

1D, 2D AND 3D MODELING OF A PAC-UPC LABORATORY CANAL BEND

Manuel Gómez, FLUMEN Research Institute. Technical University of Catalonia. Jordi Girona, 1-3. 08034 Barcelona. Spain. Phone: +00 (34) 93 401 64 75
manuel.gomez@upc.edu

Eduardo Martínez¹, FLUMEN Research Institute. Technical University of Catalonia. Jordi Girona, 1-3. 08034 Barcelona. Spain. Phone: +00 (34) 93 401 64 75
eduardo.martinez-gomariz@upc.edu

KEY WORDS

Irrigation Canals 3D, 2D and 1D computation, gate, weir, curved reach.

ABSTRACT

The present study has been carried out to analyze the hydraulic behavior of the PAC-UPC canal, located in the Campus Nord of the Technical University of Catalonia. This constitutes a model of a real irrigation canal and, since its construction in 2003, it has been used for different studies and PhD dissertations related to irrigation canals and control algorithms. The snake-shaped canal, with the aim to minimize the space to be occupied, generates some bends along itself, where pronounced upraising water surface levels are produced. Even with a subcritical regime, clearly observable whose details require a complete analysis. In order to do these different types of approach considering a 1D, 2D and 3D analysis respectively have been proposed. Codes considered were: Hec-Ras (1D), Iber (2D) and Flow 3D, performing a comparative study of the results of each code. This comparison highlights the limitations of each one. As it was known previously, 3D results offer much more information about the flow behavior, even enabling the analysis of the recirculation zones and eddies formed in the z direction. The final results, after flow analysis, suggest possible improvement in the canal for future works and studies, regarding for instance the best location of the instrumentation (Level Sensors) or modifications about the geometry of the structure.

1. INTRODUCTION AND SCOPE OF THE STUDY

The free surface flow is a type of flow widely studied in hydraulics, particularly related with canals and fluvial dynamics. Their study can be made through 1D, 2D or 3D numerical modeling, being the most common the 1D modeling to carry out the hydraulic analysis of rivers or channels. The more and more is implemented the two-dimensional analysis, even assuming a higher computational cost, but the 3D analysis is not considered too often yet. The two-dimensional analysis is carried out in areas where the velocity components in two directions dominate over the third (deltaic areas for example) and three-dimensional analysis are performed in localized areas, where clearly three-dimensional effects (eddies and complex phenomena) occur and needing a higher computational cost.

Currently, in Campus Nord hydraulic lab in Barcelona is located a canal to test control algorithms, canal PAC- UPC. This canal tries to reproduce aspects of real irrigation canals with gates and offtakes. It was built in 2003 and since then several PhD and MSc thesis related to irrigation canals were presented, and specifically with the development of canal control algorithms. These algorithms aim at the automation of the canal sluice gates, so that in every canal reach flow, water levels can be prescribed at selected points, and sluice gates move to ensure desired water flows and water levels at those selected points (offtakes).

¹ Corresponding author

From the beginning it was observed the local over-elevation on the bend zone. Because the canal aims to reproduce a real irrigation canal and the real irrigation canal control problems, these over-elevations had to be studied and the consequences over the canal facility verified.

Such flow behavior is clearly observable visually, which suggests that hydraulic phenomenon occurs even with low velocities. Flow trajectories with abrupt changes along the bend in the horizontal plane, and 90° turns of the streamlines in the vertical plane are produced, before the pass under the sluice gate. These paths indicate that the velocity components are significant in all three directions and not only prevalent in one (1D) or two directions (2D).



Figure 1: PAC-UPC laboratory canal bend in operation.

Figure 1 shows the irregularities that occur on the free surface, most next to the bend and with the influences from the downstream sluice gate and the weir.

Numerical methods applied to hydraulic problems increasingly offer more computing power and the market has a lot of variety of softwares, including many open-sources. A powerful tool for the modeling and simulation of 3D problems is the Flow-3D program. This tool has been used for the studies reported in this work and the analysis has been supplemented by open-source software such as Hec-Ras and Iber [6]. So it has been performed a detailed flow analysis, through one-, two- and three-dimensional codes.

The main objective of this study is to understand in detail the flow patterns in this canal, when the flow encounters a bend, a weir and a downstream sluice gate. Obviously this flow behavior knowledge may lead to secondary but no less important goals, such as relocation of the measuring instrument if necessary or even the rehabilitation of the bend shape if the flow behavior requires it.

The use of 1D, 2D and 3D numerical modeling tools offers the possibility to compare results of these tools and study the pros and cons of each of these types of approach. So the aims of this study can be summarized as:

- To understand in detail the hydraulic behavior of the flow that occurs in the PAC-UPC laboratory canal curved reach.
- Comparison of 1D, 2D and 3D modeling, and analysis of their suitability as needed.
- Analysis, after the hydraulic behavior of the flow, the possibility of relocating the canal instrumentation, and modify the shape of the canal's structure.

2. CANAL DESCRIPTION

Canal PAC- UPC is the acronym of "Canal de Pruebas de Algoritmos de Control (Test Canal Algorithms Control) - Universitat Politècnica de Catalunya (Technical University of Catalonia)". As its name suggests, it is a specially designed laboratory canal for basic and applied research in the area of irrigation canal control, canal instrumentation, canal modeling, water measurements, etc. It is implemented in the Laboratory of Physical Models, Campus Nord UPC.

Its construction was intended to reproduce the characteristics of real irrigation canals and thus control problems that may appear. For this reason a canal with the maximum possible length and zero slope to produce the maximum time delay was built. Limited space in the laboratory became necessary to construct the current snake shape that optimizes the area occupied. This is a 220 m canal long, rectangular cross-section of 44 cm wide and 1 m high, occupying a total floor area of 22.5 m x 5.4 m approx.

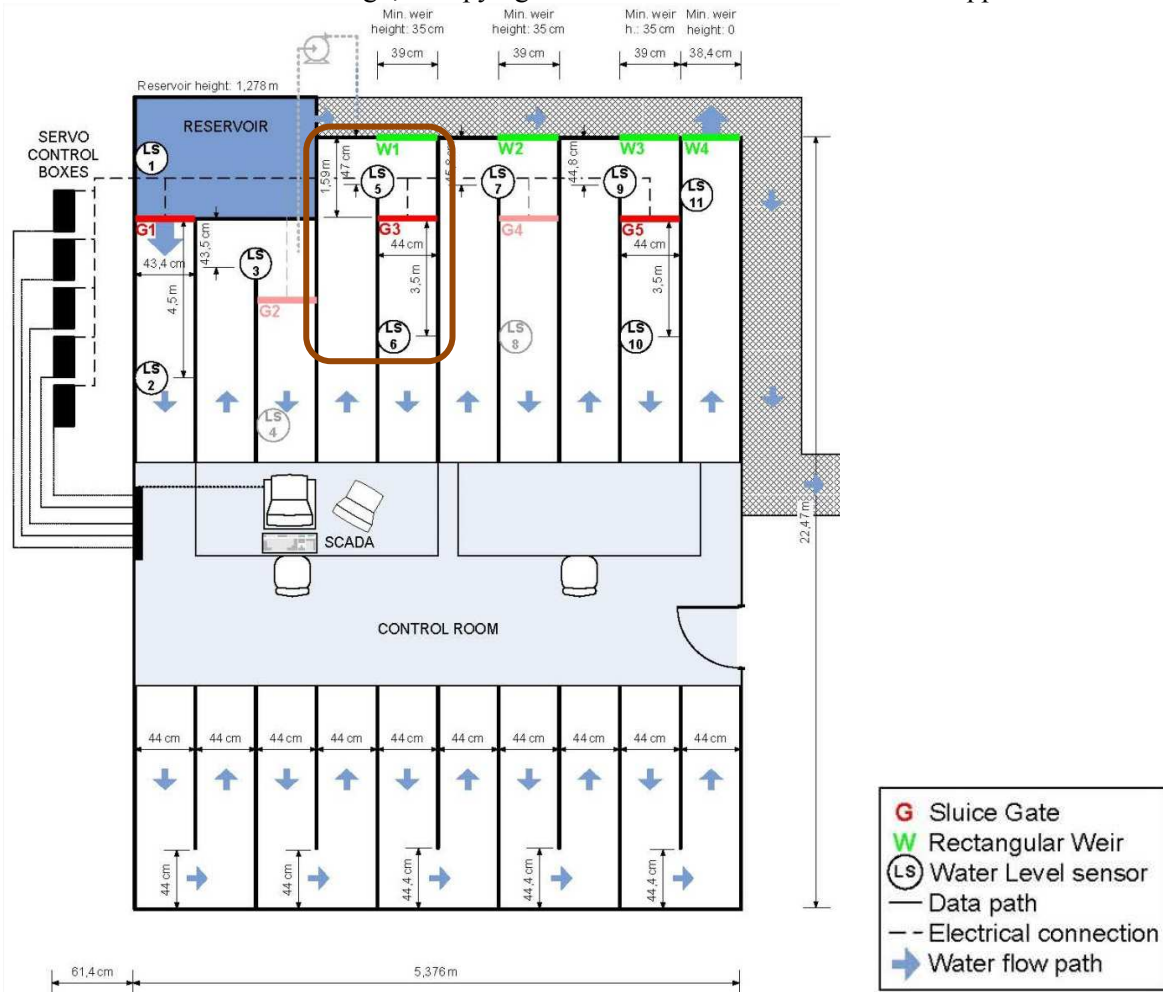


Figure 2: Detailed scheme of the whole PAC-UPC canal.

As shown in Figure 2, the elements of the installation are as follows:

- A header reservoir
- 3 vertical sluice gates (G1, G3 and G5)
- 4 rectangular weirs (W1, W2, W3 and W4)
- 9 Level sensors (LS1 to LS9)
- 1 control room

Items marked as transparent are not in operation, such as gates 2 and 4, and sensors 4 and 8, but it is possible to use if needed for future experiments. Sluice gates and weirs allow different work operations.

3. GENERAL APPROACH TO THE PROBLEM

3.1 Introduction

The study on PAC-UPC canal and more specifically in the canal bend where is located weir (W1) and sluice gate (G3), according to Figure 2.

Instrumentation needs, in particular the placement of the level sensors, took a special interest in the study of these canal bends. With this canal design it is possible to foresee how the streamlines, the more the higher the flow velocity, will perform a complex path through the canal curved reach and also water levels will be locally increased, difficult to control and determine with some accuracy unless detailed studied are made.

Based on the uncertain behavior of the flow in these bends of the PAC-UPC canal and the study of the best placement of the level sensors as accurately as possible, an experimental campaign of data collection was carried out. This consisted in the measurement of depths in 10 points throughout the canal bend for 10 different combinations (flow, sluice gate openings and height of the weir). These experimental data are basic and essential to calibrate and validate the models presented in this study.

1D, 2D and 3D numerical modeling is proposed in with the dual purpose of: 1) to study in detail this bend zone being able to determine more accurately the placement of instrumentation throughout the canal; and 2) allow the comparative analysis of 1D, 2D and 3D models results in order to determine which one offers the most and best information to this case study.

3.2 Study zone description

Four bends, three of them including a rectangular weir (W1, W2 and W3) are located in one side of the canal. Bends showing a weir are those that generate a more complex flow pattern (if the weir is operating) and so a detailed study is more interesting. In this case, the bend, where W1 weir is located, was selected. It should be mentioned that results of this study will be extrapolated to the different bends.

Figure 2 shows highlighted the study area, with water level measurements and hydraulic and instrumentation elements found there. Also noted that there is not a soft curve but it is a sudden change of direction at 90°, without rounded corners. Study area has been reproduced in 2D and 3D model drawing, which has been exported to Iber and Flow-3D program respectively. Figure 3 shows a 2D plant map with the location of the measurement points and a detailed 3D drawing.

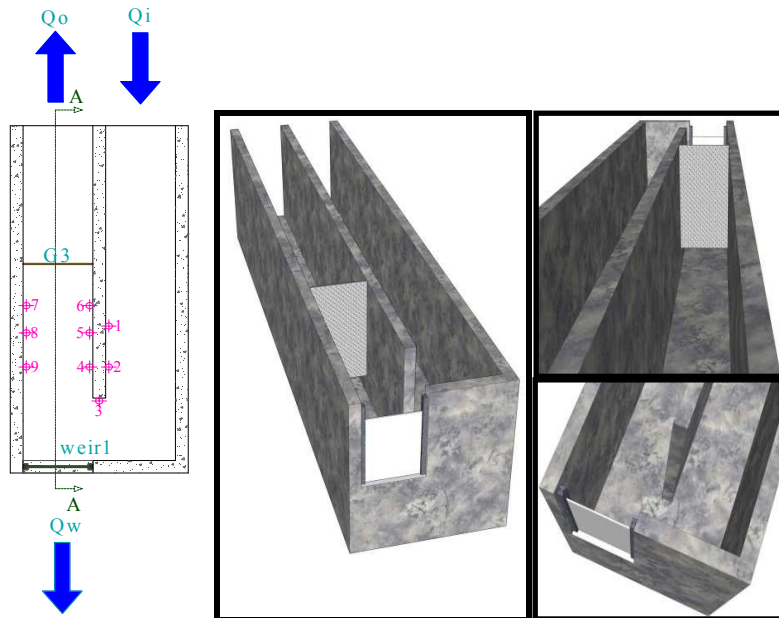


Figure 3: Study zone. Measurement points in 2D drawing (left) and 3D drawing (right).

3.3 Case Studies

The data used in the present study was acquired during 10 experiments performed in the canal in steady flow conditions. Different discharges, sluice gate openings and weir heights were combined such as the Table 1 indicates. The variables taken into account are described below:

Q_i [m^3/s]: Inflow to the study area, measured with a V-notch weir.

w [m]: W1 weir height. It is the sum of the wall concrete height (35 cm) and the added plates, less 7 mm of embedded plate in the concrete wall.

a [m]: G3 sluice gate opening.

Moreover, depth measurements were acquired at 9 locations (Figure 3), strategically distributed over the study area. Depths were measured at each of the measuring points with a ruler. Additionally LS6 level sensor readings, provides the downstream boundary condition of the study area.

Case Study	Inflow Q_i [m^3/s]	Weir Height w [m]	Sluice Gate opening a [m]	Measured Sensor Depth LS6 [m]	Depth at each point [m]								
					1	2	3	4	5	6	7	8	9
1	0.0940	0.843	0.170	0.640	0.804	0.801	0.790	0.790	0.796	0.800	0.805	0.791	0.795
2	0.0940	0.643	0.170	0.600	0.740	0.737	0.727	0.729	0.735	0.740	0.742	0.730	0.735
3	0.0983	0.443	0.170	0.545	0.620	0.621	0.608	0.609	0.623	0.620	0.615	0.613	0.608
4	0.0983	0.443	0.117	0.520	0.644	0.639	0.630	0.644	0.640	0.637	0.638	0.640	0.634
5	0.0983	0.443	0.700	0.570	0.590	0.588	0.571	0.568	0.575	0.578	0.586	0.578	0.583
6	0.0607	0.643	0.037	0.455	0.807	0.806	0.807	0.809	0.807	0.807	0.808	0.805	0.806
7	0.0660	0.643	0.073	0.510	0.767	0.763	0.762	0.765	0.769	0.765	0.766	0.768	0.765
8	0.0740	0.443	0.073	0.470	0.625	0.624	0.620	0.623	0.623	0.624	0.619	0.628	0.623
9	0.0604	0.443	0.073	0.470	0.592	0.590	0.581	0.592	0.589	0.589	0.590	0.590	0.587
10	0.0768	0.443	0.073	0.480	0.638	0.641	0.635	0.632	0.640	0.645	0.637	0.631	0.640

Table 1: Case studies. Data obtained on the PAC-UPC laboratory canal

It should be pointed out the singularities of case 1 and 5 with respect to others. As Table 1 shows for case 1, the weir height "w" exceeds values obtained in all points. This indicates that W1 weir was inoperative. On the other hand, in case 5, the sluice gate opening "a" is greater than any water depth for this case, so the gate has no influence. These two cases are necessary to further calibrate the models, since the cases in which both of the hydraulic elements (weir and sluice gate) substantially influence the flow complicate the calibration/validation of the models.

3.3 Presence of a weir and a sluice gate in the canal bend

Sluice gates and weirs are hydraulic elements studied by many different authors ([2] or [3]) but significantly different results have been obtained. It is possible to find in the literature different formulas to estimate the discharge under a sluice gate [4] or over a weir [7].

This is a clear experimental topic, since analytical studies have proposed various simplifications eventually included in the discharge coefficient. Some comments are set out below:

- The discharge coefficient assumes the velocity and contraction sluice gate coefficient uncertainties.
- The discharge coefficient is very sensitive to the sharpness degree of the sluice gate lip, and also to the different developments of the boundary layer next to the gate.
- The C_d both sluice gates as weirs, include localized energy losses.
- It is assumed that over the weir, velocities have horizontal direction. The discharge coefficient assumes the error integrating the velocity distribution between 0 and depth over the weir.
- The discharge coefficient depends generally on the weir geometry and energy head over the weir crest (h) or discharge (Q). In experimental tests in sharp-crested weirs of various geometries it has obtained values approximately $C_d=0.60$ have been obtained.

While individually the hydraulic calculation of these structures has large uncertainties, the implementation of these two close to each other, and the weir in the bend zone (no perpendicular inflow to the plane of the weir) adds even more unknowns, if possible, than the traditional study. That is why a detailed study was set out in this paper.

4. NUMERICAL MODELLING

4.1 Introduction

The experimental tests were simulated numerically using three different types of software: Hec-Ras, Iber and Flow 3D, through a 1D, 2D and 3D approach respectively. Different results are presented by each one of these, in line with their calculus limitations due to the simplifications of government equations of the problem that they do. Obviously, Flow 3D software solves the full RANS equations directly, so it offers more complete information required for this detailed analysis.

Many numerical modeling studies in 1D, 2D and 3D can be found in the hydraulic engineering literature [6]. Some of them combine the use of 1D and 3D models, the 1D analysis providing a first approximation to the problem and the 3D analysis performing a more detailed analysis. However, few comparative studies among these three approaches have been found. This can be due to the fact that they are tools designed to solve different type of problems. But performance comparison undertaken in this study is still of interest, since it can highlight the limitations and benefits of each one.

4.2 Model Calibration

1D and 2D models have been calibrated by adjusting the discharge sluice gates and weir coefficients. In the case of the 3D model, calibration was accomplished by means of the surface roughness and the adequate turbulence model. Manning's coefficient was calibrated experimentally, obtaining a value of 0.016. This value accounts for the whole dissipation energy and therefore has been included in the 1D and 2D computation. However, it is not possible to use it for the 3D computation, because of the need to input the surface roughness and turbulence instead of Manning's coefficient.

The target of the calibration is to minimize the error between experimental and numerical depths at each measuring point. The final results assure minimum achievable errors and guarantee the validity of the model, without saying this model is the only correct one for this study.

4.3 One-dimensional numerical modeling

The 1D approach has been done with the public domain code Hec-Ras. It has three types of hydraulic calculation, steady flow, variable and estimate sediment transport in moving beds. The one-dimensional Saint Venant equations are calculated only in the case of unsteady flow, directly solving the Bernoulli equation for the case of steady flow.

$$z_1 + y_1 + \alpha_1 \cdot \frac{v_1^2}{2g} = z_2 + y_2 + \alpha_2 \cdot \frac{v_2^2}{2g} + \Delta H \quad (1)$$

Where:

- z_1 and z_2 : Elevation of sections upstream and downstream, respectively, compared to an arbitrary reference plane.
- y_1 and y_2 : Depths in each section considered.
- v_1 and v_2 : Average velocity in each section. Steady flow is simply the quotient between the discharge and the flow area.
- α_1 and α_2 : Coriolis coefficients that correct the velocity distribution in each section away from the uniform.
- ΔH : Energy dissipation between cross sections 1 and 2. This term considers the continuing losses and localized as follows.

$$\Delta H = I \cdot L_{12} + \Delta H_{LOCAL} \quad (2)$$

Being I the energy gradient slope, L_{12} the distance between the two sections and ΔH_{LOCAL} are localized energy losses.

This is a steady flow study, thus Hec-Ras solves the energy balance equation using the by-step method and the canal bend has been discretized using different sections. The Hec-Ras program is intended to assume one-dimensional flow, considering negligible the other two velocity components. Also slopes are considered very small, less than 1v: 10h. In this study the canal is horizontal.

As seen in this one-dimensional analysis, the concept of turbulence does not appear. All energy dissipation is imposed on the continuous and localized losses. As said before, the Manning's coefficient was calibrated experimentally and a value of 0.016 was obtained, thus assuming that all energy dissipation was due to the energy gradient slope without considering a turbulence model. The boundary conditions for a subcritical flow are two, the discharge for upstream boundary condition and a fixed water depth (LS6 reading) for downstream boundary condition.

4.4 Two-dimensional numerical modeling

The 2D numerical modelling approach has been developed with the public domain code Iber [5]. It is a modelling system that uses high resolution schemes (FVM with TVD schemes), with a user friendly interface, compatible with GIS system. Iber modelling system has the following features:

- Simulation of free surface flow in natural and artificial channels.
- Resolution of the full Saint Venant equations 1D and 2D.
- Explicit finite volume TVD schemes.
- Wetting and Drying front with an exact conservation of water volume.
- Graphical User Interface (pre and post processing) with GiD (CIMNE.www.gidhome.com).
- GIS integration.
- Verified with different laboratory and field data.

Iber solves two-dimensional (2D) depth averaged equation, also known as 2D Shallow Water Equations (2D-SWE) or two-dimensional St. Venant equations. These equations assume a hydrostatic pressure distribution and a uniform vertical velocity distribution. It solves mass and momentum conservation equations in the two horizontal directions:

$$\frac{\partial h}{\partial t} + \frac{\partial(hu)}{\partial x} + \frac{\partial(hv)}{\partial y} = 0 \quad (3)$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -g \frac{\partial Z}{\partial x} + \frac{1}{\rho h} \frac{\partial h \tau_{xx}}{\partial x} + \frac{1}{\rho h} \frac{\partial h \tau_{xy}}{\partial x} - \frac{\tau_{bx}}{\rho h} \quad (4)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -g \frac{\partial Z}{\partial y} + \frac{1}{\rho h} \frac{\partial h \tau_{yx}}{\partial x} + \frac{1}{\rho h} \frac{\partial h \tau_{yy}}{\partial x} - \frac{\tau_{by}}{\rho h} \quad (5)$$

Being h the depth, u , v the velocity components in the horizontal x and y coordinate directions, g is gravity acceleration, z is water surface elevation, τ_{xx} and τ_{yy} , are the normal turbulent stresses in the x and y directions, τ_{xy} and τ_{yx} , are the lateral turbulent shear stresses, τ_{bx} and τ_{by} are the bed shear stresses in the x and y directions and ρ is the water density.

Iber computes the bed shear stresses by the following expressions: $\tau_{bx} = \rho C_f u |V|$ and $\tau_{by} = \rho C_f v |V|$ being $|V|$ the modulus of the velocity vector, C_f the friction bed coefficient computed by $C_f = g n^2 / h^{1/3}$ with n = Manning's roughness coefficient. And the turbulent normal and shear stresses can be formulated with different types of turbulent models: a) Constant turbulent viscosity; b) Parabolic model; c) Mixing length model turbulence; and d) k - ϵ model.

These are different models increasing in complexity to obtain turbulent viscosity and it is possible to choose the most adequate for each study. All these models are based in the formulation according to the Boussinesq's assumption to evaluate the turbulent or Reynolds' stresses. In this case, as said before, the energy dissipation is considered through the Manning's coefficient, so no turbulent model has been taken into account. The dominium has been discretized with a cell size mesh of 1 cm. On the other hand, the boundary conditions, introduced to the program, that fix the hydraulic behaviour, of the canal in subcritical regime are (the case study):

- Upstream (input): inflow (Q_i) uniformly distributed.
- Downstream (output): fixed water depth (LS6 reading) at the end of the canal bend and a weir in the bend area.

4.5 Three-dimensional numerical modeling

The three-dimensional numerical modeling has been developed through Flow-3D code [1]. It is used for numerous applications by incorporating a "multiphysics" environment and especially emphasizes its behavior for simulation of free surface flows. The numerically solved equations by Flow-3D for hydraulic studies are mass and momentum conservation equations with some additional terms:

$$\frac{\partial}{\partial x}(uA_x) + R \frac{\partial}{\partial x}(vA_y) + \frac{\partial}{\partial z}(wA_z) + \xi \frac{uA_y}{x} = \frac{R_{SOR}}{\rho} \quad (6)$$

$$\frac{\partial u}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial u}{\partial x} + vA_y R \frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z} \right\} - \xi \frac{A_y v^2}{xV_F} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + G_x + f_x - b_x - \frac{R_{SOR}}{\rho V_F} (u - u_w - \delta u_s) \quad (7)$$

$$\frac{\partial v}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial v}{\partial x} + vA_y R \frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z} \right\} + \xi \frac{A_y uv}{xV_F} = -\frac{1}{\rho} \left(R \frac{\partial p}{\partial y} \right) + G_y + f_y - b_y - \frac{R_{SOR}}{\rho V_F} (v - v_w - \delta v_s) \quad (8)$$

$$\frac{\partial w}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial w}{\partial x} + vA_y R \frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z} \right\} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + G_z + f_z - b_z - \frac{R_{SOR}}{\rho V_F} (w - w_w - \delta w_s) \quad (9)$$

Where (6) is the continuity equation and (7), (8) and (9) are the Navier-Stokes equations. In addition Flow-3D adds some components to the equations:

A_i : Fractional area open to flow in the i direction.

R : coefficient used to change Cartesian to cylindrical coordinates.

ξ : For Cartesian coordinates has a value of 0.

R_{SOR} : Mass source. You can add flow entries for example.

V_F : Fraction volume of fluid.

u_w ; δu_s : Components of relative velocities related to the mass source. For this study are not taken into account.

G_i : Part of the gravity acceleration in the i direction.

f_i : Component of the viscous forces in the i direction.

b_i : Abbreviation that only are taken into account when there are porous media. It will be 0 in calculations of this study.

Flow-3D Model calibration was made tuning parameters of energy dissipation. Such dissipation is due to surface roughness, and the effects of turbulence. Since the experimental calibration assumed that all the energy dissipation was due to the surface roughness. Manning number was calibrated, obtaining a value of 0.016. In this way the value of Manning cannot be taken into account as Flow-3D considers power dissipation by surface roughness and turbulence. In Figure 5 and 6 are shown the calibration results for roughness surface and turbulence model choice.

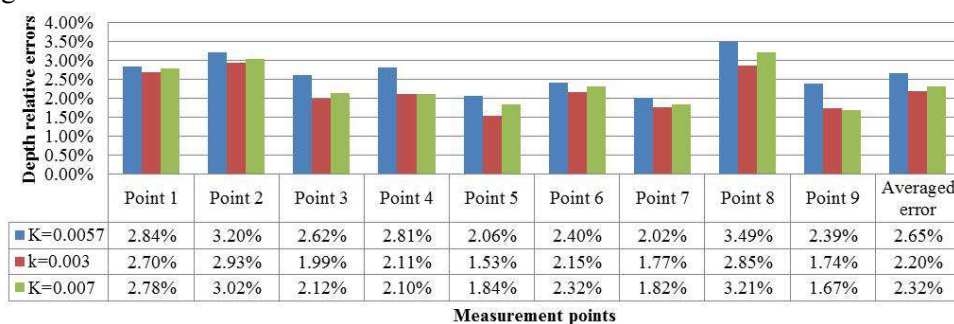


Figure 5: Roughness surface calibration in Flow 3D. Case 2 analysis.

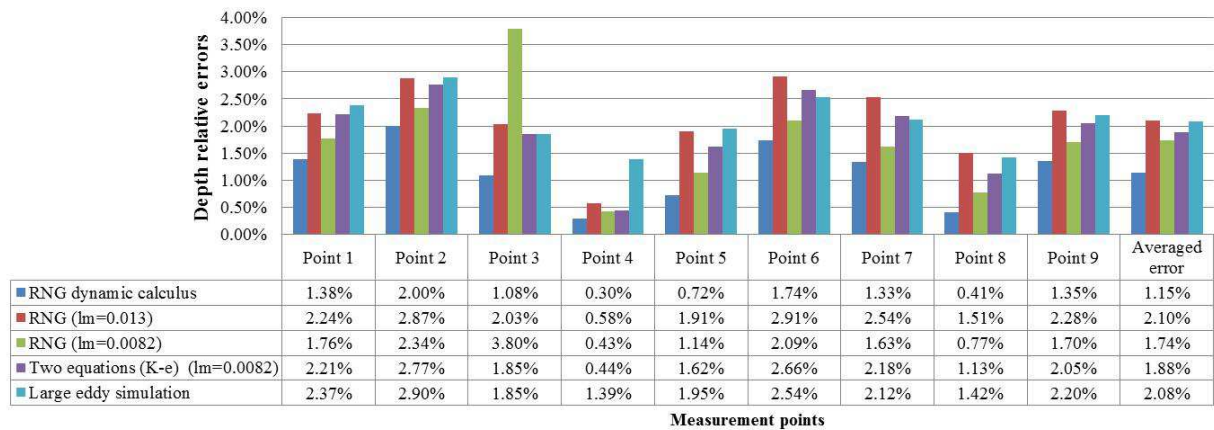


Figure 6: Model turbulence calibration in Flow 3D. Case 4 analysis.

Finally a value of 0.003 has been chosen for surface roughness and a RNG dynamic calculation as turbulence model for all the cases. It has been used three mesh blocks of cells of 1cm, 2cm and 4cm respectively. The smallest cell size (1cm) has been used exactly in the most interesting zone, from the weir to half a meter sluice gate downstream. Beside this has been constructed the 2cm cell block and finally the biggest one with 4cm cell size, close to both boundary conditions.

Concerning the boundary conditions, it has been taken into account three types of them: 1) Outflow: located in the weir; 2) Specified pressure (equivalent to fixed water level): Downstream of the study zone, using the Level Sensor 6 reading; and 3) Mass source: Located in the entrance of the study zone, with the corresponding discharge as the case study. The initial condition has been considered close to the real depth (experimental data) in each case.

4.6 Comparing results between one-, two- and three-dimensional numerical modeling

Once all results obtained, according to the three modeling approaches (1D, 2D and 3D), a benchmark exercise had been done. Each program offers the results in different formats, but it is possible to compare the obtained depths at the measurement points. A comparative analysis of the differences was performed using a histogram for each case study, obtaining a total of 10 histograms. In each of them is represented, for each measurement point, the experimental depth and numerical depths obtained with Hec-Ras, Iber and Flow- 3D.

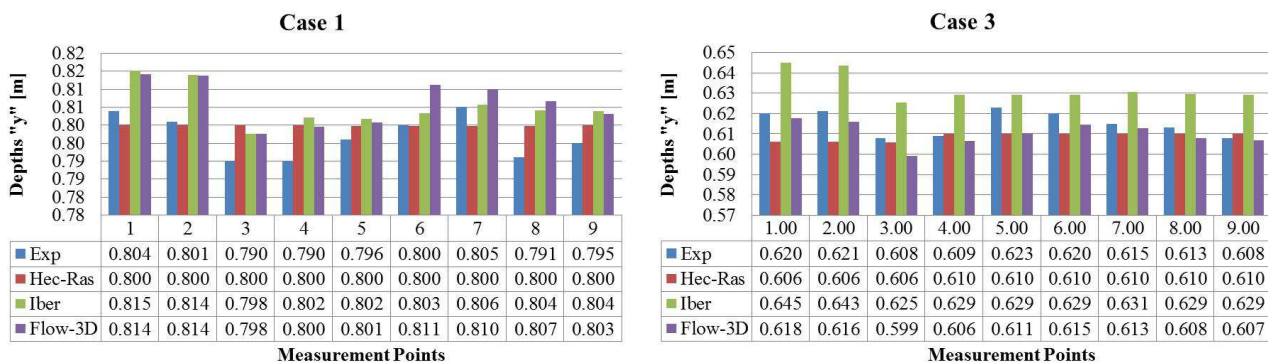


Figure 7: Comparative of the depths. Case 1 and 3.

It should be mentioned that the comparison between the results of different computer codes has not as main target to determine which program or calculation methodology is the most appropriate. Getting closer numerical depths to experimental is not indicative of a more appropriated model. In each case it is possible to compare the difference of the obtained numerical depths for each program with respect to experimental depths.

There is no defined general trend attending to the results of the histograms, but each case is independent. The obtained results by Hec-Ras in all cases are very close to experimental, but bear in mind that, in almost all points, these are average data in each section since only one depth was obtained for each couple of measurement points, and 1D approach cannot describe different water levels at the same section. So this results that in some cases differences are negative or positive depending if we are in the internal or external part of the bend. So there is no better code than another but results lead to the use of each one. Of course it should be mentioned that if the study requires only the determination of averaged water level, the use of Hec-Ras program offers an excellent results in general terms.

5. GENERAL RESULTS AND DISCUSSION

The obtained results using Hec-Ras does not provide information on the local hydraulic behavior of the study area, although the expressions of weirs and sluice gate used are well suited for a one-dimensional calculation. Iber gives much more details about the flow patterns in different cases, but neglects the vertical velocity changes, considering just average values in vertical. The study area corresponds to a clear three-dimensional behavior, where the three velocity components are important. Therefore, a 3D calculation tool as Flow-3D is extremely useful for a comprehensive analysis of the PAC-UPC canal bend.

By using a three-dimensional calculation code, it has been able to obtain the following information:

- Detail of the recirculation zones, analyzing various cross sections and observing the variation of the flow patterns. We can observe recirculation zones before the gate, next to the corners near the walls, and recirculation zones involving the three components of velocity.
 - A simulation of weirs and sluice gates, without appeal to experimental expressions of different authors, most of them 1D formula, and without a discharge coefficient which summarized the whole uncertainty of the process.
 - Detailed analysis of the streamlines, observing the different paths in the 3 spatial directions. Further measurements with a Vectrino side-looking device (Acoustic Doppler Velocimeter) will be used to confirm velocity results found with the 3D model.
- Possibility to propose and check changes in the existing structure, as well as new locations of instrumentation items if required.

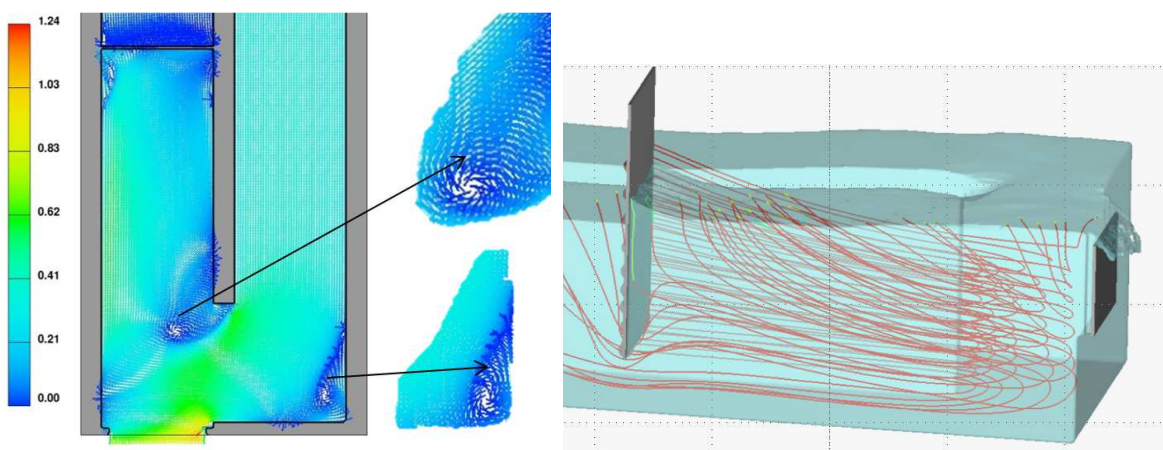


Figure 8: Velocity vector in a plant cross section, in a half depth approximately in case 3 (left) and analysis of streamlines (right) in vertical planes

6. CONCLUSIONS

This study was focused in the PAC- UPC lab canal bend, with 10 experimental case studies, varying flow, sluice gate opening, weir height, and measuring water depths at 9 measurement points strategically located in the canal.

This zone has been modeled through 1D (Hec- Ras), 2D (Iber) and 3D (Flow -3D) approaches and determining the water depth error in the different measurement points, compared with experimental data. Calibration of the 1D and 2D models has been performed based on the weir and sluice gate discharge coefficients, and using the experimentally calibrated Manning coefficient. 3D models required calibration of turbulence model used and the surface roughness value, both key concepts of the energy dissipation problem.

For 1D and 2D models there is great uncertainty regarding the weir and sluice gate discharge coefficients. These coefficients require a laborious calibration process based on reliable measures. Also Iber uses one-dimensional expressions (weir and sluice gates discharge formula), being the two-dimensional calculations performed. Flow-3D solves numerically RANS equations, which responds to a more general hydraulic case, and therefore no empirical or theoretical expressions associated with elements such as sluice gates or weirs are required.

A new proposed geometry of the bend was studied, with more rounded edges in order to a soft incorporation into the bend. It was also suggested to change the weir location since during the canal construction there was no occasion to analyze different alternatives at the time. It has been studied the different flow behavior in the bend after the change in geometry. The weir behavior is also studied in terms of changes in flow discharge, depending on new location, obtaining a 26% more discharge with the proposal location. New location is more oriented in front of the incoming flow, and rounded corners minimize the energy losses due to recirculation zones.

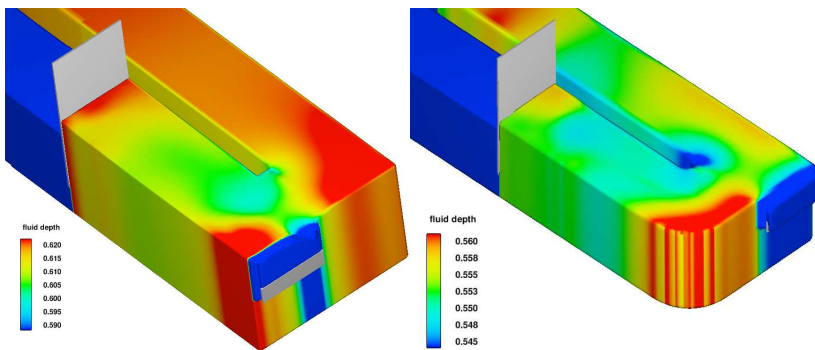


Figure 9: 3D view of fluid depth ranges. Original structure (left) and proposed structure (right).

The current sensor level 5 located at the measuring point 3 was placed there, the center wall, to protect sensor against local disturbances velocities that could distort the measurement. From the results we can conclude that the high complexity of the flow, produces a great local depression of the free surface, precisely in that area, which gives lower levels than other points in the near area. It is proposed a relocation of the sensor, closer to the gate above the gate opening, ensuring lower velocities, as shown in Figure 10.

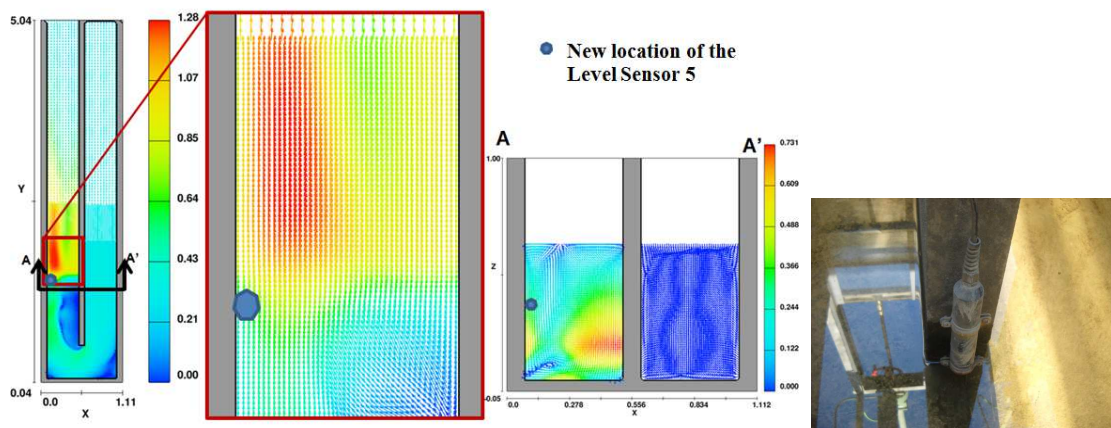


Figure 10: New location of the Level Sensor 5.

REFERENCES AND CITATIONS

- [1] Flow Science Inc, “Flow3D.” [Online]. Available: <http://www.flow3d.com/>.
- [2] Henderson, F. M., Open Channel flow. New York, USA, 1996
- [3] Henry, H. R., Albertson, M. L., Dai, Y. B., and Jensen, R. A., “Discussion of ‘Diffusion of Submerged Jets,’” ASCE, vol. 115, no. Transactions, pp. 87–694, 1950
- [4] Rajaratnam, N. and Subramanya, K., “Flow equation for the sluice gate,” J. Irrig. Drain. Eng., vol. 93, no. 3, pp. 167–186, 1967
- [5] Ros, M. de, “Modelación del flujo en un cruce de calles mediante un código de cálculo 3D.” Barcelona, Spain, p. 80, 2012
- [6] Technical University of Catalonia and Flumen Institute, “Modelo Bidimensional Iber.” [Online]. Available: <http://www.iberaula.es/web/index.php>
- [7] U.S. Department of the Interior. Bureau of Reclamation (USBR), “Water Measurement Manual.” Washington, DC, p. Chapter 7, 1997