XL CILAMCE
IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGINEERING
NOVEMBER 11-14, 2019
Praiamar Natal Hotel & Convention
Natal, RN-BRAZIL

STRUCTURAL MONITORING OF FOZ DO CHAPECÓ POWER PLANT

Gustavo Alcalá Batistela
gustavoabatistela@gmail.com
FEC - UNICAMP
R. Saturnino de Brito, 224, 13083-889, Campinas, SP, Brasil
Tiago Forti
Fabio L. G. Santos
forti@simworx.com.br
gea@simworx.com.br
Simworx Engenharia, Pesquisa e Desenvolvimento
R. Paulo Cezar Fidelis 39, Sala 306, 13087-727, Campinas, SP, Brasil
João Rodolfo Cortes Pires
jrcortes@cortesconsultoria.com.br
Cortes Consultoria
R. João Pedroso, 140, Sala 21, 13084-583, Campinas, SP, Brasil
Luis Melegari
lmelegari@cpfl.com.br
CPFL Energia
Rod. Eng. Miguel Noel Nascentes Burnier, 1755, 13088-900, Campinas, SP, Brasil

Abstract. Hydroelectric power plants represent the most important source of electricity in Brazil. Hydroelectric plants are a clean source of energy that produce no residua or pollution during its operational lifetime. However, the storage of great amounts of water may represent a risk to cities downstream. Although the chances of accidents are small, their impact may be catastrophic. Therefore, monitoring the structures is paramount to have a safe operation. This work focuses on monitoring the structural safety of Foz do Chapecó Power Plant. The plant is installed on Uruguai River in the border of states of Santa Catarina and Rio Grande do Sul, Brazil. The structure consists of a rock fill dam with asphaltic core and concrete spillways. The spillway structure is 350 meters long and 60 meters high with 15 gates. At the current stage of the research, the numerical simulation of the spillway is developed. The structure is modelled with the Finite Element Method in 3D, with the entire structure in the same mesh. Emphasis is given to mesh generation, where details such as galleries, contact with rock foundation and expansion joints are modelled. Loads are given in terms of reservoir upstream level, openings of gates and foundation sub-pressures caused by water percolation. The solver is developed using the finite element program PZ, which provides a whole family of elements morphology, different approximation spaces and mesh adaption technologies. The displacements obtained by the simulations are compared to the displacement history of the spillway measured by various sensors. This comparison is used to adjust simulation data such as the assumed material parameters. In next steps of this work, the numerical simulation will be integrated to physical monitoring of robotic total stations and other sensors yet to be installed in the structure.

Keywords: Finite elements method, Dam monitoring, Concrete structures

1 Introduction

Energy in Brazil is generated predominantly by hydroelectricity. According to Agência Nacional de Energia Elétrica [1], currently, the production capacity of this category totals 99.9 GW, which represents 60.39% of the country's total production.

The advantages of hydroelectric power include the fact that it is a clean source of energy, which means that, differently from fossil energy sources, it produces no pollutants during its operation. Also, since hydropower depends on the water cycle only, it is a renewable power source.

Nevertheless, the impoundment of great amounts of water represents a risk to the area downstream in the face of an accident. Even though the chances of rupture of a dam are small, the consequences are often catastrophic. With that being said, monitoring the structures of a dam is paramount to avoid disasters to the environment and population and ensure the safety of its operation.

This research focuses on monitoring the structure of Foz do Chapecó Power Plant. The complex has a power capacity of 855 MW. The dam is installed on the Uruguai River, between the border cities of Águas de Chapecó, in the state of Santa Catarina, and Alpestre, in the state of Rio Grande do Sul.

The impoundment structure comprises the spillway, a reinforced concrete structure with 15 gates and total outflow capacity of 62.190 m³/s, a 598 meters long, 48 meters high asphalt-core rockfill dam and a 200 meters long, 33 meters high clay-core rockfill saddle dam.

The process of monitoring these structures, as stated in the monitoring manual [2], is carried out periodically after the first year of operation. It consists in retrieving data from 51 piezometers, 19 rod extensometers, 16 flow meters, 6 triaxial joint meters, 4 concrete stress meters, 12 rebar strain meters, 13 concrete thermometers, 18 superficial marks, 3 inclinometers, 6 pressure cells and 12 settlement sensors. Additionally, visual inspections of the structure conditions take place.

To the current system, this project will add a RTS (robotic total station), capable of making measurements in real-time. Data collection will be assisted by 52 superficial marks, from which 33 are prisms on the downstream face of the main dam, 16 on the spillways and 3 on the saddle dam. Four additional prisms will be placed on geodetic marks away from the structure to allow georeferencing the system.

The collected measurements will be integrated to a web-based online monitoring system containing tools to analyze the instruments history and a digital twin of the structure. According to El Saddik [3] a digital twin is a virtual replica of a physical entity that mirrors its processes in real time since required data is transmitted seamlessly.

The digital twin exhibits the current stress-strain state of the structured, calculated using the finite elements method. Whenever the system is triggered, a new simulation takes place using updated measurements as parameters. The model is also capable of receiving data filled manually by the user to forecast the stress-strain state in hypothetical conditions.

The project is currently under development. At this point, the work is centered on the digital twin, specifically on implementing the finite element kernel and on the generation of the meshes of the spillway and the main dam. The RTS equipment is yet to be installed, so data used as parameters is gathered from the history of existing sensors.

The remainder of this article is organized as follows. Section 2 describes the elasticity problem and its weak formulation. The construction of the digital twin of both spillway and main dam are presented in Section 3. Section 4 brings simulation results and Section 5 concludes the article.

2 Finite elements method calculation

The numerical simulations of the structure are performed using the finite elements method. The finite elements method (FEM) can be defined as a systematic way of constructing approximated solutions of boundary value problems. Due to its generality, this method has been successfully applied to a broad range of problems in virtually all areas of Engineering and Physical Mathematics.

This method was first developed to be used in the analysis of aircraft structures, in the late 1950s. Its theoretical principles were established in the two following decades. The method was then effectively used to solve problems of fluid mechanics and heat transfer.

Currently, the FEM is used in commercial and academic software to simulate a wide variety of problems. In structural analysis, it is employed on steel and concrete buildings, dams, towers and on airplane and automobile structures.

The finite element method approximates the solution of partial differential equations based on the Galerkin method by systematically generating subspaces or subsets of approximation functions. In this work, the classical H1 formulation is adopted.

Tridimensional elasticity, considering pre-stress and an isotropic and isothermal material, is modelled by the following equilibrium equation:

$$div(\overleftrightarrow{\sigma} + \overleftrightarrow{\sigma_0}) + \vec{b} = \vec{0}, \text{in } \Omega$$

where $\overleftrightarrow{\sigma}$ is the Cauchy stress tensor, $\overleftrightarrow{\sigma}_0$ is the corresponding pre-stress tensor, $\vec{b} = \{b_x, b_y, b_z\}^T$ are body forces and $\Omega \subset \mathbb{R}^3$ is a closed domain with boundary $\partial\Omega$. Each component of $\vec{b}$ is a function in $L^2(\Omega)$, the space of square-integrable functions. The stress tensor is given, in linear elasticity, by the constitutive law:

$$\overleftrightarrow{\sigma} = \begin{pmatrix} \sigma_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_{yy} & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_{zz} \end{pmatrix} = \lambda\, tr(\overleftrightarrow{\epsilon})\, \overleftrightarrow{I} + 2\, G \overleftrightarrow{\epsilon}$$

where $\overleftrightarrow{\epsilon}$ is the strain tensor and λ and G are the Lamé coefficients, related to the Young's modulus E and to Poisson's coefficient ν by the following expressions:

$$\lambda = \frac{\nu\, E}{(1+\nu)(1-2\nu)}$$

$$G = \frac{E}{2(1+\nu)}$$

The infinitesimal strain tensor is given by:

$$\overleftrightarrow{\epsilon} = \begin{pmatrix} \epsilon_{xx} & \epsilon_{xy} & \epsilon_{xz} \\ \epsilon_{xy} & \epsilon_{yy} & \epsilon_{yz} \\ \epsilon_{xz} & \epsilon_{yz} & \epsilon_{zz} \end{pmatrix} = \frac{1}{2}(\nabla\vec{u} + \nabla\vec{u}^T)$$

where $\vec{u} = \{u_x, u_y, u_z\}^T$ is the displacement field.

Boundary conditions can be prescribed displacements (Dirichlet type) or external loads (Neumann type). It is also possible that both types appear in the same region, e.g., a given displacement in x and y directions and an external load acting in the z direction. A general description is a boundary condition of mixed type, in the form:

$$\overleftrightarrow{\sigma} \cdot \vec{n} = M(\vec{u} - \vec{u}_0) + \vec{g}, in\ \partial\Omega$$

where M is a matrix of scalars and $\vec{u}_0$e $\vec{g}$ are functions which each component is a function in $L^2(\Omega)$.

The FEM formulation is based on the method of the weighed residuals, which finds the solution of the equilibrium equations weighed by test functions integrated over the problem domain. The classical variational formulation $H^1$of the finite elements method (Oden 1981) is expressed by: find $\vec{u} \in U(\Omega)$, such that:

$$\int_\Omega \overleftrightarrow{\sigma} : \nabla\vec{w}\, d\Omega = \int_\Omega \vec{w} \cdot \vec{b}\, d\Omega + \int_{\partial\Omega} \vec{w} \cdot [M(\vec{u} - \vec{u}_0) + \vec{g}]\, d\omega \ \ \forall \vec{w} \in U(\Omega)$$

where:

$$U(\Omega) = [H^1(\Omega)]^3 = \{\vec{v} = \{v_1, v_2, v_3\}^T : v_i \in H^1(\Omega), i = 1,2,3\}$$

$$H^1(\Omega) = \left\{ v \in L^2(\Omega); \frac{\partial v}{\partial x_i} \in L^2(\Omega), i = 1,2,3) \right\}$$

Instead of searching for the solution $\vec{u}$ in $U(\Omega) = [H^1(\Omega)]^3$, an approximated solution $\vec{u}_{EF}$, contained in a finite subset $V^{EF} \subset U(\Omega)$ is sought. The quality of the FEM approximation is related to the capability of the adopted set V^{EF} in approximating the functions of $U(\Omega)$. In this work, polynomial functions are used, as stated in Devloo et al [4]. The approximated solution $\vec{u}_{EF}$ converges to the exact solution $\vec{u}$ as V^{EF} approximates $U(\Omega)$. The enrichment of V^{EF} is obtained by the division of mesh elements into smaller elements (h-refinement) or by increasing the polynomial order of the approximating functions (p-refinement).

2.1 Computational implementation

The simulations of this project are performed using the finite elements library PZ, described in Devloo [5]. The library is written in the programming language C++ and implements a variety of FEM approximation spaces and numerical technologies. The PZ library is structured in a modular way, which allows new functionalities to be added easily.

3 Digital twin

A tridimensional model of the spillway and main dam is constructed (Fig. 1). The original blueprints and other documents were consulted to create a geometry as realistic as possible. Details of expansion joints and internal galleries (Fig. 2) are taken into consideration in the model.
The contour lines of the valley were used to shape the bottom of the dam whereas the spillway foundation was modelled in conformity to the building plans. These regions define the boundary of the problem, where displacements are restricted.

Figure 1. Model and real[1] structures

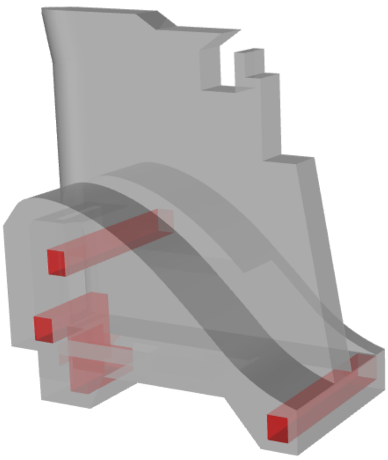

Figure 2. Details of the internal galleries on spillway block 5

[1] https://turismo.aguasdechapeco.sc.gov.br/equipamento/index/codEquipamento/2881

The generation of the mesh to be used with FEM was carried out by the software Gmsh, proposed by Geuzaine and Remacle [6]. Gmsh is a free open-source software capable of generating meshes of different geometry and with various advanced options to enhance element quality.

The resulting spillway mesh (Fig. 3) contains 130092 elements, from which 100597 are tetahedra, with sizes ranging from 0.017 m to 1.913 m and 29495 are triangular faces where boundary conditions are defined. The main dam mesh (Fig. 4), in turn, consists of 60649 elements, such that 47536 are tetrahedra, with sizes varying from 0.062 m to 2.559 m and 13113 are triangles in the boundary. Using an approximation order of 2, the spillway simulation contains 584662 equations and the dam simulation contains 230958 equations.

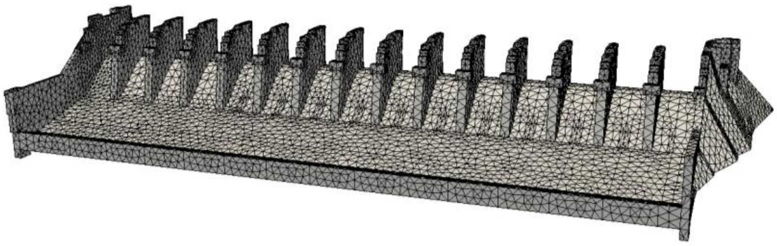

Figure 3. Spillway mesh

Figure 4. Main dam mesh

Hydrostatic pressures are applied to the downstream and upstream faces of the structures. The pressure values vary according to the water levels of the reservoir. These parameters can be set at each simulation to mimic the current state of the structure. The gates are not included in the mesh. Instead, loads equivalent to the water pressure on the gates are applied directly to the regions that support their hinges. This equivalent load is dependent on the upstream water level and on the height of the gates openings.

The spillway and the main dam are simulated separately. The interface between the dam and the spillway are included differently in each simulation. In the spillway simulation this region receives an earth pressure from the dam. Conversely, in the dam simulation it is considered as part of the foundation, meaning the displacements are restricted in this region through the application of a Dirichlet boundary condition in the direction normal to the contact surface.

It is considered that the Young's Modulus (E) of the concrete along all of the spillway is 21800 MPa and the Poisson's coefficient (ν) is 0.20. The material parameters of the different dam sections (Fig. 5), obtained from the design documents [7] are given in Table 1.

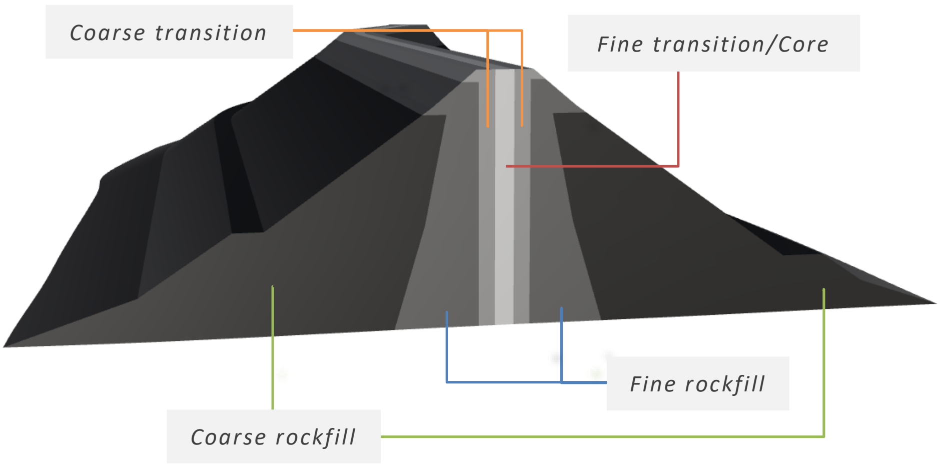

Figure 5. Different sections of the main dam

Table 1. Material parameters of different dam sections

Section	E (MPa)	ν
Fine transition/Core	100.0	0.20
Coarse transition	100.0	0.20
Fine rockfill	50.0	0.25
Coarse rockfill	35.0	0.25

The asphaltic core layer is neglected. Even though its presence is of utmost importance to ensure the impermeability of the dam, results from the design records show that this layer has little influence on the overall displacements. It was also considered that due to its small thickness, modelling this layer would yield elements with bad aspect ratio around this region. Instead, the region that comprises the asphaltic core and the two adjacent fine transition layers material are merged into a single volume and the properties of the fine transition are adopted throughout this layer.

4 Results

At this point, simulations are carried out manually in order to validate the model and the considered loads.

The stress state associated with the self-weight of the structures is used as pre-stress. This is necessary since the sensors measurements are given by the difference between the current state of the complex and the complex at the end of construction, in which the structure is deformed by the self-weight but the reservoir is yet to be filled.

The simulations are done considering a downstream water level of 264.0 m and an upstream water level of 223.0 m. The resulting displacements of the spillway are shown in the Figures 6 and 7. Figure 8 shows the displacements of the main dam. The stress in the z-direction is shown in Fig. 9.

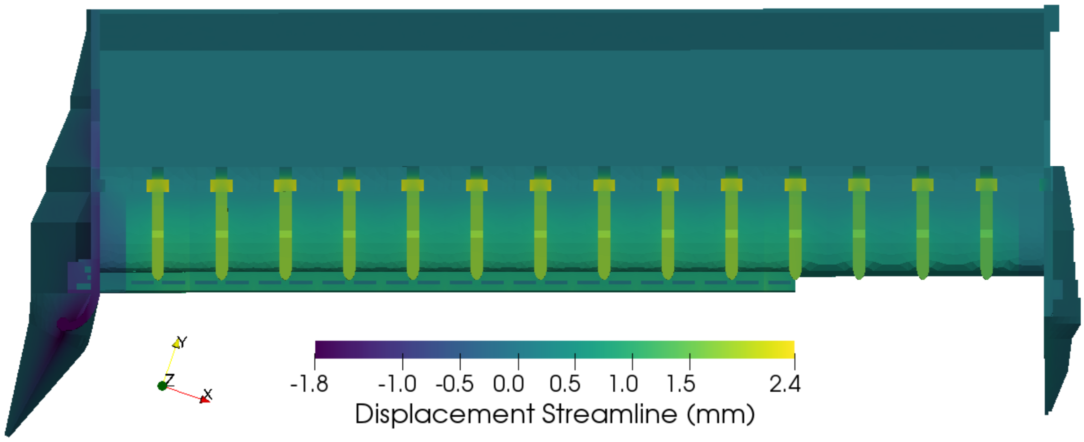

Figure 6. Displacements of the spillway in the streamline direction

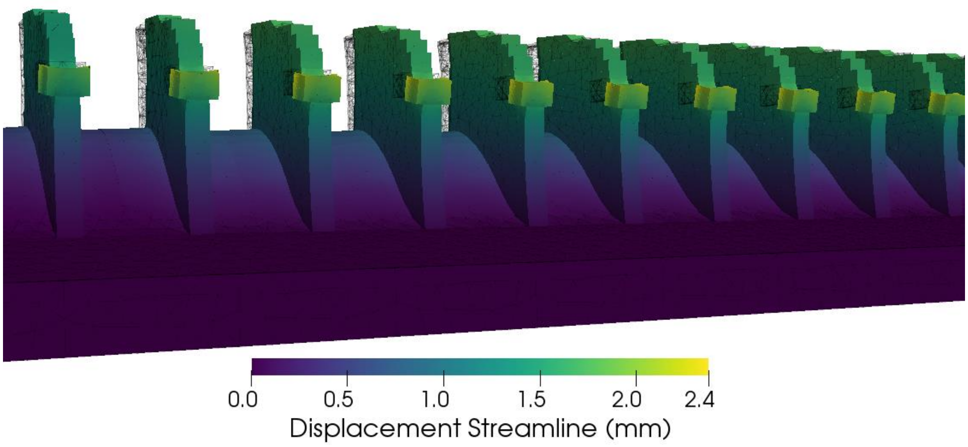

Figure 7. Detail of the deformed shape of the spillway blocks

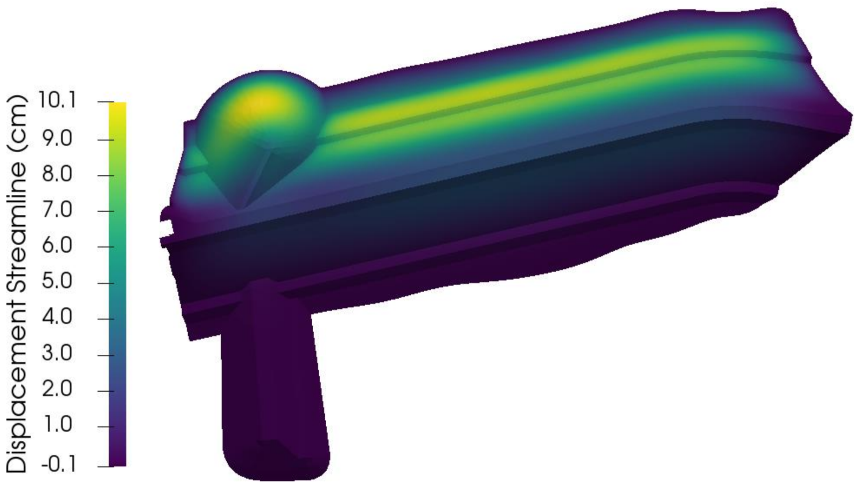

Figure 8. Displacements of the main dam in the streamline direction

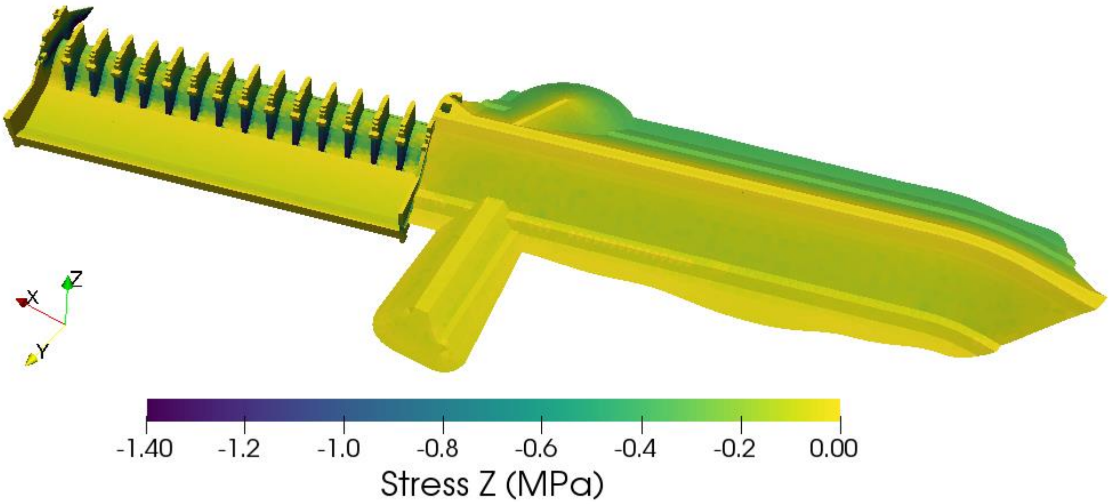

Figure 9. Stress in z-direction along the complex

Figure 10 shows displacements obtained from the inclinometer IN-D, installed on the central region of the dam, for an upstream level of 265 m and the corresponding calculated displacements at this region.

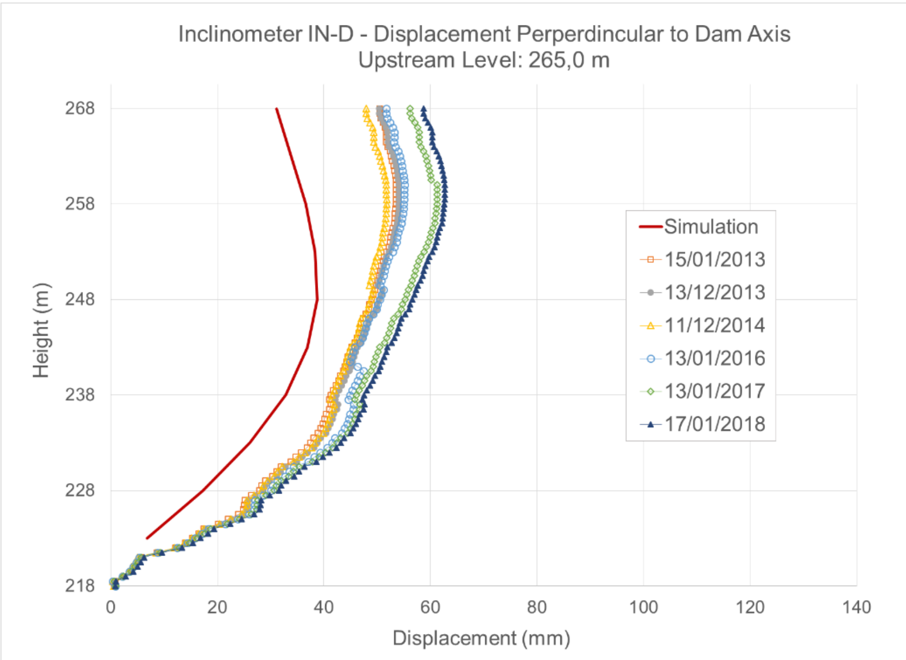

Figure 10. Comparison between calculated and measured displacements at dam center

While the distribution of both read and calculated displacements are similar, the latter have smaller values. This may be due to initial displacements, occurred during the consolidation of the rockfill, that are not taken into consideration in the model, since the region is modelled in a simplified manner as a homogeneous elastic material.

5 Conclusion

This project is under development. At this point, the digital twin yielded satisfactory results in comparison to the measurement history of an existing inclinometer. Still, most parameters were taken from the design documentation and blueprints, meaning that as-built properties may be different.

In the next steps, the system will be coupled to more sensors and to the RTS to be installed. During the monitoring period, we will be able to observe the structure behavior according to changes in the reservoir level, comparing the measured displacement to simulation results to better adjust input data, such as elasticity modulus of materials. A well-adjusted digital model would then be used in the monitoring the structure and to foresee the structures behavior in hypothetical conditions.

Acknowledgements

The work reported in this paper was performed as part of an interdisciplinary research and development project. The authors acknowledge the financial funding and support of the companies Ceran – Companhia Energética Rio Das Antas S.A., Enercan – Campos Novos Energia S.A., Foz Do Chapecó Energia S.A., under supervision of ANEEL - The Brazilian Regulatory Agency of Electricity, Project number PD-00642-1304/2018.

References

[1] Agência Nacional de Energia Elétrica. *BIG - Banco de Informações da Geração*, 2019. [online] Available at: http://www.aneel.gov.br/aplicacoes/capacidadebrasil/capacidadebrasil.cfm. [Accessed 10 Jul. 2019].

[2] UHE Foz do Chapecó. *Barragem Principal e de Fechamento - Manual de Monitoramento e Inspeções Visuais - HFC-MO2P-BPC31-2001*, 2010.

[3] A. El Saddik, Digital Twins: The Convergence of Multimedia Technologies. *IEEE MultiMedia*, vol. 25, n. 2, pp. 87-92, 2018.

[4] P. R. B. Devloo, C. M. A. A. Bravo, E. C. Rylo. Systematic and generic construction of shape functions for p-adaptive meshes of multidimensional finite elements. *Computer Methods in Applied Mechanics and Engineering*, vol. 198, pp. 1716-1725, 2009.

[5] P. R. B. Devloo. PZ: An object oriented environment for scientific programming. *Computer Methods in Applied Mechanics and Engineering*, vol. 150, pp. 133-153, 1997.

[6] C. Geuzaine and J. F. Remacle. Gmsh: A 3-D Finite Element Mesh Generator with Built-in Pre- and Post-Processing Facilities. *International Journal for Numerical Methods in Engineering*, vol. 79, n. 11 pp. 1309 – 1331, 2009.

[7] UHE Foz do Chapecó. *Barragem Principal – Estudo de Tensões e Deformações da Barragem de Enrocamento com Núcleo Asfáltico – Memória de Cálculo - HFC-MC2P-BPC02-2003*, 2009.