket.

Table 1. Comparison of velocities in physical and numerical model, right channel

Training	Average speed, m/s (Q=4,700 m^3/s)	
	Physical model	Numerical model
0+120	19.32	19.05
0+311,60	25.76	25.08
0+370,79	29.28	29.22
0+381,06	28.98	28.87
Flip bucket	28.88	25.62

Table 2. Comparison of velocities in physical and numerical model, left channel

Training	Average speed, m/s (Q=4,700 m^3/s)	
	Physical model	Numerical model
0+120	19.20	18.80
0+311,60	25.79	26.00
0+370,79	29.60	29.24
0+381,06	29.59	29.75
Flip bucket	28.79	26.53

Comparison between original and current geometry

The numerical simulation was performed with the original geometry of the emergency spillway (Figure 13). It can be noticed that due to the horizontal curvature without slope, the flow was concentrated in the left margin, resulting in minimal depths in the right margin and a high initial discharge; resulting in poor hydraulic performance. Numerical modeling including the design proposals showed a satisfactory hydraulic behavior is observed.

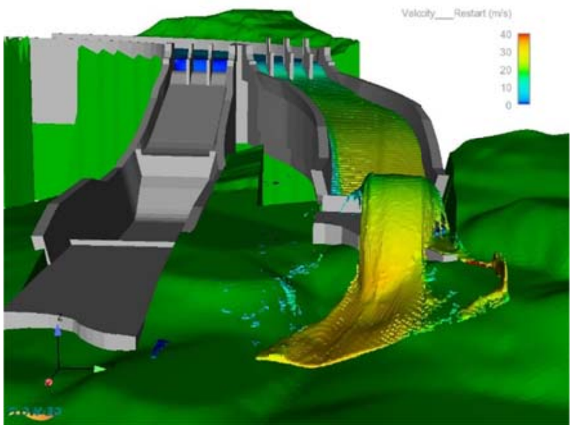

Figure 13. Numerical modeling with the original geometry

Figure 14 presents the comparison of the hydraulic operation of the numerical and physical model in original conditions, observing the same behavior of the flow lines and the flow concentration in the left margin.

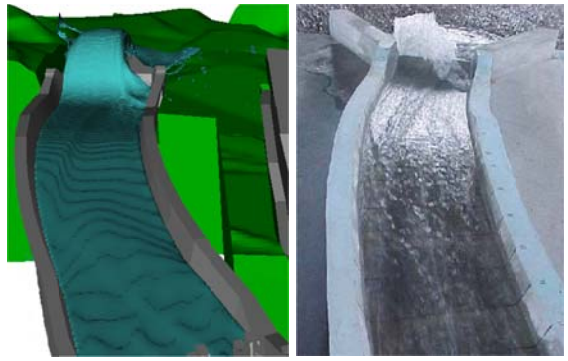

Figure 14. Hydraulic operation in original conditions numerical and physical model

Figure 15 displays the comparison of the streamlines with the original arrangement and with the proposed mitigation design. It is clearly seen in the original geometry, the concentration of the flow in the left margin, generating a lift of the depth in this margin, due to the horizontal and vertical curve that follows the trace of the discharge channel; in addition to the initial discharge was very high. This concentration of flow in the left margin also caused that the energy was concentrated in a smaller space and it avoids energy dissipation.

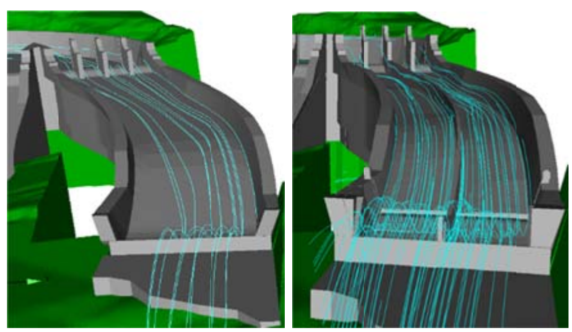

Figure 15. Comparison of original and current geometry current lines

Conclusions

The hydraulic operation of the emergency spillway with respect to the mitigation actions proposed by the CFE have the following positive effects:

- The discharge of the spillway will be channeled in a more uniform way due to the implementation of the intermediate wall. In addition, the construction of a transversal slope for each channel counteracts the inertial forces due to the horizontal curve.
- The placement of vertical deflectors in both the side walls and the central wall allows the energy dissipation from the impact of the water when leaving the terminal structure on the discharge site, which will help avoid scouring effects on the channel.
- Initial discharge for operation will be reduced, will be uniform flow in the width of the channel, due to the over-elevation of the bottom.
- The hydraulic behavior of the emergency spillway will be improved and will increase its functionality.
- The safety of the dam will be higher.

With the construction of the mitigation actions, the emergency spillway would be in optimal conditions of operation, it increases the capacity and security of the dam.

Regarding the numerical modeling, it accurately represented the flow along the emergency spillway, acceptable values for the depths, velocities and pressures. With the analysis of the numerical results, it was possible to analyze the hydraulic behavior of the spillway.

From the analysis of the results, it can be concluded that the numerical modeling is an appropriate resource to optimize the design and revision of hydraulic structures; also it is possible to make geometric changes or design adjustments in a reduced time and cost; and even better if such numerical results are validated by experimental or prototype data.

Likewise, the numerical model of a hydraulic structure yields variables that will be useful for other disciplines to realize their design, for example, the dynamic pressures and flow velocities for the structural and geotechnical design. An example are the dynamic pressures exerted in the bucket, knowing these, the durability of concrete, resistance, erosion, safety, among other parameters must be checked for the life of the project.

The importance of this study and the results obtained from the numerical modeling of the emergency spillway of the Netzahualcoyotl dam, will stablish future decisions to guarantee the management of the Grijalva river and operational and emergency policies to save the integrity of the elements downstream of the dam, such as the Grijalva tunnels, the Peñitas dam and the safety of the city of Villahermosa, Tabasco.

Bibliography

1. Arreguín, F. & Alcocer, V. (2011) *Spillways Hydraulic design*. Mexico.

2. Camacho, J. (2000) *C.H. Netzahualcoyotl. Hydraulic operation of the spillway with flip bucket, in physical model.* Cuernavaca, Mor., Mexico. Hydraulic Laboratory. Federal Electricity Commission.

3. Capella, A. (2012) *Study of the necessary modifications to the Malpaso Dam spillways, Chiapas.* Engineering Institute. UNAM. Mexico.

4. Electricity Federal Commission (1981) *Civil Structures Design Manual*. Hydrotecnia. A.2.10. Excedence Structures. Mexico.

5. FlowScience (2009) *FLOW-3D. Hydraulic Training Class*. Santa Fe, NM.

6. FlowScience (2009) *FLOW-3D. Hydraulic Exercise Sessions*. Santa Fe, NM.

7. Gómez, A. (2016) *Numerical simulation of the dam Netzahualcoyotl emergency spillway validating it with the hydraulic operation physical model.* UNAM.

8. Gómez, A. & Sánchez, F. (2015) *Dam Netzahualcoyotl, Numerical Simulation of the hydraulic operation of the emergency spillway*. Cuernavaca, Mor., Mexico. Hydraulic Laboratory. Federal Electricity Commission.

9. González E. (2012) *C.H. Malpaso. Netzahualcoyotl Dam, Chis. Spillways Operation, in hydraulic model.* Cuernavaca, Mor., Mexico. Hydraulic Laboratory. Federal Electricity Commission.

10. González E. (2014) *C.H. Malpaso. Spillways Functioning for the executive project, in physical model, Le = 65.* Hydraulic Laboratory. Cuernavaca, Mor., Mexico. Federal Electricity Commission.

11. Hidalgo, I. (2004) *Spillway with flip bucket operation, in hydraulic model. Study extension.* Laboratory of Hydraulics. Cuernavaca, Mor., Mexico. Hydraulic Laboratory. Federal Electricity Commission.

12. National Water Commission (1982-1994) *Dams of Mexico*. Mexico.

13. Hydraulic Resources Secretariat (1976) Great Dams of Mexico. Mexico.