



A CFD ACCOUNT OF THE MEAN FLOW AND TURBULENCE OVER BREACHED EMBANKMENT DAM



Rui M.L. Ferreira¹
CERIS - Instituto Superior
Técnico - Universidade de
Lisboa



Nuno M Martins
CERIS - Instituto Superior
Técnico - Universidade de
Lisboa,



Teresa Alvarez
CERIS - Instituto Superior
Técnico - Universidade de
Lisboa



Teresa Viseu
Laboratório Nacional de
Engenharia Civil

ABSTRACT

We measure and compute the flow over a breached dam, identifying the main topological features of the mean flow (separation and stagnation surfaces, attached vortexes), secondary currents and boundary layer development.

The Navier-Stokes solver is based on the RANS with a $k-\omega$ SST closure model and a rough-wall correction, implemented in OpenFoam.

The geometry of the breached dam corresponds to a fixed instant of a model dam tested at LNEC. It depicts a homogeneous dam at the transition between the first and second erosion stages (the breach has started to deepen). Validation data is provided by PIV measurements obtained at IST.

We show that the development of the boundary layer over the crest is unlikely to be described by the Kármán momentum integral, as it is influenced by the breach crest wall curvature, secondary currents and strong lateral flow convergence.

Three-dimension separation is observed in the plunging pool. Two different attached vortices develop at the interface between the dam body and horizontal fixed wall. The first is associated to an adverse pressure gradient at the lower end of the boundary layer. The second seems originated by the flow that spreads outward from the stagnation point of the plunging flow. The attached vortices rotate in opposite directions but may both contribute to underscoring. Strong turbulence production is observed in the plunging pool. A counter rotating vortex pair is observed in the flow exiting the dam breach channel.

We discuss the significance of these structures for hydraulic erosion and underscoring. We also provide recommendations for CFM modelling of dam breaching.

Keywords: CFD, $k-\omega$, dam-breach, 3D-separation.

¹ruimferreira@tecnico.ulisboa.pt, Portugal

1. INTRODUCTION

Earth dams and dikes are prone to breaching following overtopping. Water flowing over the crest results in an erosion process that may lead to a significant or complete destruction of the dam. Erosion process is a complex process, with a strong transient character. It includes hydraulic erosion (Amaral et al. 2020) and the detachment of earth blocks that will add another layer of complexity to the process. Experimental techniques are needed to collect evidence of the relation between flow hydrodynamics and erosion processes. The characterization of the flow field, in particular, is paramount.

Velocity measurements in dam breach flows are not trivial due to lack of optical access and the impossibility to deploy intrusive instrumentation. Imaging-based techniques are mostly used to measure velocity fields on the free surface (Bento et al. 2017, Aleixo et al. 2022). Due to these limitations and the transient character of the flow, it is often impossible to experimentally investigate in detail the different phenomena acting during the breaching process.

To address this problem, we propose CFD modelling as a means to generate the 3D flow field over the breached dam and within the erosion cavity. The CFD model must, however, be shown to reproduce the actual flow field, which leads to the problem of validating the code for a flow for which there is no available detailed velocimetry data.

This paper provides a possible solution. We developed and employed a simplified experimental facility, which replicates the fundamental hydrodynamic characteristics of the phenomenon while maintaining control over critical variables. The facility generates velocimetry data sets, used to confront a Reynolds-Averaged Navier-Stokes (RANS) based Computational Fluid Dynamics (CFD) model, implemented in OpenFoam.

The simplified experimental facility features a scaled breached dam model, that preserves the shape of a live overtopped erodible dam at a specific instant (Alvarez et al. 2022). The fixed bed model allows for optical access and for a comprehensive experimental characterization of the flow field for that specific instant. The key aims of this paper are thus to compare the computational solution with the experimental data and to use the CFD results to highlight the flow structures that may influence the erosion process.

2. EXPERIMENTAL APPROACH

2.1. The fixed dam model

A dam with 0.45 m height and 1.2 m wide was constructed in a facility at the Portuguese Laboratório Nacional de Engenharia Civil (LNEC), as a part of an experimental setup detailed in Amaral et al. (2019) or Amaral et al. (2023). It was tested in overtopping conditions, under an approximately constant hydraulic head. In the transition between the initial and second erosion stages, the test was interrupted in an almost instantaneous way. This allowed for breach characterization with a Kinect sensor for Xbox One from Microsoft. Kinect is a depth sensing device composed by two internal cameras (RGB and IR-camera) that allows simultaneously to acquire colored and depth images. The post-processing of the acquired data allowed the creation of points clouds, which, led to 3D reconstructions of the embankment. The interruption is required because the water flow over the breach is turbid, due to suspended sediments, and this low transparency disturb the Kinect acquisitions (Alvarez et al. 2019).

The post processing of the depth images was performed with the software CloudCompare. It included the point cloud alignment to the same coordinate system; removal of all points that are not the breach dam (e.g., channel walls); merging of the point clouds and noise removal and smoothing of the final cloud. This procedure allowed for the reconstruction of the breach surfaces and 3D characterization of the erosion cavities (Figure 2).

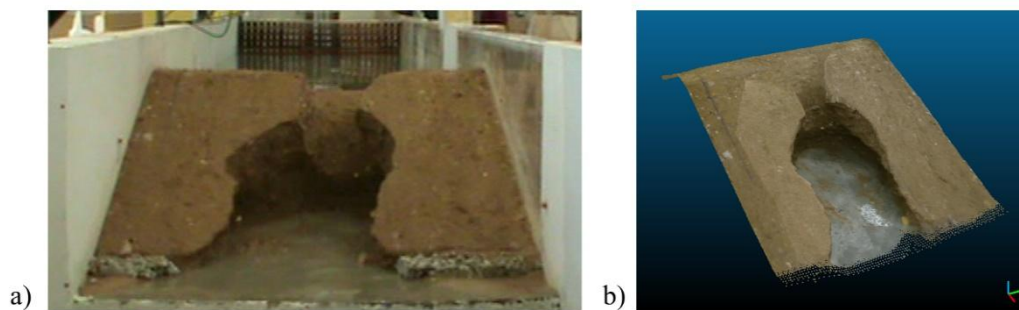


Fig. 1 – a) Downstream view of the live breached dam. b) Perspective of the 3D model.

The surface data depicting the 3D geometry of the breached dam was then converted in a solid mesh to be 3D-printed, to obtain a non-erodible scaled model (Figure 2). This scaled model as used in a smaller flume at Instituto Superior Técnico, under controlled flow conditions and susceptible to be studied in detail with optical access and without the constraints associated with transient flows. To allow the optical flow for Particle Image Velocimetry (PIV) having observed that the flow was essentially symmetric, only half of the dam was printed. The cut section was selected considering the thalweg of the breach channel (Figure 2).

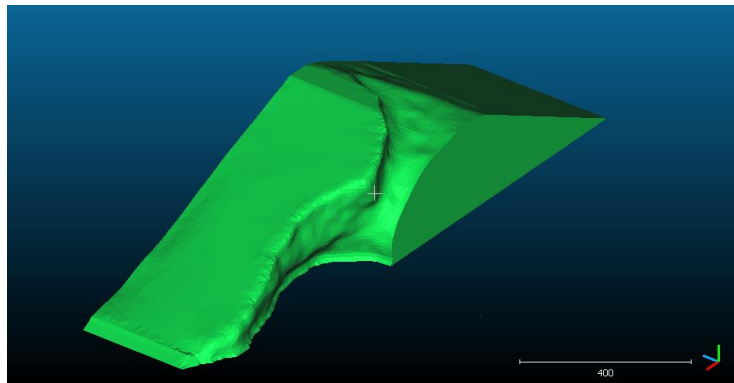


Fig. 2 – View of the printable 3D model.

2.2. Experimental facility and PIV instrumentation

The measurements were carried in a close circuit flume of the Hydraulics Laboratory of Instituto Superior Técnico. The flume is 5.8 m long, 0.395 m wide and 0.40 m height. The channel can be divided into 3 sections: 1) settling chamber, 2) test section and 3) exit section. The settling chamber has a length of 1.230 m. and the transition for the test section is made by a honeycomb like structure to break the large turbulence eddies and to allow a more stable flow. The test section has a length of 3.630 m and the glass walls allow the optical access to the flume. Finally, the exit section has a length of 0.94 m. The water was evacuated from the channel by means of two orifices in the channel exit section. These orifices are connected to the flume's reservoir by means of two independent flexibles pipes, each one having a valve allowing to adjust the water level in the channel. The bottom of the channel in the test section is made of Perspex allowing optical access to the flume from the bottom, whereas in both settling chamber and exit section, the bottom is made of PVC. Water from a reservoir placed under the channel was pumped by a frequency-controlled pump, allowing a fine tuning of the flow rate. The flow rate was measured by means of an electromagnetic flowmeter placed in the channel feeding circuit and it was set at $Q = 2.75$ l/s. Figure 3 depicts the flume section used in the experiments with the PIV laser light sheet positioned above the channel and the camera placed parallel to the channel window.

To measure velocity field, a Particle Image Velocimetry (PIV) system by Dantec was used. This system consisted in a Solo pulsed laser, a camera with a resolution of 1600×1200 pixels and timer box to synchronize the laser and the camera. The camera was equipped with a Nikon 60 mm/f2.8 lens. The system was controlled by the Dantec's software DynamicStudio 7.3. The laser was placed above the channel attached to a moving frame (Figure 3), allowing to sweep the channel in the x and y directions (details on the PIV system of the IST Hydraulics lab can be found in Ferreira et al. 2011). The water was seeded with polyamide particles with $50 \mu\text{m}$ of diameter.

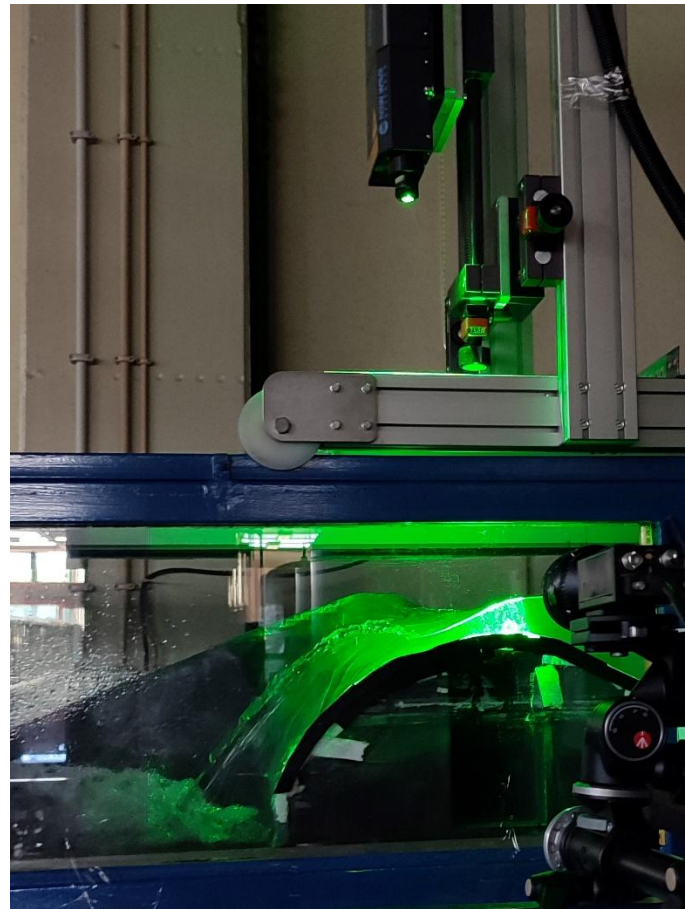


Fig. 3 – Flume and dam breach model used in the experiments. Above the channel, the laser source is placed in a rigid frame on a moving chariot.

2.3. Data acquisition procedure

It was not possible to observe all the region of interest (ROI) in the camera as this would make it impossible to resolve appropriately the seeding particles, resulting in peak-locking. Therefore, the region of interest was divided into smaller regions, hereafter referred as sample regions, and the system laser+camera was moved accordingly to sweep all the ROI. An overlap of at least 10% of the image areas was considered between adjacent sample regions. Figure 4 depicts the scheme used to cover the ROI above the crest of the dam.

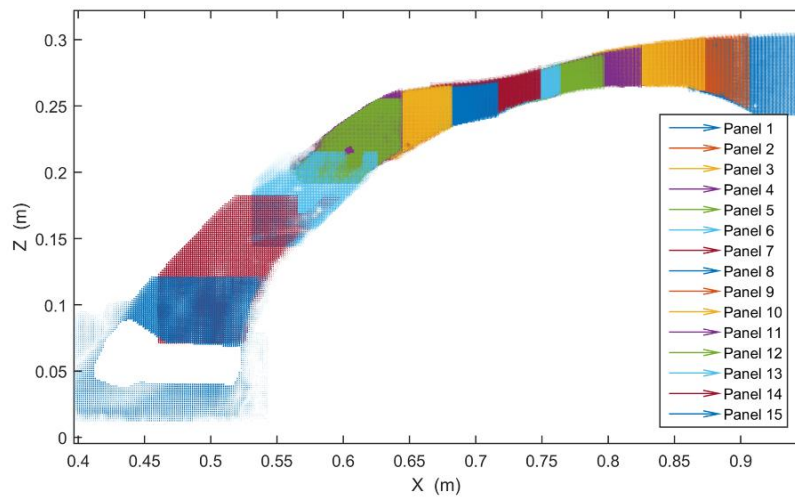


Fig. 4 – Segmentation of the ROI in sample regions and respective measurements.

The final interrogation window size was 16×16 pix corresponding to a spatial resolution between 0.6 and 1.5 mm. Each PIV acquisition was made with at least 4500 images pairs and repeated 3 times. PIV is nowadays an established technique described in detail in the literature, as in Adrian and Westerweel (2011) and references therein

Image calibration was performed using a checkerboard, with 2×2 mm squares (Figure 5). Since the distortion observed in the images was negligible, a linear conversion algorithm was applied to obtain the final velocity fields in metric units.

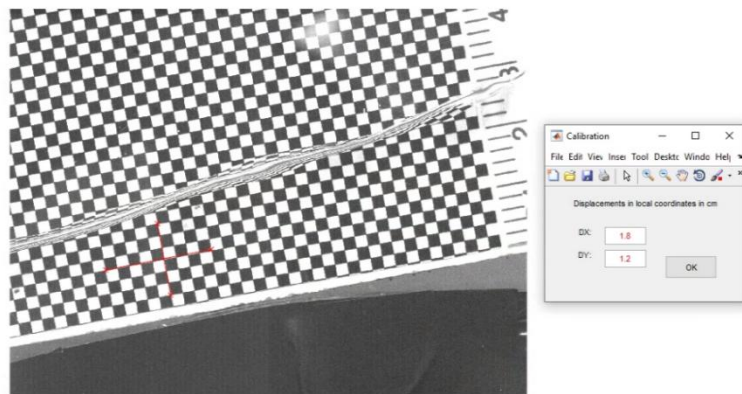


Fig. 5 – Example of calibration image over the breached dam crest.

After image calibration, masks were applied with the two following goals: (i) to restrict the analysis to the ROI, thus reducing the processing time; and (ii) to remove areas where the velocity measurements could not be trusted due to the excessive aeration and turbulence of the jet. In particular, near the plunging jet impact area, the air bubbles block the visibility of the seeding particles (Figure 6).

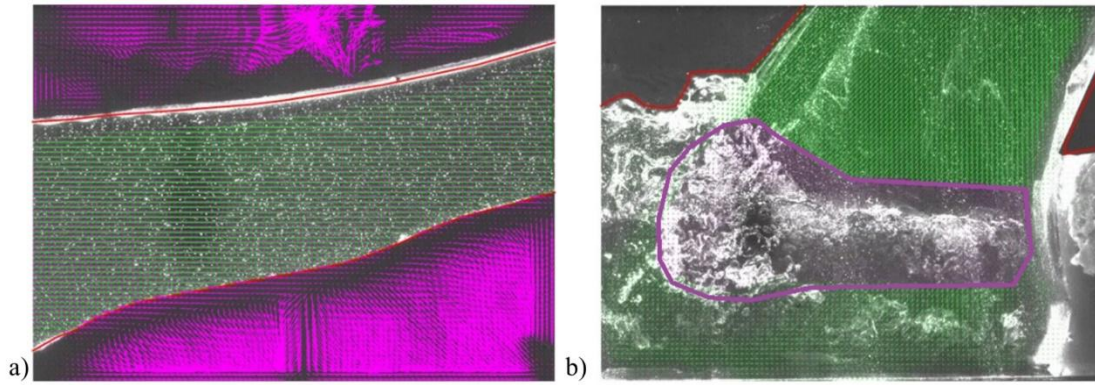


Fig. 6 – Flume and dam breach model used in the experiments. Above the channel, the laser source is place in a rigid frame on a moving chariot.

For the jet impact area, panel 15, the Reynolds shear stresses were computed as

$$\tau_{xz} = -\rho \overline{u'_z u'_x} \quad (1)$$

where ρ is the water density, u'_i velocity fluctuations in x and z directions.

3. THE CFD APPROACH

3.1. Conservation and closure equations

Our approach is based on Reynolds-Averaged Navier-Stokes (RANS) equations, as a widely used solution for simulating turbulent flows. The Reynolds stresses require modelling of turbulent stresses to close the system of equations. We opted for the $k-\omega$ Shear Stress Transport (SST) turbulence model, a two-equation eddy-viscosity model that combines the robustness of the $k-\omega$ model near walls with the free-stream independence of the $k-\epsilon$ model. Its hybrid nature makes it particularly suitable for flows with adverse pressure gradients and separation, as envisaged in this application.

The governing RANS equations can be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0, \quad (1)$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - \frac{\partial \tau_{ij}}{\partial x_j}, \quad (2)$$

where ρ is the fluid density, u_i are the velocity components in a cartesian frame $\{x, y, z\}$, where z is the vertical coordinate, against the gravity vector, p is the pressure, μ is the dynamic viscosity, and $\tau_{ij} = -\rho \overline{u'_i u'_j}$ represents the Reynolds stresses.

The $k-\omega$ SST model introduces two transport equations for the turbulent kinetic energy k and the specific dissipation rate ω :

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = P_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (3)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1-F_1) \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, \quad (4)$$

where P_k is the production of turbulent kinetic energy, and $\mu_t = \frac{\rho k}{\omega}$ is the turbulent viscosity.

The blending function F_1 ensures a smooth transition between the $k-\omega$ and $k-\epsilon$ formulations. It is formulated as

$$F_1 = \tanh \left(\min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\rho \sigma_\omega k}{CD_{k\omega} y^2} \right]^4 \right). \quad (5)$$

The $k-\omega$ SST model is implemented in OpenFOAM using the RAS turbulence modelling framework. Having been subjected to extensive validation, $k-\omega$ SST approach has been shown to provide credible solutions in 3D flows with separation, recirculation, and wall-bounded turbulence.

3.2. Considerations on discretization and numerical solution

The discretization of the governing equations is performed using a finite volume method (FVM) on a computational grid composed of structured hexahedral elements. Hexahedral meshes, implemented in OpenFOAM, are preferred due to their higher numerical accuracy, efficiency, and robustness in resolving flow fields. This is particularly important in flows with strong gradients, such as boundary layers and recirculation zones, expected in the flow in the erosion cavity of the dam.

In the finite volume framework, the computational domain is divided into discrete hexahedral control volumes, and the governing equations are integrated over each volume. Fluxes of mass, momentum, and energy are evaluated at the faces of the control volumes using appropriate interpolation schemes, such as linear upwind or central differencing. The face-normal gradients are computed using second-order accurate schemes to ensure consistency and accuracy. Hexahedral grids also facilitate structured alignment with the flow direction, minimizing numerical dissipation and improving the resolution of anisotropic turbulence structures.

The numerical mesh is built around the digital model of half the dam, to mimic the experimental conditions (Figure 7).

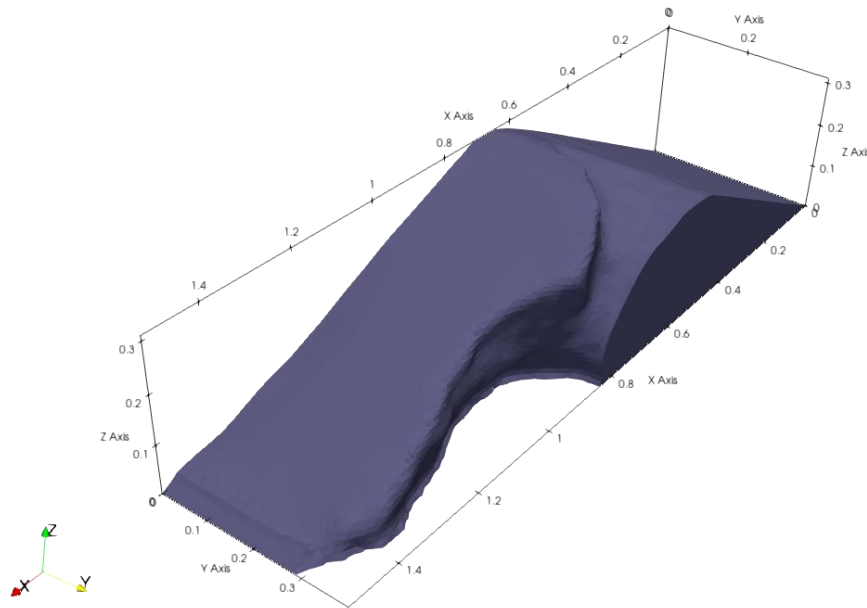


Fig. 7 – Digital model of the breached dam.

OpenFoam includes a combination of interpolation techniques, robust time-stepping methods, and boundary conditions while ensuring the numerical stability and accuracy required to resolve complex hydrodynamic phenomena.

Interpolation and differentiation schemes determine the accuracy and stability of the numerical solution. For convective fluxes, we employed linear upwind differencing (LUD) to balance accuracy and stability, particularly in regions with steep gradients. It uses flux-limiting functions to prevent spurious oscillations near discontinuities. For diffusive fluxes, central differencing as employed, providing second-order accuracy by averaging values at cell faces. Cell-face values are reconstructed using appropriate linear or non-linear interpolation techniques.

Time-stepping is handled using a semi-implicit scheme based on the backward Euler method and the second-order implicit Crank-Nicholson scheme, unconditionally stable and well-suited for (URANS) transient simulations. The time step was adjusted to ensure numerical stability.

The boundary treatment is critical to ensuring the accuracy of the numerical solution and maintaining physical fidelity. An inner region wall function was employed with near-wall grid refinement. For the inlet, we prescribe the water flux as well as turbulence quantities (e.g., k and ω). A zero-gradient Neumann boundary condition is used to allow free outflow. Cyclic boundary conditions are applied when the flow geometry or physical configuration allows, reducing computational cost and ensuring consistent treatment of periodic domains.

4. EXPERIMENTAL DATA AND NUMERICAL RESULTS

The mean velocity field over the dam crest, measured with PIV is shown in Figure 8. Dimensionless velocity profiles along the crest are plotted in Figure 8a) showing the development of the boundary layer. Figure 8 shows that: i) the thickness of the flow layer increases from upstream to downstream (right to left in Figure 8b); ii) the mean velocity increases from upstream to downstream; and iii) the thickness of the boundary layer increases along the breach channel.

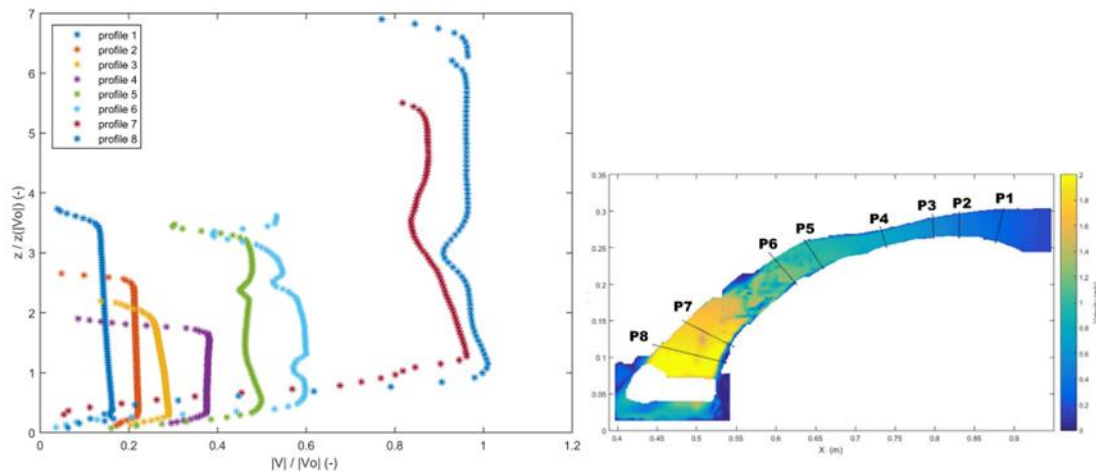


Fig. 8 – a) Mean velocity field along the breach channel; b) velocity profiles in the 7 cross-sections identified in the left panel.

As the flow depth increases, the pressure gradient becomes adverse, eventually past the threshold of separation. The last two profiles in Figure 8a) already show a curvature indicating that the flow is near the onset of separation. Separation eventually takes place near the end of the breach channel and comes to occupy a region adjacent to the bottom of the flume. Figure 9 reveals the existence of an anti-clockwise rotational structure inside the separated region (note that the flow is from the right to the left in Figure 9).

The flow in Figure 9 corresponds to the plunging jet impact area. The central part has been masked out since air entrainment would reduce the quality of the velocity data. Below the region where air entrainment is the dominant feature, the flow encounters the bottom, generating a stagnation point. The flow curls clockwise downstream (to the left in the figure), to exit the dam cavity. Upstream the stagnation point (to the right, in the figure), the flow circulates in the separated region.

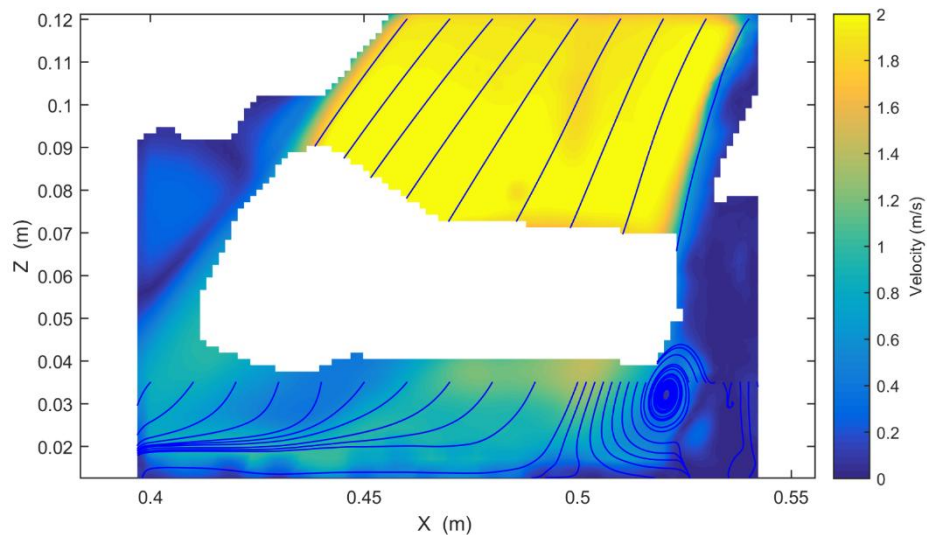


Fig. 9 – Mean velocity and stream lines at Panel 15, at the impact area of the plunging jet. Note vortical structure inside the separated region and the stagnation point at $x \approx 0.5$ m.

Reynolds shear stresses in this region are shown in Figure 10. High values of shear stresses are found on the separated region and in the developing boundary layer downstream the stagnation point. Around the locus of the stagnation point turbulence is strongly suppressed, as the pressure gradients are very high, rendering the lowest values of the Reynolds shear stresses.

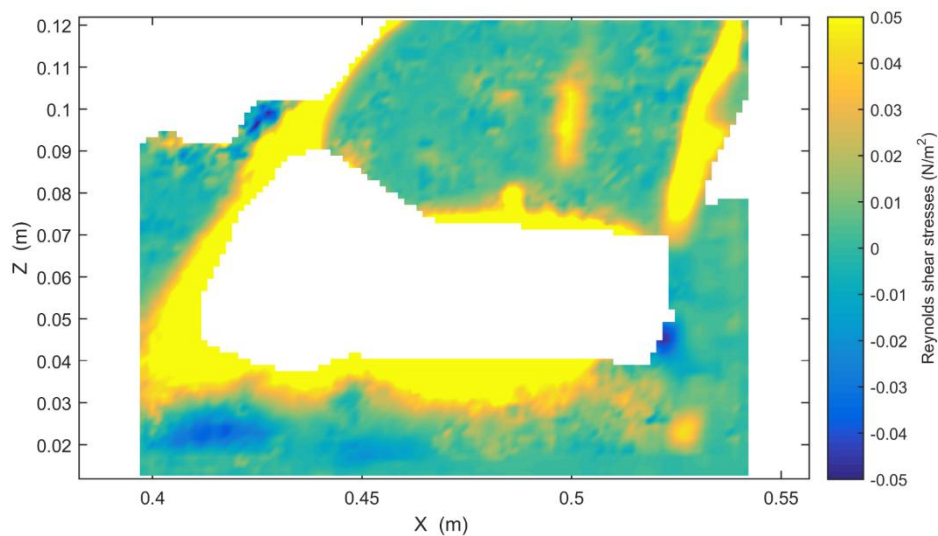


Fig. 10 – Reynolds shear stresses in the plunging jet impact area.

We compared the results of the CFD numerical simulations with the experimental results in order to assess the potential of the unsteady RANS solution to describe the dam-breach flow.

Complete verification and validation, with mesh sensitivity analysis or mesh convergence, and analysis of spatial error distribution is out of the scope of this work.

An overview of the solution is depicted in Figure 11. The simulation correctly captures the convergent flow into the breach, including the increase of flow depth. The latter feature is revealed to be the consequence of the diversion of a substantial part of the discharge in the from the breach wall to the centre of the breach. The laterally diverted flow pushes upwards the flow in the centre of the breach (represented in the simulation as the symmetry no-friction wall).

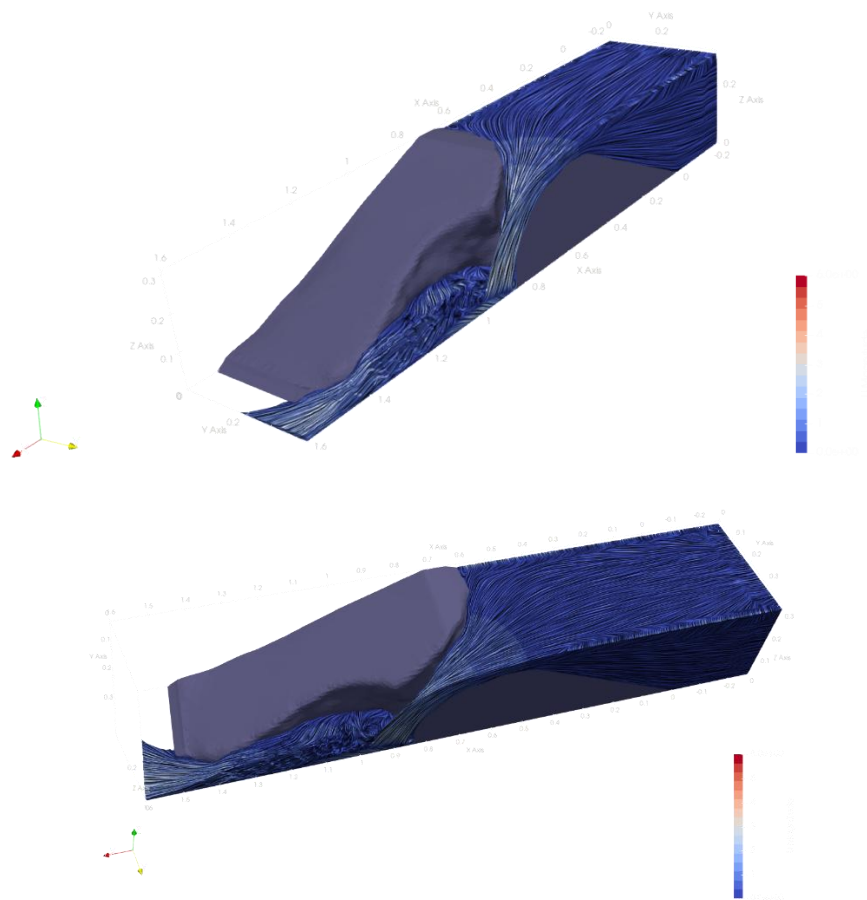


Fig. 11 – General aspect of the velocity field obtained with the RANS-based CFD solution (velocity magnitude and streaklines)

A closer look at the structure of the flow in the breach channel, for instance in the cross-section depicted in Figure 12, reveals that the lateral diversion of the flow creates a complex topology. The detailed flow is shown in Figure 13. The lateral diverted flow curls to the channel wall to form an attached vortex but is also directed upwards to the free-surface. This is the result of the interaction with the flow nearest to the symmetry wall, that develops similar structures. The result is saddle point topological structure that separates the flow adjacent to the cavity wall and adjacent to the symmetry wall and forms a stagnation surface between the upward and

downward flow, encompassing the later separated cells. This topological structure has not been confirmed by observations.

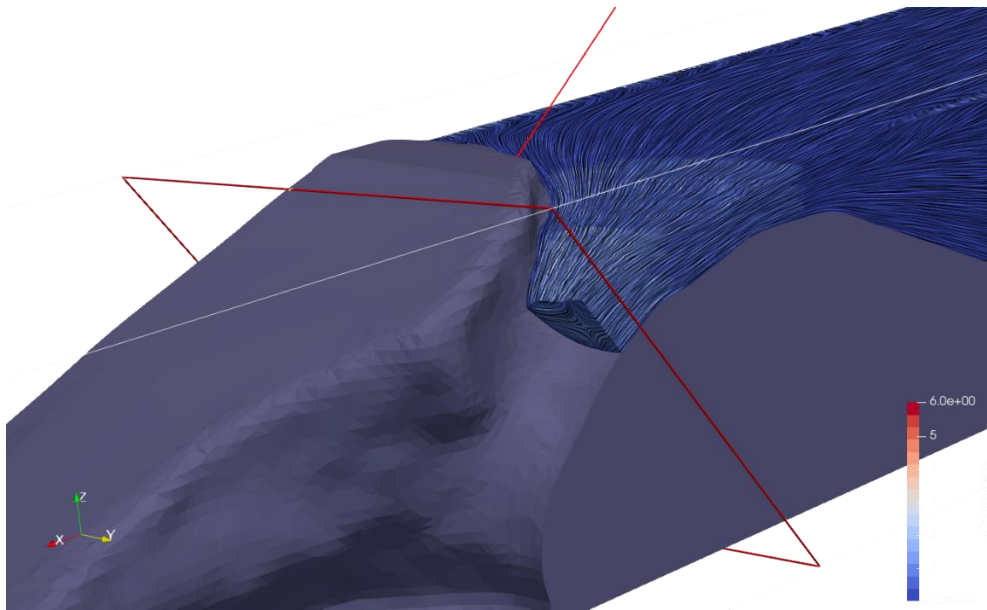


Fig. 12 – Cross-section in the flow in the breach channel

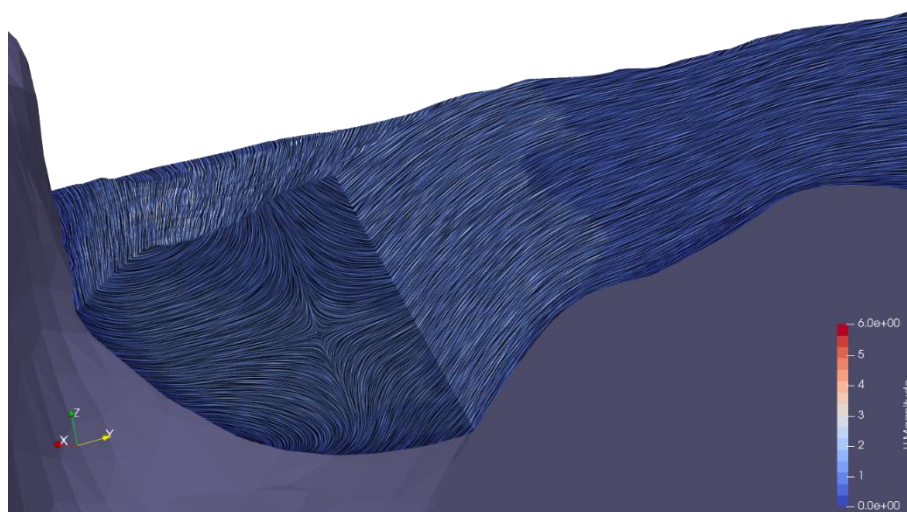


Fig. 13 – Detail of the cross-sectional flow in the breach channel. Note the formation of a saddle point.

As a result of the complex topological structure, the development of the boundary layer over the crest is unlikely to obey to the Kármán momentum integral, as it is influenced by the breach crest wall curvature, secondary currents and strong lateral flow convergence.

The plunging jet is, as seen, concentrated around the centre of the breach channel. As it hits the channel bottom wall, it spreads laterally, downstream and upstream. There is thus a stagnation point, associated to a maximum of pressure in the bottom wall. The stagnation point in the numerical solution can be seen in Figure 14.

Above the stagnation point, the flow topology features a saddle point, not confirmed in the PIV measurements. It should be noticed that the saddle point occurs at the symmetry wall while the PIV measurements were taken at 3 mm from the physical channel glass wall.

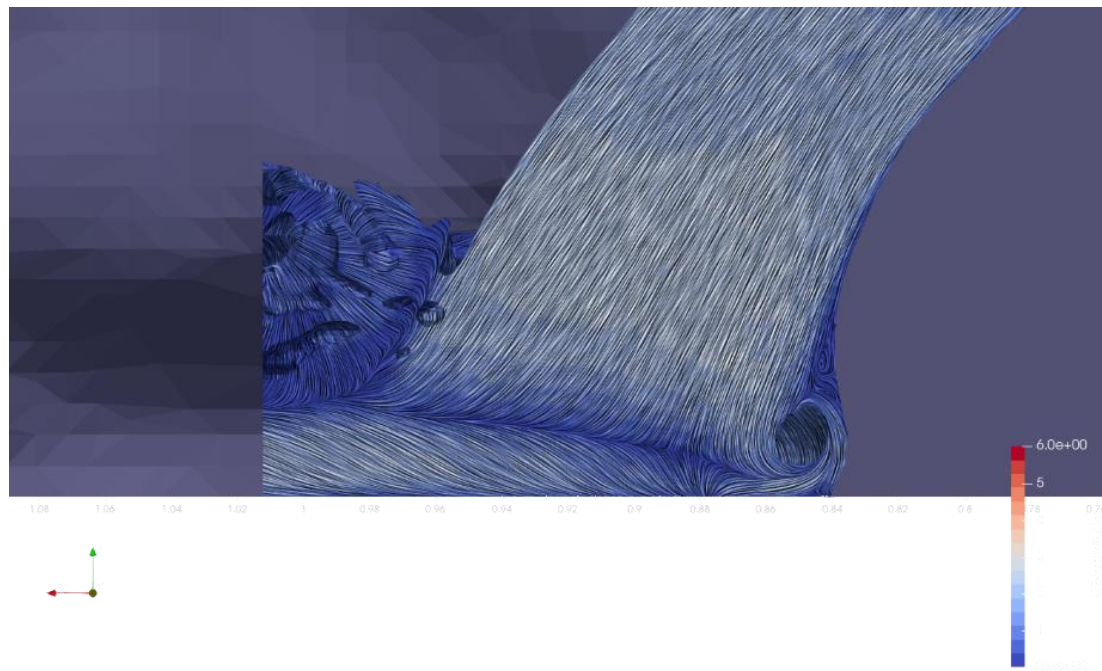


Fig. 14 – Separated region and stagnation point in the CFD solution for the plunging jet region.

In the upstream direction, the flow curls into a separated region, the same observed experimentally, as seen in Figure 14. The separation of the boundary layer along the breach channel is resolved with great detail. The numerical solution suggests that the separated zone is populated by three vortical structures. From upstream to downstream, the flow separates and forms a small counter-clockwise rotating cell, followed by a compatibility cell (clockwise rotating). Lastly, adjacent to the channel bottom, the large counter-clockwise rotating cell occupies most of the separated region.

Downstream the stagnation point, there is evidence of a longitudinal of a helical cell (the streaklines detach from the bottom wall to attach to the separation line issued from the saddle point), presumably a part of counter-rotating vortex pair conducting flow out of the erosion cavity. The experimental rests confirm the formation of a helical cell, since the stream lines seem to converge to a separation line (Figure 9). The helical cell seems less strong in the measurements though.

Figure 15 shows a cross-section of the flow in the erosion cavity. A detailed depiction of the cross-section is shown in Figure 16.

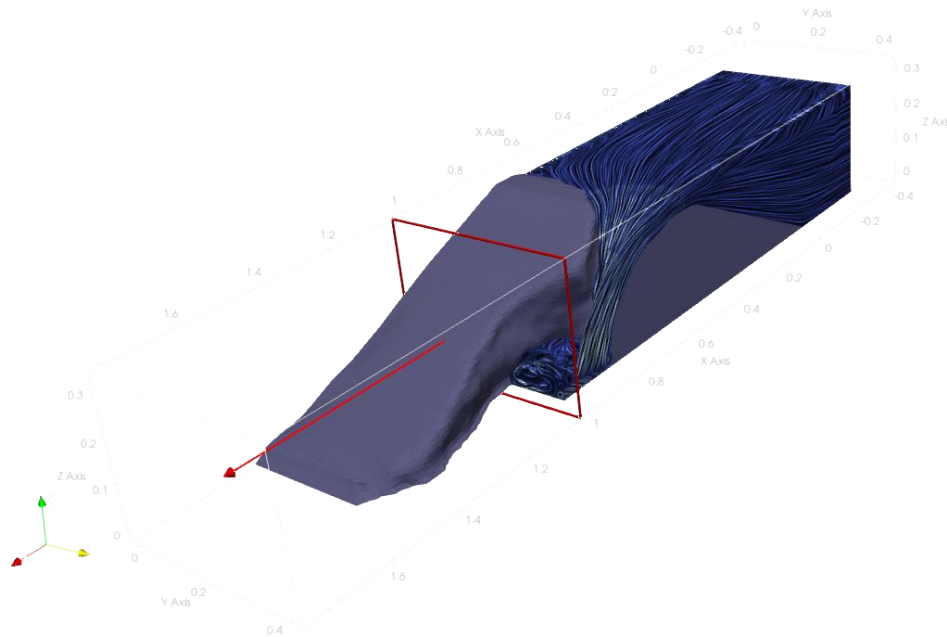


Fig. 15 – General aspect of the flow and cross-section to show the flow in the erosion cavity.

Figure 16 shows that the left half of counter-rotating vortex pair is clearly visible attached to the symmetry wall. The main trend of the flow is from the symmetry wall to the dam cavity wall. As it hits the cavity wall, the flow curls back and forms a large counter-clockwise rotation flow, driving the flow out of the cavity.

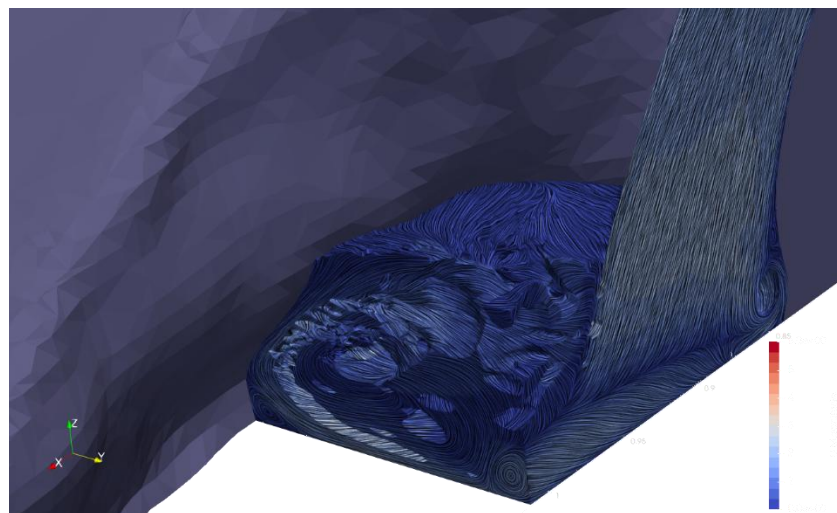


Fig. 16 – Flow in the erosion cavity. Notice the main helical cell generated by the curling back of the lateral flow, cell attached to the symmetry wall the and the separated cell around the rim of the cavity.

However, Figure 16 also reveals that the later flow separates before reaching the cavity wall. There is apparently a separated region running along the rim of the erosion cavity with clockwise rotating cell. This structure is unlikely to have been originated in the boundary layer separation cell at the end of the breach channel, since the rotation is opposite.

This jet impinging on the plunging pool generates a complex flow pattern of vortices with a strong air-water emulsion. This complex flow pattern is the responsible for the erosion of the dam's downstream bank. The simulation results suggest that erosion is mostly driven by the separation structures at the end of the breach channel and around the cavity, both contributing to undercutting and upstream and lateral progression of the erosion cavity. The lateral flow towards the cavity walls should also be responsible for significant erosion in the cavity walls.

5. CONCLUSIONS

We have used CFD simulation results to identify structures that may be relevant to influence the erosion process of the overtopped dams. Measuring live dam breach flows is currently beyond technical capacities, hence the need for CFD approach. But the need to validate the CFD simulation tool still remains. We thus used a simplified 3D printed model of a breach dam to generate a stationary flow representing one instant of the transient flow over the breach crest. We performed long-duration measurements using PIV allowing to determine the mean velocity field and the Reynolds stresses. We employed the results to compare with the numerical simulations.

Overall the agreement between the observations and the results of the numerical simulations is good. We further used the simulation results to highlight the complexity of the flow dynamics in the breach channel and erosion cavity. We showed that the flow in the breach interacts with the symmetry wall, forming complex topological structures, including saddle points and attached vortices. The strong lateral gradients in the breach channel, influenced by boundary curvature and attached cells should defy classical models like the Kármán momentum integral to describe the evolution of the boundary layer.

The boundary layer along the breach channel separates as it reaches the bottom wall. The plunging jet impacts the bottom wall of the breach channel, creating a pressure maximum, in the stagnation point, that drives lateral and longitudinal flow, upstream and downstream. In the erosion cavity, the flow topology becomes increasingly intricate, with helical cells and counter-rotating vortices. The flow from the symmetry wall curls towards the cavity wall, forming large counter-clockwise rotations and a detached cell along the cavity rim. These structures contribute significantly to the erosion process. The simulation suggests that erosion is predominantly influenced by the separation structures at the end of the breach channel and around the cavity, emphasizing the role of lateral flow in undercutting and cavity wall erosion.

These findings underline the interplay between flow dynamics and non-hydraulic erosion mechanisms in overtopped dams.

ACKNOWLEDGEMENTS

This work was partially supported by national funds through Portuguese Foundation for Science and Technology (FCT), project DikesFPro PTDC/ECI- EGC/7739/2020, strategic funding UIDB/04625/2020 for the research unit CERIS (DOI: 10.54499/UIDB/04625/2020) and the Doctoral Grant PD/BD/127801/2016.

REFERENCES

- [1] Adrian, R.J. & Westerweel, J. (2011). Particle Image Velocimetry, Cambridge: Cambridge University Press.
- [2] Aleixo, R. Amaral, S., Mendes, S.V., Viseu, M.T., Alvarez, T., Fraga Filho, C.A. & Ferreira, R.M.L. (2022). Challenges of Surface Image Velocimetry - The dam-breach example. 39th IAHR World Congress, Granada, 19-24 June 2022.
- [3] Alvarez, T., Valente, S., Amaral, S., Viseu, T., & Ferreira, R. M. L. (2019). Sensitivity analysis to initial conditions in dam breach by overtopping. 38th IARH World Congress – Water Connecting the World, Panama, 1-6 September 2019.
- [4] Amaral, S.; Caldeira, L.; Viseu, L. and Ferreira, R.M.L. (2020) Designing Experiments to study Dam Breach Hydraulic Phenomena. Journal of Hydraulic Engineering. 145(4):: 04020014-1-04020014-17 [https://ascelibrary.org/doi/10.1061/\(ASCE\)HY.1943-7900.0001678](https://ascelibrary.org/doi/10.1061/(ASCE)HY.1943-7900.0001678)
- [5] Amaral, S.; Viseu, T. and Ferreira, R.M.L. (2019) Experimental methods for local-scale characterization of hydro-morphodynamic dam breach processes. Breach detection, 3D reconstruction, flow kinematics and spacial surface velocimetry. Journal of Flow Measurement and Instrumentation. Volume 70, December 2019, Article number 101658. <https://doi.org/10.1016/j.flowmeasinst.2019.101658>
- [6] Amaral, S., Viseu, T., Zhao, G., Ferreira, R.M.L. (2023). Recent Advances in Instrumentation and Monitoring Techniques Applied to Dam Breach Experiments. In: Chastre, C., Neves, J., Ribeiro, D., Neves, M.G., Faria, P. (eds) Advances on Testing and Experimentation in Civil Engineering, pp. 299–324. Springer Tracts in Civil Engineering, Online ISBN 978-3-031-05875-2. Springer, Cham. https://doi.org/10.1007/978-3-031-05875-2_13
- [7] Bento, A.M.; Amaral, S.; Viseu, T.; Cardoso, R. & Ferreira, R.M.L. (2017) Direct Estimate of the Breach Hydrograph of an Overtopped Earth Dam. Journal of Hydraulic

Engineering. 143(6), 06017004-1: 06017004-13,
[http://dx.doi.org/10.1061/\(ASCE\)HY.1943-7900.0001294#sthash.UWJRuUjg.dpuf](http://dx.doi.org/10.1061/(ASCE)HY.1943-7900.0001294#sthash.UWJRuUjg.dpuf)

- [8] Ferreira, R.M.L. (2011) Turbulent flow hydrodynamics and sediment transport. Laboratory research with LDA and PIV. In Pawel Rowinski, editor, "Experimental Methods in Hydraulic Research", GeoPlanet-Earth and Planetary Sciences, Polish Acad Sci, Inst Geophys. pp. 67-111. ISBN: 978-3-642-17474-2, DOI: 10.1007/978-3-642-17475-9_4.