# Iber applications basic guide

Two-dimensional modelling
of free surface shallow water flows
www.iberaula.es

UNIVERSIDADE DA CORUÑA

Luis Cea Gómez
Ernest Bladé i Castellet
Marcos Sanz Ramos
María Bermúdez Pita
Ángel Mateos Alonso

# Iber applications basic guide

## Two-dimensional modelling of free surface shallow water flows

Luis Cea Gómez
Ernest Bladé i Castellet
Marcos Sanz Ramos
María Bermúdez Pita
Ángel Mateos Alonso

A Coruña
2019

Servizo de Publicacións
Universidade da Coruña

# Iber applications basic guide. Two-dimensional modelling of free surface shallow water flows

CEA GÓMEZ, Luis (https://orcid.org/0000-0002-3920-0478)
BLADÉ I CASTELLET, Ernest (https://orcid.org/0000-0003-1770-3960)
SANZ RAMOS, Marcos (https://orcid.org/0000-0003-2534-0039)
BERMÚDEZ PITA, María (https://orcid.org/0000-0003-3189-4791)
MATEOS ALONSO, Ángel
A Coruña, 2019
University of A Coruna Press
Number of pages: 105
Contents, pages: 5-7

ISBN: 978-84-9749-717-6
DOI: https://doi.org/10.17979/spudc.9788497497176

CDU: [519.62:004.4][556.53/.54:627.13](035)*IBER
IBIC: TNF | UM | 4GE

EDITION
University of A Coruna Press
Universidade da Coruña, Servizo de Publicacións <http://www.udc.gal/publicacions>

# Contents

# 1. Introduction

This is a basic tutorial of Iber, aimed at new users who want to acquire a basic knowledge of the software. It is not intended to cover all the software's capabilities, but rather to give a flavour of the typical applications of Iber and its main elements, so that new users can evaluate whether the software is appropriate for their needs.

The basic steps to set up a computation from scratch are described in detail in each chapter, as well as the input data needed to run the model. The case studies described include Iber's hydraulic, sediment transport, and water quality modules.

Details about the equations and models implemented in Iber are not given in this document. Such information can be found in the Iber's Hydraulic Reference Manual, as well as in Bladé et al. (2014), in Cea and Bladé (2015), in Cea et al. (2016), and in the references contained in those documents.

Specific examples included in this document are the following:

- Practical Example 1 – Street intersection
- Practical Example 2 – River flood inundation
- Practical Example 3 – Bridges and culverts
- Practical Example 4 – Dam break
- Practical Example 5 – Sediment transport: bedload
- Practical Example 6 – Wastewater discharge of CBOD and ammonia in a river
- Practical Example 7 – Sewage spill in an estuary

Some examples need some digital input data to be completed, as Digital Elevation Models, ortophotos, or boundary conditions. These data is supplied together with this tutorial, and can also be retrieved from the model's webpage.

Further basic and advanced training of Iber is given via the Iber online courses (www.ibercursos.com)

### *Model references*

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, E., Dolz, J., Coll, A. (2014a). Iber: herramienta de simulación numérica del flujo en ríos. Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería, Volume 30, Issue 1, 2014, Pages 1-10, ISSN 0213-1315, DOI: 10.1016/j.rimni.2012.07.004

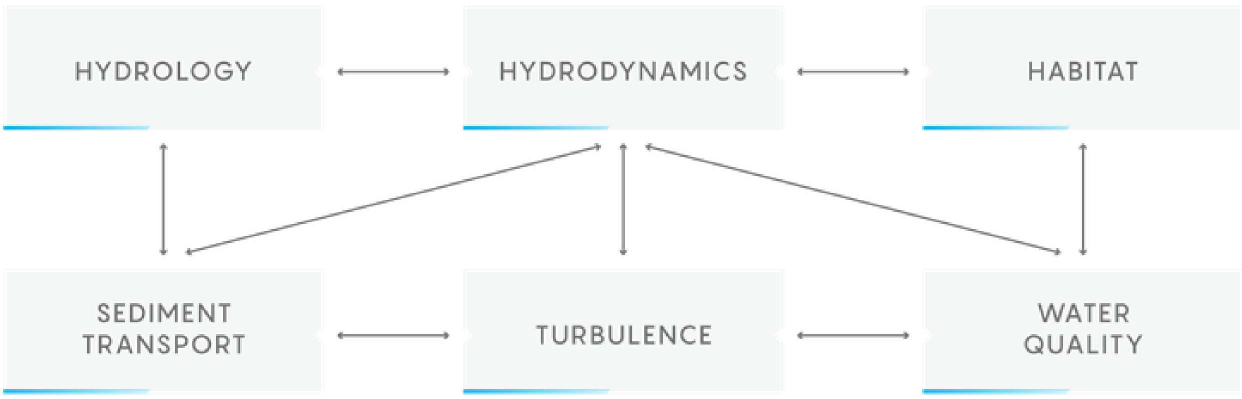
Cea, L., and Bladé, E. (2015). A simple and efficient unstructured finite volume scheme for solving the shallow water equations in overland flow applications. Water Resources Research, 51, 5464-5486. DOI: 10.1002/2014WR01654

Cea, L., Bermudez, M., Puertas, J., Blade, E., Corestein, G., Escolano, E., Conde, A., Bockelmann-Evans, B., and Ahmadian, R. (2016). IberWQ: new simulation tool for 2D water quality modelling in rivers and shallow estuaries. Journal of Hydroinformatics 18, 816–830. DOI: 10.2166/hydro.2016.235

# 2. Iber

Iber is a software package for simulating unsteady free-surface turbulent flow and transport processes in shallow water flows. The hydrodynamic module of Iber calculates the depth-averaged two-dimensional shallow water equations (2D Saint-Venant Equations). A turbulent module allows the user to include the effect of the turbulent stresses in the hydrodynamics. These are evaluated with different depth-averaged turbulence models for shallow waters of different complexity. Additional capabilities include sediment transport modelling, water quality modelling, and rainfall-runoff modelling. All the equations of the model are provided in a finite-volume non-structured mesh made up of triangle and quadrilateral elements.

A more detailed description of the model can be found in Bladé et al. (2014), Cea and Bladé (2015), Cea et al. (2016), and in the references contained in those documents. Other references, journal publications, congress proceedings and BSc, MSc and PhD theses can be found in the model's webpage.




Learn more about Iber and download it for free:

**www.iberaula.com**

Have a look to our high-level and quality online and classroom trainings:

**www.ibercursos.com**

# 3. Practical Example 1: Street intersection

## 3.1. Objectives

This chapter introduces the Iber interface (GiD) and the basic aspects of the hydrodynamic module. It illustrates the basic tools for creating and editing the geometry of a model. Using a street intersection example, we will see elements related to the definition of the geometrical properties and the importance of these regarding the mesh generation process. This exercise will also serve to illustrate the importance of a 2D model as a means of accurately reproducing certain hydraulic phenomena, such as hydraulic jumps and crossed waves.

## 3.2. Description of the case study and input data

The example consists of two perpendicular channels that represent two scaled streets with different slopes. The streets intersect on a horizontal surface (Figure 1).

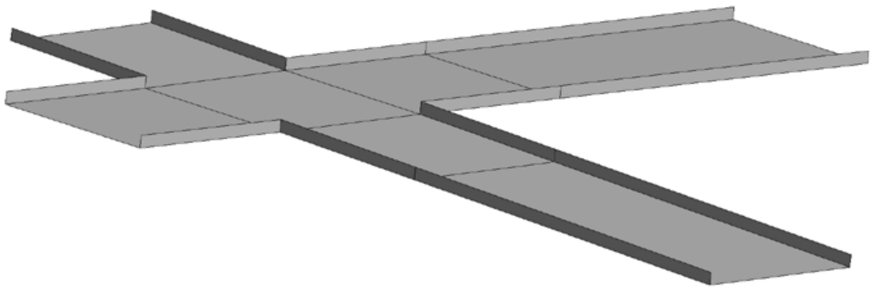


**Figure 1. Representation of the street intersection.**

This example is a laboratory case study used to analyse the flow distribution between two streets (Nanía Escobar, 1999) and to validate Iber's hydrodynamic module (Bladé, 2005). The area under study involves three flow regimes (supercritical, critical and subcritical) due to the different slopes of the streets. A hydraulic jump is formed in the intersection area.

The geometry is detailed in Table 1. Please note that all units used in Iber are defined in the International System Units (SI). Coordinates are in metres.

**Table 1. Geometric description of the streets (coordinates in meters).**

| Point | X | Y | Z |
|---|---|---|---|
| 1 | 0 | 0 | 0 |
| 2 | 0 | 1.5 | 0 |
| 3 | 1.5 | 0 | 0 |
| 4 | 1.5 | 1.5 | 0 |
| 5 | -1.5 | 0 | 0.015 |
| 6 | -1.5 | 1.5 | 0.015 |
| 7 | 0 | 3 | 0.03 |
| 8 | 1.5 | 3 | 0.03 |
| 9 | 3 | 0 | -0.015 |
| 10 | 6.3 | 0 | -0.05 |
| 11 | 3 | 1.5 | -0.015 |
| 12 | 6.3 | 1.5 | -0.05 |
| 13 | 0 | -1.5 | -0.03 |
| 14 | 1.5 | -1.5 | -0.03 |
| 15 | 0 | -4.8 | -0.1 |
| 16 | 1.5 | -4.8 | -0.1 |

The two streets are oriented by X and Y Cartesian directions, and each street has a different inlet discharge and slope (Figure 2).

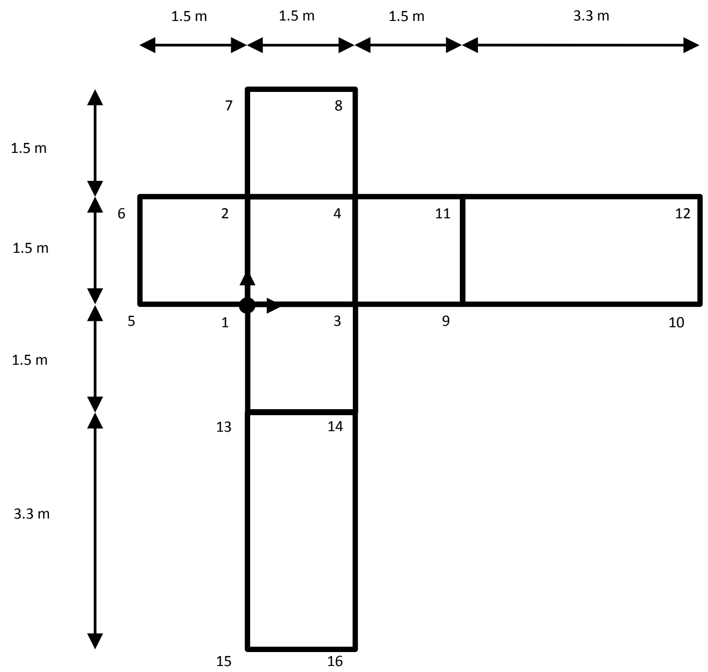



**Figure 2. Schematic representation of the model.**

## 3.3. Model set-up

### *3.3.1. Geometry*

The first step when starting a new simulation from scratch is to save the model (**Files>>Save**).

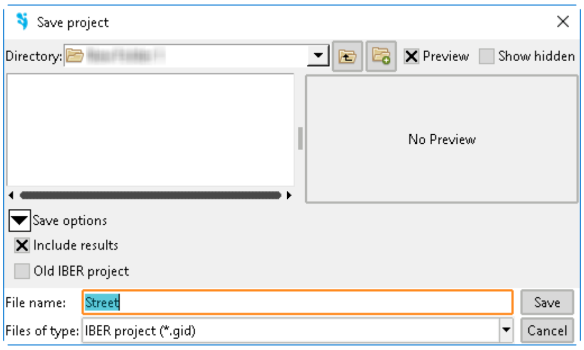



**Figure 3. Save project window.**

The geometry of the model will be generated with squares and rectangles (Figure 2). Using points, lines and surfaces we will obtain the complete geometry of the streets.

First, we introduce the coordinates of the points that define the intersection area, using **Geometry>>Create>>Point** (points 1 to 4). On the **Command** line (located at the bottom of the interface) we insert the coordinates of point 1 (Table 1).

Once the coordinates of point 1 have been introduced, we press ESC, and we repeat this procedure for points 2, 3 and 4[1].

We then join these points with lines. Using **Geometry>>Create>>Straight line**, we click on point 1. The **Create point** procedure window (Figure 4) will appear. We want to use an already existing point (number 1) as the first point of the line, so we click **Join**.

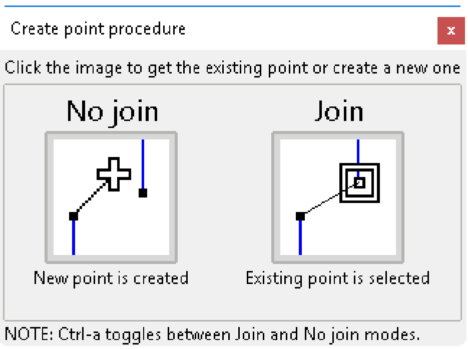



**Figure 4. Creation point procedure window.**

[1] Note that the format of the coordinate system is "*a*,*b*,*c*", where *a*, *b* and *c* represents the values of the coordinates X, Y and Z, respectively. The comma is the coordinate separator, and the decimal separator is based on SI (point).

We repeat this action to create the other lines (2-4, 3-4, 1-3) and press ESC to end this action.

Finally, we create a NURBS surface[2], using the **By Contour** option in the **Geometry>>Create>>NURBS Surface** menu. Select all the lines and press ESC. The result of those operations is shown in Figure 5[3].

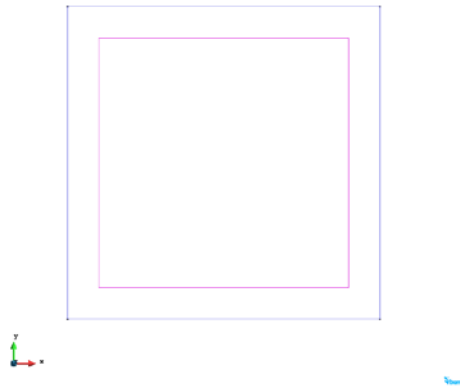


**Figure 5. Surface created.**

With **Geometry>>Create>>Straight line** we can now create the lines 1-5, 5-6 and 2-6 using the coordinates shown in Table 1 directly. Note that when we create the lines, Iber asks if we want to use an already existing point (**Join** option). Once the lines are created, we need to create a surface defined by these lines that form a closed polygon.

We turn now to the Copy menu (**Utilities>>Copy**). This allows us to perform different kinds of operations. In this case, we are going to copy line 2-4 in order to generate lines 2-7 and 4-8. In **Entities type** we choose **Select lines**. The first point is point number 2 and the end point is point number 7 (see coordinates on Table 1). Click on **Select** and choose line 2-4. Then press ESC or click **Finish**. Join points 2-7 and 4-8 with their respective lines. Finally, we can create the surface.

Using the **Copy** tool, we can create new lines and the surfaces within them using the **Do Extrude** option. Following the steps previously described, we generate line 9-11 using line 3-4 and line 4-11 using line 3-9. The surface will be created automatically. Copy line 3-4 from point 3 to 9, and in the **Do Extrude** option choose **Surface**. Select the translation as operation, insert the first and the second points, then select the line (3-4). Press ESC to finish.

Repeat this action with line 9-11 to create lines 10-12, 9-10 and 11-12 and the corresponding surface. Do the same with line 1-3 and then with line 13-14.

The result of all operations is shown in Figure 6.

---

[2] Non-Uniform Rational Basis-Splines are the kind of shapes used by GiD to represent the surfaces.

[3] GiD represents the geometry of the model by points, lines and surfaces. By default, points are coloured black, lines are dark blue, and surfaces are purple. The surface representation has a smaller shape than the closed polygon that encloses it.

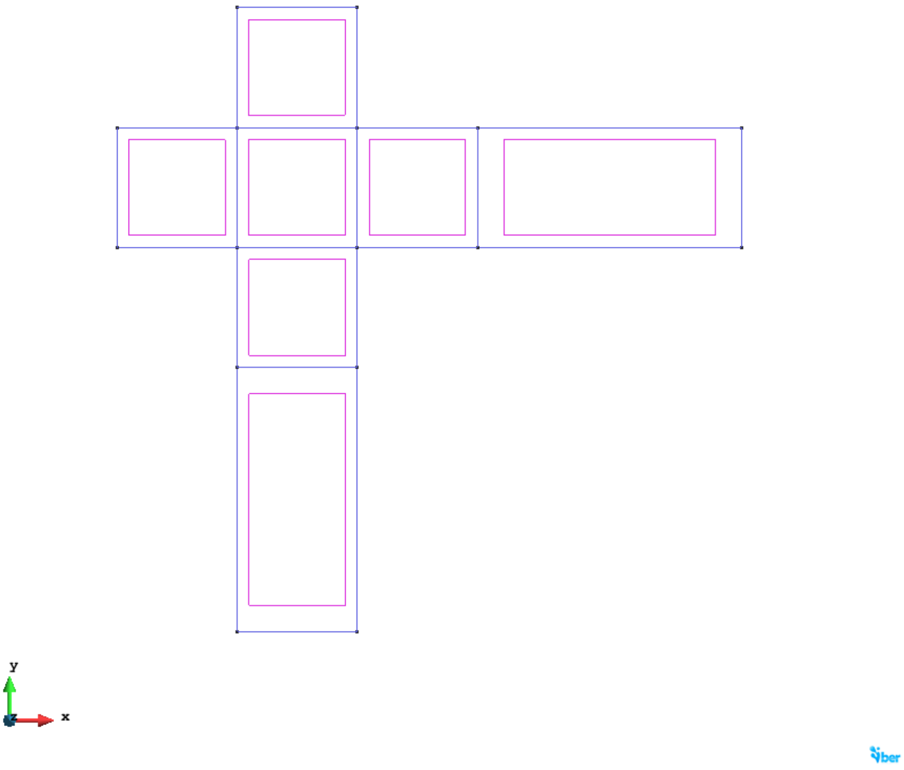



**Figure 6. Geometry of the model.**

### *3.3.2. Hydrodynamics*

The next step is to assign the hydrodynamics boundary and initial conditions. The model has two inlets and two outlets (which must be defined as boundary conditions). The simulation starts with the streets being dry (initial condition). All the hydrodynamics conditions are defined in the **Data** menu.

Boundary conditions are assigned in **Data>>Hydrodynamics>>Boundary conditions**. A constant discharge of 0.04286 $m^3/s$ on line 5-6 will be assigned using the **2D Analysis window** (Figure 7). In the **Inlet** field we choose **Total Discharge**. In the **Inlet Condition** field we choose the **Critical/Subcritical** regime. In **Total Discharge** we expand the table ( ) and introduce a unique value of discharge at the time 0 seconds[4]. Introduce the value “0.04286” in the “Q [m3/s]” column and **Assign** this condition to the line 5-6. The **Inlet Num** is 1.

Repeat this action with line 7-8, but here the inlet is defined by a **Specific Discharge** and **Supercritical Flow Regime**. The value of this boundary condition is a specific discharge of 0.0667 $m^2/s$ and a water elevation of 0.0771 m, corresponding to a discharge of 0.1 $m^3/s$.

---

[4] A constant value is introduced in Iber from time $t_i$ to $t_f$. If the simulation time is greater than $t_f$, Iber considers this value to be constant from $t_f$. to the end of the simulation.

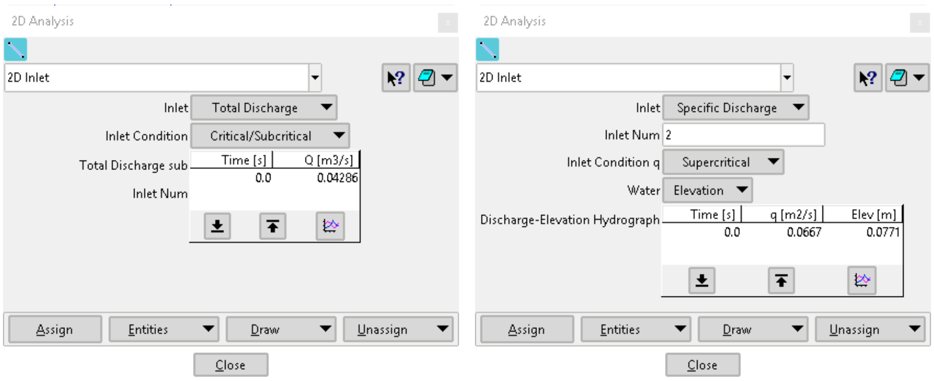



**Figure 7. 2D Analysis window. Characteristics of the inlets on line 5-6 (left) and on line 7-8 (right).**

The characteristics of the laboratory experiment produce a mixed regime flow (subcritical and supercritical). The outlet conditions of lines 10-12 and 15-16 are supercritical flow (Froude number greater than 1). We use the drop-down menu to change between **Inlet** and **Outlet** conditions. Note that the **Outlet Num** must be 1 and 2, respectively.

As mentioned previously, the initial condition is a dry channel, meaning that water depth is zero. Thus, using **Data>>Hydrodynamics>>Initial condition** we assign a water depth equal to zero to all surfaces.

Finally, the model needs a roughness coefficient. We define this in **Data>>Roughness>>Land use** (Figure 8). Iber has a database of land uses with common Manning roughness coefficient values. Use the drop-down menu and choose **Concrete**, which has a Manning value of 0.018 $s \cdot m^{-1/3}$. Assign this to all surfaces and press ESC.

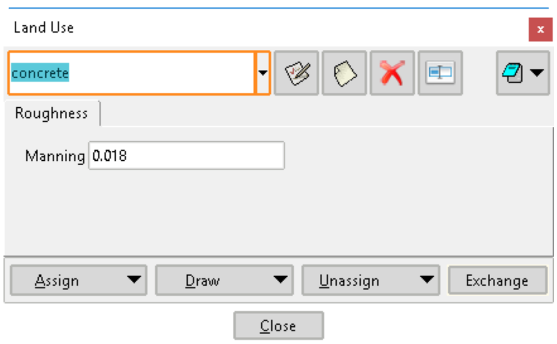



**Figure 8. Land Use window, where the Manning coefficient is defined.**

### *3.3.3. Mesh*

The mesh is the discretization of the space domain where Iber solves the Shallow Water Equations (SWE). Iber can work with structured and non-structured meshes with triangular or/and quadrilateral elements. Mesh options are defined from the **Mesh** menu.

In this example, we will use a structured grid formed by quadrilateral elements. Using **Mesh>>Structured>>Surfaces>>Assign size** we define the size of the mesh elements. In this case, all elements will have the same length. Select all surfaces and press ESC. A new window will ask for the elements size. Introduce 0.1 metres and then select **Assign** to all the lines. Press ESC and close the window.

We now generate the mesh (**Mesh>>Generate mesh**). The Mesh Generation window asks for an element size. We have already defined this, so the value introduced here will not be taken into account. Press OK.

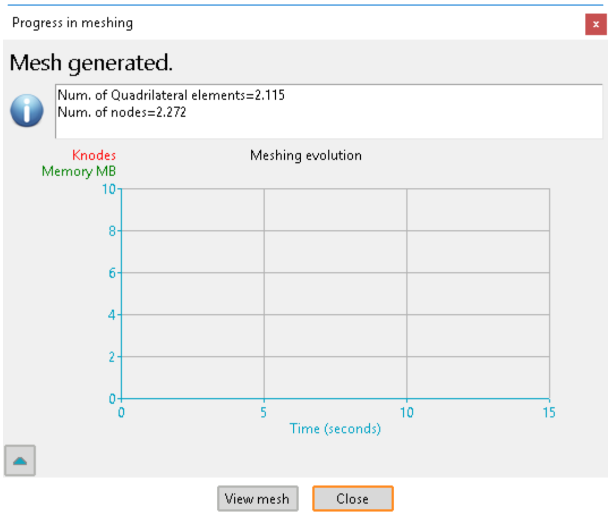



**Figure 9. Progress in the Meshing window, in which some properties of the created mesh are shown.**

The **Progress in meshing** window automatically appears (Figure 9). This window shows the number and kind of elements generated (triangular or quadrilateral), the number of nodes, and the memory used in this process. In this case a mesh of 2115 quadrilateral elements has been created.

We can view the mesh by clicking on **View Mesh** (Figure 10).

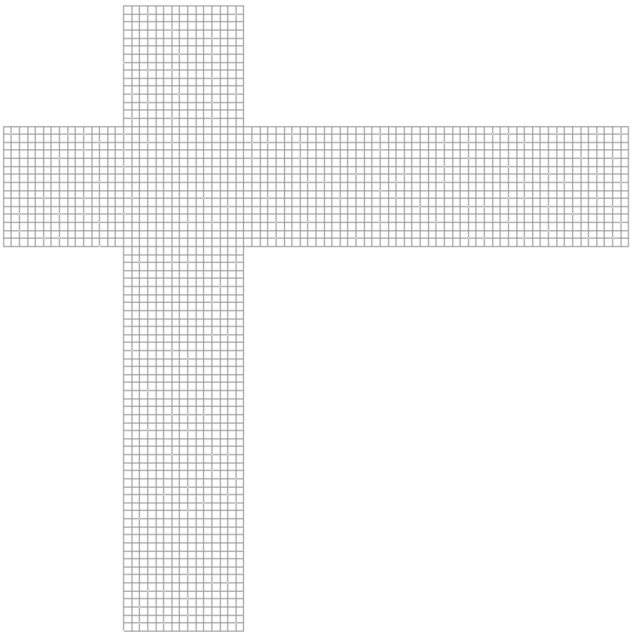


**Figure 10. View of the mesh of the model.**

### *3.3.4. Calculation data*

The **Data** settings define the general and numerical parameters of the simulation (time, results, calculation modules, etc.).

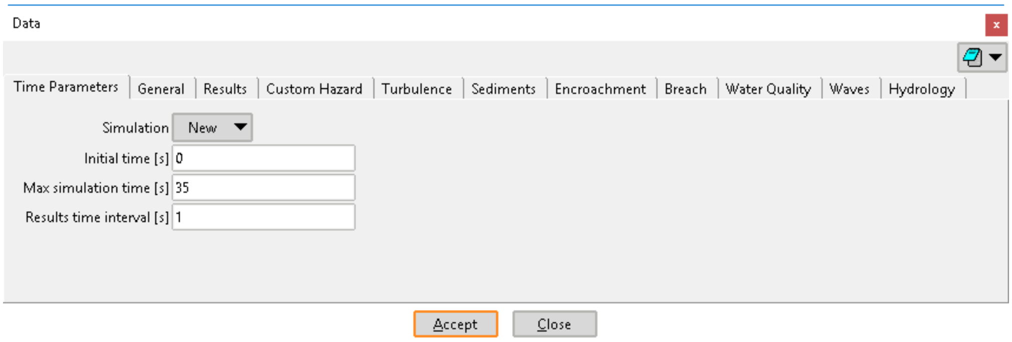



**Figure 11. Data window. Time parameters tab.**

In this example we only need to define the **Time Parameters**, using **Data>>Problem data**. We start the simulation at a time equal to zero seconds (**Initial time**) and finish it at 35 seconds (**Max simulation time**). We want results to be saved every second, so we define the **Result time interval** as 1 (Figure 11).

Press **Accept** to save these parameters.

### *3.3.5. Calculation process*

We have already built the model, and now we need to calculate it.

Through **Calculate>>Calculate** we can begin the simulation. When the calculation process has finished, the **Process info** window will appear (Figure 12). Press OK to maintain the preprocess interface or press **Postprocess** to see the results.

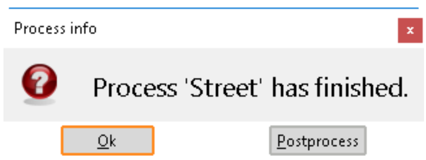



**Figure 12. Process info window.**

It is possible to toggle between pre- and postprocess view using the **File** menu or the  button.

Before we view the results, we will check if the simulation process has been successful. In the **View process info** window (**Calculate>>View process info** window), Iber shows some information about the calculation process (name of the project, time starting, Iber version, etc.). This window also shows warnings and error messages. These alert the user to any issues that have occurred during the computation process (e.g. a missed condition, problems with the mesh, a duplicated boundary condition, non-permitted boundaries or internal conditions, etc.).

If the computation has been performed successfully, the message *COMPUTATION FINISHED SUCCESSFULLY!* will appear at the bottom of the **View process info** window, along with the data and the time when it finished.

## 3.4. Results

In **Window>>View results** we can choose the different ways to show the results (Figure 13). We will represent the depth field at the last time calculated (35 seconds). Select **Contour fill** in **View**, **Hydraulic** in **Analysis**, 35 in **Step**, then **Depth** as the result to show. Press **Apply** and the results will be drawn on the elements (Figure 13).

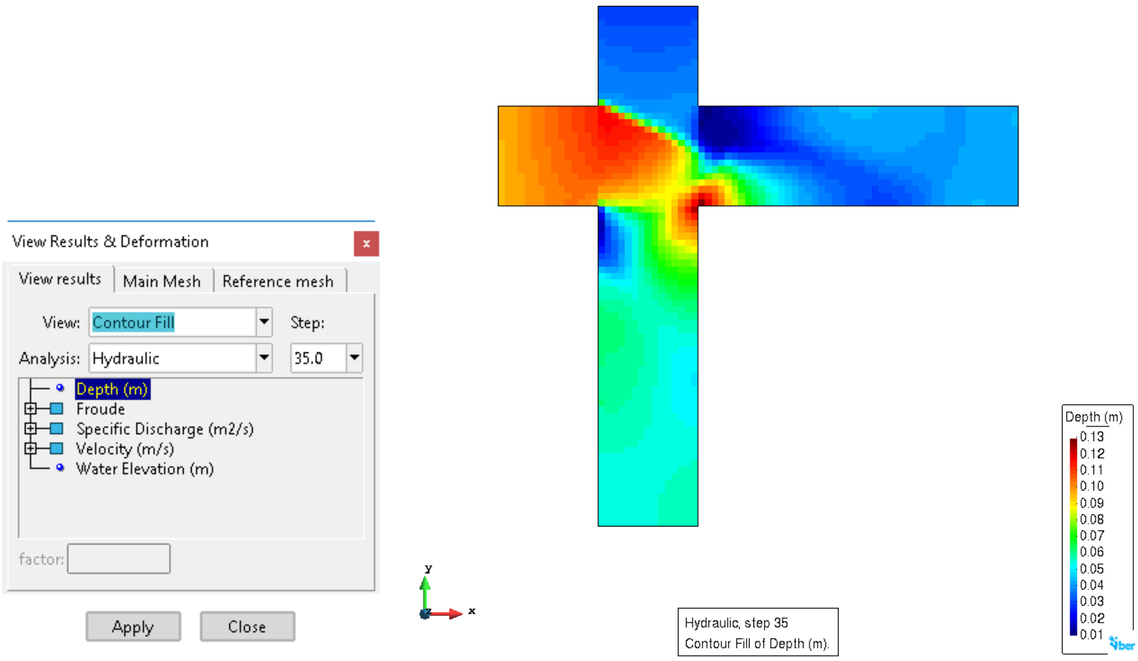



**Figure 13. Contour fill representation of the depth result at 35 seconds.**

**Contour fill** represents the results using one colour on each element. If we want to represent a "non-pixelated" representation, we can use the **Smooth Contour fill View** (Figure 14).

We can observe upstream of the intersection that on the X-oriented street the water depth is greater than on the Y-oriented street. In the intersection the two flows converge, forming a hydraulic jump. Crossed waves are then generated on the X and Y streets downstream of the confluence.

In order to reach a better representation of the hydraulic jump, we can represent the Froude number. A hydraulic jump is generated when a flow changes its hydraulic regime from supercritical (Froude less than 1) to subcritical (Froude greater than 1). Select the **Froude module (|Froude|)** result in the **Hydraulic Analysis** as a **Smooth Contour fill**, **View** at 35 s, and **Apply**. We can see that X-street has a subcritical flow (dark blue on Figure 15 left) and Y-street has a supercritical flow (dark red on Figure 15 left).

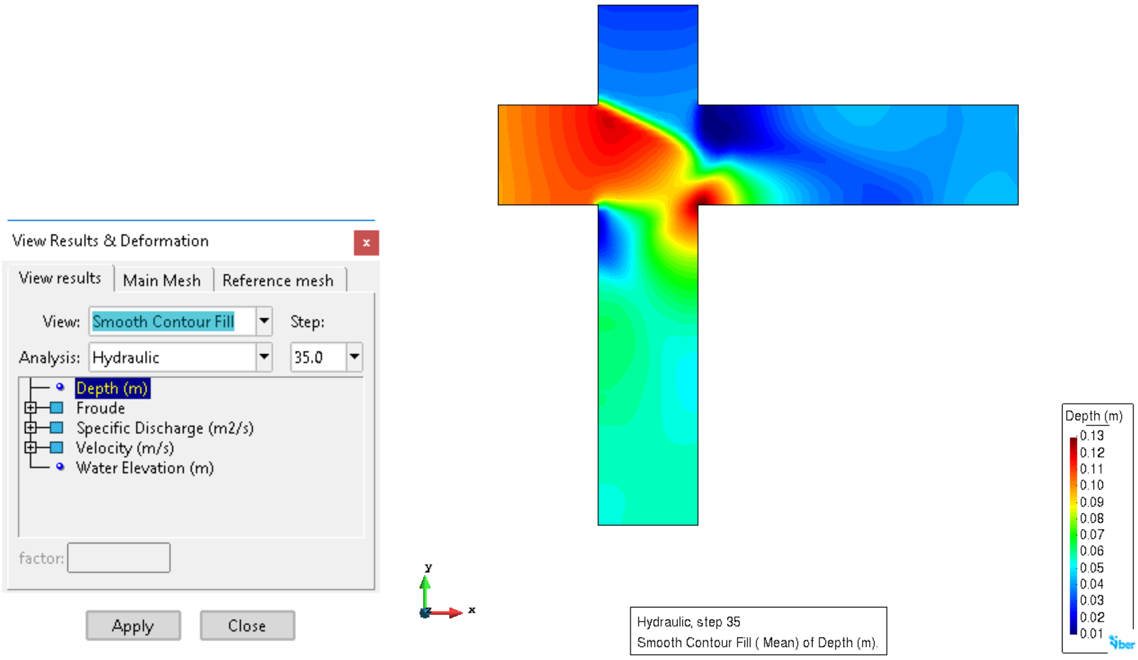



**Figure 14. Smooth Contour fill representation of the depth result at 35 seconds.**

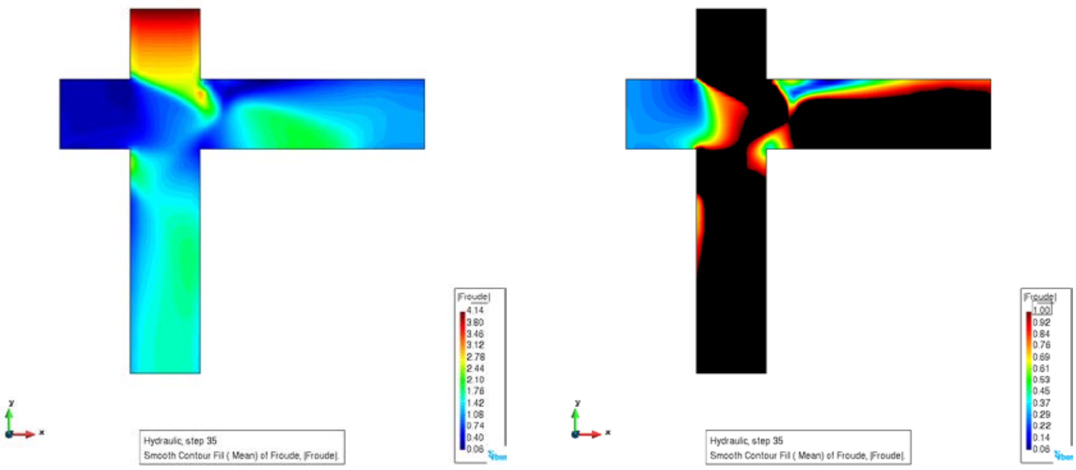


**Figure 15. Froude number representation. Smooth Contour fill representation of the Froude value at 35 seconds (left) and Smooth Contour fill representation of the Froude value limited to 1 at 35 seconds (right).**

We can set the maximum and minimum values to be shown using **Utilities>>Set contour limits>>Define limits** (Figure 16). Set the maximum value to 1, and **Apply**. In Figure 15 (right) we can see the subcritical flow in colours and the supercritical flow as black colour. To finish this result representation, unmark “Max” and then apply, or using Utilities>>Set contour limits>>Reset limits menu.

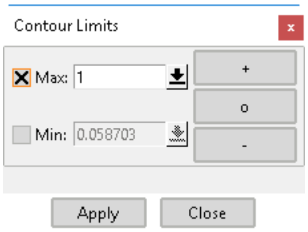



**Figure 16. Contour Limits window. Maximum value fixed at “1”.**

Another common way of representing results is by using Vectors. This is only available for vector variables as specific discharge and velocity.

Using the **View Results & Deformation** window (**View results** tab), select **Display Vectors View**, the **velocity module** (**|Velocity|**) at the last **time step** (35 s), and **Factor** equal to 1. Select **Apply**.

The velocity is shown as multicolour arrows, but they are overlapping (Figure 17 left). We can modify the details of the representation using **Utilities>>Preference**. First, open the Postprocess>Vectors options and select Monochrome as Colour mode, then change Vector size type to Fixed, and then apply. The results of these operations are shown in Figure 17 (right).

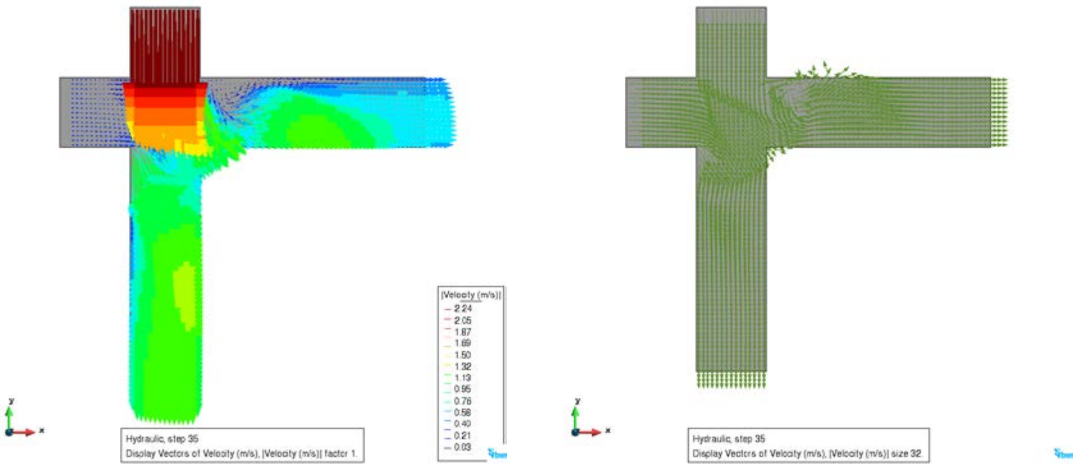


**Figure 17. Vector representation of the velocity module.**

With this representation we can observe a recirculation area on Y-street, downstream of the street intersection. We can also observe that the discharge on the Y-street is greater than on the X-street. If we represent the two components of the velocity we see that, in general, the Y component is greater than the X component (Figure 18).

Another kind of available results are graphical representation by Graphs, which can be used to represent the time evolution of a variable, and to create cross sections or hydrographs using the **Window>>View graphs** options.

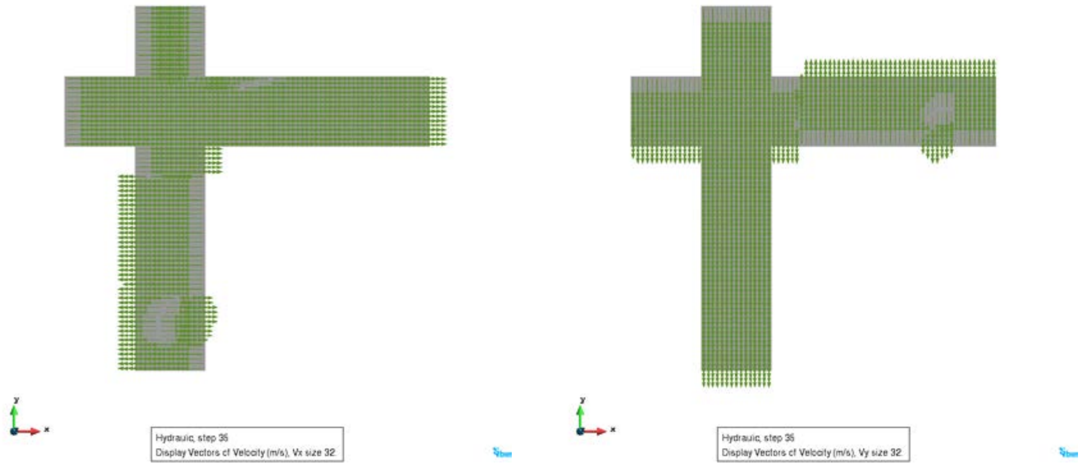



**Figure 18. Vector representation of the X (left) and Y (right) velocity.**

We start by plotting the time evolution of the water elevation at the control point located in the middle of the intersection area ([0.75,0.75]). On the **Create tab** we select **Point evolution View**, **hydraulic Analysis** and any **time step** (for the present example, this parameter is not needed). On the X-axis (for **All steps**) of the graph we do not select anything, and for the Y-axis we choose **Water Elevation**. After we select **Apply**, Iber asks for the coordinates of the point. Introduce "0.75,0.75" in the **Command** line (at the bottom of the interface) and press **Finish** or ESC (Figure 19).

It might also be of interest to show the water level at a specific time step. In this case, the last time step simulated represents a steady state situation. We will create another graph-set here, using the button Create ( ). In the **Create** tab select **Line Graph View** of the Water elevation at step 35 s. Select **Line Variation** and **Water Elevation** for the X-axis and the Y-axis respectively. Select **Apply** and indicate the line which serves to plot the water surface. The first point is [0.75,3] and the last point is [0.75,-4.8]. In the same graph we will plot the bottom elevation. We follow the same steps, but changing the **Analysis** to **Topography**, the **Step** to "0", and the Y-axis value to **Elevation**. The result is shown in Figure 20.

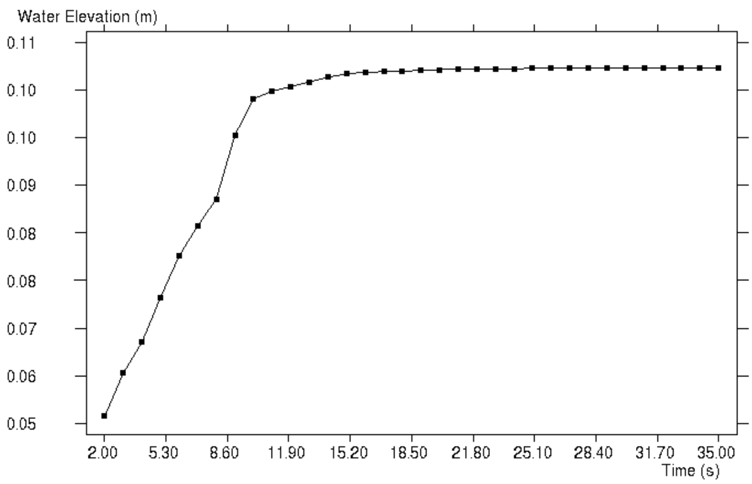



**Figure 19. Graph of the water elevation evolution in time.**

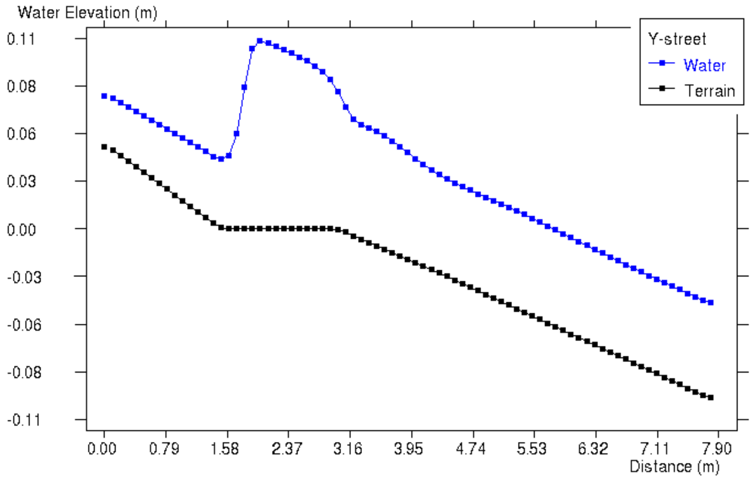



**Figure 20. Graph of the water elevation along the Y-direction.**

## 3.5. Conclusions

In this example we have introduced the user to the Iber's interface. The basic tools and geometry options have been shown by creating a simple model. It has also been shown how to introduce the basic flow conditions (boundary and initial conditions, roughness, etc.). A structured mesh criteria has been used to discretize the spatial domain. The results show a hydraulic jump and crossed waves, characteristic of a 2D shallow water flow.

## 3.6. References

Bladé, E. (2005). Modelación del flujo en lámina libre sobre cauces naturales análisis integrado con esquemas en volúmenes finitos en una y dos dimensiones. Tesis Doctoral. Universitat Politècnica de Catalunya. Retrieved from http://tesisenred.net/handle/10803/6394

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, E., Dolz, J., Coll, A. (2014a). Iber: herramienta de simulación numérica del flujo en ríos. Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería, Volume 30, Issue 1, 2014, Pages 1-10, ISSN 0213-1315, DOI: 10.1016/j.rimni.2012.07.004

Nanía Escobar, L. S. (1999). Metodología numérico-experimental para el análisis del riesgo asociado a la escorrentía pluvial en una red de calles. Tesis Doctoral. Universitat Politècnica de Catalunya. Retrieved from http://upcommons.upc.edu/handle/2117/93707

# 4. Practical Example 2: River flood inundation

## 4.1. Objectives

The aim of this example is to learn how to generate the geometry and mesh of a typical river inundation simulation using georeferenced information. It will be also shown how to import a digital elevation model (DEM) to create the geometry of the problem.

## 4.2. Description of the case study and input data

The case study is the final reach of the Bisbal River, which flows into the Mediterranean Sea.

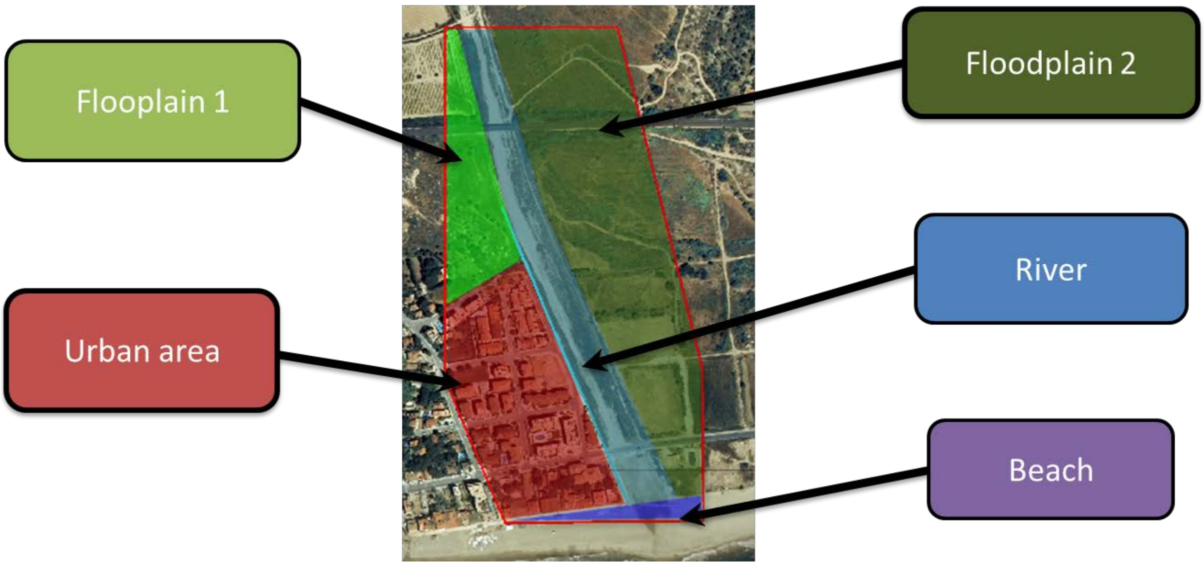



**Figure 21. Schematic picture of the study area.**

The following material is provided for this practical exercise:

- Digital Elevations Model (DEM) of the study area – bisbal.txt
- Digital Elevations Model (DEM) of a levee – wall.txt
- Hydrograph for a return period of 500 years – discharges.txt
- Georeferenced orthophoto – orthobisbal.jpg and orthobisbal.jgw

The orthophoto has a red polygon that defines the study area. This area coincides with the extension of the DEM. The topography is defined by a RASTER file. This file is made up of 1-meter cells. It also has the altitude of each cell.

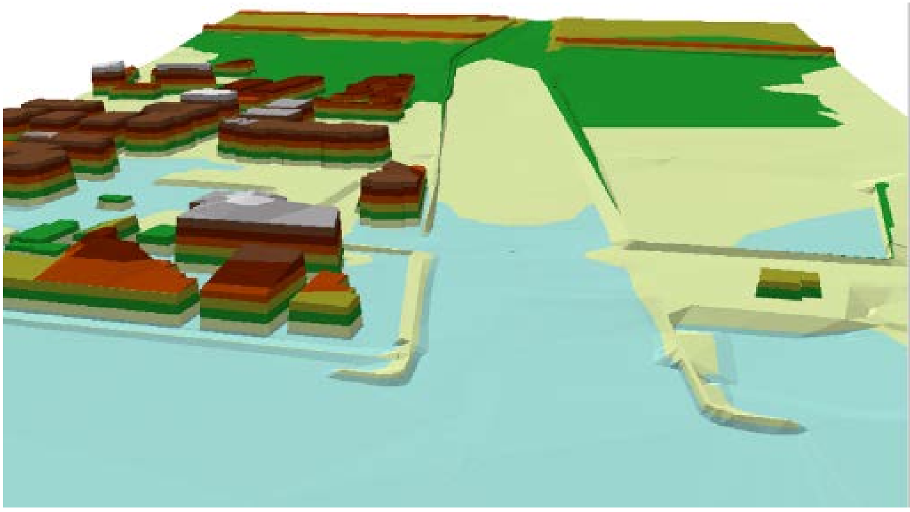


**Figure 22. DEM file display on GIS software.**

The boundary conditions are set out in a spreadsheet that contains the hydrograph of the standard project flood. In this file there is a table with flows for every time point.

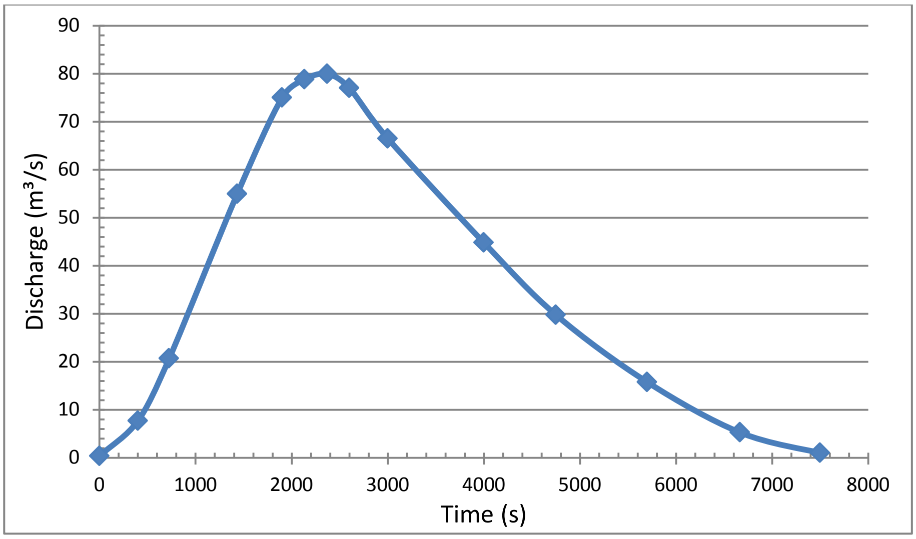



**Figure 23. Hydrograph of standard project flood for a return period of 500 years.**

## 4.3. Model set-up

### *4.3.1. Geometry*

The first step towards the simulation process is to save the model (**Files>>Save**).

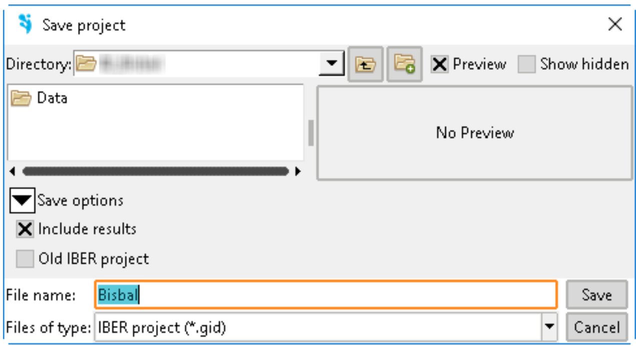



**Figure 24. Save project window.**

The next step is to put the orthophoto in the background by **View>>Background image>>Real size** menu. If the image is not shown, refresh the screen or make a zoom frame (located at the left toolbar). This background image will help us to draw the geometry of the model.

To create the geometry, 5 polygons need to be drawn to define the areas with different Manning coefficients. To make the polygons that define the different areas, select the **Create line** tool (located at the left toolbar). In our orthophoto, a red polygon defines the DEM extension.

We begin with the urban area. The lines must be created within the red box.

**Figure 25. How to make the lines.**

When we are finishing the polygon, we need to select the first vertex of the polygon. The program will then show a dialogue box asking us whether we want to join the lines or not.

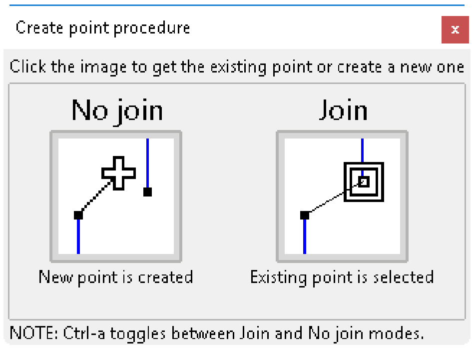



**Figure 26. Create point procedure.**

Once the polygon is made, we create the surface. To do this we will use the **Create NURBS surface** tool  (located at the left toolbar). The next step is to select all the lines and press ESC.

A pink box will now appear inside the red polygon. If this does not happen, please verify that the polygon is closed, and that there are no double lines.

**Figure 27. NURBS surface.**

Another way to make the NURBS surface is using the **Search** tool, which allows us to make NURBS surfaces automatically.

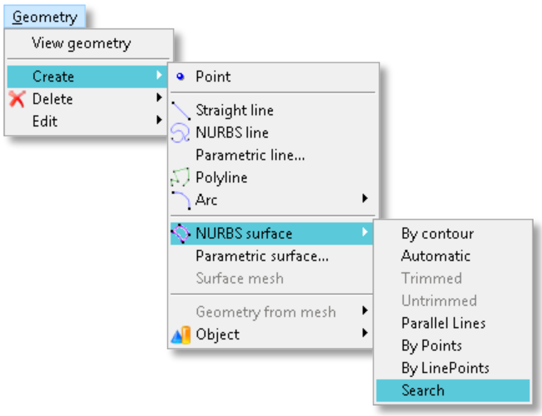



**Figure 28. How to use "Search" tool.**

We need to refine the geometry to avoid points outside of the red box or other errors. To move a point, use the **Move point** tool: **Geometry>>Edit>>Move Point**.

Following this procedure, we can make the remaining areas. It is important use the lines that are already drawn to make the other polygons, to avoid double lines.

**Figure 29. Study area discretized by surfaces, with (left) and without background image (right).**

### *4.3.2. Hydrodynamics*

The next step is to assign the hydrodynamic and initial conditions. This model has one inlet and one outlet. The inlet is on the top of the river stream (north of the model) and the outlet is on the beach (south of the model).

These boundary conditions are assigned in the **Data>>Hydrodynamics>>Boundary conditions** menu. In the **Inlet** field, we select **Total Discharge**. In the **Inlet condition** we select **Critical/subcritical**. Following this, open discharges.txt file. Copy the whole table and paste it in the table of the **Boundary conditions**.

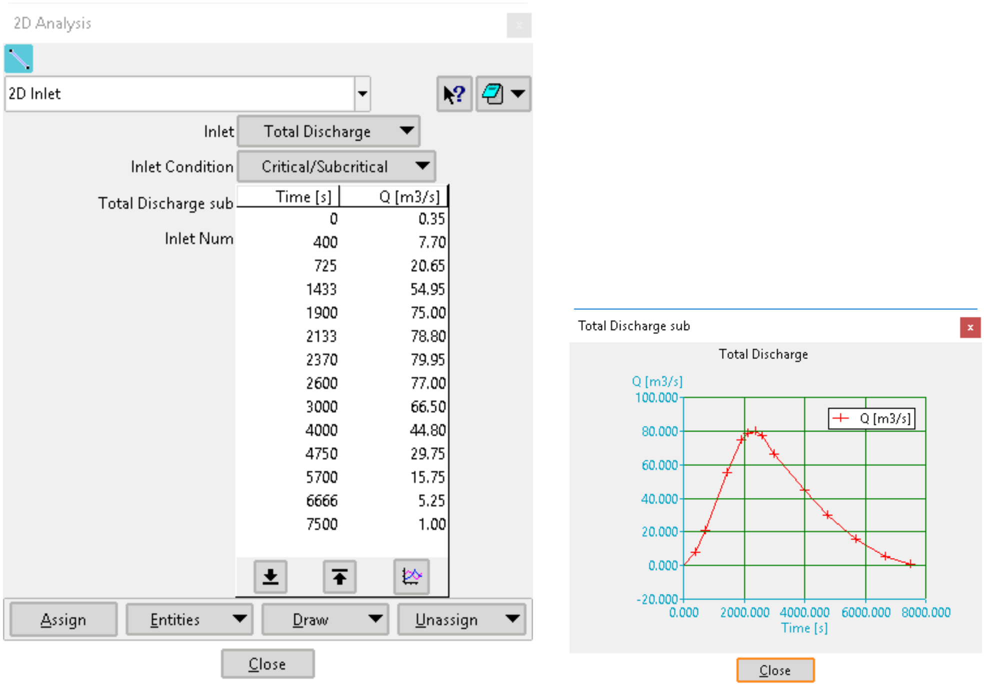



**Figure 30. Boundary condition at its representation.**

By pressing the key, we see the chart in Figure 31 below. This will help us to verify if we have copied the right table.

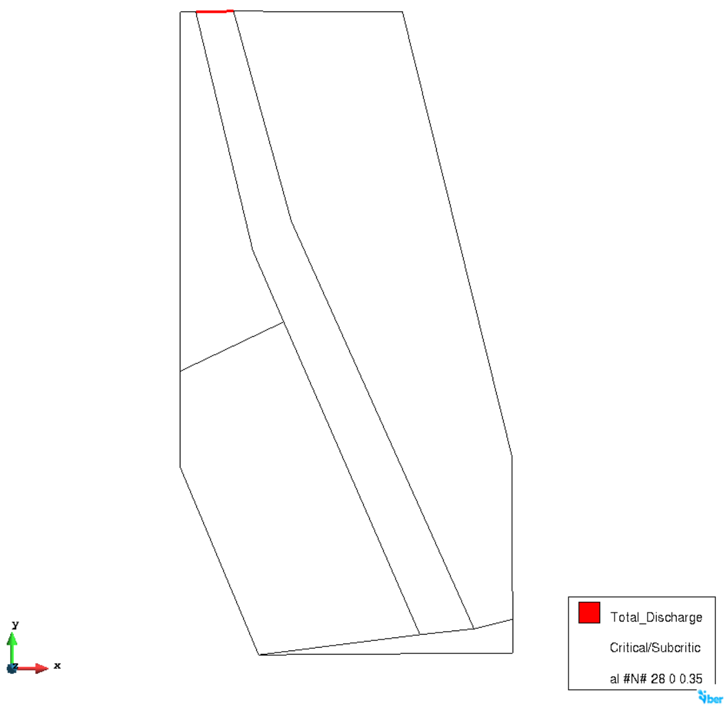



**Figure 31. Location of the inlet condition.**

We assign this inlet to the top line on the river stream (Figure 31).

In the same window we can define the outlet. In the **Outlet** field we select **Subcritical**. In the **Outlet** condition we select **Given level**. The elevation is zero because the outlet is at sea level and the river slope is very slight. We must remember to assign this condition to the lowest part of the beach.

The simulations begin with the dry model. For this, we assign a depth of zero to all the surfaces: **Data>>Hydrodynamics>>Initial condition**.

The final step is to assign roughness coefficients. In this case there are different areas with different Manning roughness coefficients. Use **Data>>Roughness>>Land use** for this. Iber has a database containing many roughness coefficients. These depend on the land use (Table 2).

**Table 2 Land uses assigned to each area identified in Figure 21.**

| Area | Land use | Manning coefficient |
|---|---|---|
| River | river | 0.025 |
| Urban area | residential | 0.15 |
| Floodplain 1 | brushland | 0.05 |
| Floodplain 2 | brushland | 0.05 |
| Beach | sand/clay | 0.023 |

Assign the corresponding land use to each surface following the steps detailed in the Practical Example 1: Street intersection. Briefly, select the land use, define the Manning coefficient value and then assign to the surfaces. To verify the coefficient assignments, we can draw the areas according to their Manning coefficient by **Draw>All materials**.

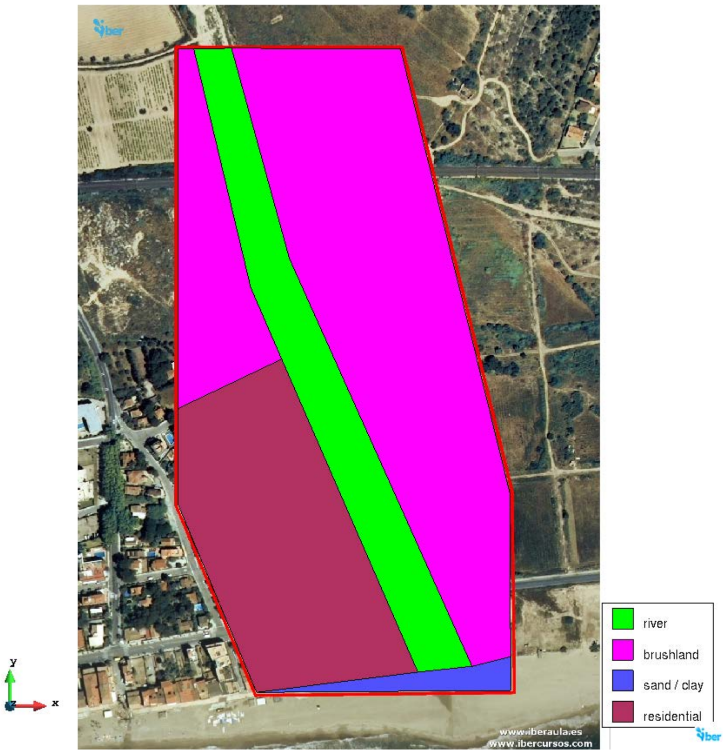



**Figure 32. Surfaces according to land uses.**

### *4.3.3. Mesh*

In this example, we will assign a mesh criterion to each surface. In this way, our mesh will be generated from the discretization of the surfaces. **Mesh options** are on the **Mesh** menu.

The river course and the beach need a mesh of a medium size in order to study the water flow. The mesh elements will have a size of 6 m. The urban area requires a fine mesh because we need to discretize the houses and streets topography. This mesh will have elements with a characteristic size of 4 m. The floodable areas do not need as much detail. Their mesh will have elements with a characteristic size of 15 m.

Our mesh will be unstructured. This mesh type is marked by having elements with no order and no orientation. For this, use **Mesh>>Unstructured>>Assign sizes on the surfaces**. The window that appears asks us to introduce the size of the elements. We enter 6 (**River and beach** surfaces) and press **Assign**. Now we need to select the river and the beach surfaces, then press ESC to confirm the assignment. The previous window will now appear again. We repeat the procedure for the urban area (4 –m-size). The floodable areas do not have a size element assignment because we assign to them in the mesh generation. Close the window.

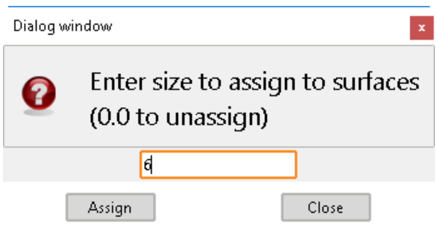



**Figure 33. Enter value window.**

To verify that we have assigned the mesh criterion correctly: **Mesh>>Draw>>Sizes>>Surfaces** (press ESC key to finish the action).

Finally, we generate the mesh: **Mesh>>Generate mesh**. During this step, we enter the mesh size of elements that were not assigned before. Recall that the floodable areas do not have mesh size assignment. In the window, enter 15 and press **Accept**.

When the generation has finished, press **View mesh** to see it. Figure 35 shows the different mesh sizes according to our needs.

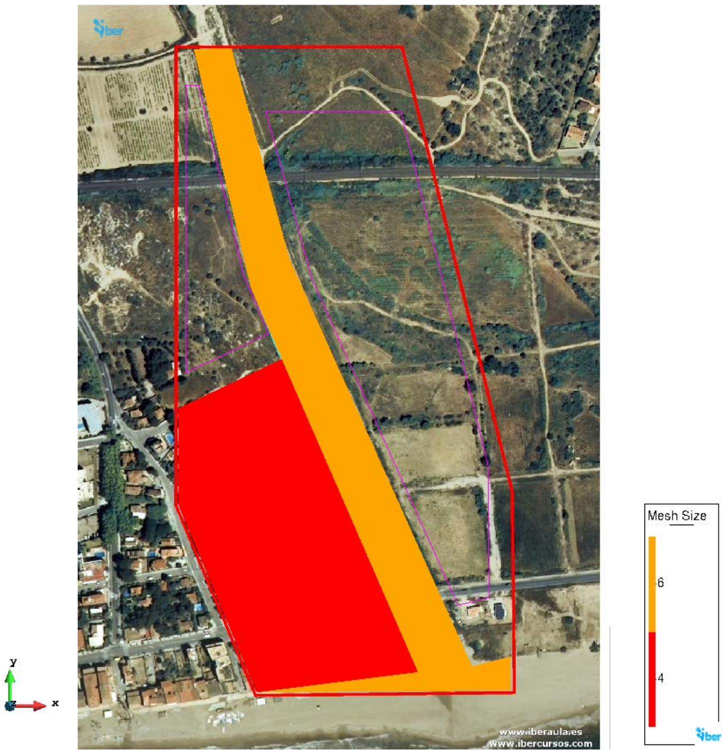



**Figure 34. Drawn surfaces by mesh size.**

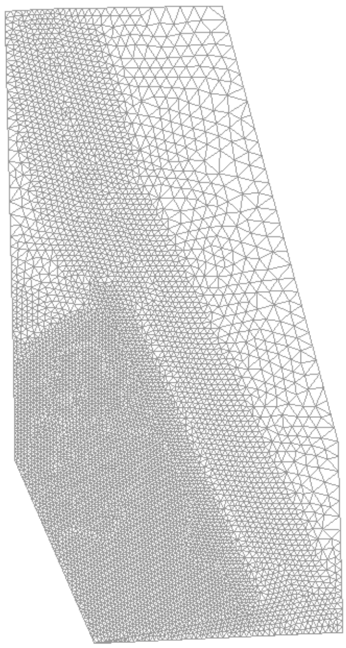


**Figure 35. Mesh (without elevation data).**

We now have a 2D mesh, because draw all the geometry without elevation data (all model has Z = 0 m). However, a 3D mesh is needed to study the problem. For this, we set the mesh elevation from the DEM file, using **Iber tools>>Mesh>>Edit>>Set elevation from file**. In the window, choose “bisbal.txt”.

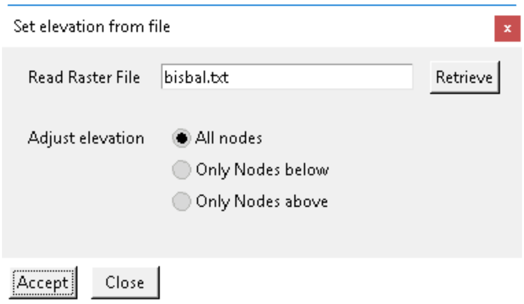



**Figure 36. "Read raster file" window.**

If we use the **Rotate trackball** we can now view our 3D Mesh ( ). We can view again the XY plane view (or other types of views) by View>>Rotate menu.

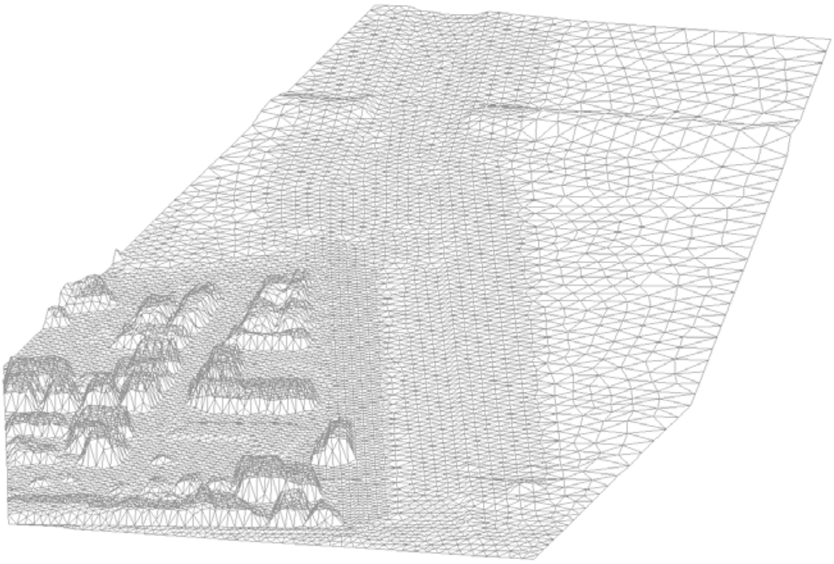


**Figure 37. Mesh (with elevation data).**

### *4.3.4. Calculation data*

The final step before simulation is setting problem data, using **Data>>Problem data**. In the **Time Parameters** tab, set 7600 seconds for **Max simulation time** and 60 seconds for **Results time interval**.

Iber allows us to define which results will be shown after the simulation. We can set these in the **Results** tab. For the current example we select **Hazard RD9/2008 & ACA**. This item will show us hazard maps, and we will be able to classify the urban area accordingly.

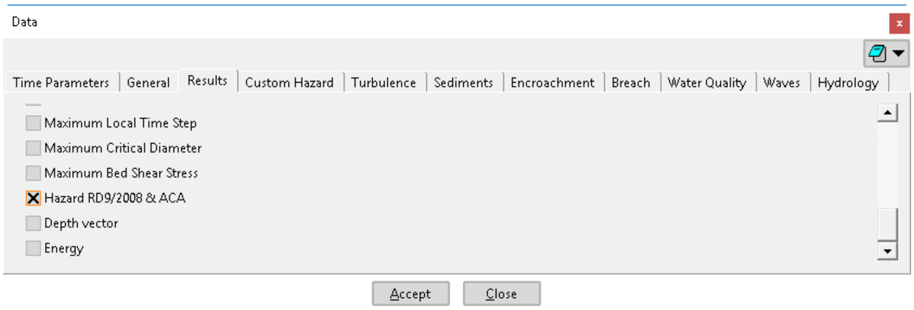



**Figure 38. Hazard RD9/2008 & ACA.**

### *4.3.5. Calculation process*

Having built the model, we now need to calculate it.

Through **Calculate>>Calculate** we can start the simulation. When the calculation process has finished, the **Process info** window will appear (Figure 39). Press OK to maintain the preprocess interface or select **Postprocess** to see the results.

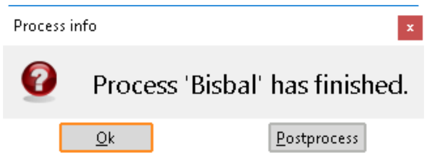



**Figure 39. Process info window.**

You can toggle between pre- and post-process view using the **File** menu or the button.

Before viewing the results, we can check if the simulation process has been successful. In the **View process info** window (**Calculate>>View process info window**), Iber provides information about the calculation process (name of the project, time starting, Iber version, etc.). This window also shows warnings and error messages, if there are any. These alert the user to issues that have occurred in the computation process (e.g. a missed condition, problems with the mesh, a duplicated boundary condition, non-permitted boundary or internal condition, etc.).

If the computation is successful, at the bottom of the **View process info** window the message *COMPUTATION FINISHED SUCCESSFULLY!* will appear, plus the data and the time when it was completed.

## 4.4. Results

The first result that we are interested is the flooding map. This map will tell us whether it is necessary build a wall to protect the urban area. For this, the **View Results & Deformation** window should appear like this:

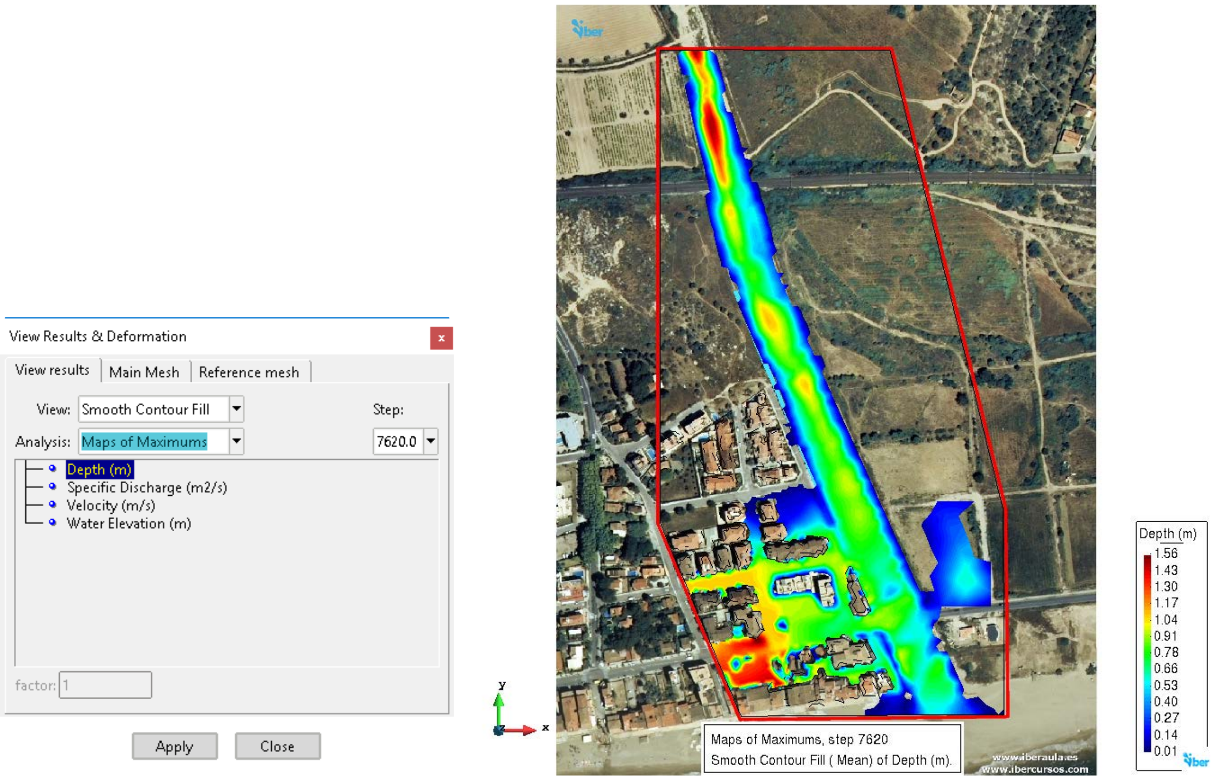



**Figure 40. Map of maximums (depth).**

When we select **Maximum depth map**, Iber looks for the moment in time when the depth is at its maximum. This depends on the steps we have taken before.

In this example, we can see that the depth in the urban area is higher than a meter.

The hazard criterion depends on the prevailing legislation. This criterion sets a value of depth and/or fluid velocity that establish the hazard level. The three most common criteria are:

- Depth (Maximum Level on the elements)
- Flow velocity (Related to dragging people and vehicles)
- Both (Person instability)

Iber, since version 2.1, allows us to define the hazard according to Spanish Legislation (*Real Decreto 9/2008 de Modificación del Dominio Público Hidráulico*) and the comparable technical guidelines in Catalunya (*Agència Catalana de l´Aigua, 2003. Guia Tècnica: Recomanacions techniques per als estudis d´inundabilitat d´àmbit local*). However, user can define their own criteria by Custom hazard tab.

In the **View Results & Deformation** window we can select these maps. For example, by selecting **Maximum Hazard ACA** (Figure 41) we are shown three hazard levels; high, moderate and no hazard (Contour ranges View). If we choose **Maximum Severe Hazard RD9/2008** (Figure 41) we can see that the whole flooded area shows severe hazard.

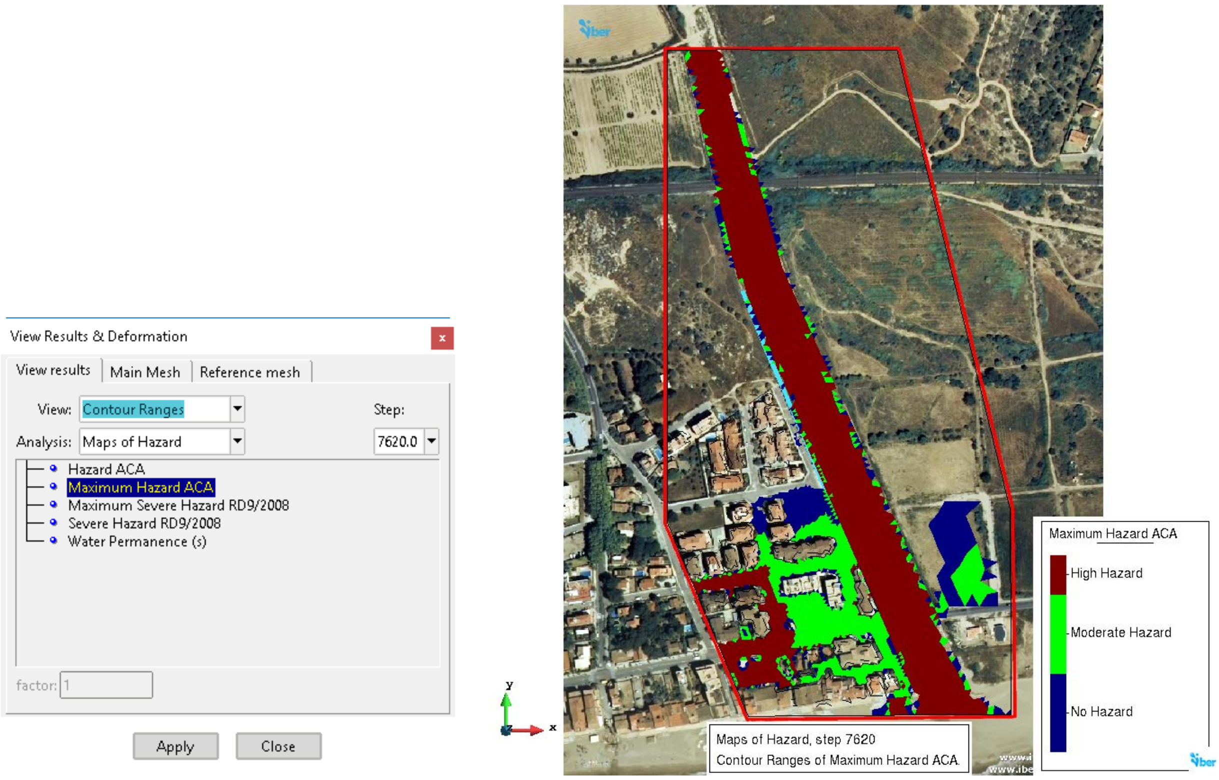



**Figure 41. Maximum Hazard according to ACA.**

As can be seen, in this case it is necessary to build a wall to protect the urban area. Returning to preprocess, we will now upload the DEM file with the wall elevations.

In the preprocess, we deactivate the background picture so we can view the wall area.

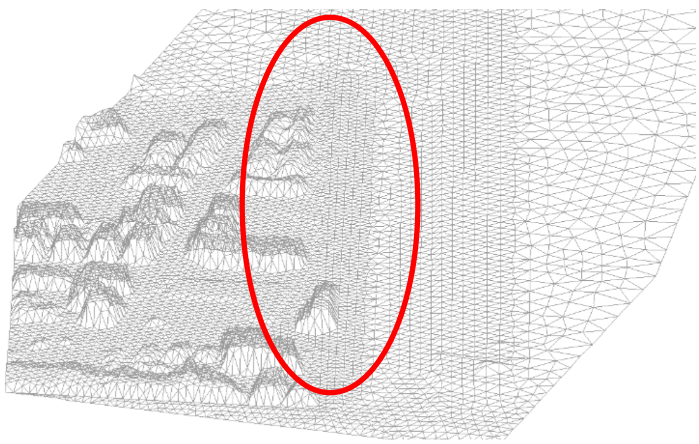


**Figure 42. Wall area.**

In the same way that we set the elevation of the mesh, we upload the wall file: **Iber tools>>Mesh>>Edit>>Set elevation from file**. Select "wall.txt".

When we can see the wall area, we note that there are gaps. This is because the mesh size in this area is greater than the wall width.

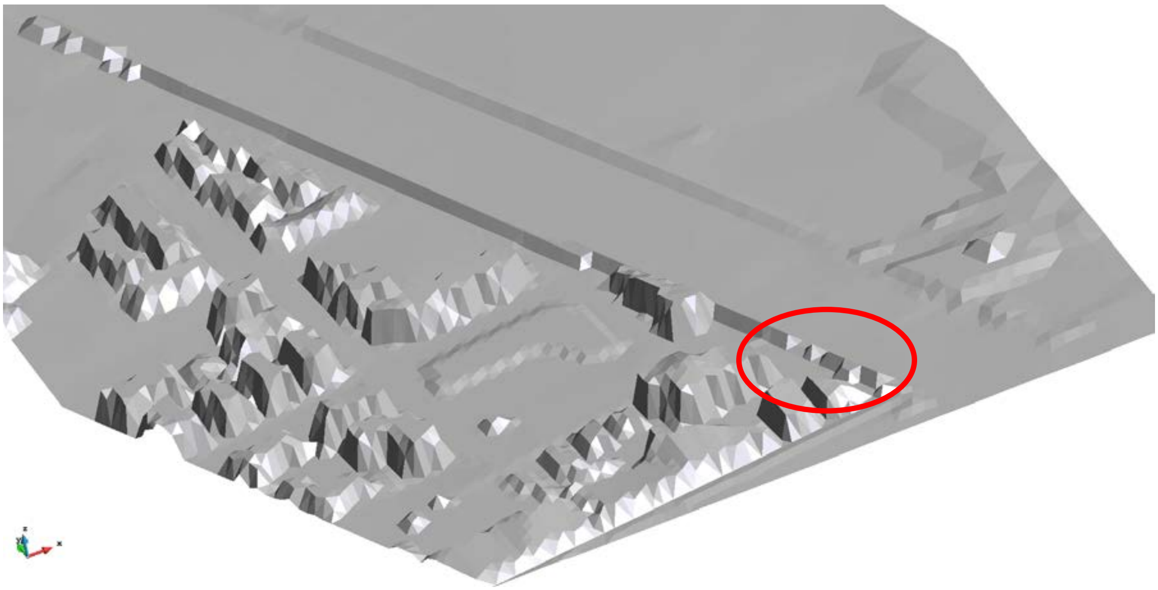


**Figure 43. Gaps in the protection wall.**

To solve this problem, we need to refine the mesh in these areas. We use **Mesh>>Edit mesh>>Split elements>>Triangle>>Triangle**, selecting the elements that we want to split. Having done this, press ESC. In Figure 44 we can now see the elements as they are now.

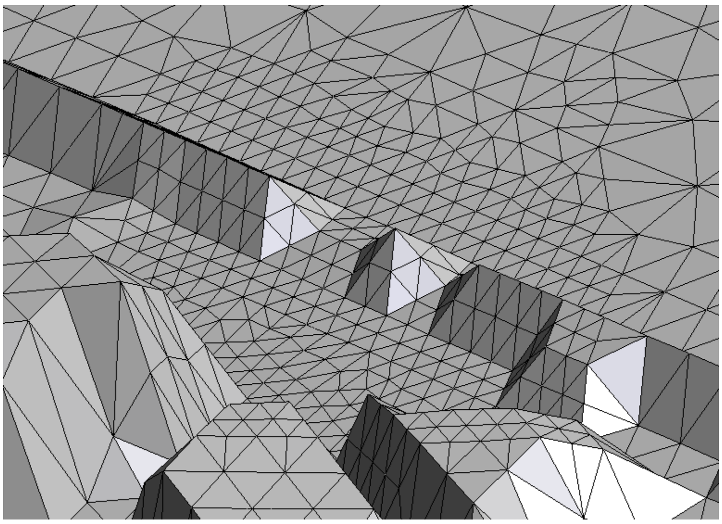


**Figure 44. Split elements area.**

The next step is to set the elevation wall file again, using **Iber tools>>Mesh>>Edit>>Set elevation from file**. Select “wall.txt”.

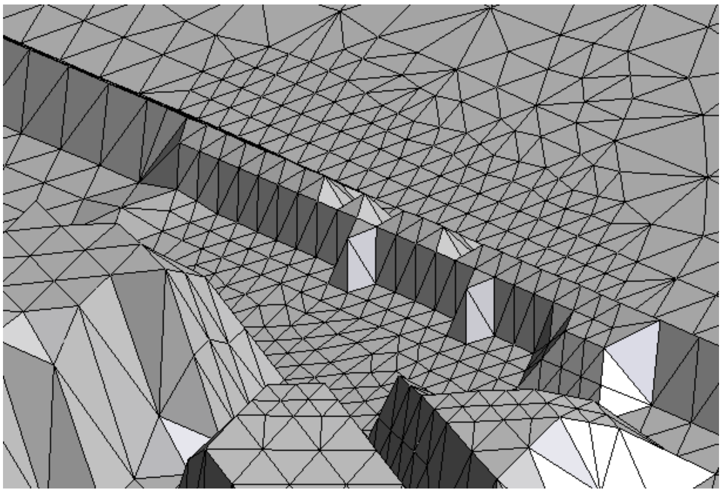


**Figure 45. Wall without gaps.**

This operation must be repeated on all the areas of the wall that show gaps in their geometry.

We can now run the simulation again.

If we select the **Maximum Depth map**, we can see that the urban area does not show a flood.

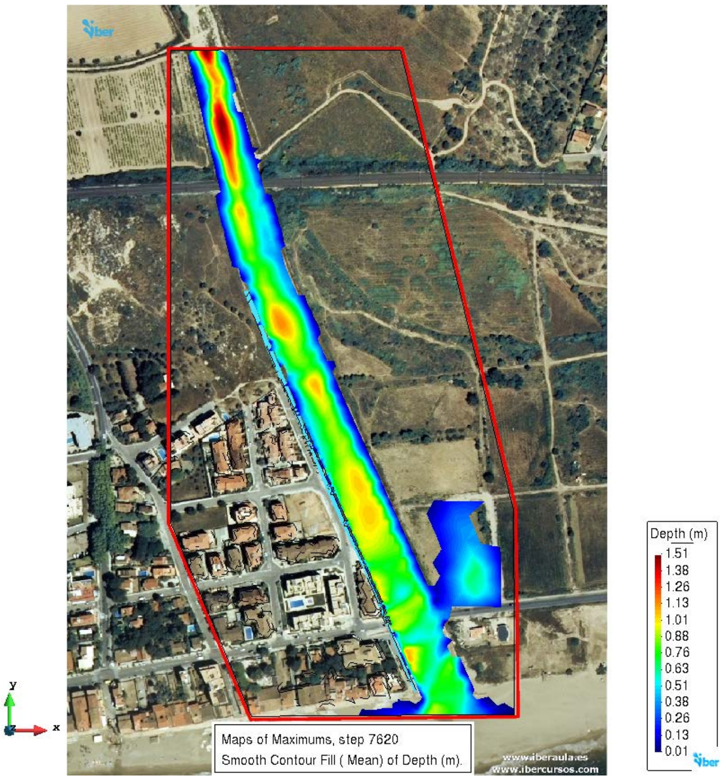



**Figure 46. Urban area is protected by the wall.**

## 4.5. Conclusions

With this exercise we have built the 2D geometry of the problem manually. We have then assigned land uses to 5 areas. We also built a no-structured mesh with different mesh sizes in which we set the real elevations.

Finally, we have refined an area of the mesh to include the geometry of a protection wall.

## 4.6. References

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, E., Dolz, J., Coll, A. (2014a). Iber: herramienta de simulación numérica del flujo en ríos. Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería, Volume 30, Issue 1, 2014, Pages 1-10, ISSN 0213-1315, DOI: 10.1016/j.rimni.2012.07.004

Bladé, E., Cea, L., Corestein, G. (2014b). Modelización numérica de inundaciones fluviales. Ing. del agua 18, 68. DOI: 10.4995/ia.2014.3144

Cea L, Puertas J. Flood modelling with the software Iber. In: II International Congress on Water: Floods and Droughts. Ourense. 27-28 Octubre 2016; 2016.

González-Aguirre JC, Vázquez-Cendón ME, Alavez-Ramírez J. Simulación numérica de inundaciones en Villahermosa México usando el código IBER. Ing del Agua. 2016;20(4):201. DOI: 10.4995/ia.2016.5231

# 5. Practical Example 3: Bridges and Culverts

## 5.1. Objectives

This example looks at the implementation of the two most common types of structures in flood studies: bridges and culverts. We will see how to define a bridge, including the piers and culverts, as a means of transferring flow from one point to another. We will analyse the hydrodynamic behaviour of the bridge, taking in account that the flow here is clearly two-dimensional. Furthermore, this example will serve as a brief review of the properties and criteria of the mesh as described in previous chapters, will provide further opportunities to create graphs, and will introduce Iber (Bladé et al., 2014b) tools for exporting results available in the postprocess and the layer options in the preprocess.

## 5.2. Description of the case study and input data

This example involves the analysis of the hydraulic behaviour of a bridge located in the northeast of Spain, very close to the village of Besalú.

This bridge is part of the transport infrastructure (road) that allows the Fluvià River to be crossed. The road itself links to another important road in the area, the A-26 highway. The bridge is 290 m long, with 5 symmetrical decks (58 m) with 4 rectangular piers (2 m wide) and 4 culverts on the left riverbank (3 m diameter). Figure 47 shows a scheme of the whole infrastructure.

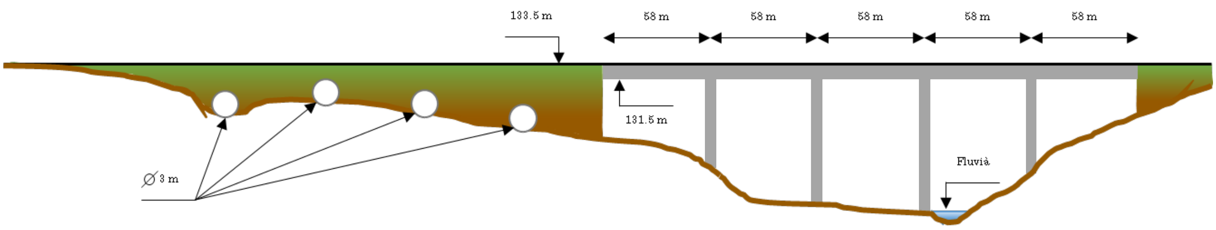



**Figure 47. Scheme of the bridge (upstream view).**

The bridge is located on a widening part of the river, including an old meander.

The data provided is the Digital Terrain Model (DTM.txt, 1x1 m cell size), the hydrograph (Discharge.txt), the orthophoto from a satellite image (Sat_image.jpg and Sat_image.jgw) and a basic model which includes the piers. Note that the hydrograph does not correspond to any return period; it has been created for this example.

The geometry of the model, as well as the bridge, will be set up after a topographical analysis of the study area.

## 5.3. Model set-up

### *5.3.1. Geometry*

Open the model provided (Model.gid) and save it with another name (e.g. Fluvia.gid). Note that "Fluvià" has an accent on the last letter, but it is recommended do not use special characters such as accent marks, symbols, etc., to name the models.

This model has been created with an old version of Iber. If you are using a more recent version, you will need to transform it to the new problem type (**Data>>Problem type>>Iber**); select **Transform to new problem type** and save the model. Note the ✓ has to be visible in **Iber problem type**.

After saving the project, we load the Sat_image.jpg image as a background image (**View>>Background Image>>Real size**)[5].

In this map (Figure 48) we can see the study area. The river crosses from west to east, and there are 4 surfaces which represent the 4 piers of the bridge. Also, on the left bank there are 4 circular culverts (Figure 48, bottom-right).


**Figure 48. Study area. View of the background image (left), the geometrical discretization of the piers (top-right) and the culverts (bottom-right).**

We will prepare two different models in order to consider the piers as a part of the model (Case 1) and as a hole in the mesh (Case 2). But before doing this we will create the geometry of the model, because there are some parts of the models which are common to both cases.

An interesting feature of Iber is that it is possible to define specific properties in specific areas (surfaces, lines or points). This can be done easily with the **Layers** menu (**Utilities>>Layers** or the button). If we open the **Layers** window (Figure 49) we can see that there is only one layer called "Piers". We can create ( ) and delete[6] ( ) layers, and do other things such as change the name ("Name"), change the

[5] The georeferenced file has to have the same name as the image file and has to be located in the same folder.

[6] Only void layers, without any entities (points, lines and surfaces).

colour (“C”), show/not show the layer (“I/O”), block/unblock the layer (“F/U”) and make the layer transparent (“Tr”).

For the current example, we will use this tool to create some new layers and to define specific properties (mesh criteria and land uses). The 6 new layers are “Road”, “Fields”, “River”, “Industry”, “Mining” and “Brushland”.

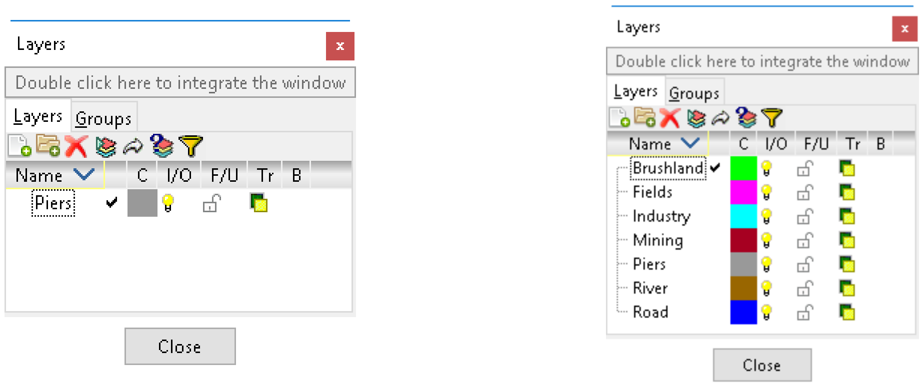



**Figure 49. Layers window. On the left the original layers and on the right the specific layer created for this study.**

We begin by defining the “Road”. Double click on a layer to activate it[7]. Using the **Create** line tool we can now create a polygon that represents the road and the bridge. Bear in mind that we must join this polygon with the existing polygons that represent the piers of the bridge. Then, create the surfaces (Figure 50). Note that the other road (located on the south) is the limit of the model, so it is not necessary to include it.


**Figure 50. Representation of the “Road” geometry. On the right side we can observe how the new layer (“Road”) has been joined with the existing one (“Piers”).**

[7] Note that the ✓ must be on the layer that we want to use.

We continue creating new surfaces corresponding to each layer. The “Mining” area is located downstream of the bridge and the “Industry” area is upstream (it is not necessary to include all its area, see Figure 51, left).

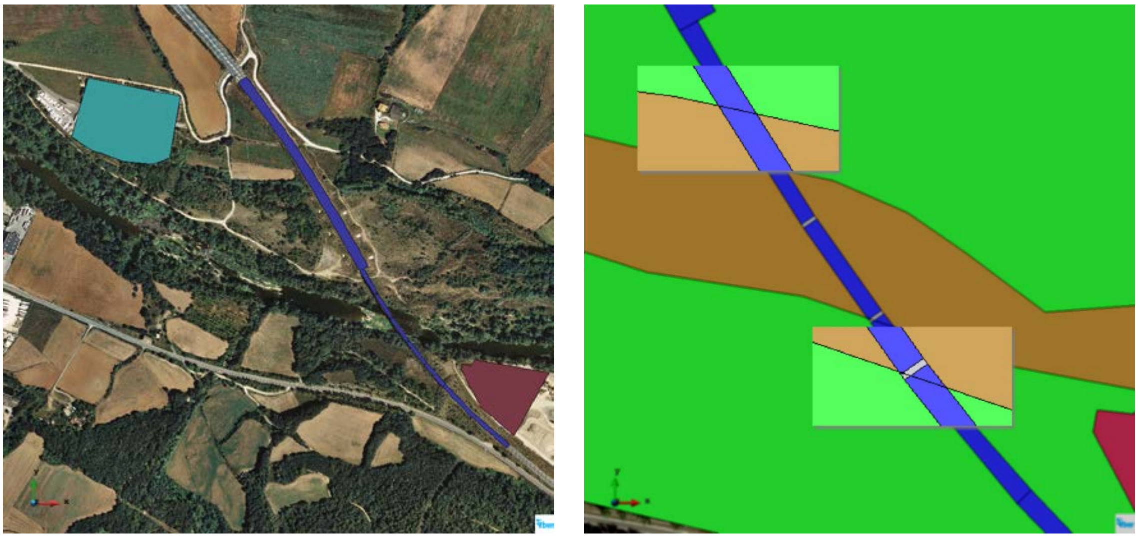


**Figure 51. Representation of the “Mining” and the “Industry” geometry (left) and the intersection area of the “River” layer with “Road” and “Piers” layers.**

The “River” should represent only the actual reach (as shown in the background image). Note that we have to pass over existing layers (“Bridge” and “Piers”). We need to intersect the lines and redraw the geometry to take in account the river discretization (important for the definition of land uses). To do this we use **Geometry>>Edit>>Intersection>>Lines** and select all the lines of “River” that pass over other layers. Delete the surfaces that have been intersected and redraw the geometry taking in account the layers criteria. Finally, we can create the surfaces that represent the river (Figure 51, right).

The “Fields” layer represents some agricultural areas in the study area (on both river banks). To define it, we repeat the same steps shown above. The rest of the model is considered part of the “Brushland” layer.

The geometrical discretization is shown in Figure 52. Each layer represents a specific part of the model, and will serve to define both the mesh criteria and the land uses (one Manning coefficient per surface).

In this model[8] we are going to define all the properties, and then define the mesh criteria, as a means of considering both Case 1 and 2.

[8] The data provided include the final model, which includes all steps carried out thus far.

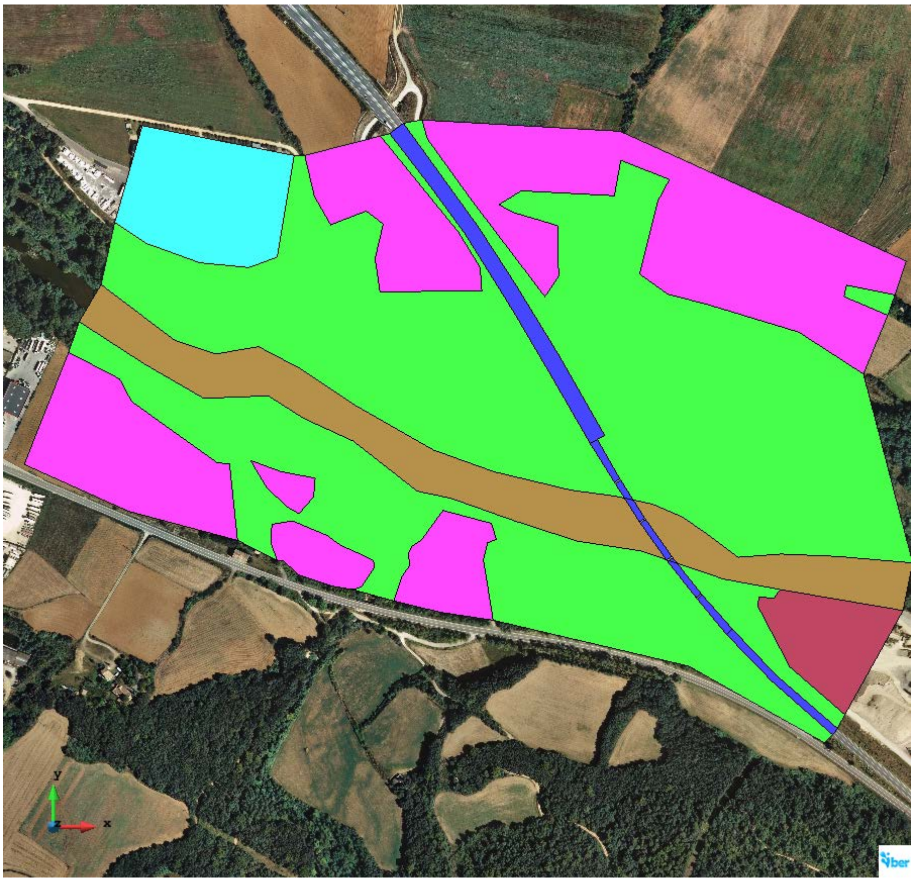



**Figure 52. Final geometrical discretization of the study area.**

### *5.3.2. Hydrodynamics*

We introduce the inlet as a Total discharge (subcritical flow) on the "River" line located on the west part of the model. The outlet condition (supercritical/critical flow condition) has been implemented on the east part of the model and must include the whole width of the river and the riverbanks.

We will now define the land uses using the geometrical discretization. The Manning coefficient of each layer is defined as follows: "Road" 0.02 $s \cdot m^{-1/3}$ (corresponding to "infrastructure"); "Mining" 0.075 $s \cdot m^{-1/3}$ (new creation); "Brushland" 0.05 $s \cdot m^{-1/3}$ (corresponding to "brushland"); "Fields" 0.028 $s \cdot m^{-1/3}$ (new creation); "Industry" 0.1 $s \cdot m^{-1/3}$ (corresponding to "industrial"); "River" 0.035 $s \cdot m^{-1/3}$ (corresponding to "river", although the value here must be modified); and "Piers" 0.02 $s \cdot m^{-1/3}$ (corresponding to "infrastructure").

To create a new Land use you only have to press "New land use" button ( ), introduce the name of the new land use and then define the Manning coefficient. Finally, it is necessary to Update the changes by the button .

Note that the land uses of the decks of the bridge must be defined as part of the terrain because the water will flow under the bridge. For this reason, we will consider all the decks of the bridge as land uses of the "river" and the "brushland". Figure 53 shows the land use discretization.

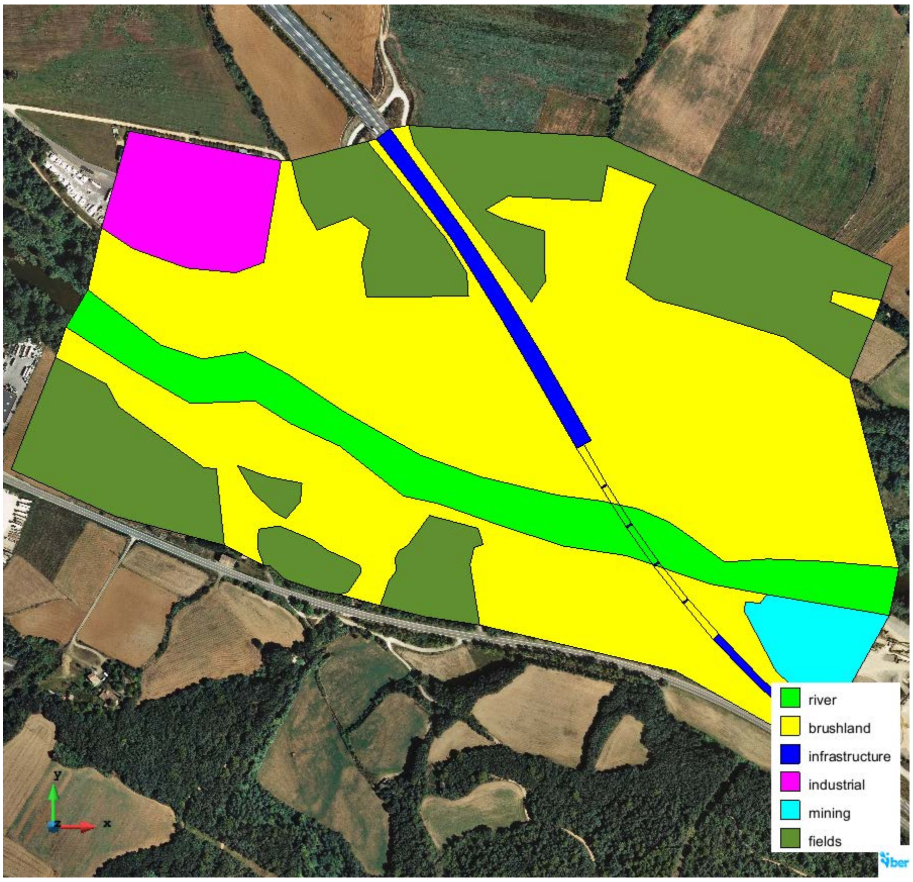



**Figure 53. Land uses discretization.**

### *5.3.3. Culverts*

Iber has a specific tool for introducing culverts. We understand culverts as hydraulic structures that allow for the transfer of water flow from one point to another. These structures are usually located under roads and rail lines in order to prevent the accumulation of water upstream when a bridge or other structure has been built on a platform higher than the natural topography.

This tool has been designed to simulate the working of rectangular and circular culverts (Figure 54). We can introduce one or more culverts using **Data>>Hydrodynamics>>Structures>>Culverts**. In this new windows we can create, delete and rename culverts. The start and end point of a culvert can be introduced manually or by using the button , selecting any point on the model (the coordinates will be acquired from the topography).

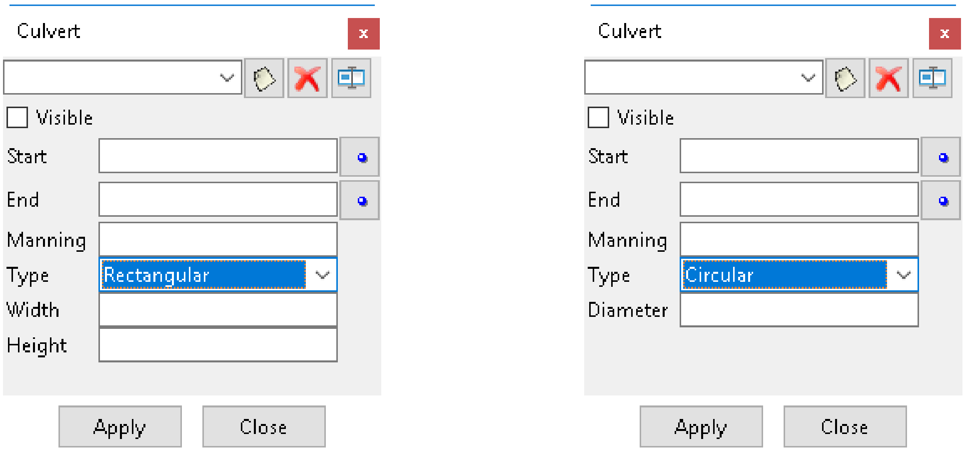



**Figure 54. Culvert window. Rectangular (right) and circular (left) parameters.**

In this case, we will define 4 culverts on the left bank of the river, under the road platform. The characteristics of the culverts are as follows:

**Table 3. Culvert characteristics.**

| Culvert | Start [m] | | | End [m] | | | Manning | Diameter |
|---|---|---|---|---|---|---|---|---|
| | X | Y | Z | X | Y | Z | [s·m$^{-1/3}$] | [m] |
| 1 | 477119 | 671757 | 125.24 | 477181 | 671753 | 125.18 | 0.015 | 3 |
| 2 | 477146 | 671718 | 126.14 | 477200 | 671716 | 124.54 | 0.015 | 3 |
| 3 | 477166 | 671673 | 125.31 | 477227 | 671672 | 125.42 | 0.015 | 3 |
| 4 | 477195 | 671629 | 126.10 | 477252 | 671625 | 125.69 | 0.015 | 3 |

The methodology for implementing culverts is, first, to create the culvert itself using the button . We then need to indicate all the parameters, then **Apply**. A blue line, with the name of the culvert ("culvert-X" by default, where X means the number of the culvert) will be automatically drawn to represent the culvert (Figure 55).


**Figure 55. Culvert representation.**

### 5.3.4. Bridges

Bridges are one of the most common structures in flood studies. With Iber, a bridge can be introduced as an internal condition in order to modify the shallow water equations accounting for the bridge discharge equations. The bridge condition must be defined on the upstream part of the bridge.

We can introduce a bridge using **Data>>Hydrodynamics>>Structures>>Bridge**. This window (Figure 56) asks for the parameters of the bridge, including the deck elevation (lower and upper part), the bridge opening (if we want to consider obstructions), and the flow coefficients ($C_d$ over the deck; free and submerged pressurized $C_d$ under the deck).

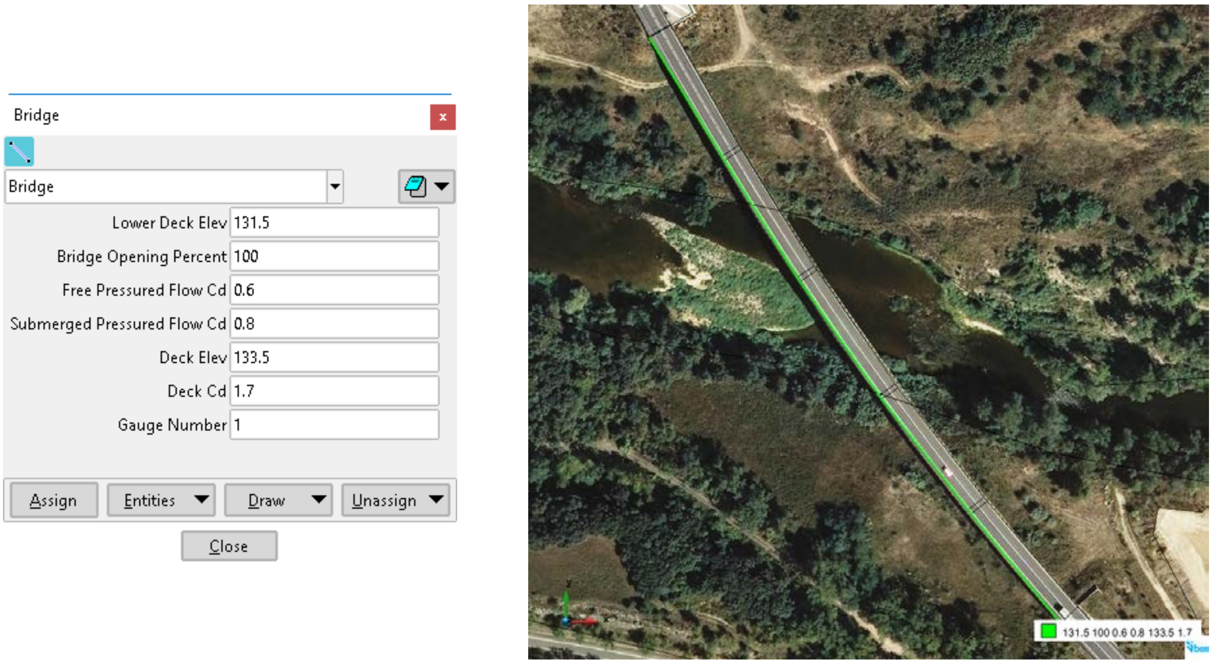



**Figure 56. Bridges definition. Parameters of the bridge (left) and green lines considered as a bridge (right).**

The bridge has a constant elevation (133.5 m) and a height of 2 m (131.5 m on the lower deck). We do not consider obstructions (bridge opening 100 %) and we use the discharge coefficients by default.

Once the parameters have been introduced, we **Assign** on the upstream line that defines the bridge (including the piers part), as shown in Figure 56 (left).

### 5.3.5. Mesh

Before creating the mesh, we save the model with another name, so that we can simulate two different cases. Case 1 considers the piers as a part of the models, and this implies that we need to introduce the piers as an elevation in the mesh. Case 2 considers the piers as a hole in the mesh, which implies that we should not mesh the area that represents the piers.

Nevertheless, some properties of the mesh are common to the two cases. Thus, we are going to assign different mesh sizes. For the "River" layer, this will be 5 m, for the "Roads" layer 3 m, for the "Piers" layer 2 m, and for the rest of the model 10 m. To do this, we use **Mesh>>Unstructured>>Assign sizes** in the **Surfaces** menu.

## Case 1

The first step is to generate the mesh and to load the DTM file (DTM.txt) in order to assign the bed elevation to the mesh elements (**Iber tools>>Mesh>>Edit>>Set elevation from file**). We obtain around 24,000 elements.

Second, we need to elevate the piers to the upper part of the bridge deck (133.5 m). To do this, we use **Tools>>Mesh>>Edit>>Set elevation constant** and assign the elevation to the "Piers" elements (perimeter and interior nodes).

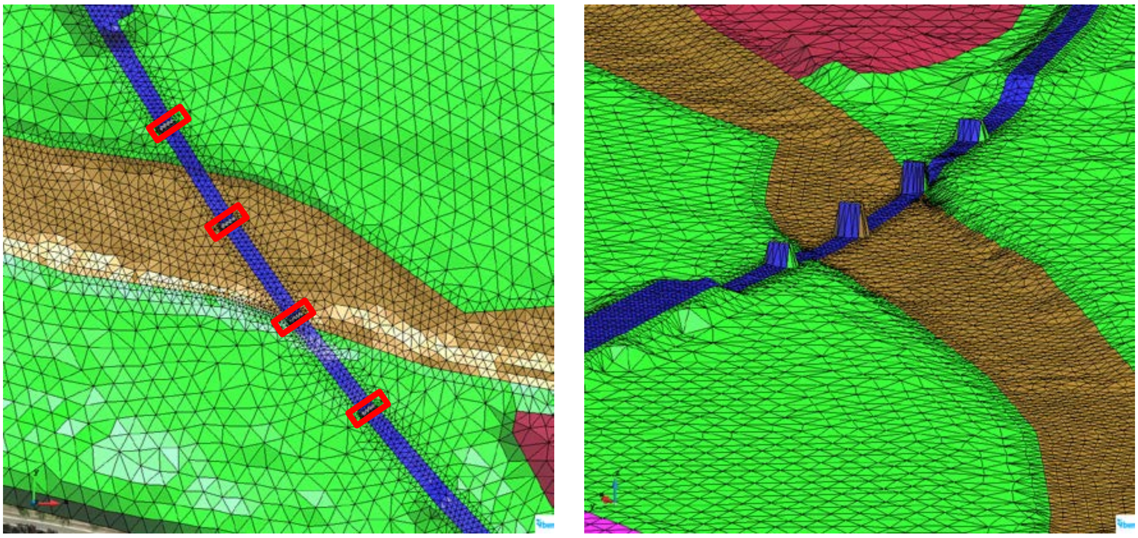


**Figure 57. Piers elevation process. Inside the red rectangle are the elements to modify their elevation (left) and the 3D visualization of the topography (right), including the new piers elevation.**

## Case 2

In the second example, we have to indicate in the model that the "Piers" surfaces must not be meshed. For this we use **Mesh>>Mesh criteria>>No mesh** in the **Surfaces** menu and select the appropriate surfaces (Piers").

Remember that we previously defined the bridge condition in the piers, but this condition is an internal condition, and now it becomes a boundary condition, because the piers have not been meshed. Hence we need to unassign the bridge condition. Use the **Bridge** menu (**Data>>Hydrodynamics>>Structures>>Bridge**) then repeat the mesh process (generate and elevate the mesh).

### *5.3.6. Calculation data*

In the **Problem data** window (**Time parameters** tab) we define the maximum simulation time (7200 s) and the results time interval (120 s). Then **Accept** the changes.

### *5.3.7. Calculation process*

We have now built the model, and we simply need to calculate it (for both cases). Using **Calculate>>Calculate** we can begin the simulation.

## 5.4. Results

### *5.4.1. Hydrodynamic results*

Here we will compare the two cases as calculated.

First, we check whether the maximum water extensions are the same in both cases. Figure 13 shows the maximum inundation, where no significant differences are observed, except in the flow around the piers. These differences are because of the different discretization of the piers, one through the mesh modification (elevation) and the other through a hole in the mesh.

In Case 1 there are inclined elements (Figure 57, left) around the piers that cause a different interaction between the piers and the flow. Due to this, a high-water elevation is produced upstream of the piers.

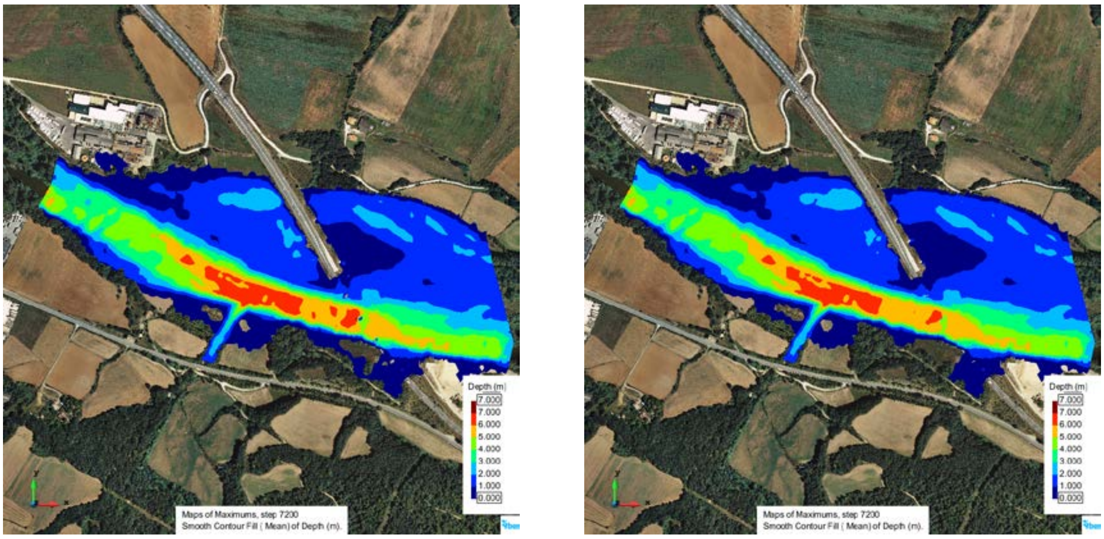



**Figure 58. Maximum inundation. Case 1 (left) and Case 2 (right).**

In this sense, something similar occurs with the velocity field. For example, if we plot the maximum velocity (Figure 59) we can see how in Case 1 we have higher velocities upstream and lower velocities downstream of the piers than in Case 2.

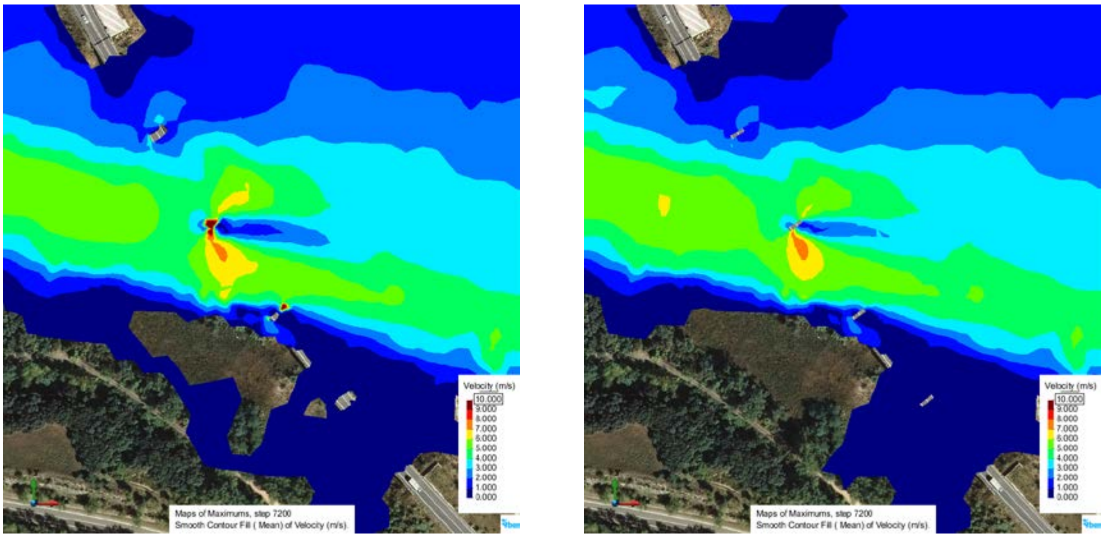



**Figure 59. Maximum velocities. Case 1 (left) and Case 2 (right).**

### *5.4.2. Bridge results*

In order to compare the two cases and to see if the water under the bridge reaches the decks, we will now create the maximum water elevation and export it for Case 1. Then, we will plot the same variable in Case 2, in order to compare them.

We create a profile using the **Graph** representation (**Window** menu). Create a new graph. Select **Line Graph** of the **Maps of Maximums** at the end of the simulation (7200 s), and the choose **Line Variation of the Water Elevation (m)**. **Apply** and then, in the **Command line** (located at the bottom of the interface), write the coordinates P1 [477228.0,671598.0] and P2 [477396.0,671362.0]. The maximum water elevation has now been plotted (Figure 60).

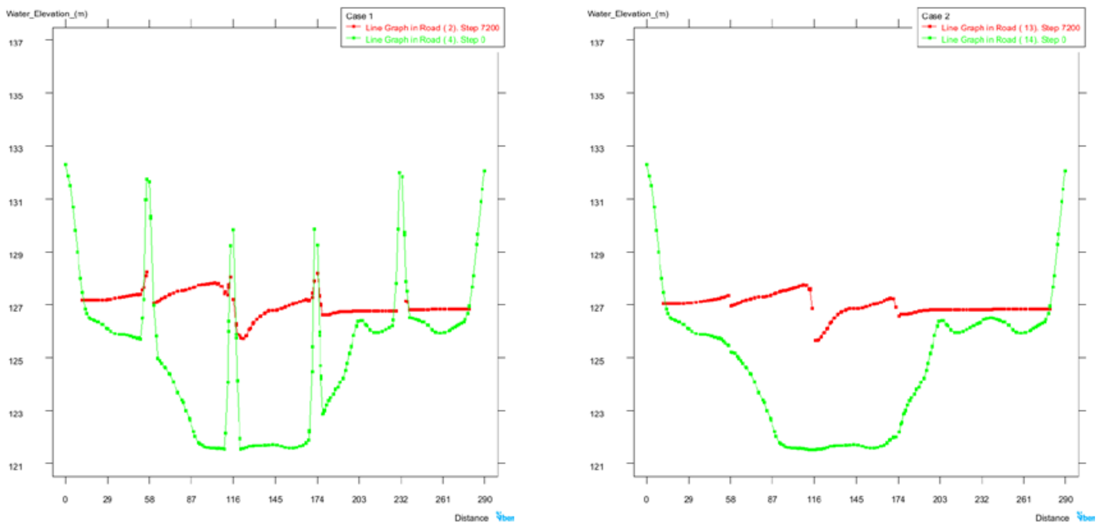



**Figure 60. Maximum water elevation under the bridge. Case 1 (left) and Case 2 (right).**

For an accurate comparison of the two cases, we will export the graph of Case 2 and import it into the graph set of Case 1. Go to **Files>>Export>>Graph** and select **Line Graph in Road. Step 7200**. Iber asks for a name to save it. Then, through Case 1 model we import it (note the graph shown in Figure 60 must be created and open). The data of Case 2's maximum water elevation is automatically plotted in the same graph set as Case 1.

We can now observe the differences between the water elevations (Figure 61). In general, the water level in Case 1 is higher than in Case 2, because in Case 1 the width of the piers is also greater than in Case 2 (in Case 1 we represented the piers as an elevation in the mesh, thus there is a gradual transition between the top elevation of the pier and the topography)[9].

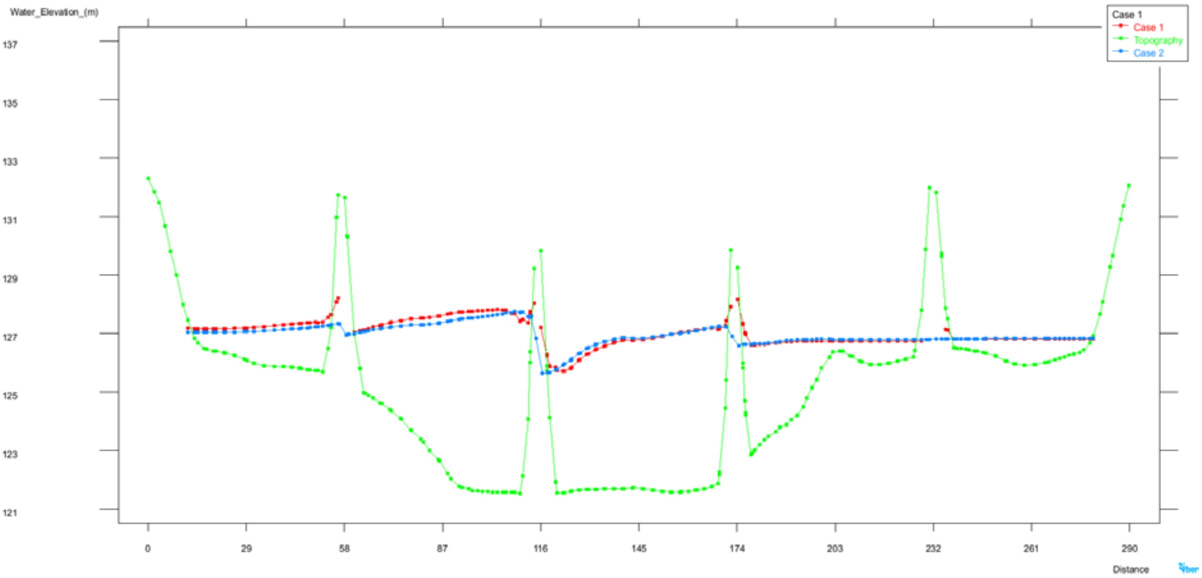



**Figure 61. Comparison of the maximum water elevation under the bridge in Case 1 and in Case 2.**

### *5.4.3. Culver results*

Iber allows the user to see which part of the flow is transferred by the culverts. Since version 2.4.3 of Iber, these results are saved in a file called "Culverts.grf" (previously they were saved in "Culverts.rep", and this file can be opened in Notepad or any other text-based application).

We can observe (Figure 62, top), in both cases, how the culverts transfer a notable amount of water from upstream to downstream of the bridge (around 125 $m^3/s$). The closest culverts to the river (Culverts 3 and 4) transfer less water than Culverts 1 and 2. This is because there is a depression on the left bank of the river due to the presence of an old meander.

This meander affects the culverts and also the flow field. We can see how the water flows from downstream to upstream on the left bank during the first time steps of the simulation.

[9] Note that the name and colours of the lines have been changed using the **Options** tab.

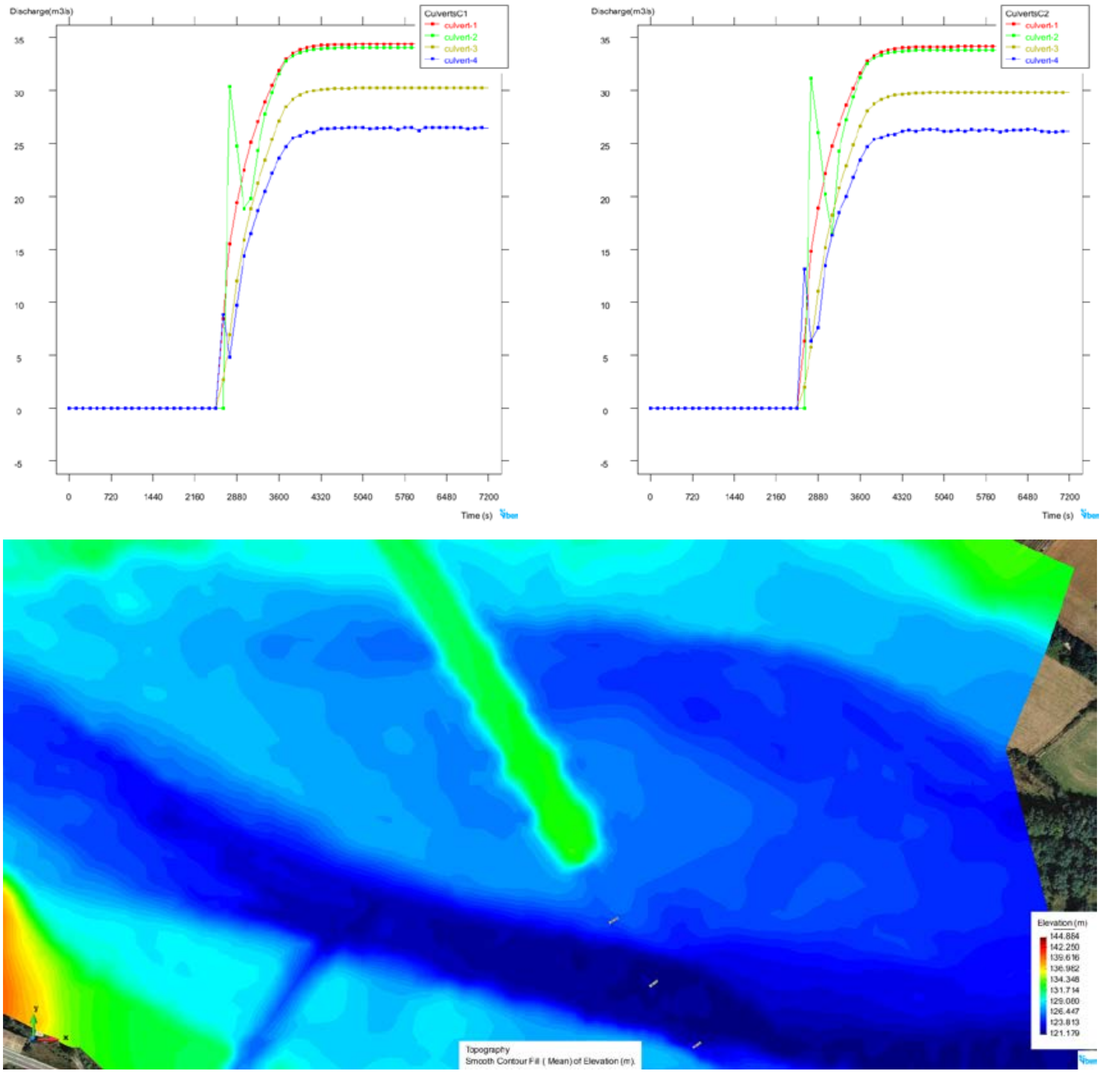



**Figure 62. Evolution of the flow through the culverts in Case 1 (top-left) and Case 2 (top right). Topography elevation map (bottom).**

## 5.5. Conclusions

In this example we have introduced some methodologies and modelling techniques to improve flood studies. The implementation of bridges and culverts allows us to reproduce more accurately the reality of a given situation (Bladé et al., 2014a).

Bridges in Iber are treated as an internal condition. This condition considers two dimensions of the space, and means that if the flow passes obliquely to the condition, Iber will respect this flow condition and will interact and modify the hydrodynamics to take in account the 2D phenomenon (e.g. the water elevation is greater than the deck elevation).

Two different ways for representing the piers of the bridge (Case 1 as a part of the model, and Case 2 as a hole in the mesh) have been shown. Both cases are valid in these conditions (water elevation is less than the deck elevation of the bridge), but only Case 1 is valid when the water elevation reaches and exceeds the bridge deck.

Culverts is an interesting means of allowing the transfer of flow between two points. Commonly used on road and train lines to allow small reaches to continue or for avoiding flooding upstream, we need to be able to see how these work in both case studies. The amount of flow transferred depends on the water level and the characteristics of the culvert (size, slope and roughness).

This chapter has also served to introduce the layers management tool and how to use it for defining certain properties. In the postprocess, we have seen how to export and import results, and to compare them in the same graph set.

## 5.6. References

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, E., Dolz, J., Coll, A. (2014a). Iber: herramienta de simulación numérica del flujo en ríos. Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería, Volume 30, Issue 1, 2014, Pages 1-10, ISSN 0213-1315, DOI: 10.1016/j.rimni.2012.07.004

Bladé, E., Cea, L., Corestein, G. (2014b). Modelización numérica de inundaciones fluviales. Ing. del agua 18, 68. DOI: 10.4995/ia.2014.3144

Cea L, Puertas J. Flood modelling with the software Iber. In: II International Congress on Water: Floods and Droughts. Ourense. 27-28 Octubre 2016; 2016.

González-Aguirre JC, Vázquez-Cendón ME, Alavez-Ramírez J. Simulación numérica de inundaciones en Villahermosa México usando el código IBER. Ing del Agua. 2016;20(4):201. DOI: 10.4995/ia.2016.5231

# 6. Practical Example 4: Dam break

## 6.1. Objectives

This example introduces the user to dam break analysis using Iber. The Breach tool will be explained, which allows us to define the shape and other properties of the breach formation process. We will analyse the water depth evolution in a reservoir and downstream of a dam, and also the maximum water elevation in the village located downstream. The way to represent the breach formation and its effects on the modelling process will be shown, as well as how to create hydrographs. This chapter will serve as a brief review of mesh properties and other criteria dealt with in previous chapters, and also to introduce Iber's tools for creating graphs in the postprocess.

## 6.2. Description of the case study and input data

This example involves the analysis of a dam break and the effects provoked by flood inundation downstream.

The Bayco u Ortigosa reservoir was built to avoid floods on the Ortigosa River. The reservoir is located in the southeast of Spain, occupies 111.54 ha and has 6.2 $hm^3$ of storage capacity. The dam is 648 m long and 43 m high.

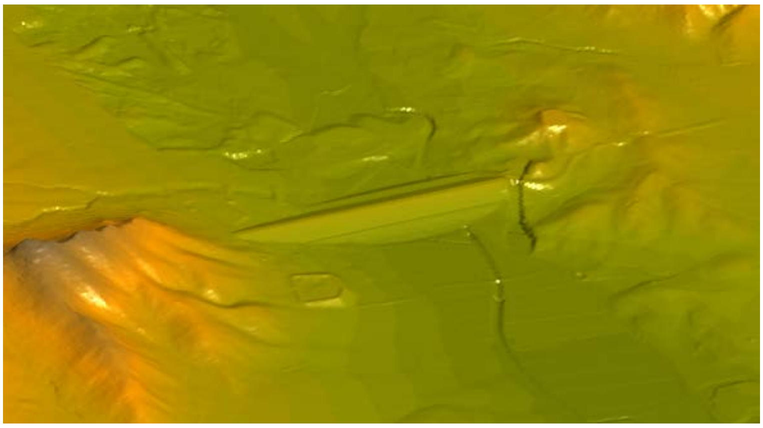


**Figure 63. Representation of the dam.**

The presence of a village (Ontur, at south) is the reason for a dam break analysis.

The data provided consist of a Digital Terrain Model, which has a 5x5 m cell size resolution (DTM.txt) and includes the dam (Figure 63), and a background images (maps of the study area, Map.jpg and Map.jgw).

The geometry of the model will be set up after a topographical analysis of the study area, and will be refined after considering the model results.

## 6.3. Model set-up

### *6.3.1. Geometry*

After saving the project, we will load the map as a background image Map.jpg (**View>>Background Image>>Real size**). This image is a georeferenced map located in the real UTM coordinates[10].

Though this map (Figure 64) we can see the reservoir located to the north (in light blue), Ontur village to the south (in pink), plus the contour topography lines (in brown).

The reservoir limit and the contour lines will serve to define the geometry of the model.

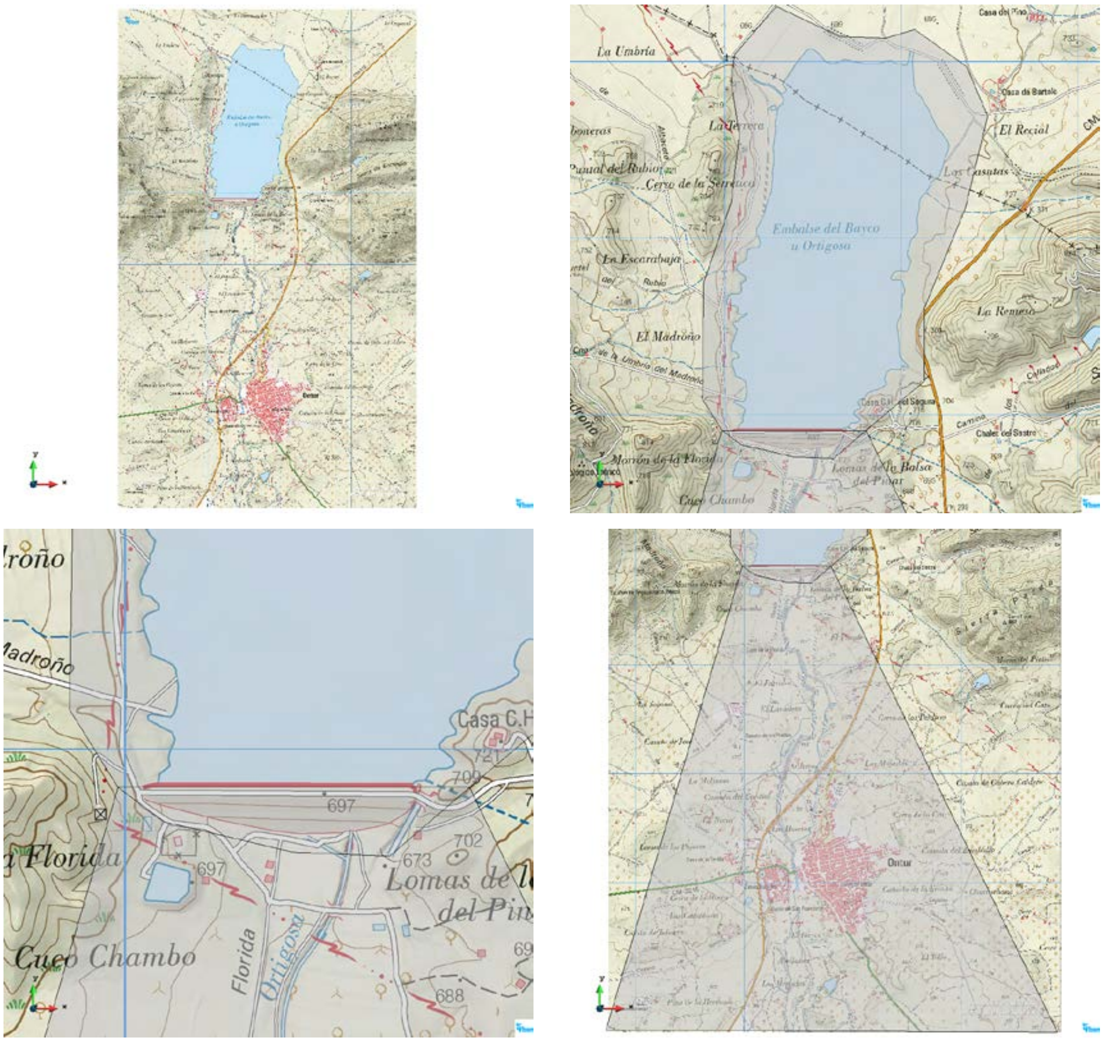



**Figure 64. Study area. View of the background image (top-left), the geometrical discretization of the reservoir (top-right), the dam (bottom-left) and the rest of the model (bottom-right).**

[10] The georeferenced file must have the same name as the image file, and has to be located in the same folder.

Using the geometry tools, we will create three surfaces (reservoir, dam and the rest of the model). First, we define the reservoir. Zoon in ( ) to the reservoir and draw the perimeter with a generous offset (Figure 64, top-right). For the dam, we will define another polygon, bearing in mind that the dam has a variable width downstream (Figure 64, bottom-left). Finally, for the rest of the model we extend this area from the corners of the dam to the southernmost part of the model[11] (Figure 64, bottom right).

It might be useful to define a more accurate mesh in the village and its neighbourhood. For this reason, we will create another surface on the village.

We delete the surface located downstream of the dam (Figure 64, bottom-right) and create another polygon with the village inside it. We create the surfaces by selecting, first, the external and the internal lines, in order to consider the village area as a hole in the external surface (Figure 65, left). Then we create the village surface (Figure 65, right).

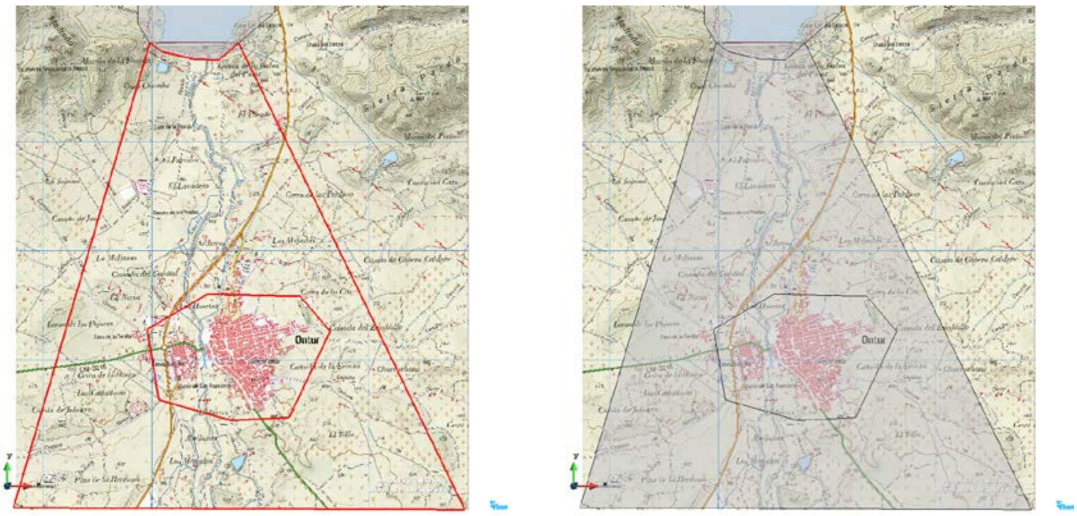


**Figure 65. Village surface definition. View of the selected lines of the rest of the model polygon (left) and the result of this surface creation process (right).**

### *6.3.2. Hydrodynamics*

We define as the outlet boundary condition the southernmost line of the model. A priori, we do not know what the flow condition on this boundary is. Nevertheless, our study area is limited to the village, thus we can assign a supercritical flow, because the outlet is far away from the study area.

This model does not have a defined inlet boundary condition[12].

The initial condition will be assigned only on the reservoir surface (Figure 64, top-right). The value of this condition is 687 metres (**Elevation**).

We use "river" as a land use (0.025 $s\cdot m^{-1/3}$) for the whole model except for the Ontur village (residential, 0.15 $s\cdot m^{-1/3}$).

[11] The DTM extension is no longer than the image. Do not overdo it.

[12] Iber can work without boundary conditions (e.g. the inundation process of a channel considering only an initial water volume).

### *6.3.3. Mesh*

The geometrical definition will serve to define different element sizes. We have two zones where the hydrodynamics must be more accurate than for others. The dam and the village are important for the study because the breach will be formed in this area and we need more detailed results in the village.

We will now define the following element size (**Mesh>>Unstructured>>Assign sizes on surfaces**):

- Reservoir: 200 m
- Dam: 5 m
- Village: 15 m
- Rest of the model: 50 m

We can check the mesh size assigned using **Mesh>>Draw>>Sizes>>Surfaces** (Figure 66 left). Once the mesh criteria have been checked, we generate the mesh. The value requested during Mesh generation is 250 m. The model will have around 35000 elements (Figure 66).

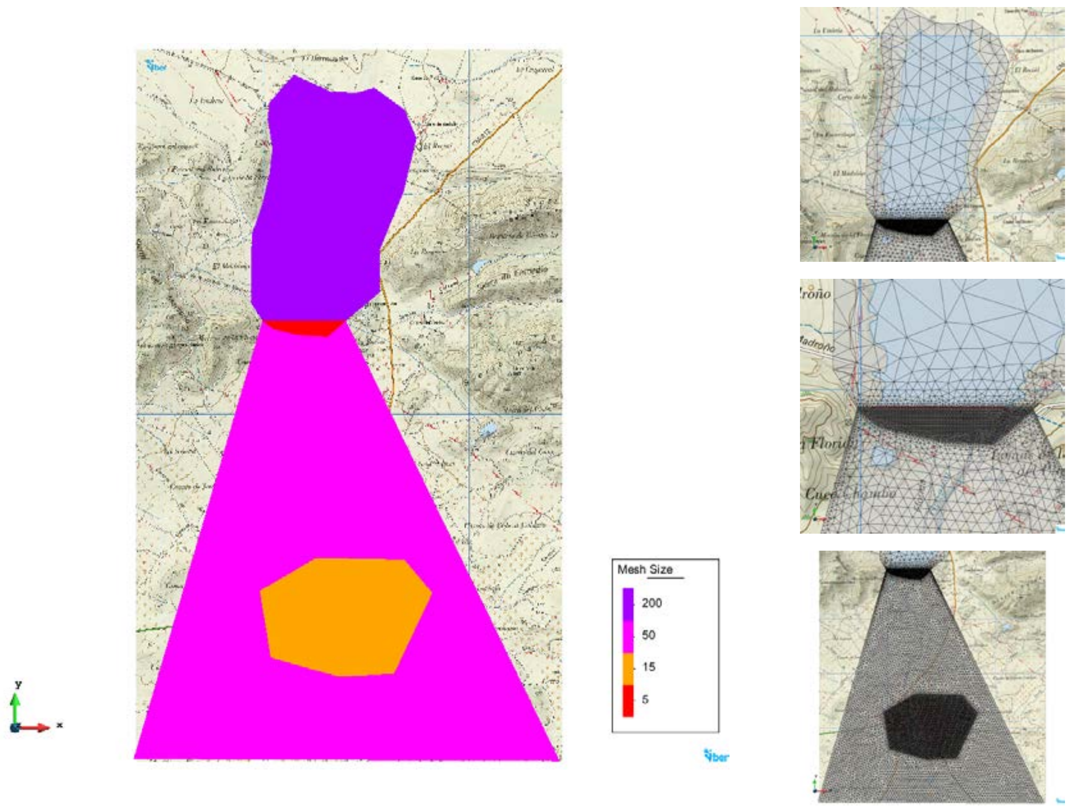



**Figure 66. Mesh criteria. Mesh size of each surface (left) and mesh model in each zone (right).**

Once the mesh has been generated, we load the DTM file (DTM.txt) to assign the elevation on the mesh elements (**Iber tools>>Mesh>>Edit>>Set elevation from file**).

### *6.3.4. Breach*

Iber has a specific tool for introducing the breach parameters. It will read these and will make a mesh deformation in order to reproduce the evolution of the breach, allowing the exit of the water over the breach.

Iber lets the user to define two types of breach: one according to the Spanish Technical Guide (MMA, 1996); the other a trapezoidal shape according to the parameters defined by the user. For each methodology the user must define the parameters needed. Iber allows you to define more than one breach using both methodologies. In **Data>>Breach>>Breach definition** the user can introduce the breach parameters (Figure 67).

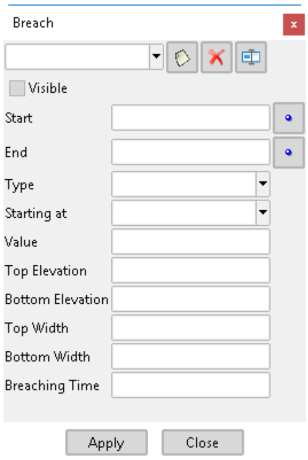



**Figure 67. Breach window.**

Having completed the previous step to introduce the breach parameters, the breach is created using the button (we can rename, , or delete, , each breach). For this example, we will use the **Trapezoidal breach type**.

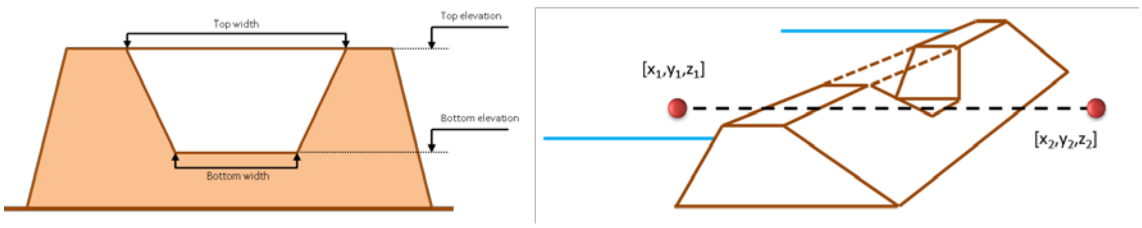



**Figure 68. Definition of the breach parameters.**

Once created ( ), we use the button to define the first ([$x_1$,$y_1$,$z_1$]) and the second ([$x_2$,$y_2$,$z_2$]) points of the breach line axis[13] (Figure 68). Choose one point upstream and another one downstream of the dam, as shown in Figure 69. The breach will be created at a time equal to 1200 s. The geometry of the breach is defined thus: Top elevation 697 m; Bottom elevation 672 m; Top width 200 m; and Bottom width 50 m. The breaching time defines the time needed for the breach formation. In this case we define 1 hour (3600 s) because the dam is built with granular materials.

[13] The Z value is needed at both points in order to define the full coordinates, but for the breach formation it is not needed. The bottom elevation defines the lowest part of the breach.

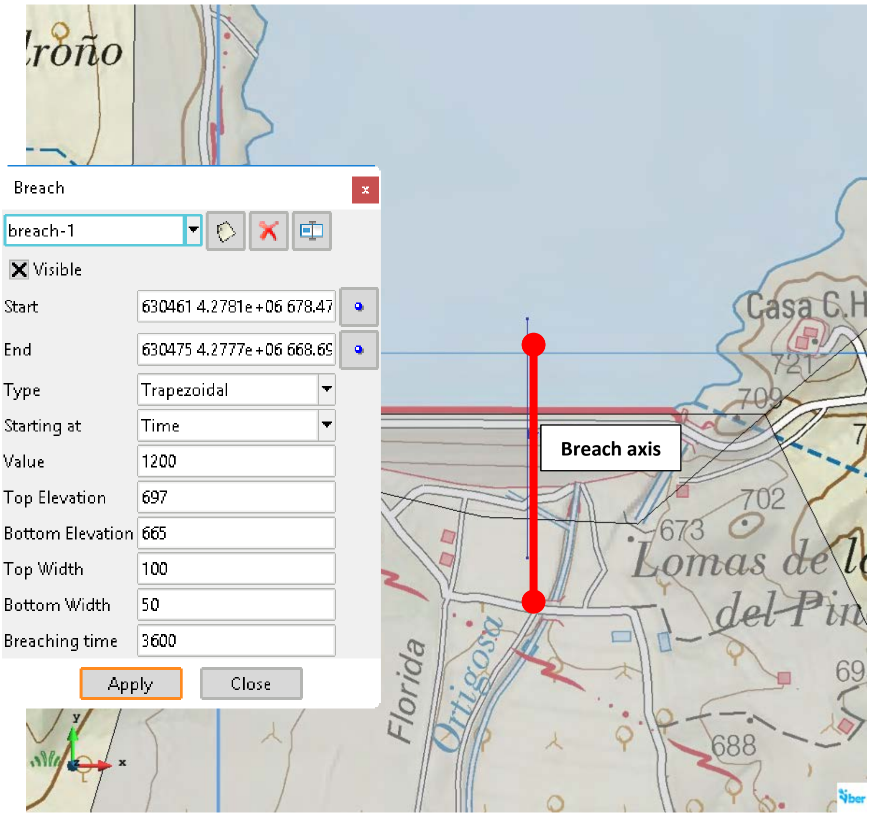



**Figure 69. Breach definition. Location of the breach axis (read line).**

### *6.3.5. Calculation data*

The calculation data settings define the parameters of the simulations (time, results, calculation modules, etc.). For this simulation, we use the hydrodynamic module (by default) and the breach formation module. The activation of the latter is required.

Taking into account the breach time formation (1 hour), the maximum simulation time will be defined as four hours (14400 s). The results will be written each 5 minutes (300 s). We define this as Time Parameters (**Data>>Problem data**).

On the **Breach** tab, enable the **breach formation** (Figure 70). Finally, **Accept** the changes.

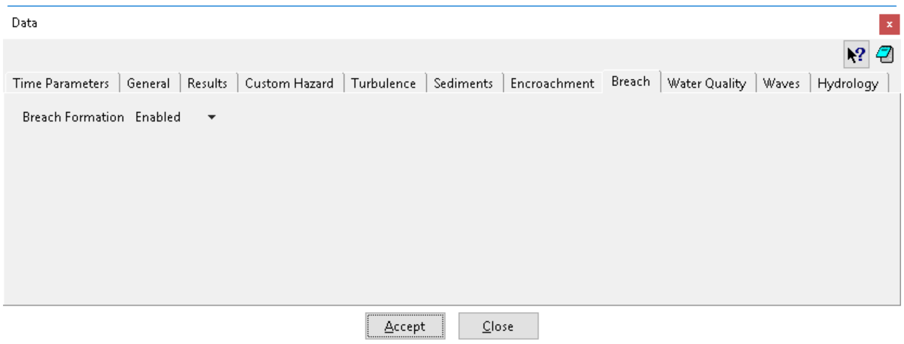



**Figure 70. Data window. Breach tab.**

### *6.3.6. Calculation process*

We have now built the model, and we simply have to calculate it. Using **Calculate>>Calculate** we start the simulation.

## 6.4. Results

### *6.4.1. Hydrodynamic results*

Once the calculation process is finished, we check the results.

Using the **View results** windows, we set the water elevation at the beginning of the simulation (Figure 13, left) and compare it to the water elevation at the end of it (Figure 13, right). We can observe that the water flows from the dam to downstream, passing through Ontur village, and at the end of the simulation the reservoir is nearly at its lowest storage capacity.

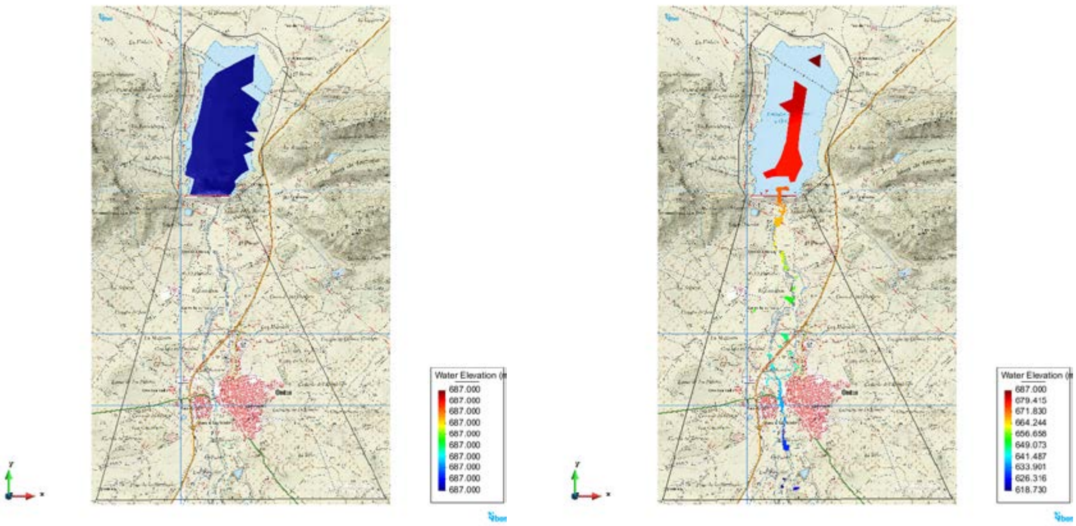



**Figure 71. Smooth contour fill representation of the water elevation. Results at the beginning (left) and the end (right) of the simulation.**

The inundation process is shown in Figure 72. These images show the evolution of the water depth across the simulation. We can see that the water flows downstream when the breach reaches the water level.

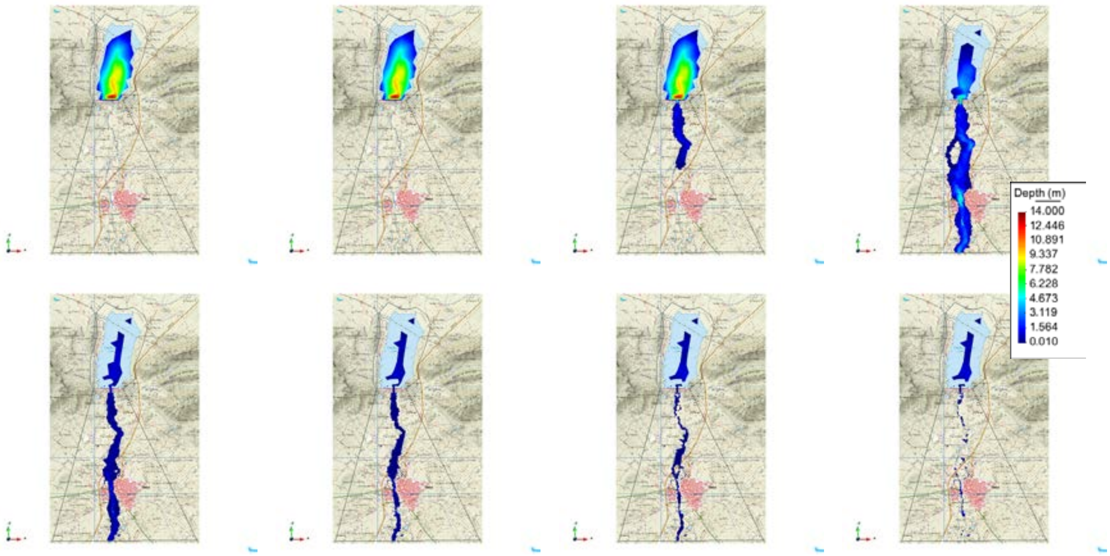



**Figure 72. Smooth contour fill representation of the water depth. From left to right, and from up to down, water depth at times 1800 s, 3600 s, 5400 s, 7200 s, 9000 s, 10800 s and 14400 s.**

We will now show the maximum extension of the flood using **Maps of maximum** results. In **Analysis** we select **Maps of Maximums** at the end of the simulation (14400 s) and the result **Depth**. This yields interesting results in the analysis of the flooding of Ontur village (Figure 73).

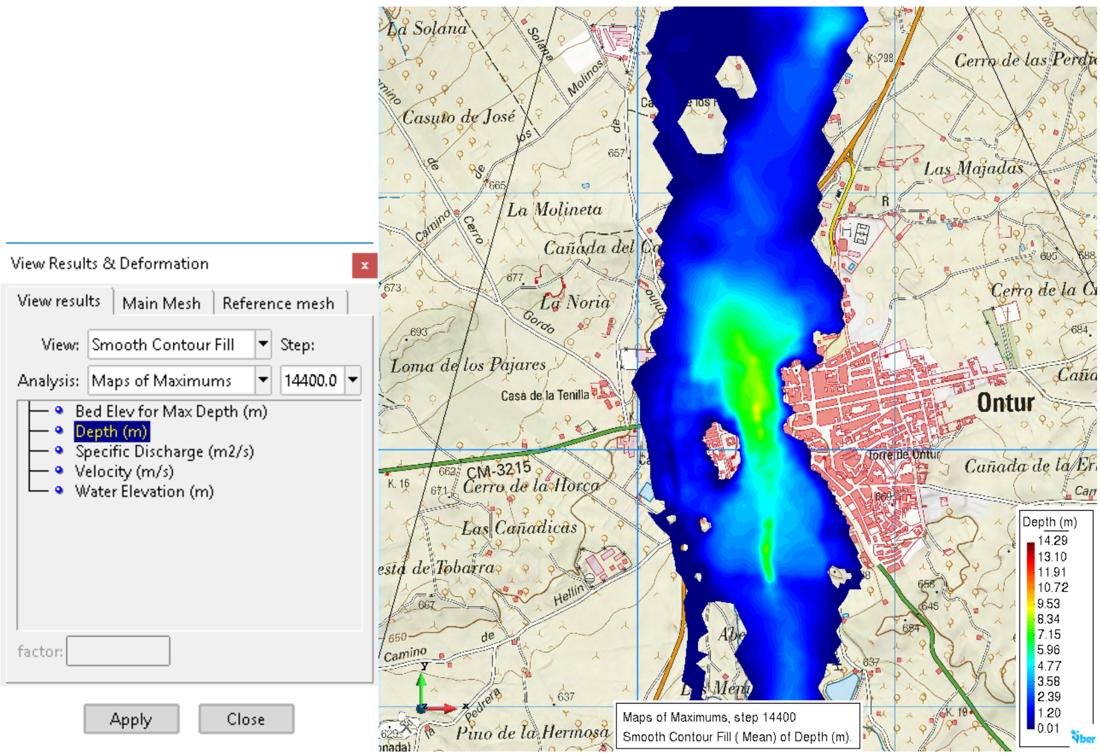



**Figure 73. Maximum inundation provoked by the dam break.**

### 6.4.2. Dam break results

As noted above, Iber modifies the mesh topography to create the breach. This modification allows the movement of the water from one part of the model (upstream) to another (downstream). We can observe the breach formation using the **Sediments Analysis**[14]. This kind of analysis allows the user to observe the breach formation process through **Bed elevation** or **Erosion** results[15].

The breach formation starts at 1200 s and ends 1 hour later, so we represent the breach formation process from 1200 s to 4800 s, every 1200 s (Figure 74).

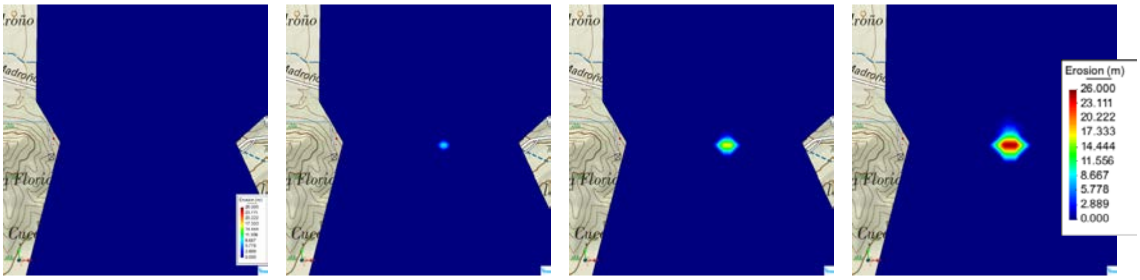



**Figure 74. Smooth contour fill representation of the erosion. From left to right, erosion at times 1200 s, 2400 s, 3600 s and 4800 s.**

Finally, we will analyse the hydrograph of the dam break. The hydrographs ($m^3/s$) are the integration through a line of the specific discharge ($m^3/s/m$) over time. So, we must define the cross section where we want to plot the hydrograph, that is, the integration line.

Change the visualization of results to the **Maximum water depth**. The integration lines must be defined from left to right in the flow direction. Through **Geometry>>Cut 2D polygonal** we can define the cross-section. Zoom in on the dam area and make the cut (from left to right in the flow direction) downstream of the dam with extra length on each side (Figure 75). If the length of the cut is smaller than the maximum water extension, Iber does not take it in account for the hydrograph generation.

Then, open the **View graphs** window (**Window** menu). On **Create tab** choose **Integrate Vector Normal View**, **Hydraulic Analysis** and the **Specific discharge** as a result (the time definition is not needed for this graph). Click **Apply** and select the cross section previously created. The hydrograph will be plotted automatically (Figure 76).

We can observe that the maximum discharge is produced at 4800 s and the peak discharge value is around 4500 $m^3/s$.

In the same way, we can plot the hydrographs at Ontur village and at the end of the model in order to assess the propagation process of the dam break discharge. To do this, we create two new cuts, one in the middle of Ontur and the other just before the boundary of the model (at the south). Thus, we will repeat the steps previously described.

---

[14] The breach formation is a modification of the mesh. It does not generate sediment transport if this calculation module is not activated, but the mesh modification is defined in the results as a bed modification in the **Sediment Analysis**.

[15] Erosion results show sedimentation processes as negative values and erosion processes as positives.

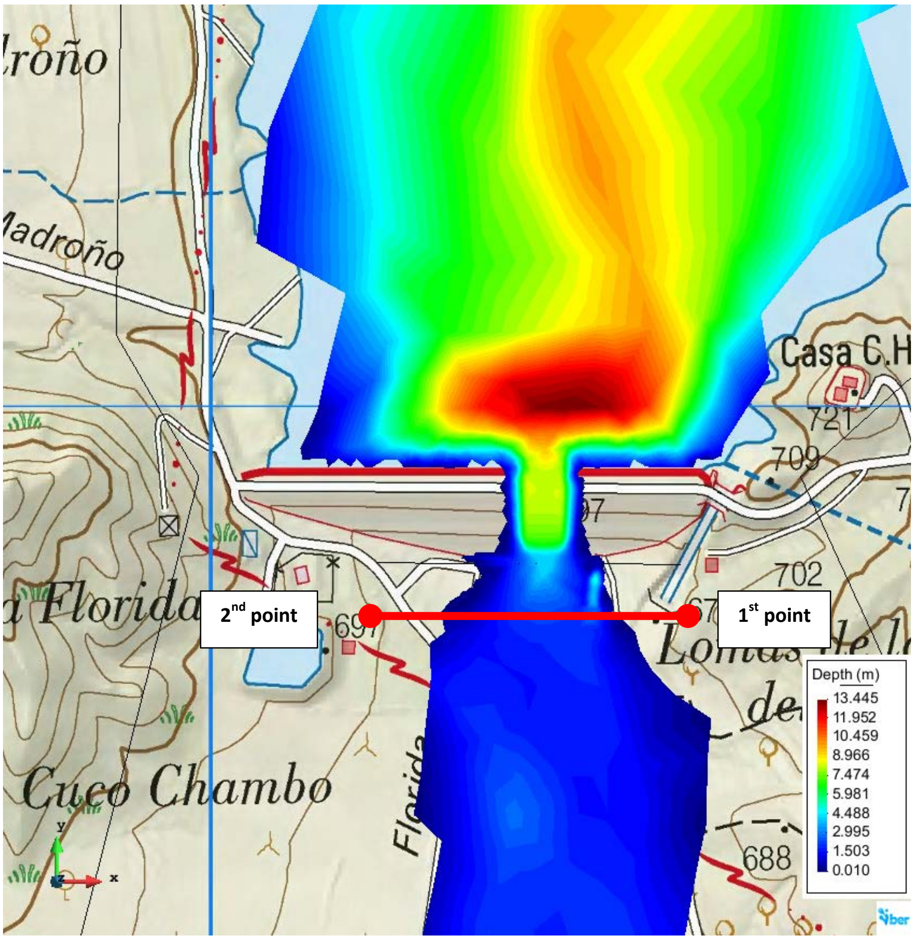



**Figure 75. Cut definition. The red line defines the length of the cut.**

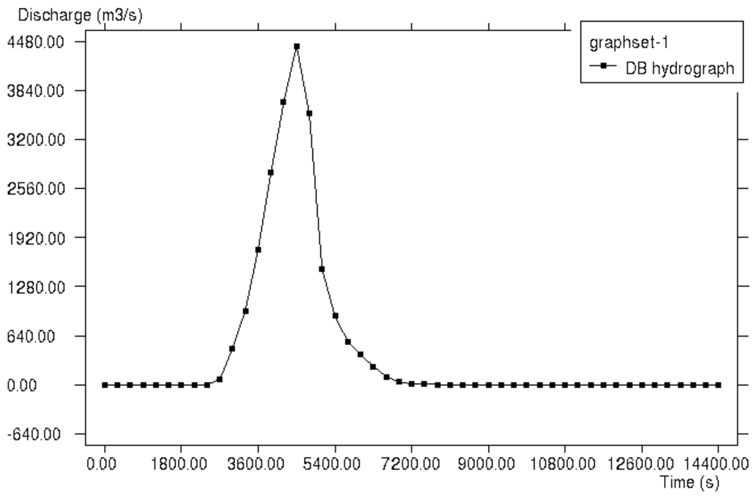



**Figure 76. Hydrograph downstream of the dam.**

Each graph will be placed in the same graph-set with a different colour (Figure 77). We can see that the peak discharge reduces to around 300 $m^3/s$ when the flow arrives at Ontur village, and to around 400 $m^3/s$ at the end of the study area.

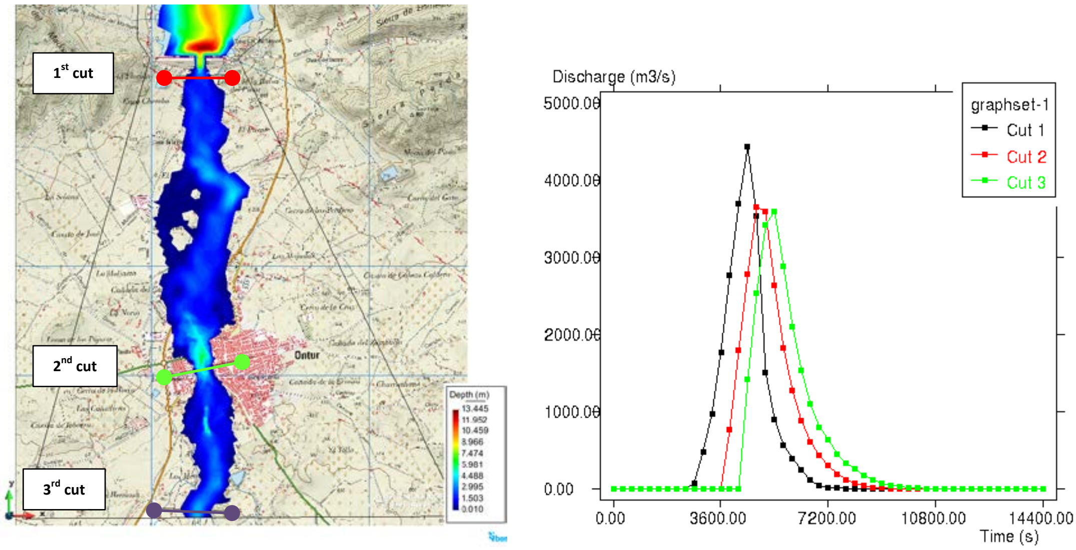



**Figure 77. Hydrograph representation. Location of the cuts (left) and the hydrograph plot of each cut (right).**

## 6.5. Conclusions

A dam break is a complex phenomenon, involving many hydraulic, structural and geotechnical process. Iber allows the user to define a hypothetical breach in a dam in order to assess the inundation process.

Iber incorporates two different breach formation methodologies: one based on the Spanish Technical Guide, and another in which the user defines all the parameters.

This exercise has served to introduce the user to Iber's breach formation module. It has shown how the breach formation modifies the topography and how this has allowed the water movement from inside the reservoir to the flood plains located downstream.

It has also explained how to create a hydrograph and how to analyse the topography modification caused by the breach formation process.

## 6.6. References

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, E., Dolz, J., Coll, A. (2014a). Iber: herramienta de simulación numérica del flujo en ríos. Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería, Volume 30, Issue 1, 2014, Pages 1-10, ISSN 0213-1315, DOI: 10.1016/j.rimni.2012.07.004

Cea, L., Bladé, E., Corestein, G., Fraga, I., Espinal, M., Puertas, J., (2014). Comparative analysis of several sediment transport formulations applied to dam-break flows over erodible beds. Proceedings in: EGU General Assembly 2014, Held 27 April - 2 May, 2014 in Vienna, Austria, id.7000.

MMA (1996). Clasificación de presas en Función del Riesgo Potencial - Guía Técnica. Ministerio de Medio Ambiente. Dirección General de Obras Hidráulicas y Calidad de las Aguas; 1996. 64 p.

# 7. Practical Example 5: Sediment Transport: Bedload

## 7.1. Objectives

This example will serve to explain how to calculate the elevation variation of bottom canals. Specifically, we will look at a rectilinear canal that has a constriction in its central area.

The width variation often generates a scour of the channel bottom. We see this phenomenon in bridge areas where the bridge abutments effectively narrow of the river.

There are three types of erosion around bridges: General erosion, squeeze narrowing, and local erosion on bridge bents and abutments.

The local erosion on bents and abutments is usually simulated by empirical expressions, since the water flow around these elements is strongly three-dimensional and non-uniform. However, in this example we will consider erosion by narrowing, with two sediment sizes on the channel bottom.

## 7.2. Description of the case study and input data

We have a rectangular channel with a horizontal bottom. The inlet discharge is 15 $m^3/s$. This discharge does not have any sediment. The outlet depth is 1 m.

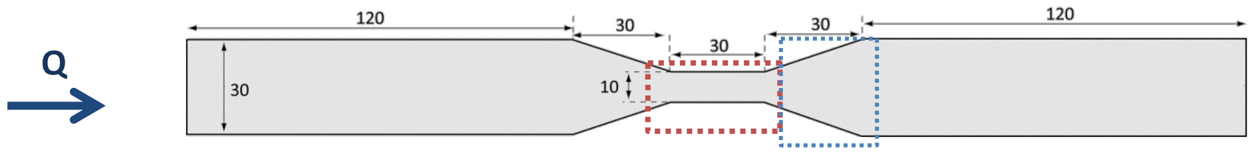



**Figure 78. Channel geometry.**

In this example we have two different areas where the geometry changes. The channel contraction will cause an increase in velocity and consequently an increase in sediment transport capacity. Then, there is a channel expansion that will cause the velocity to decrease. At the same time, the eroded material will be deposited, and the bottom elevation will increase.

In our example we will consider two types of sediment.

First we will consider a sediment with the following characteristics:

- Diameter: 5 mm
- Porosity: 0.5
- Internal angle of friction: 0.55 rad

Then, we will repeat the study with sediment of a 10 mm diameter. The porosity and the internal angle of friction will not vary.

## 7.3. Model set-up

### *7.3.1. Geometry*

The first step is to create the geometry. We introduce the coordinates shown in Figure 79. When the lines are made, we need to make surfaces from them.

The channel bottom has a zero elevation.

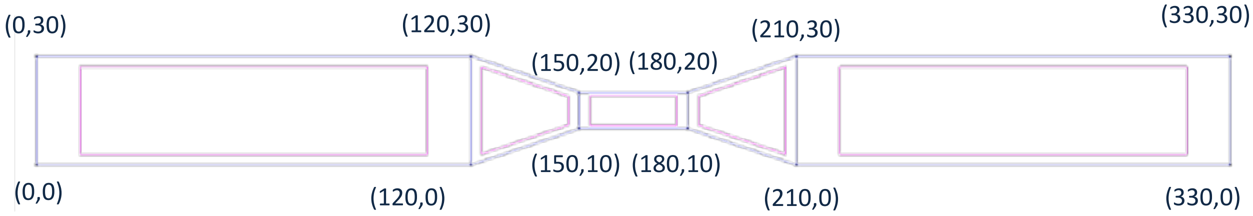



**Figure 79. Geometry of the channel.**

With the geometry defined, we set the initial and boundary conditions of the problem.

### *7.3.2. Hydrodynamics*

The inlet flow is 15 $m^3/s$, which we set using **Data>>Hydrodynamics>>Boundary Conditions>>2D Inlet>>Total Discharge**.

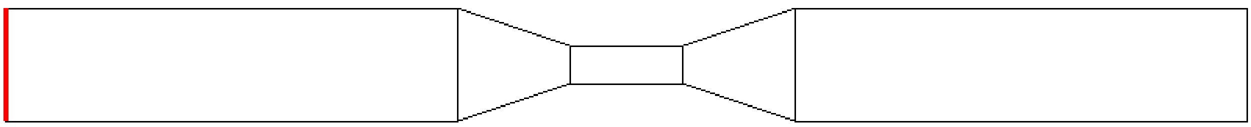


**Figure 80. 2D Inlet (red line highlighted).**

Due to the sediment transport, we need to define the bedload boundary conditions, using **Data>>Sediment transport>>Bedload Boundary Conditions**.

If we open the **Solid Discharge** list, we can select from four types of conditions:

- **None (Clear Water)**. Zero solid discharge on the boundary.
- **Bedload Capacity**. The solid discharge will be calculated by the transport expression that we have selected in **Problem Data**. Using **Data>>Problem Data>>Sediments** we can turn the suspended sediment and bed transport on or turn off. There are different expressions to simulate these transports.
- **Time dependent**. The user defines the solid discharge variety over time.
- **ql/qs.** The user defines the ratio of liquid discharge to solid discharge.

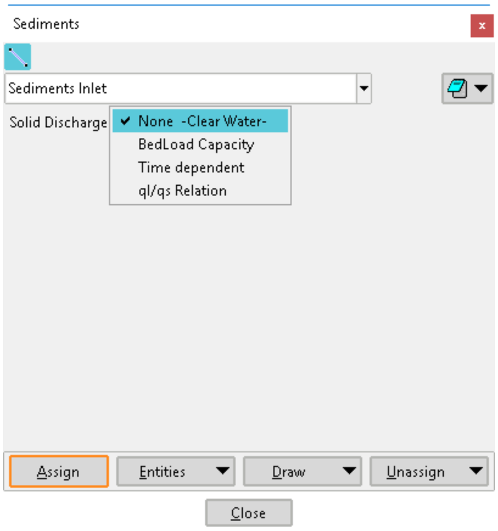



**Figure 81. Boundary conditions for Bedload sediment transport.**

We select **None (Clear Water)** because the inlet flow does not have any sediment; use **Assign** and select the inlet boundary.

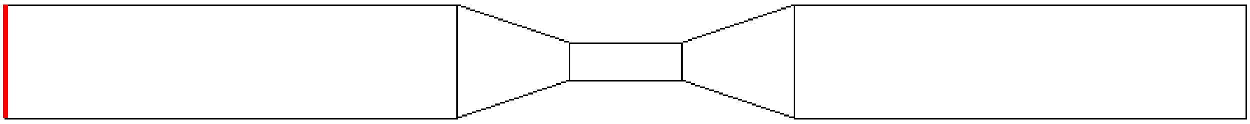


**Figure 82. How to assign sediments initial conditions.**

For the outlet boundary we assign 1 m. depth, using **Data>>Hydrodynamics>>Boundary Conditions>>2D Outlet>>Subcritical>>Given level**. It is not necessary assign any boundary condition for sediment transport because the model will calculate it automatically.

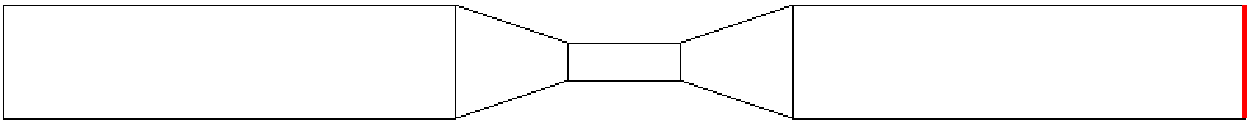


**Figure 83. How to assign 2D Outlet.**

Now we need to assign the initial condition to all the surfaces. We will give them a depth of 1 m, using **Data>>Hydrodynamics>>Initial conditions**. The solid transport initial condition is only necessary when the user turns on suspended sediment transport. In this example we only have bed transport.

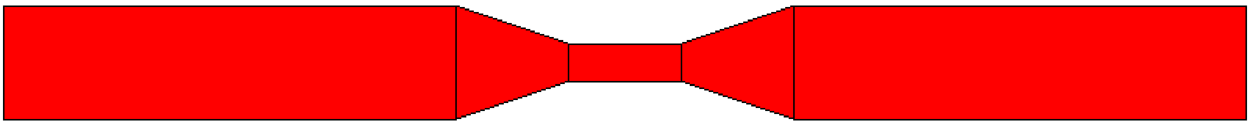


**Figure 84. Initial condition.**

For the roughness parameters we select the channel bottom not having any form. Due to this, all friction will be made by the grain ($n=n_s$). The $n_s$ parameter is calculated by the Manning-Strickler equation.

$$n_S = \frac{K_S(m)^{1/6}}{25}$$

- $Ks=2.5\ D_s$
- $D_s=5$ mm
- $n=0.0193\ s \cdot m^{-1/3}$

We now use **Data>>Roughness>>Land Use**. Iber's list of land uses does not include calculated Manning coefficient. Thus we have to create one from the list (*channel*) and assign our own coefficient. We modify River land use, then immediately assign it to all surfaces.

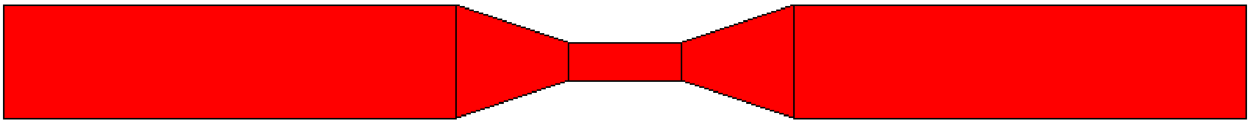


**Figure 85. Land use.**

### *7.3.3. Mesh*

For this example, we will perform a structured mesh. We will increase the resolution at those areas where the channel section changes.

We will use the element sizes shown in Figure 86.

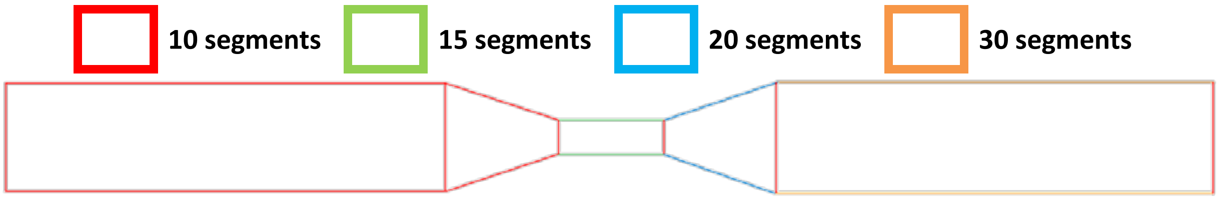



**Figure 86. Mesh criteria. Elements sizes.**

Using **Mesh>>Structured>>Surfaces>>Assign number of cells** we enter the number of cells.

When we have finished, we can generate the mesh, using **Mesh>>Generate mesh**. In Figure 87 we can see that the mesh has 850 elements. If your mesh it is not the same, you may have entered some bad parameters.

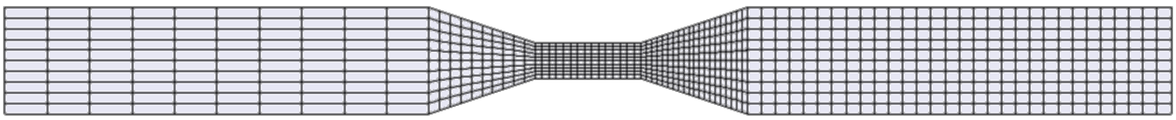


**Figure 87. Example mesh.**

### 7.3.4. Sediment transport

The next step is to define the sediment parameters, using **Data>>Data problem** and selecting **Sediments**.

In this window we need to turn off the **Suspended sediment** option and activate **Bed transport**.

We then introduce the following parameters:

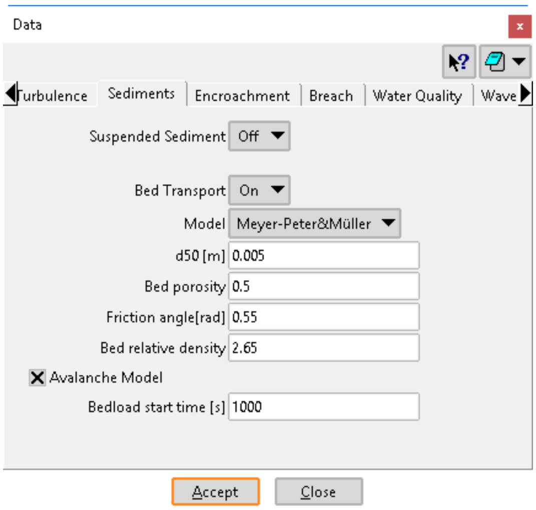



**Figure 88. Sediment transport options.**

The Avalanche model allows us to choose whether we want to know the results when the bottom slope overtakes the friction angle. In our example, the results do not have any variations because the bottom slopes are small.

The final parameter is the Bedload start-time. It allows Iber not to simulate sediment transport in the first instants. A value of 1000 s is enough to achieve a stable state.

### 7.3.5. Calculation data

Now we define the time parameters, using **Data>>Problem Data**. In the **Time Parameters** tab we establish the following:

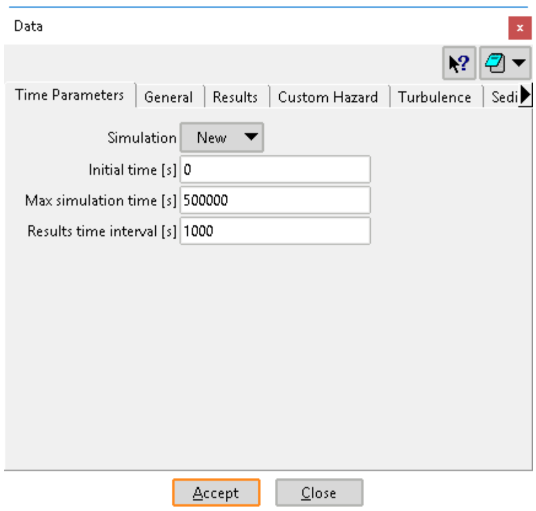



**Figure 89. Time parameters.**

In the **General** tab (in **Problem Data**), we select the numerical scheme that we want for our simulation, in this case **2nd Order** and a CFL value of 0.6.

The final parameters to establish are the results we want to know. Using the **Data>>Data Problem and Results** tab, we select **Critical Diameter** and **Bed Shear Stress**, and its corresponding maximums.

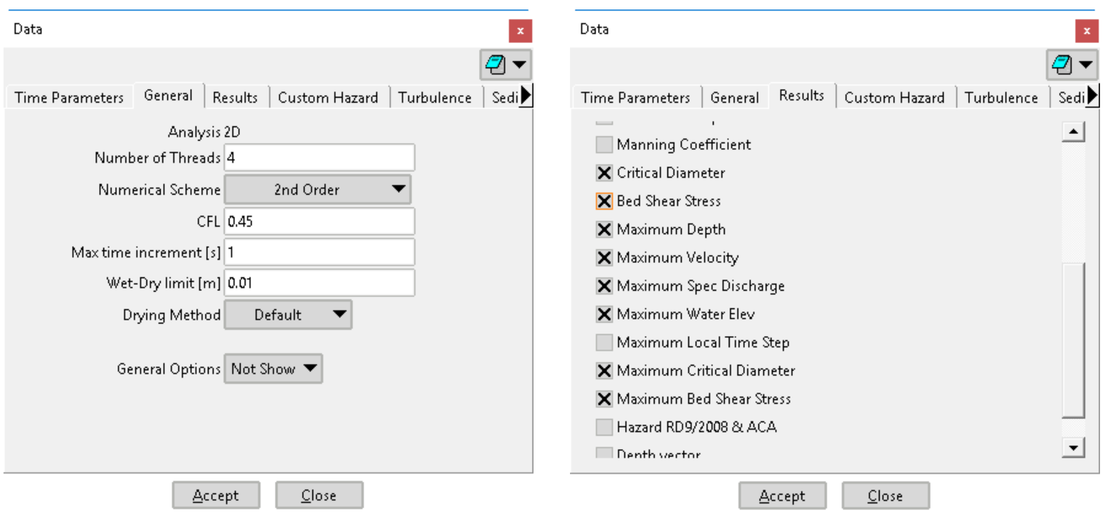



**Figure 90. How to define the numerical scheme and the results parameters.**

We can see that the simulation time is 6 days. This is very high due to fact that the erosion and sedimentation processes are very slow. To obtain results quickly (the simulation can last 3 hours) we will make a mesh with large-sized elements and we will not consider turbulent flow.

The way to get better results here is to use smaller elements at those areas where the erosion and the sedimentation are highest.

As noted above, in this example we are going to use two sediment diameters (5 and 10 mm), and we will simulate both cases.

We now launch the 5 mm size simulation. Meanwhile we save the project with a different name (Estrechamiento_10mm) and change some parameters of the case.

In **Data>>Data Problems** (**Sediment** tab) we change the d50 parameter to 0.01 m. We also have to change the Manning coefficient, because we have changed the diameter. We reassign the new Manning coefficient to all surfaces.

Then save the new project and launch the simulation with the new parameters.

Using **Calculate>>Calculate window** we can check that the two processes are being calculated by Iber.

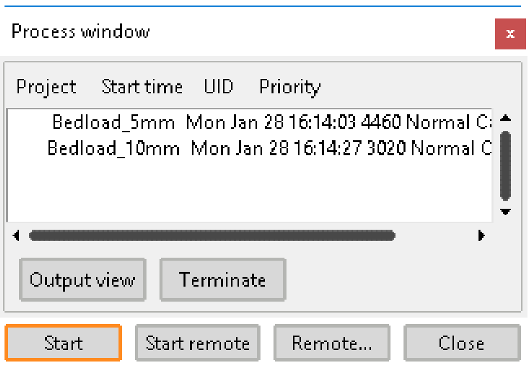



**Figure 91. Two simulations running.**

## 7.4. Results

When the simulations finish, we go to **Postprocess** to see the results, using **Window>>View results**.

If we select **Smooth Contour Fill**, there are three different results to choose Sediments: **Bed Elevation**, **Bedload transport** and **Erosion**.

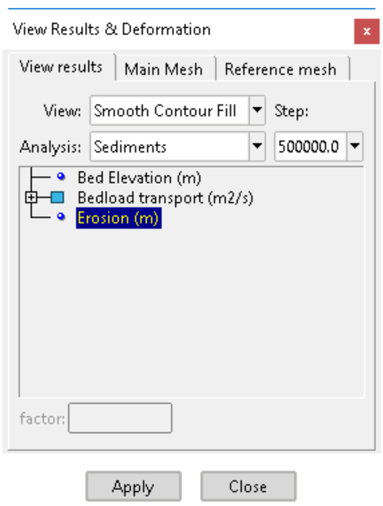



**Figure 92. Sediments Analysis.**

We will view the Erosion analysis at step 500,000 (the final one). Figure 93 corresponds to sediment of 5 mm (Note, the legend of the results has been fixed between -0.51 and +0.51 m).

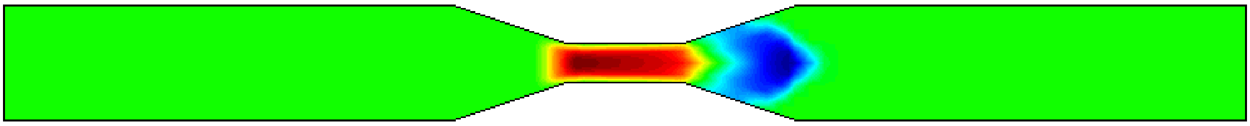


**Figure 93. Erosion with 5 mm sediment.**

We can see that the erosion in the narrowing area is caused by the increase of velocity. The material that the water flow drags is deposited on the expansion area where the velocity has decreased.

This tool allows us to make a comparison between the two cases. We can show both results and see information about them. The following images illustrate the comparison between the two cases.

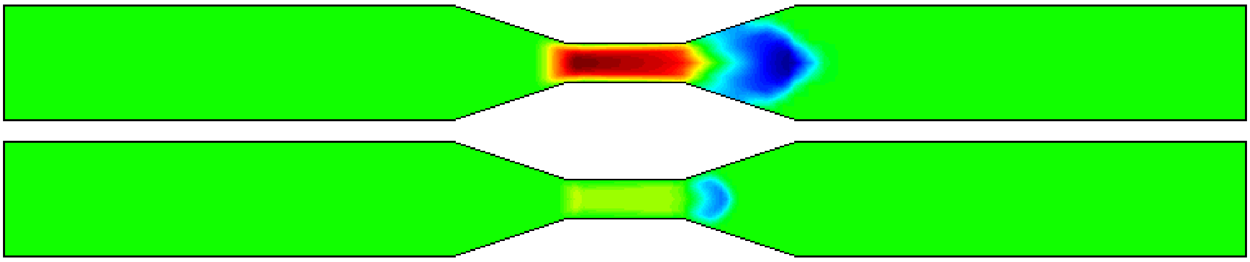


**Figure 94. Comparison between two cases. Erosion for 5 mm (above) and for 10 mm (below).**

If we compare the results, we can observe that the 5 mm sediment is transported a higher distance than that of 10 mm diameter. Likewise, the maximum erosion is in the 5 mm sediment case.

We can now perform a longitudinal profile to achieve the best results. This profile will start before narrowing and will finish after widening. It is defined by two points, A(100,15) and B(260,15).

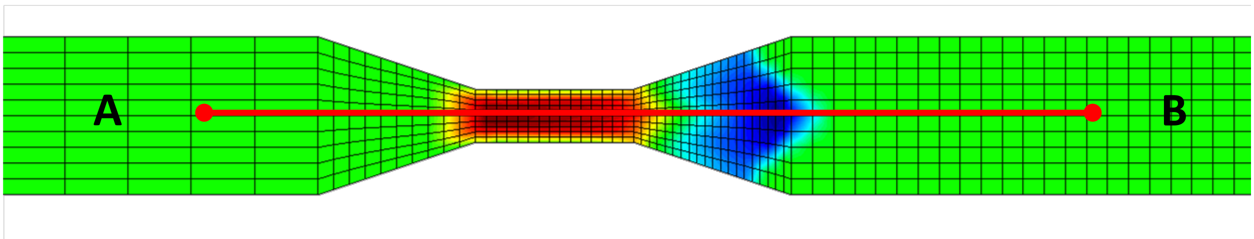



**Figure 95. Scheme of longitudinal profile.**

To make a longitudinal profile we have two options. The first is by using **Geometry>>Cut 2D Polygonal**. The second way is by selecting **Create profiles** on the Iber toolbar ( ).

Having chosen one of the options we can select the points that define the profile or introduce the coordinates of these on the Command line. The best way is by introducing the coordinates on the command line.

When we have done the polygonal cut, we need to change a display style in order to view the cut properly. The best option here is showing **boundaries** (Windows>>View style>Global settings>Style). The result is shown in Figure 96.

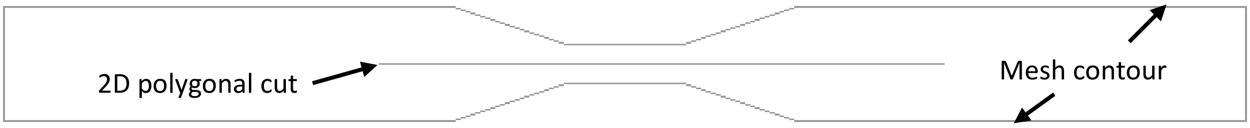



**Figure 96. Appearance that should be displayed by Iber. In the middle of the channel will appear the 2D polygonal cut.**

The next step in making the longitudinal profile involves going to the **Graphs** window and selecting **Border graph** in the **Create** tab. For the X-axis we need to select **Line variation**, and for the Y-axis **Bed elevation**. The step time must be the last (500000 s). Press **Apply** and select the cut that we have made previously.

In the resulting profile we can distinguish the two different areas, Figure 97.

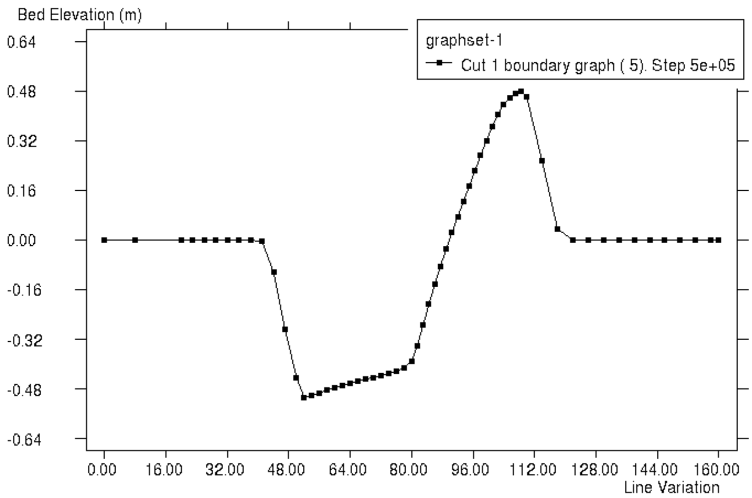



**Figure 97. Bed elevation profile for 5 mm study case.**

We will now show the same cut, but changing the sediment size to 10 mm. The first step here is to rename the graphic set and the extracted series.

It is important to specify a name for graphic set, so that different information can be distinguished when we import it from other projects. You can set the name of the graph-set by the button "Rename set of graphs" (  ). The following window will appear. In this case, the set name will be "Profile".

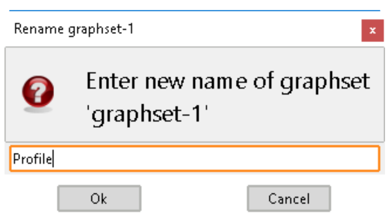



**Figure 98. Window where we have to write the new name.**

We will specify the series name that is included in the graphic set. By selecting each graph-set and it(s) series, we can modify the name, the legend limits, the line style, etc., of each series.

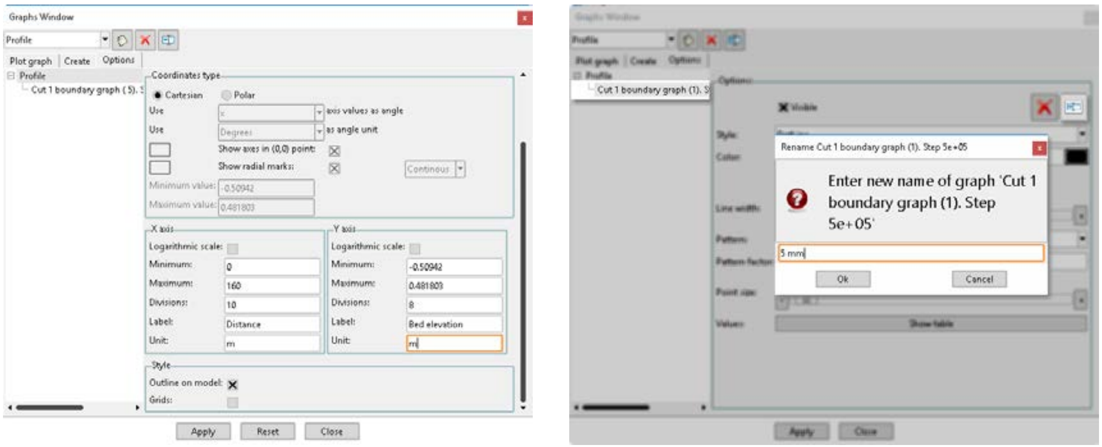


**Figure 99. Window where we can change the series name.**

The customization of the graph-set and its series could be as shown in Figure 100, in which negative values correspond to erosion and positive values to sedimentation process.

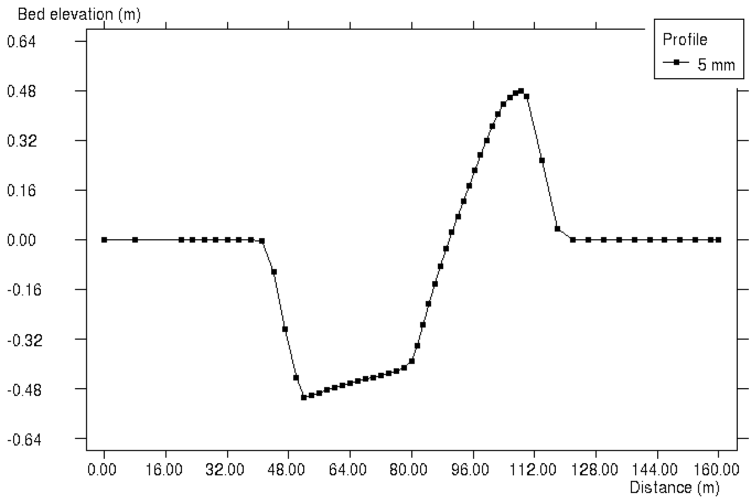



**Figure 100. Window where we can change the series name.**

The order in which the series appears is the one in which we have previously created them.

When we have finished this process, we can export the graphs in order to combine them with the 10 mm sediment sections, using **Files>>Export>>Graph>>All graphsets**. Then, in the next window, we specify the folder where we want to save the graph and we given them a name. The graph is saved as *.grf file, which can be imported by Iber or other kind of graphs processors.

Then, we have to repeat the same process to create the profile of 10 mm sediment case. In this case, as the previous one, the graph-set name must be "Profile" and the series "10 mm".

We now import the new sections from the D=10mm case, using **Files>>Import>>Graph**.

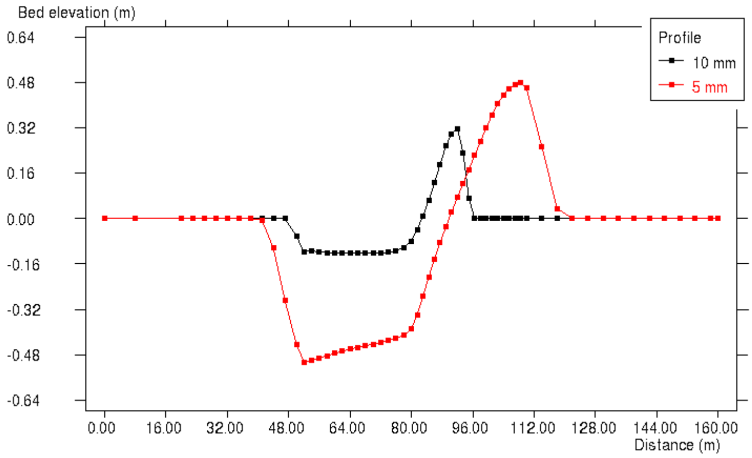



**Figure 101. Combination of the longitudinal profiles.**

In Figure 101 we can see that in the case with a sediment of 10 mm diameter-size the accumulation of sediments is produced closer to the narrowing. In the case with a sediment of 5 mm diameter-size the eroded area is higher.

Iber allows the user to analyse the influence of the erosion on the hydrodynamics. To do this we draw a longitudinal profile of the water sheet at t=1000 s (when the erosion process has not yet started) and at t=500000 s (the final step).

We are going to use the 5 mm sediment-size case. Using the "Profile" graph-set, **Create** the following graphs series:

- Bed elevation at 1,000 s
- Water elevation at 1,000 s
- Water elevation at 500,000 s

Performing some changes at Options tab (line style, line colours, line width, etc.), we can represent different situations of the channel:

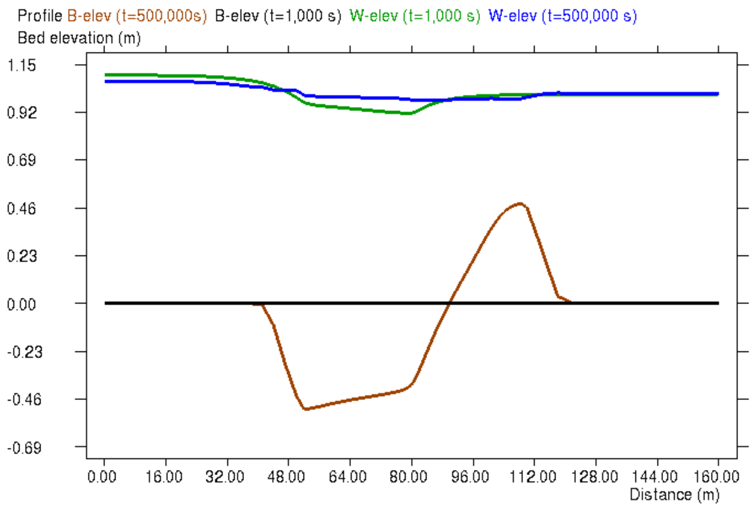



**Figure 102. Effects of the erosion/sedimentation on the hydrodynamics.**

As we can see in Figure 102, the channel contraction causes a rising of the water elevation at the beginning of the simulation. When the simulation is coming to an end, the water elevation is almost horizontal, because the erosion counters the narrowing effect.

Now we will see the temporal evolution of the Bottom Elevation on three points, doing so for the two cases. The first point will be located at the middle of the narrowing, and the other two will be after this.

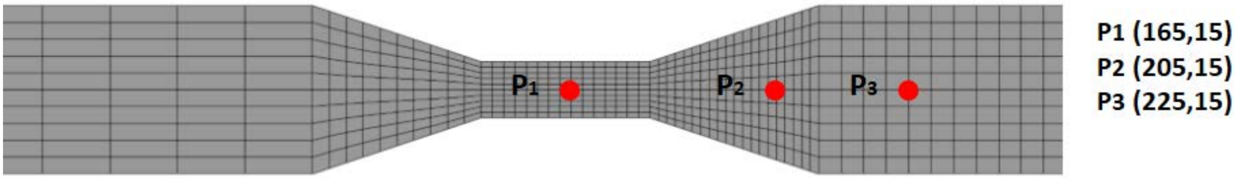



**Figure 103. Points coordinates.**

In the **Graphs Window** go to the **Create** tab and choose **Point Evolution**. On the **Sediment** option select **Bed Elevation** and enter the coordinates of the three points.

Thus we want to compare the different graphs here, we must create a new graph-set and change the name of the graph-set and series. The new name of the graph-set will be “Temporal Evolution” and the series will be called “Bottom Elevation” followed by the point (P1, P2 or P3) and the diameter (D=5mm).

At P1 the erosion is practically 50 cm, at P2 the sediment is accumulated quickly, and at P3 nothing happens.

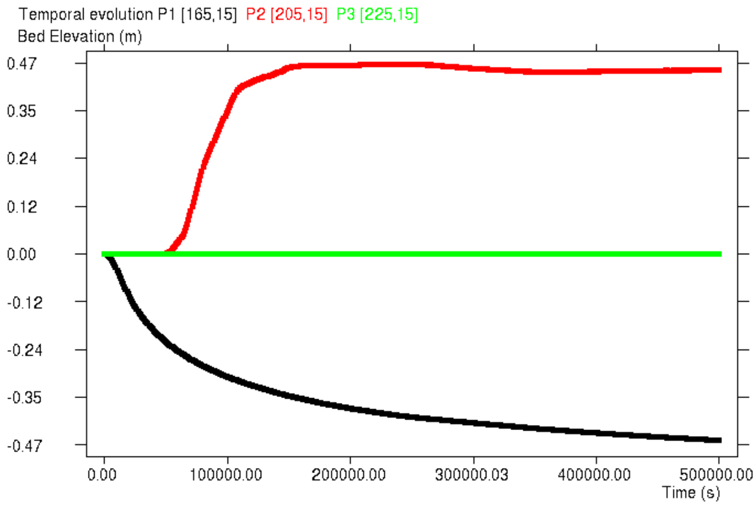



**Figure 104. Temporal evolution D=5 mm.**

Iber allows to analyse the maximum sediment size needed to avoid erosion, which is the critical diameter. By using window results, and selecting the **Map of Maximums** and **Critical diameter** we can plot the sediment-size.

Figure 105 shows how the maximum critical diameter for the 5 mm study case. There are areas where the critical diameter is less than 5 mm, meaning that the stress generated by the flow was not enough to move the sediment with a diameter of 5 mm (legend limits 0 and 0.0225 m).

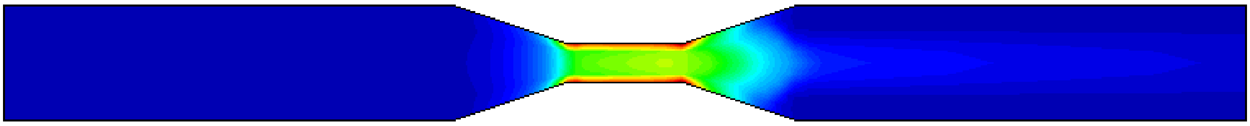


**Figure 105. Critical Diameter variation.**

Now, if we set a minimum value of 0.005 m to legend, we can see the area where the diameter must be bigger than this value to avoid erosion.

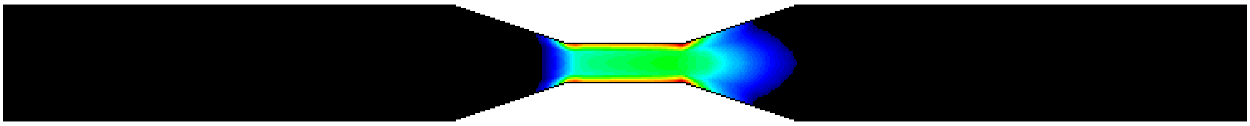


**Figure 106. Area where the critical diameter is bigger than 5 mm.**

We can check that the diameter is bigger than 5 mm, using **View>>Label>>Select on>>Result** ( ). In the central area the critical diameter varies between 7 and 13 mm.

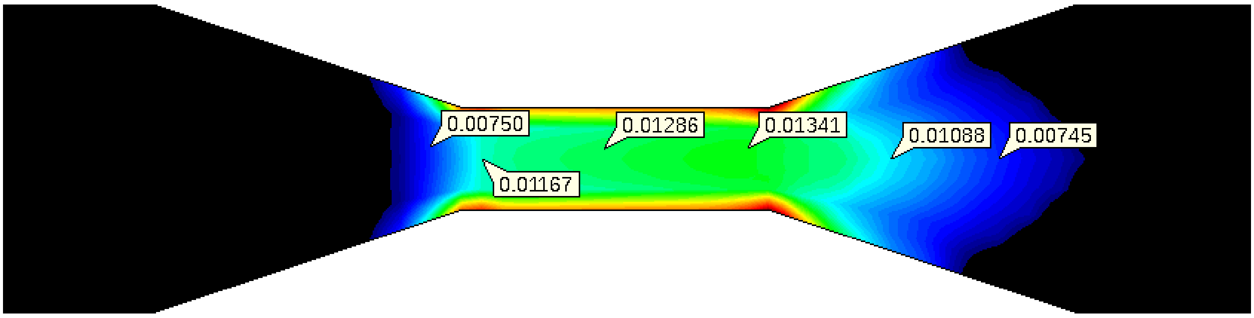



**Figure 107. Checking of critical diameter.**

## 7.5. Conclusions

In this example we have analysed erosion at channel narrowing.

We have seen that we need more time to be able to get results of erosion and sediment transport. This is because achieving a steady state is slower than other hydrodynamic processes.

In these analyses the combination of the hydrodynamic process and the solid transport means that the depth and velocity fields change during the erosion process. This is because the bottom elevation affects the hydrodynamics.

The result of the critical diameter analysis allows us to estimate what sediment size is necessary to avoid erosion on the channel bottom. To get this it is not necessary to execute a sediment transport simulation. A hydrodynamic analysis is enough.

This type of analysis is very important in areas with bridges, due to the bents and abutments that often narrow the riverbed. The current example shows one of the erosive processes in such structures. The local erosion on these supports should not be simulated with 2D models because the water flow has a strongly 3D character.

## 7.6. References

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, E., Dolz, J., Coll, A. (2014a). Iber: herramienta de simulación numérica del flujo en ríos. Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería, Volume 30, Issue 1, 2014, Pages 1-10, ISSN 0213-1315, DOI: 10.1016/j.rimni.2012.07.004

Corestein G, Bladé E. Validación del módulo de transporte de sedimendos de fondo - Modelo Iber (2013). Proceedings in: III Jornadas de Ingeniería del Agua : la protección contra los riesgos hídricos. JIA 2013; 2013. p. 27–34.

Corestein, G., Bladé, E., (2013). Validación del módulo de transporte de sedimentos de fondo - Modelo Iber. Proceedings in: V Jornadas de Ingeniería del Agua, Córdoba (España). JIA 2013

Forn M, Bladé E, Dolz J. Modelación numérica bidimensional de la dinámica sedimentaria del río Ebro en Castejón. Proceedings in: V Jornadas de Ingeniería del Agua, A Coruña (España). ISBN: 978-84-9749-670-4; 2017.

# 8. Practical Example 6: Wastewater discharge of CBOD and ammonia in a river

## 8.1. Objectives

This example introduces the options available in Iber's nitrogen cycle, dissolved oxygen cycle, and organic matter modules (Cea et al., 2016). We will look at the evolution of discharge from a wastewater treatment plant into a river reach. This plant releases a continuous discharge of organic matter and nitrogen in the form of ammonia and nitrate. We will analyse the levels of DO (dissolved oxygen), NH3-H (ammoniacal nitrogen), $NO_3$-N (nitrate-nitrite nitrogen) and CBOD (carbonaceous biochemical oxygen demand) in the river once a steady state is achieved. We will then compare the concentrations predicted in Iber with the observed concentrations presented in Thomann & Mueller (1987).

## 8.2. Description of the case study and input data

The example consists of a river with an annual discharge of 100 $ft^3/s$ (2.83 $m^3/s$) in which a wastewater treatment plant discharges effluent containing organic matter, ammonia and nitrates. We want to analyse the evolution of dissolved oxygen, organic matter and nitrogen over 50 km of the river under steady state conditions.

The example is presented in Thomann & Mueller (1987) (Figure 108).

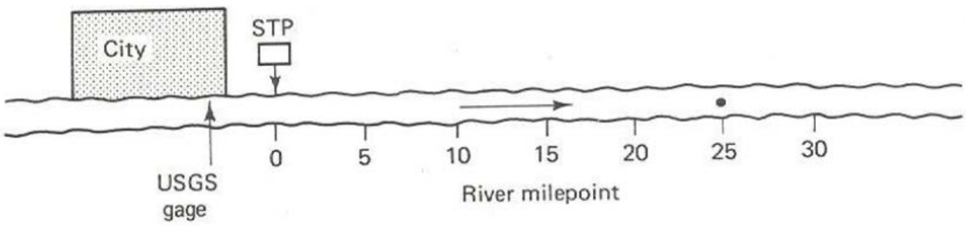



**Figure 108. Schematic diagram of the case study from Thomann & Mueller (1987).**

Since the length of the river reach is much larger than its width, the geometry will be conceptualized as a straight channel with a rectangular cross section, with uniform slope and dimensions along the reach, as proposed in Thomann & Mueller (1987). Figure 109 schematically represents the geometry of this example. Note that in this figure the scales on the longitudinal and transverse directions are different.

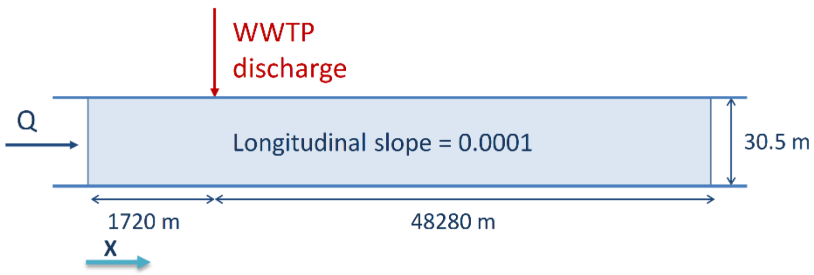



**Figure 109. Schematic diagram of the case study from Thomann & Mueller (1987).**

The complete scheme for this exercise, with the characteristics of the effluent from the wastewater treatment plant and the river, is shown in Figure 110. Please note that the variable units in the scheme are different to those used in the Iber model.

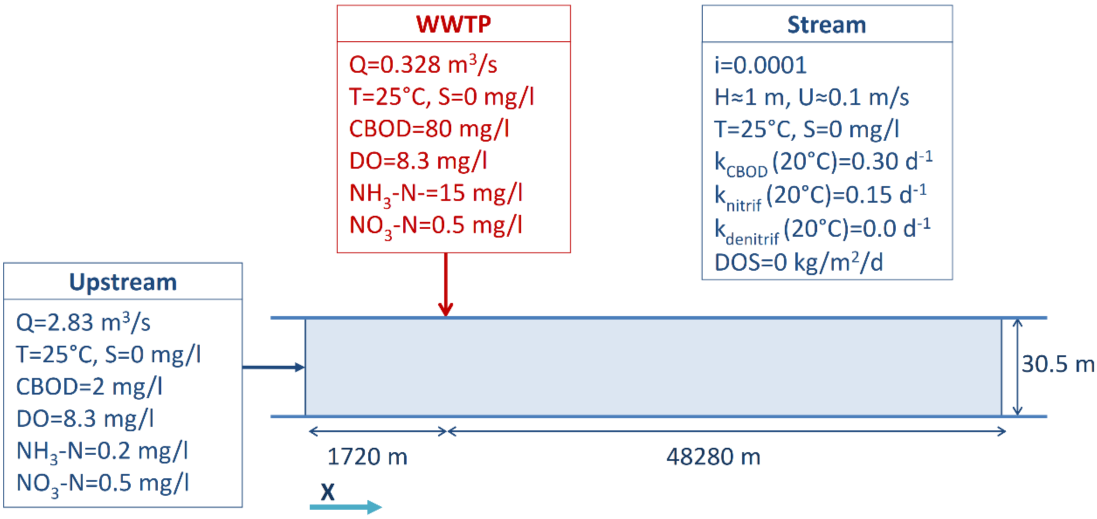



**Figure 110. Schematic diagram of the case study.**

We will ignore the denitrification process ($k_{denitrif}$=0 $d^{-1}$), since it only occurs in anoxic conditions, and dissolved oxygen concentrations are relatively high throughout the reach (we will later verify in the results that this is the case). Values for the CBOD degradation rate ($k_{CBOD}$) and the nitrification rate ($k_{nitrif}$) are taken directly from Thomann & Mueller (1987).

The salinity and temperature equations will not be calculated. Instead, we will directly assign the values of these variables (S and T in Figure 110), assuming they are uniform along the reach and constant over time.

The observed in-stream concentrations of DO, CBOD, $NH_3$-N and $NO_3$-N downstream from the discharge as presented in Thomann & Mueller (1987) provide a benchmark comparison for the numerical results obtained with Iber. These point measurements are shown in **Table 4** and are included in the file field_data.grf.

**Table 4. Observed in-stream concentrations along the reach (distance X is measured as indicated in Figure 110**).

| X (m) | CBOD (mg/l) | NH3-N (mg/l) | N-NO3-N (mg/l) | DO (mg/l) |
|---|---|---|---|---|
| 1300 | 2.4 | 0.3 | 0.5 | 7.6 |
| 4500 | 9.8 | 1.6 | 0.6 | 6.4 |
| 8300 | 7.0 | 1.7 | 0.6 | 6.5 |
| 12800 | 5.8 | 1.5 | 1.1 | 5.4 |
| 19000 | 6.0 | 1.0 | 0.9 | 6.1 |
| 26900 | 4.4 | 0.9 | 1.1 | 6.3 |
| 35100 | 3.4 | 1.1 | 1.7 | 6.3 |
| 47700 | 0.6 | 0.8 | 1.7 | 7.2 |

## 8.3. Model set-up

### 8.3.1. Geometry

After launching the model and creating a new project (**Files>>Save**) we are ready to set up the model. The basic steps are described in this section. First, we create the geometry of the model. This consists of a single rectangular surface with a slope of 0.0001 in the longitudinal direction.

We manually introduce the coordinates of the start and end point of each line (**Geometry>>Create>>Straight line**). The elevation of the inlet and outlet boundaries are Z=5 m and Z=0 m, respectively. Once the four sides of the rectangle have been defined according to the scheme shown in Figure 111, we create a rectangular NURBS surface (**Geometry>>Create>>NURBS surface**).

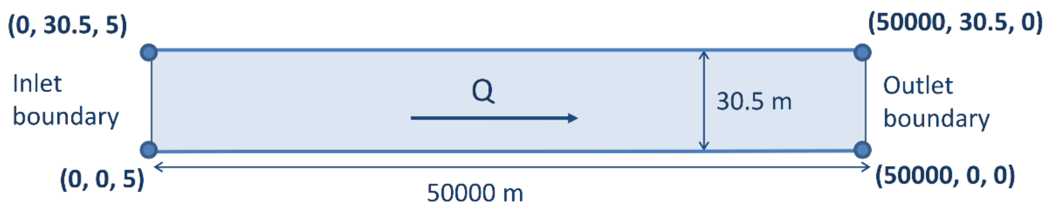



**Figure 111. Definition of the rectangular surface in the model.**

### 8.3.2. Hydrodynamics

The next step is to assign the hydrodynamic boundary and initial conditions (**Data>>Hydrodynamics>>Boundary Conditions**). At the upstream boundary, we assign a discharge of Q=2.83 $m^3$/s. At the downstream boundary we set a water surface elevation 1 m. As the initial condition we impose a water depth to 1 m throughout the domain (**Data>>Hydrodynamics>>Initial Conditions**).

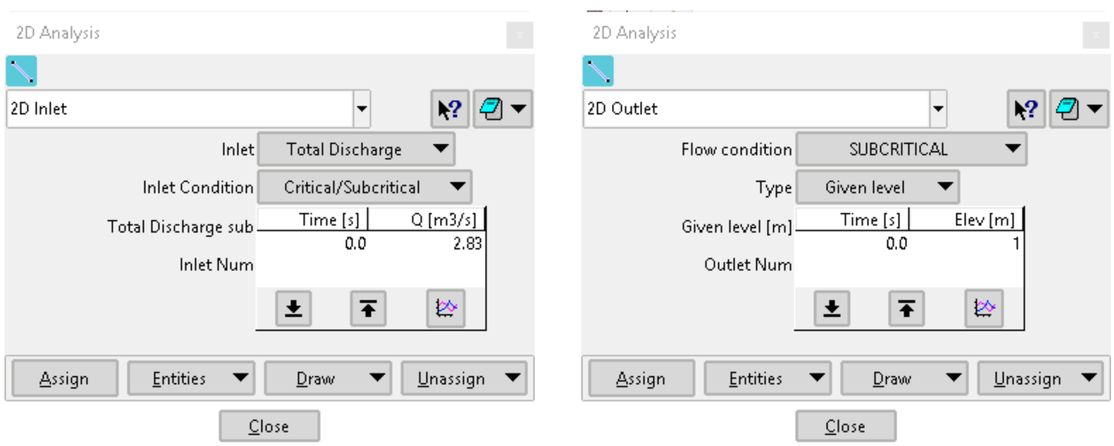



**Figure 112. Boundary conditions tab to introduce the upstream discharge and the downstream water level.**

As indicated in Figure 110, the water depth and velocity in the reach are approximately 1 m and 0.1 m/s. In order to satisfy these values of velocity and depth, we have to define a suitable Manning roughness coefficient (n), according to:

$$U = \frac{1}{n} \cdot R_h^{3/2} \cdot i^{1/2} = \frac{1}{n} \cdot h^{3/2} \cdot i^{1/2} \qquad [1]$$

where $U$ is the water velocity, $R_h$ is the hydraulic radius, $h$ is the depth and $i$ is the slope. For U=0.1 m/s, h=1 m, given the slope of i=0.0001, we have to set a Manning coefficient of 0.10 s·m$^{-1/3}$ throughout the domain (**Data>>Roughness>>Land Use**). These values correspond to the lowest part of the river reach, downstream of the WWTP discharge. At the point where the discharge occurs there will be a local variation of depth and velocity.

### *8.3.3. Water quality*

We now introduce the water quality data. We first define the boundary conditions of the model (**Data>>Water Quality>>Boundary Conditions**). We assign separately the conditions referring to the dissolved oxygen, the CBOD and the nitrogen cycle, only at the upstream boundary (vertical left line of the channel). In accordance with Figure 110, we introduce the following values in the corresponding tabs (Figure 113): CBOD=0.002 kg/m$^3$, DO=0.0083 kg/m$^3$, $NH_3$-N=0.0002 kg/m$^3$ and $NO_3$-N=0.0005 kg/m$^3$.

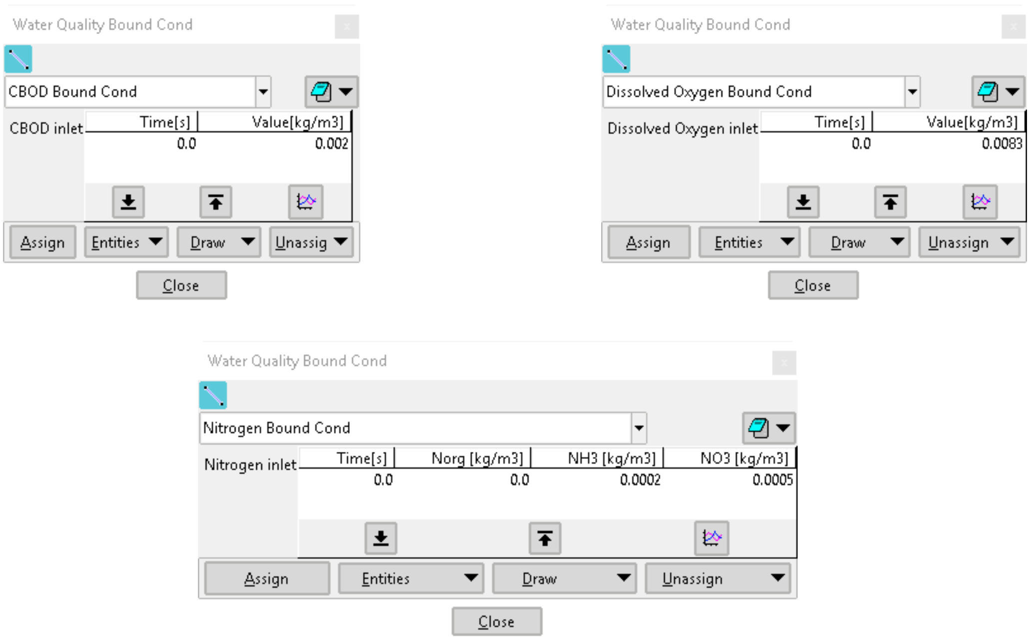



**Figure 113. Water quality boundary conditions tab to introduce the upstream DO, CBOD, $NH_3$-N and $NO_3$-N concentrations.**

We then specify the initial water quality conditions in the whole reach (**Data>>Water Quality>>Initial Conditions**). As with the previous step, we introduce separately the conditions for the dissolved oxygen, the CBOD and the nitrogen cycles (Figure 114). We use the same values as for the upstream boundary condition. However, this choice is not relevant for the current example, since we are only interested in the final results, when the steady state is reached, these being independent of the initial conditions.

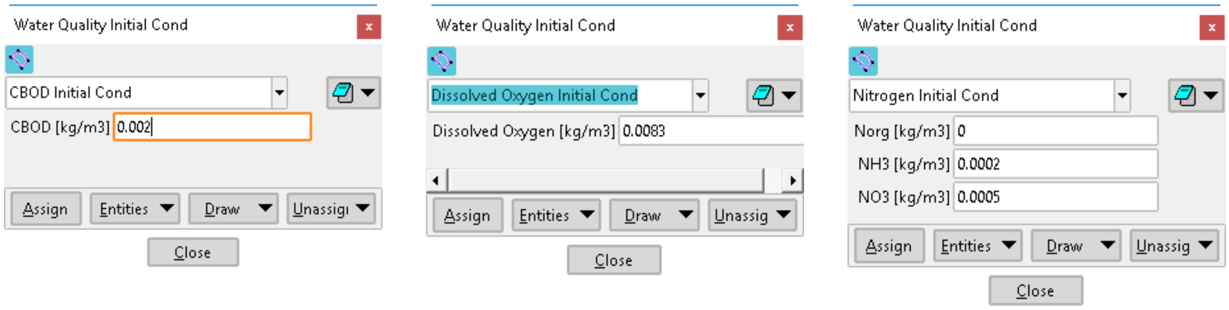



**Figure 114. Water quality initial conditions tab to introduce the DO, CBOD, $NH_3$-N and $NO_3$-N initial concentrations.**

We are not interested in the sediment oxygen demand, and therefore we set a value of 0 kg/m$^2$/d throughout the domain (**Data>>Water Quality>>Sediment Oxygen Demand**) (Figure 115).

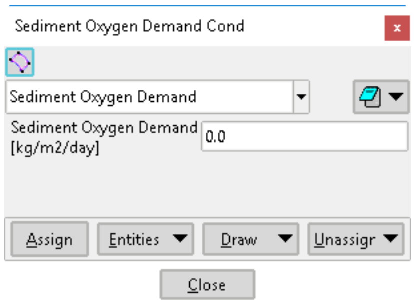



**Figure 115. Sediment oxygen demand tab.**

Finally, we introduce into the model the discharge of the wastewater treatment plant. To do this, we create a new discharge (**Data>>Water Quality>>Discharges**) and manually specify its location. According to Figure 110, the discharge is located at X=1720 m. In the transverse direction, we introduce the discharge in the middle of the river (Y=15.25 m). This is in fact not relevant here, since we will create a mesh with a single element in the transverse direction, therefore assuming complete mixing in this direction. Given that Iber is a depth-averaged model, the location of the discharge on the Z axis is also not relevant here. We arbitrarily set it at Z=0 m.

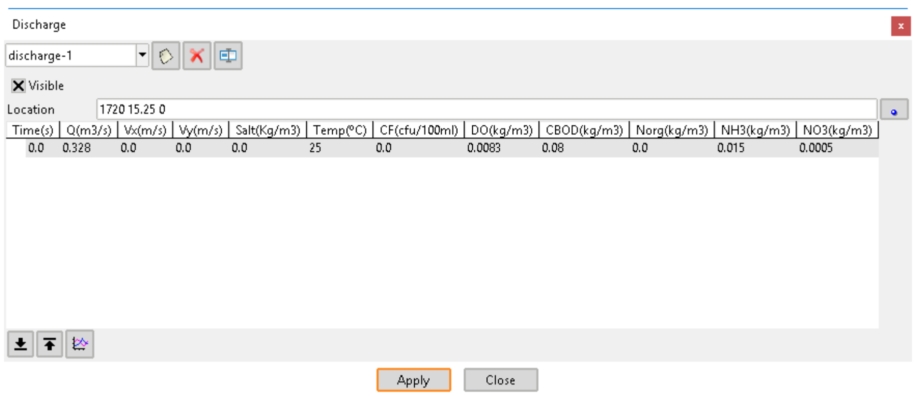



**Figure 116. Discharge tab to introduce the location and water quality characteristics of the effluent of the wastewater treatment plant.**

We must also introduce the flow rate and the water quality characteristics of the effluent of the wastewater treatment plant (Figure 116). According to Figure 110, we introduce the following values: Q=0.328 $m^3/s$, CBOD=0.08 $kg/m^3$, DO=0.0083 $kg/m^3$, $NH_3$-N=0.015 $kg/m^3$ and $NO_3$-N=0.0005 $kg/m^3$.

Once the discharge data has been introduced, its location will be displayed in the model geometry (with points on the X and Y coordinates indicated above).

### *8.3.4. Mesh*

The next step is to create a computational mesh to spatially discretize the domain. We use a structured mesh made of rectangular elements. Since the length of the river reach is three orders of magnitude larger than the river width, the mesh has a single element in the transverse direction. In the longitudinal direction, the spatial domain covers a total distance of 50 km, and we select a mesh size of 100 m. We therefore assign the following number of cells in the surface: 1 cell in the transverse direction and 500 cells in the longitudinal direction (**Mesh>>Structured>>Surfaces>>Assign number of cells**). When we generate the mesh (**Mesh>>Generate**) we obtain 500 elements of 30.5 m x 100 m.

### *8.3.5. Calculation data*

Before running the model we need to set the computation parameters (**Data>>Problem Data**). First, we select the time parameters (Figure 117). We choose a maximum simulation time of 7 days (604,800 s) to allow the model to reach a steady state. Bear in mind that the total length of the reach is 50 km and that velocities are in the order of 0.1 m/s, so that the time it takes the water to traverse the reach is around 500,000 s. This value is an approximation, since the concentration of organic matter, dissolved oxygen and nitrogen depends not only on its advection by the mean flow along the channel, but also on the interaction processes between these components. Therefore, we need to check in the post-processing that a steady state has really been achieved. We will use time evolution graphs of the different variables for this. Given that we are not interested in the transient results, we choose a wide time interval of 10,800 s for storing the results (i.e., one result every 3 hours).

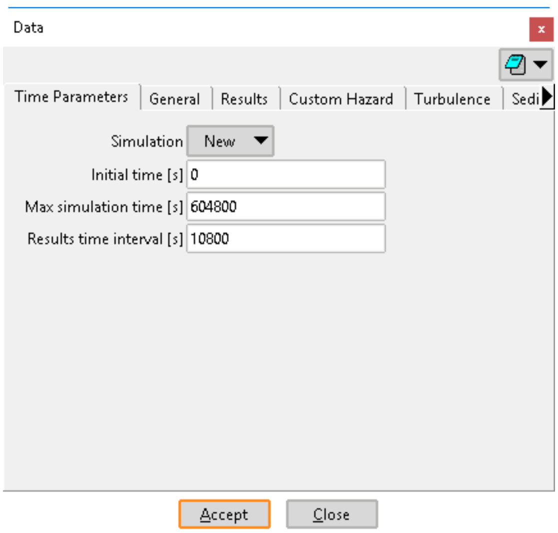



**Figure 117. Time parameters tab to assign the initial time, maximum simulation time and results time interval.**

We then establish the **parameters and kinetics** related to **water quality**. First, we set the temperature and salinity values at 25 °C and 0 kg/m$^3$, respectively (Figure 118). Second, we activate the dissolved oxygen, CBOD and nitrogen modules and define the reaction kinetic constants (Figure 119 and Figure 120). As indicated in Figure 110, the first order kinetic coefficients for organic matter degradation, nitrification and denitrification are $k_{CBOD}$=0.30 d$^{-1}$, $k_{nitrif}$=0.15 d$^{-1}$ and $k_{denitrif}$=0 d$^{-1}$.

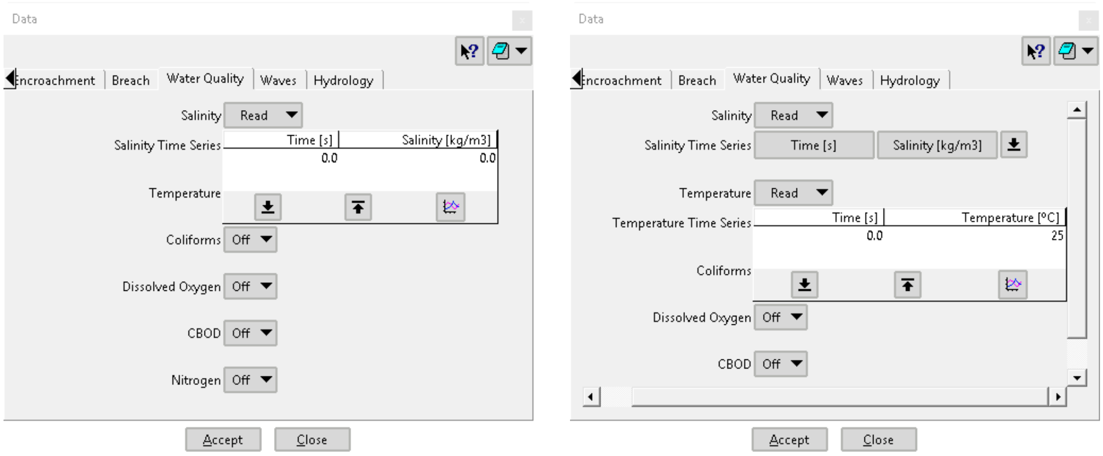



**Figure 118. Water quality tab to assign the salinityand temperature values.**

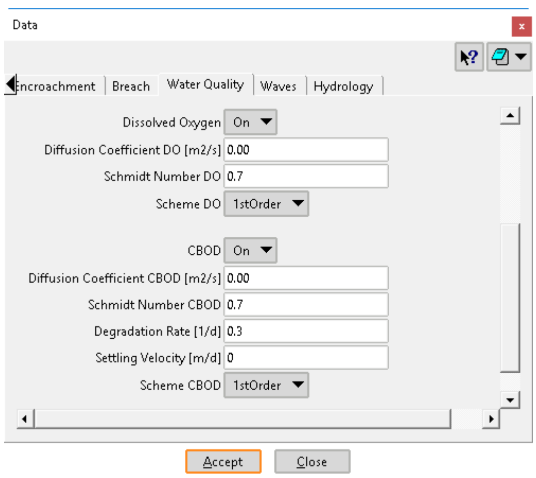



**Figure 119. Water quality tab to define the DO and CBOD kinetics.**

The organic nitrogen hydrolysis rate does not influence the results, since the concentration of organic nitrogen is zero throughout the domain. The same is true for the settling velocity of organic nitrogen. We can thus leave the default values for the ammonification rate and the organic nitrogen setting velocity. The settling for the organic matter also has little influence on the CBOD concentrations compared to the degradation process. We can thus ignore this process and enter a setting velocity of 0 m/d (Figure 119).

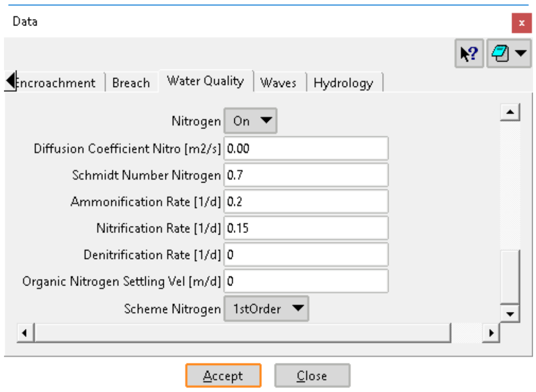



**Figure 120. Water quality tab to define the nitrogen kinetics.**

We are now ready to run the model (**Calculate>>Calculate**). Although the mesh has only 500 elements, we will simulate 7 days, so the calculation may take a few minutes. Once the calculation is finished, we can go to the post-processing tab to analyse the results.

## 8.4. Results

### *8.4.1. Initial checks*

The results are first checked to ensure that a steady state is achieved at the end of the simulation. We plot the time evolution of all variables (DO, CBOD, $NH_3$-N and $NO_3$-N) at a point close to the downstream boundary (e.g., X=49050 m) (Figure 121). In order to do this, we open the **View graphs** window and click on the **Create** tab. Select **View: Point Evolution, Analysis: Water Quality**. Then choose the variable to plot on the Y-axis, starting for example with the dissolved oxygen concentration (DO). Press **Apply** and introduce the coordinates of the point in the command line (e.g., 49050,15). Once we have repeated the procedure for the other variables (CBOD, $NH_3$-N and $NO_3$-N), we obtain a figure like the one shown in Figure 121. We can see how a steady state is achieved for all variables around t=500,000 s (dotted line in Figure 121).

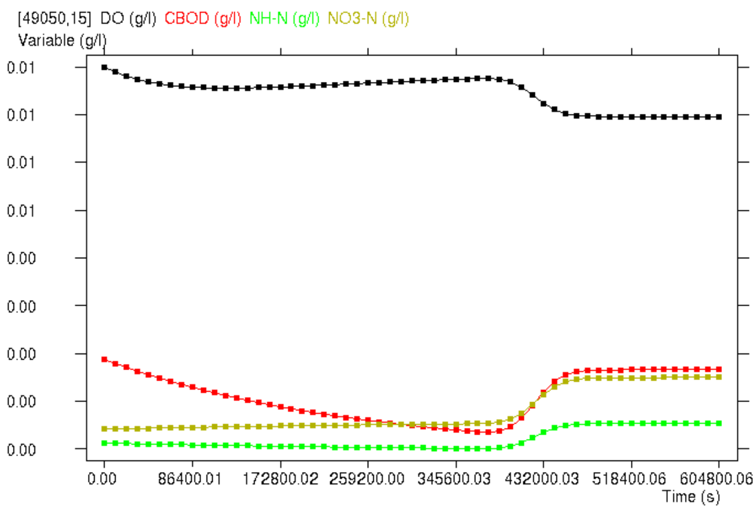



**Figure 121. Temporal evolution of DO, CBOD, $NH_3$-N and $NO_3$-N concentrations at X=49050 m.**

### 8.4.2. *Hydrodynamic results*

Before analysing the water quality results, it is a good idea to check the hydrodynamic results. Water velocity and depth in the river should be in the order of 0.1 m/s and 1 m, respectively. We create a longitudinal profile using the **create profiles** icon ( ) on the Iber toolbar (located at the left part of the interface), and manually enter the coordinates of the start and end points of the profile, as shown in Figure 122.

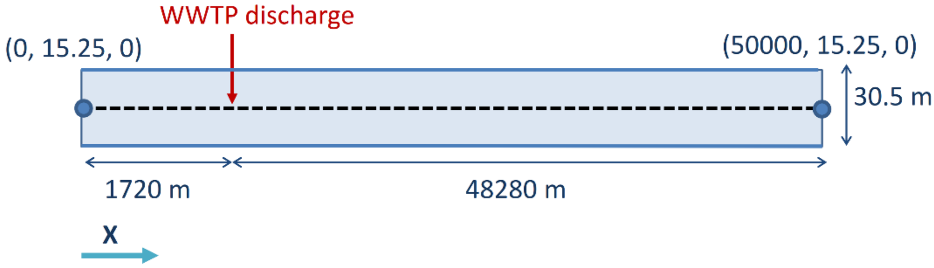



**Figure 122. Definition of the longitudinal profile of the river.**

Once the profile is created, we open the **Graph** window (**Window>>View graphs>Create**). With the **Border graph** tool, we plot the water depth and velocity along the profile at the last time step of the simulation. The depth and velocity values should be in the range of 0.92 - 1 m and 0.090 - 0.096 m/s, respectively, downstream of the discharge (Figure 123).

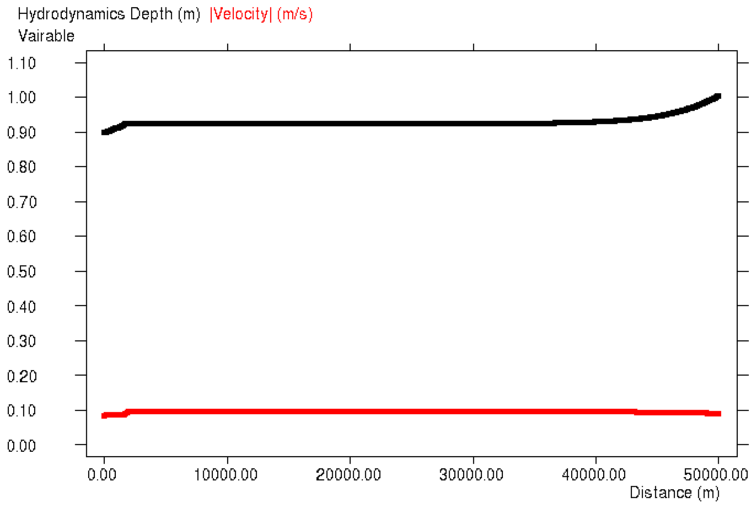



**Figure 123. Depth and velocity profiles along the longitudinal axis of the river at the end of the simulation.**

### 8.4.3. *Water quality results*

We now begin the analysis of the water quality results. We plot the longitudinal profiles of DO, CBOD, $NH_3$-N and $NO_3$-N concentration, using the profile that we created in the previous step (Figure 122). We select the last time step of the simulation to plot the results under steady state conditions. The results should look like the image below (Figure 124).

The WWTP discharge has higher CBOD and $NH_3$-N concentrations than the river in its natural state (the upstream boundary conditions). Therefore, the WWTP discharge produces a sudden change in $NH_3$-N and CBOD concentrations. The same does not apply for DO and $NO_3$-N, given that the effluent from the WWTP and the river have the same concentrations.

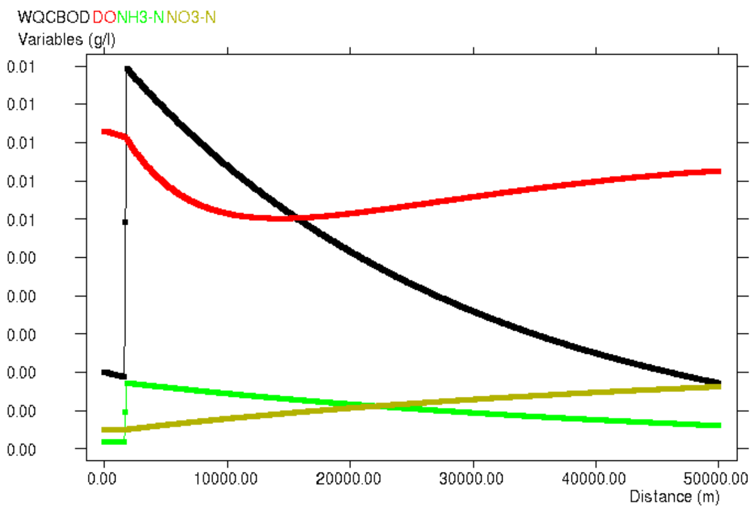



**Figure 124. Concentration profiles of DO, CBOD, $NH_3$-N and $NO_3$-N along the longitudinal axis of the river at the end of the simulation.**

The discharge of CBOD and ammonia induces a decay in the concentration of DO along the first 10 km downstream of the discharge due to the oxygen demand of the processes of biodegradation and nitrification. DO concentration reaches a minimum value of around 6 mg/l. Further downstream the biodegradation and nitrification rates diminish due to the lower concentration of CBOD and ammonia, and the concentration of DO increases progressively by surface reaeration. However, the DO at the downstream end of the reach has not yet recovered to the original DO levels.

As explained in the input data section, we have ignored the denitrification process due to the relatively high DO concentrations throughout the reach. We have now verified that this was indeed the case by means of the DO profile shown in Figure 124.

We will now compare the concentrations predicted by the model with the observed concentrations presented in Thomann & Mueller (1987) (shown in **Table 4**). In order to do so, we must change the name of the graph-set to “WQ” and then import the file field_data.grf (**Files>>Import>>Graph**). Note that the values shown in Table 4 have been converted to the International System of Units.

We compare the simulated and measured concentrations using graphs. We plot the simulated concentrations as continuous lines and the measured concentrations as data points (Figure 125).

It can be seen how the numerical model correctly reproduces the observed spatial evolution of the four variables considered, despite the simplifications made in the geometry of the river. There are of course some errors and dispersion in the field measurements, but the trend over the analysed section is well captured by the numerical model.

The differences between the model data and the field data are to some extent due to the simplification of a 50 km river reach to a rectilinear channel of a rectangular section with a constant width and slope.

In this exercise we have taken directly the values of the reaction constants proposed in Thomann & Mueller (1987), which have been previously calibrated. In a real case we would need to calibrate these to achieve a good level of agreement between the numerical results and the measurements.

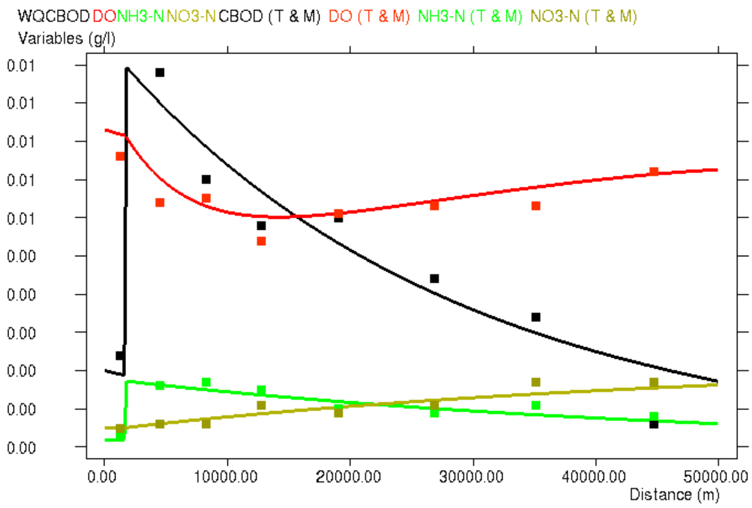



**Figure 125. Longitudinal concentration profiles of DO, CBOD, $NH_3$-N and $NO_3$-N obtained with Iber and comparison with measured concentrations.**

## 8.5. Conclusions

In this exercise we have analysed the evolution of dissolved oxygen, organic matter and nitrogen in a 50 km river reach. Due to the length of the reach, we have simplified the geometry to a rectilinear channel of a rectangular cross section, and we have assumed that there is complete mixing in the transverse direction of the river. The calculation mesh in Iber thus has a single element in this direction. The reaction kinetic constants have already been calibrated for this test case in Thomann & Mueller (1987).

Despite the simplifications made, the model correctly reproduces the observed evolution of the water quality variables along the reach.

## 8.6. References

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, E., Dolz, J., Coll, A. (2014a). Iber: herramienta de simulación numérica del flujo en ríos. Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería, Volume 30, Issue 1, 2014, Pages 1-10, ISSN 0213-1315, DOI: 10.1016/j.rimni.2012.07.004

Cea, L., and Bladé, E. (2015). A simple and efficient unstructured finite volume scheme for solving the shallow water equations in overland flow applications. Water Resources Research, 51, 5464-5486. DOI: 10.1002/2014WR01654

Thomann, R. V. & Mueller, J. A. 1987. Principles of Surface Water Quality Modeling and Control. Harper & Row, New York, USA.

# 9. Practical Example 7: Sewage spill in an estuary

## 9.1. Objectives

In this example, Iber is used to model faecal coliform contamination. We will look at the impact of wastewater discharges on an estuary (the receiving water body), analysing the evolution of bacterial concentration over two days. Based on the results, we can analyse compliance with regulations regarding concentrations of faecal indicator bacteria.

We will define the decay rate of bacteria by introducing a time series of T90, this being the time required to reach a 90% reduction in concentration. Alternatively, we can also use the empirical formulations proposed by Mancini (1978) or Canteras et al. (1995) to compute the bacterial decay rate, based on temperature, salinity, turbidity and solar radiation levels.

## 9.2. Description of the case study and input data

The case study site is the estuary of Ferrol, in northwest Spain. The total surface of this estuary is approximately 25 $Km^2$, with a length of 16 km and a width of around 350 m (Figure 126). The estuary is connected to the sea by a narrow channel, 400 m wide. The volume of water in the estuary varies between 0.21 $km^3$ at low tide and 0.29 $km^3$ at high tide. The discharge of local rivers is very low and is not significant compared to the tidal flow, the water in the estuary being mainly of marine origin.

The port of the city of Ferrol is located on the northern shore of the estuary. In the inner part of the estuary, a bridge connects the northern and southern shores (marked with an orange circle in Figure 126). In this example we will ignore the effect of the bridge piers on the flow, due to their small dimensions.

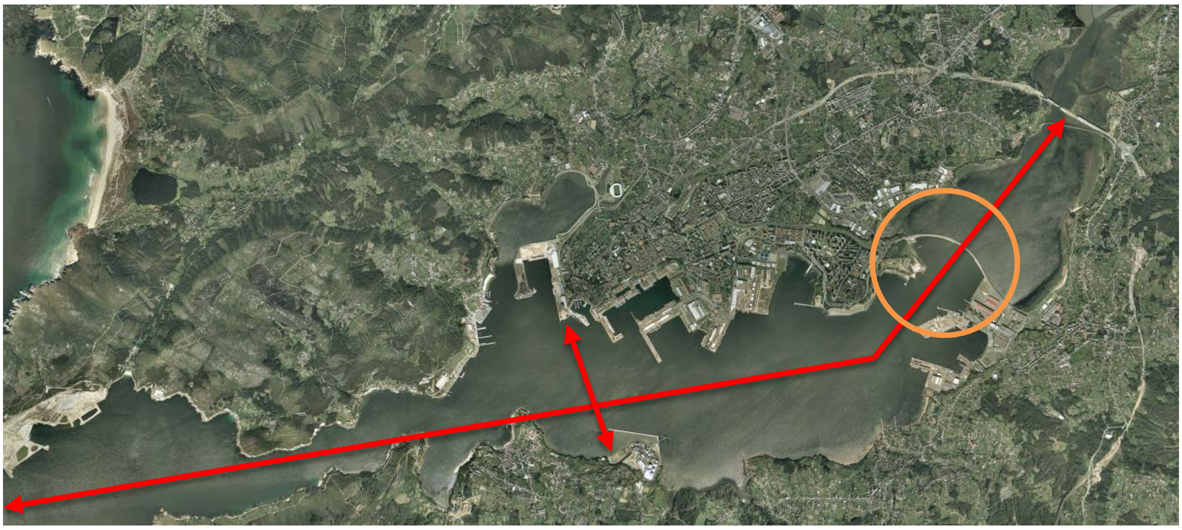


**Figure 126. Aerial image and dimensions of the estuary. The location of the bridge that connects the two shores is circled in orange.**

We will consider two sewage spills with a constant flow discharge of 0.1 $m^3/s$ and a coliform concentration of $10^7$ cfu/100 ml. These inflows do not correspond to real flows from sewage treatment works in this estuary, and are used only as simplified examples. These two inflows will be the only sources of coliform contamination.

The tidal level in the open sea varies from 0.42 m to 5.06 m, while the maximum water depth at the mouth of the estuary is approximately 25 m (Figure 127). The river that flows into the estuary has an average discharge of about 3 $m^3/s$, which is negligible with respect to the flow generated by the tide, thus we will not take it into account in the calculation.

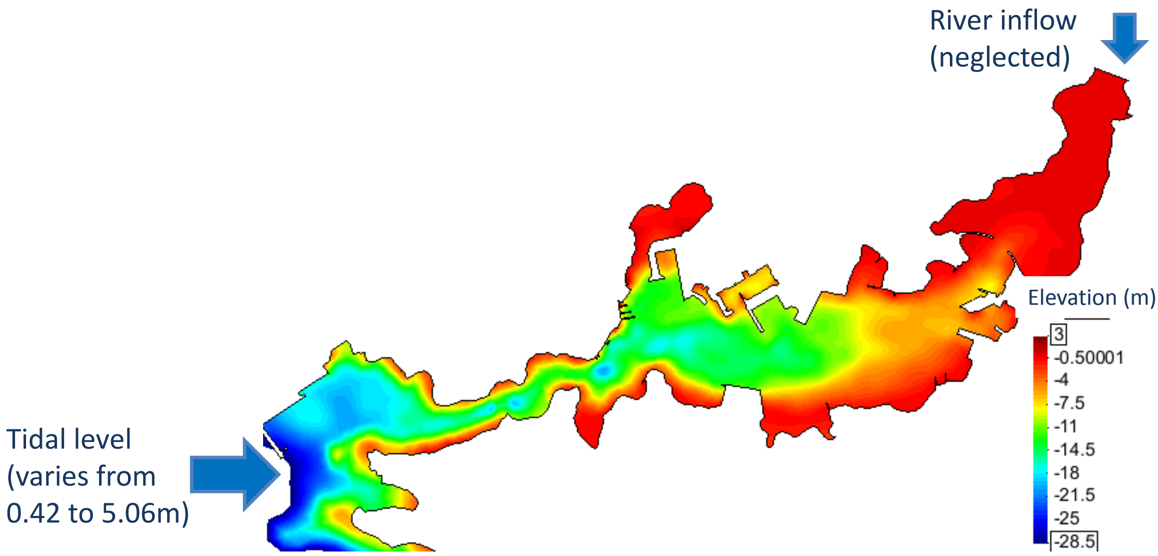



**Figure 127. Bathymetry of the estuary used in the numerical model, relative to mean sea level in Alicante.**

We will use the following input data files:

- Georeferenced orthophoto (Ferrol.jpg and Ferrol.jgw)
- Boundary of the estuary in CAD format (Boundary_Ferrol.dxf)
- Bathymetry of the estuary in raster format (DTM_Ferrol.asc)
- Time series of tidal levels at the mouth of the estuary (Tide_Ferrol.txt)

## 9.3. Model set-up

### *9.3.1. Geometry*

After launching the model and creating a new project (**Files>>Save**), we are ready to define the geometry of the model. We start by loading the orthophoto Ferrol.jpg using the menu **View>>Background Image>>Real size**.

The boundary of the estuary is provided in CAD format (Boundary_Ferrol.dxf). We import this file (**Files>>Import>>DXF**). Whenever we import a geometry file, it is advisable to collapse the geometry (**Geometry>>Edit>>Collapse>>Model**), but in this case we enable the option “Collapse allow more tasks” available when enable the automatic collapse options.

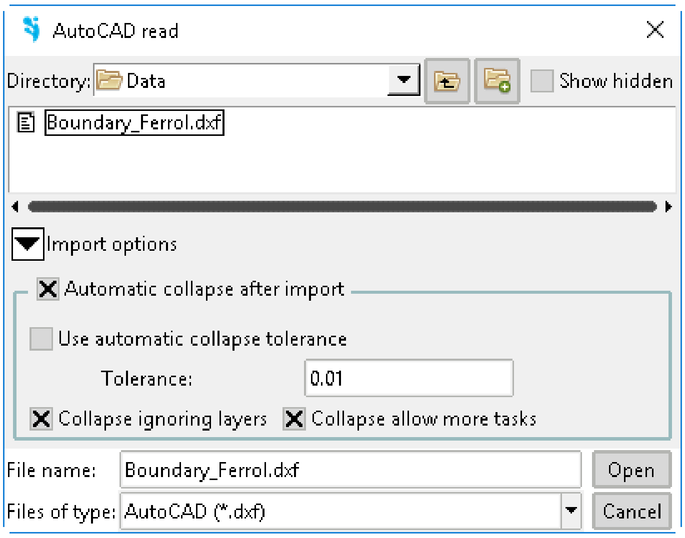



**Figure 128. DXF file import options.**

The next step is to create the three lines shown in Figure 129, and then to generate the four NURBS surfaces that define our model domain. The easiest way of creating the NURBS surfaces is to use the search tool (**Geometry>>Create>>NURBS surface>>Search**). After completing this step, the geometry should look like Figure 130.

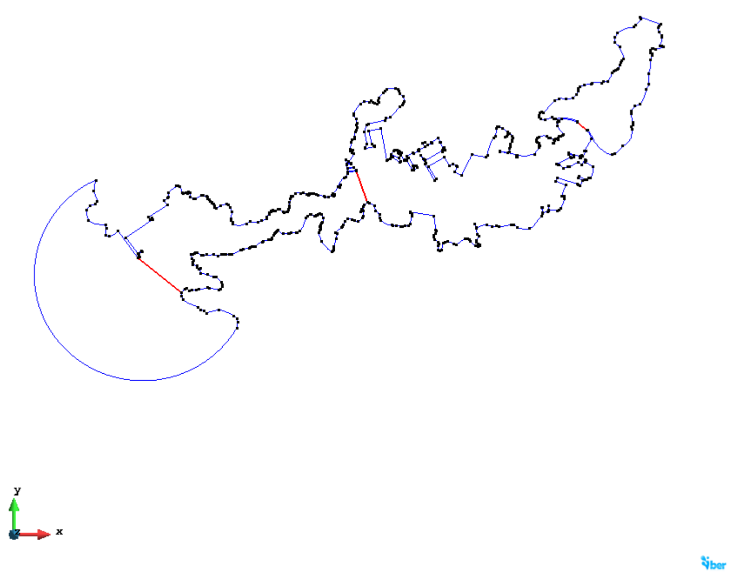



**Figure 129. Lines to be created in Iber.**

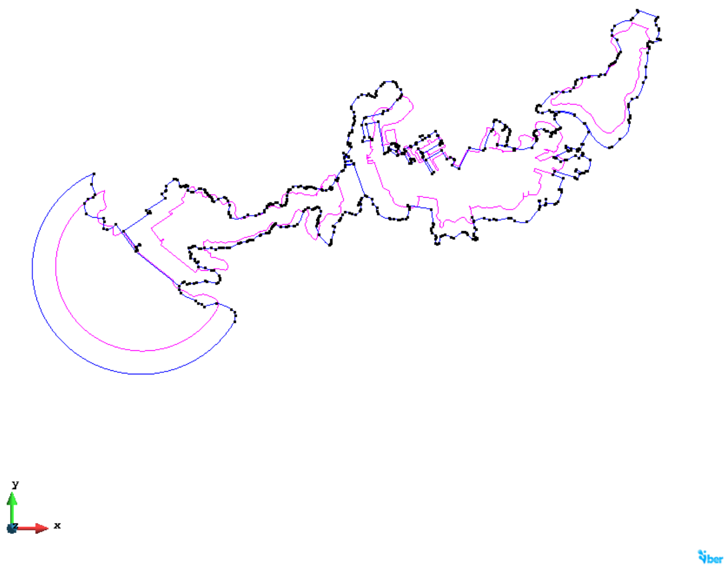



**Figure 130. Geometry of the model once the NURBS surfaces have been created.**

### *9.3.2. Hydrodynamics*

The only boundary condition to be imposed in this case is the tidal level at the mouth of the estuary (Figure 131). The remaining boundaries are closed, since the runoff inflows are negligible. To assign this boundary condition we go to **Data>>Hydrodynamics>>Boundary conditions>>2D Outlet** and we choose a **Subcritical** flow condition of the type **Given Level**. We introduce the tide level series (Tide_Ferrol.txt file) and assign it to the sea boundary of the model. We can copy all the values directly from the text file and paste them into the Iber window.

Although we have set the tidal condition as a given level at an outlet boundary, the water can enter or exit the domain through that boundary, depending on the water level at the mouth of the estuary and the tidal level imposed at each time step. Iber will automatically detect at each time step whether the water level at the boundary is higher or lower than the water level at the mouth of the estuary and, as a function of this, it will correctly calculate the inflow or outflow through that boundary.

As an initial condition we set a water level for the whole estuary equal to the tide level at time t = 0 s, that is, 2.72 metres (**Data>>Hydrodynamics>>Initial conditions**). We impose a Manning coefficient of 0.025 s·m$^{-1/3}$ throughout the estuary (**Data>>Roughness>>Land use**).

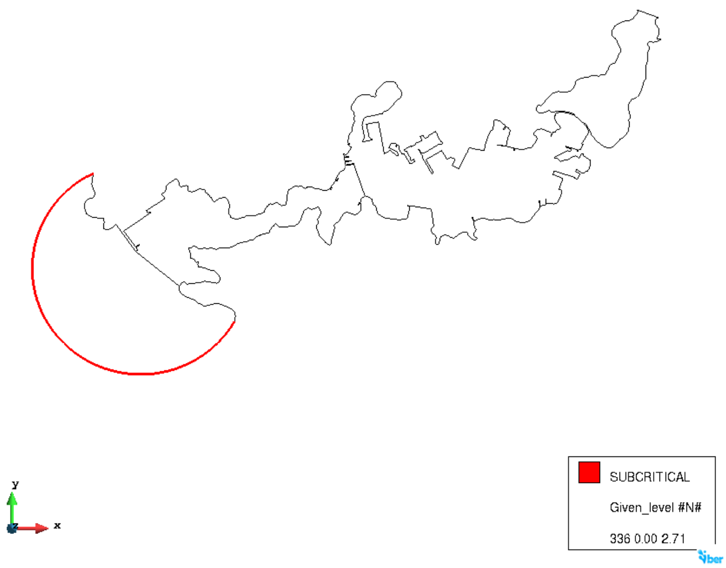



**Figure 131. Location of the open boundary of the model.**

### *9.3.3. Water quality*

We now define the water quality initial conditions. In this case, we define the initial concentration of coliforms in the estuary, which we set to zero for the entire domain, using **Data>>Water Quality>>Initial Conditions>>Coliforms Initial Cond**, as shown in Figure 132. We assign a value of 0 cfu/100 ml to the four NURBS surfaces.

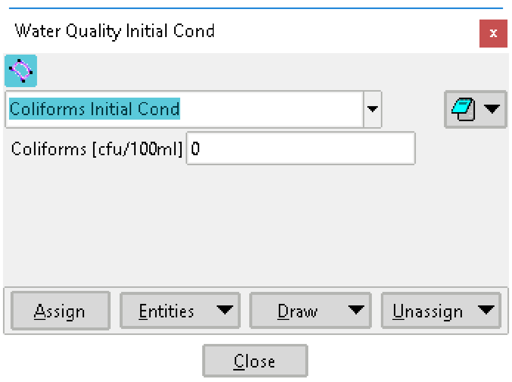



**Figure 132. Water quality initial conditions window.**

We now define the water quality boundary conditions of the model, using **Data>>Water Quality>>Boundary Conditions**. In this case there is only one open boundary, to which we assign a concentration of coliforms equal to 0 cfu/100 ml (Figure 133).

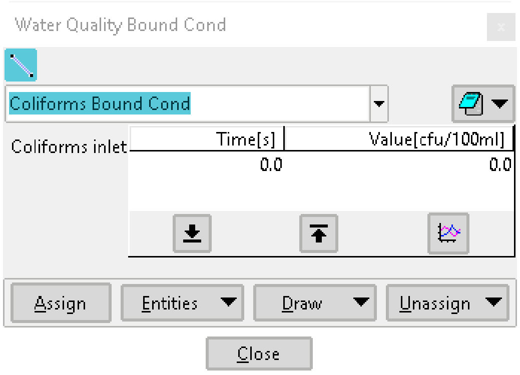



**Figure 133. Water quality boundary conditions window.**

Finally, we introduce the two sewage spills in the model in the positions indicated in Figure 134. To do this, we create a new discharge (**Data>>Water Quality>>Discharges**) and manually specify its location through the button . We must make sure that we place the spills inside the model domain, and not on the outer boundaries. We define a water temperature (Temp) of 20 °C, a flow discharge (Q) of 0.1 $m^3/s$ and a coliform concentration (CF) of 10000000 cfu/100 ml (Figure 135), and keep the default values for the rest of the parameters, since they will not be included in the calculation. Once the discharge data has been introduced, its location will be displayed in the model geometry (with a point at the corresponding location).

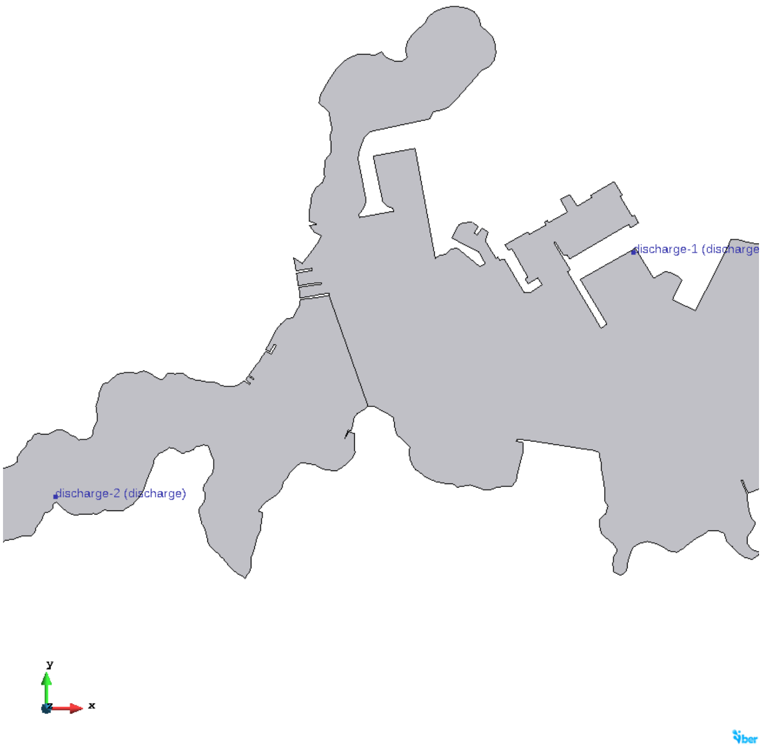



**Figure 134. Location of sewage spills in the model.**

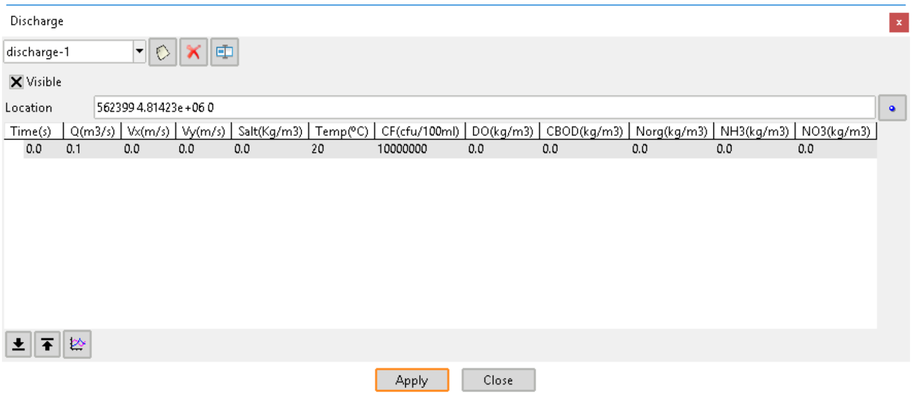



**Figure 135. Discharge tab to introduce the location and water quality characteristics of the spill.**

### *9.3.4. Mesh*

The next step is to create a computational mesh to spatially discretize the domain. We generate an unstructured mesh made of triangular elements, with a mesh size of 250 m in the outer part of the estuary and 100 m in the inner part, as shown in Figure 136. Mesh sizes can be defined in **Mesh>>Unstructured>>Assign sizes on surfaces**.

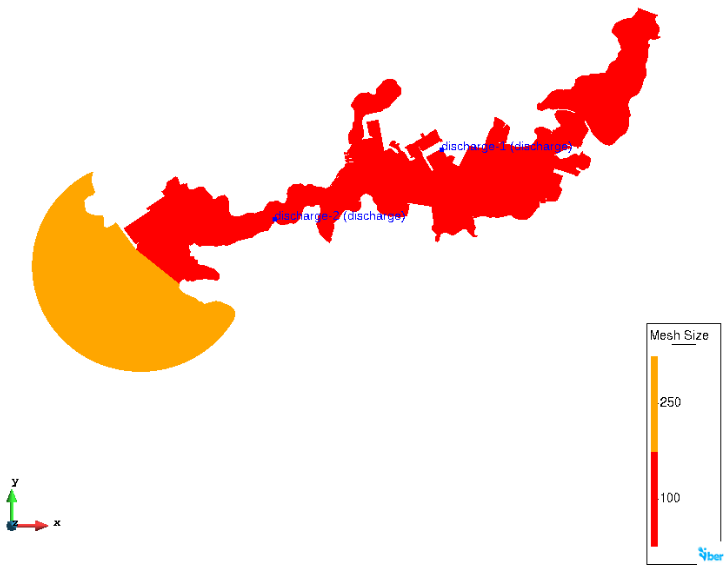



**Figure 136. Mesh sizes defined.**

We set the following meshing preferences in **Utilities>>Preferences>>Meshing**:

- Automatic correct sizes: Normal

- Unstructured size transitions: 0.7
- Regular transition near boundary: Disabled

With these options, we generate a mesh (set 250 m as mesh value for mesh generation) of approximately 8,000 elements (**Mesh>>Generate mesh**) (Figure 137).


**Figure 137. View of the mesh of the model.**

Finally, we interpolate the bathymetry in DTM_Ferrol.asc using **Iber Tools>>Mesh>>Edit>>Set Elevation From File**.

### *9.3.5. Calculation data*

Before running the model, we need to set the computation parameters (**Data>>Problem Data**). First, we choose a maximum simulation time of 172,800 s (approximately 4 tidal cycles) and a time interval of 900 s to analyse the results in sufficient detail.

We define the following parameters in the **Data>>Problem Data>>General** tab, so that the calculation time is not too long:

- Numerical scheme: First Order
- CFL: 0.6
- Wet-dry limit: 0.01 m
- Friction on walls (Show General Options): No Friction

In the **Water Quality** tab (**Data>>Problem Data**), we activate the calculation of coliforms. We choose the T90 decay model for faecal coliforms (Figure 138). As noted previously, T90 is the time required to reach a 90 % reduction in bacterial concentration. We impose a constant T90 value of 10 hours during the entire simulation (T90 Time Series). The value of T90 actually varies with variables such as solar radiation and water temperature, but we will assume a constant value in this simplified example. We select the default options for the remaining parameters.

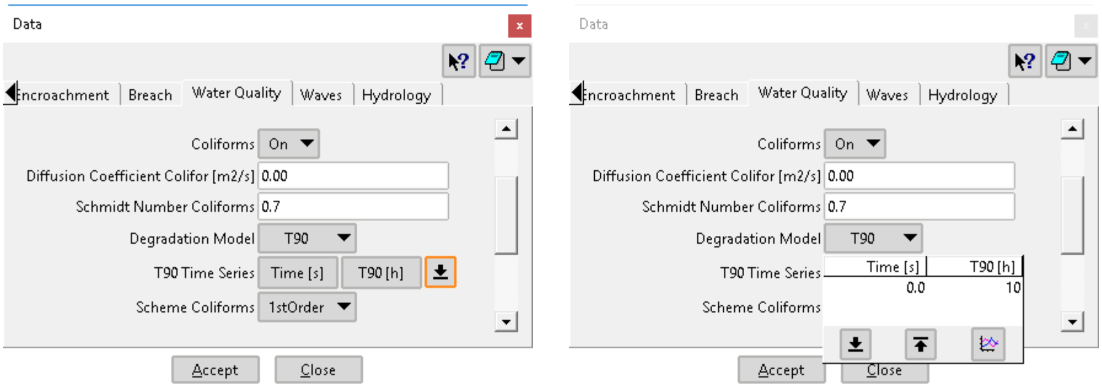



**Figure 138. Definition of T90 values.**

Finally, we launch the calculation (**Calculate>>Calculate**).

## 9.4. Results

Once the calculation has finished, we plot the concentration of coliforms in the final time step (172,800 s), using the **View Results** window (**Window>>View Results**). The results should look like the following image (Figure 139). We can set the range of values represented (maximum and minimum coliform concentration).

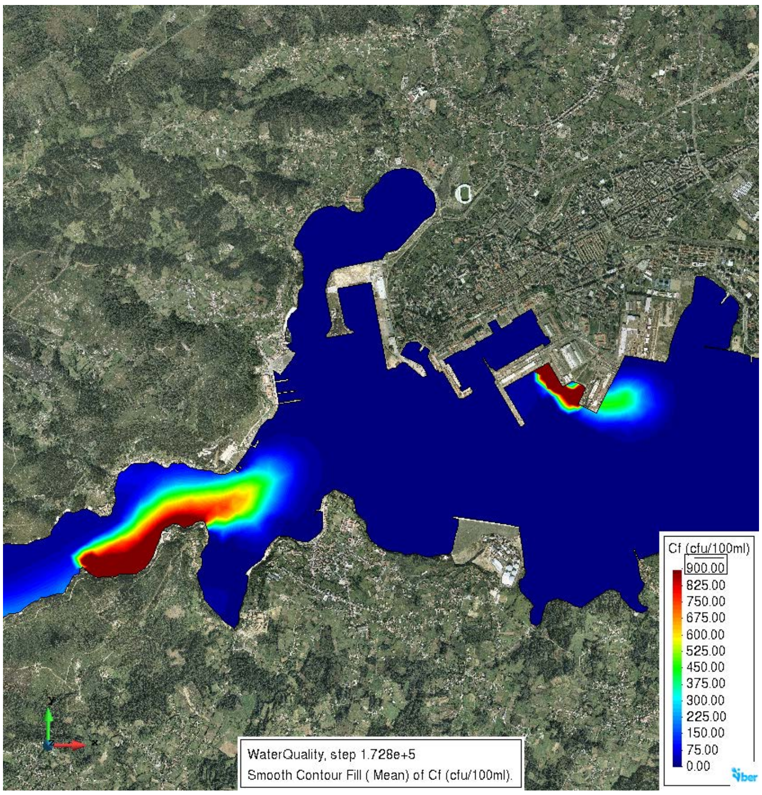



**Figure 139. Concentration of coliforms in the estuary in the last time step.**

We will now plot the temporal evolution of the coliform concentration at two specific points. Open the **View graphs** window and click on the **Create** tab. We select **View: Point Evolution, Analysis: Water Quality** and the concentration of faecal coliforms as the variable to plot (Y-axis). Press **Apply** and choose two points within the model domain (see Figure 140). We should obtain results similar to those shown in Figure 141.

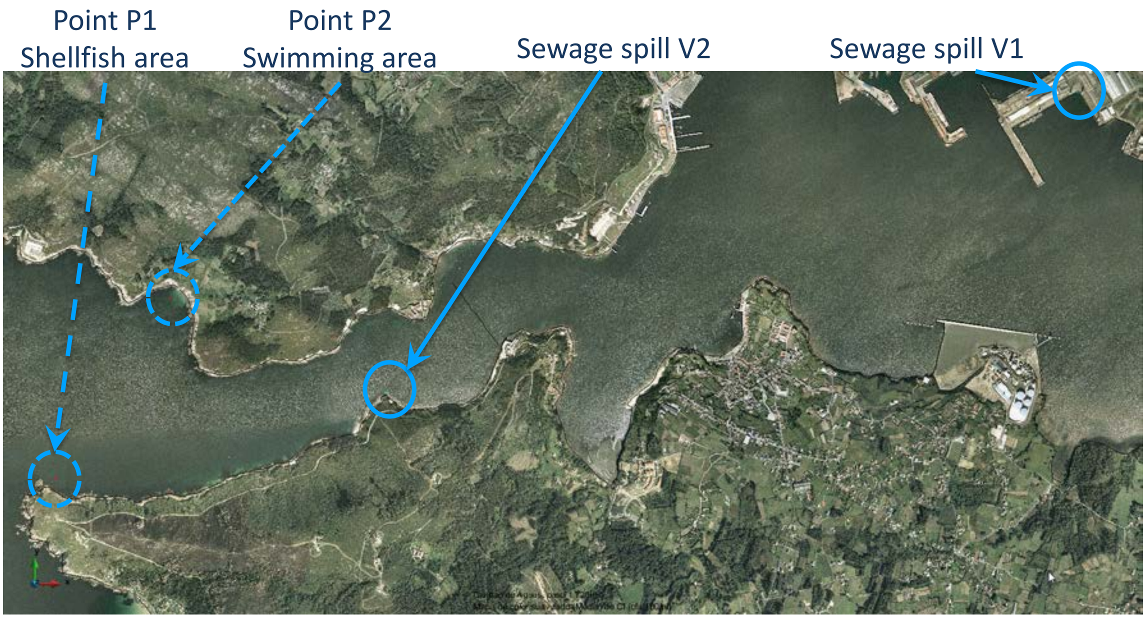



**Figure 140. Location of control points and sewage spills.**

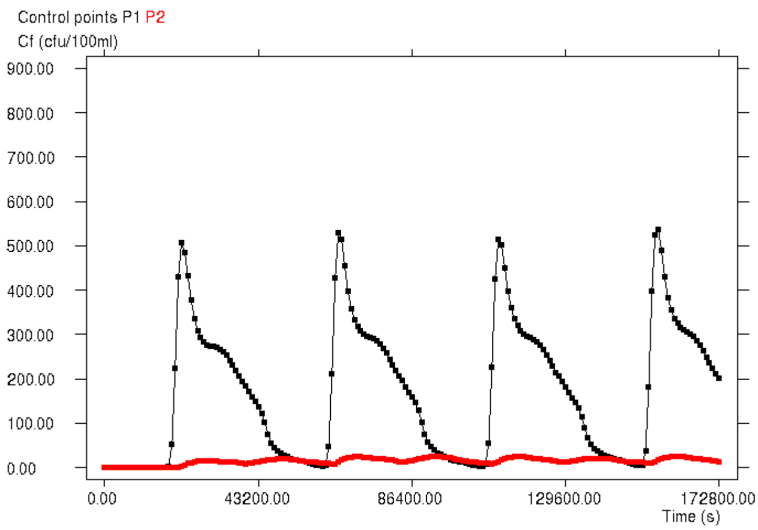



**Figure 141. Time series of coliform concentration at the two control points.**

We can export the time series to analyse them with other software, using Show table in the Options tab (Figure 142). A table with the results for each time step will appear. We can simply copy and paste these results into a spreadsheet, for example, to analyse them in more detail. Please note that in these data the decimal separator is the period "." symbol.

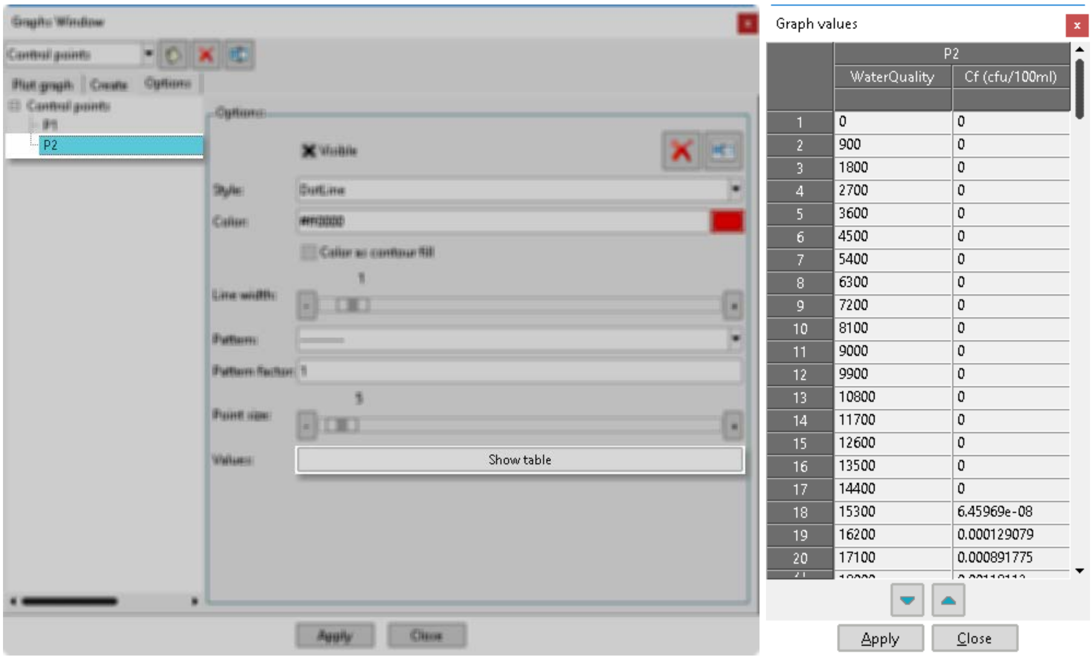



**Figure 142. Time series of coliform concentration P2 (table of values).**

Once we have copied the two time series of concentrations in a spreadsheet, we can sort the values from highest to lowest, as shown in Figure 143. In this way we can see the number of time steps in which the coliform concentration exceeds the threshold value. At point P1, the threshold of 300 cfu/100 ml is exceeded for 34% of the time. At point P2, the coliform concentration is always below 100 cfu/100 ml.

| | A | B | C | D | E | F |
|---|---|---|---|---|---|---|
| 1 | | | | | | |
| 2 | | | | | | |
| 3 | | Shellfish area | | | Swimming area | |
| 4 | | % >300: | 34% | | % >100: | 0% |
| 5 | | FC | Time | | FC | Time |
| 6 | | 931.259 | 110700 | | 49.1999 | 113400 |
| 7 | | 819.177 | 153900 | | 48.885 | 114300 |
| 8 | | 817.818 | 65700 | | 47.0956 | 156600 |
| 9 | | 749.325 | 111600 | | 46.4377 | 68400 |
| 10 | | 704.152 | 153000 | | 45.9169 | 115200 |
| 11 | | 702.791 | 64800 | | 45.7897 | 155700 |
| 12 | | 650.08 | 109800 | | 45.0932 | 67500 |
| 13 | | 593.174 | 154800 | | 44.8389 | 157500 |
| 14 | | 591.56 | 66600 | | 44.2159 | 69300 |
| 15 | | 567.577 | 23400 | | 43.6859 | 116100 |
| 16 | | 491.637 | 24300 | | 41.2328 | 112500 |
| 17 | | 455.297 | 112500 | | 40.3257 | 158400 |

**Figure 143. Analysis of the time series in a spreadsheet.**

If we repeat the same process for the water surface elevation, we can add the corresponding time series to the spreadsheet. This allows us to analyse how the concentration of coliforms varies with the tidal level, by plotting the two time series in the same figure (Figure 144).

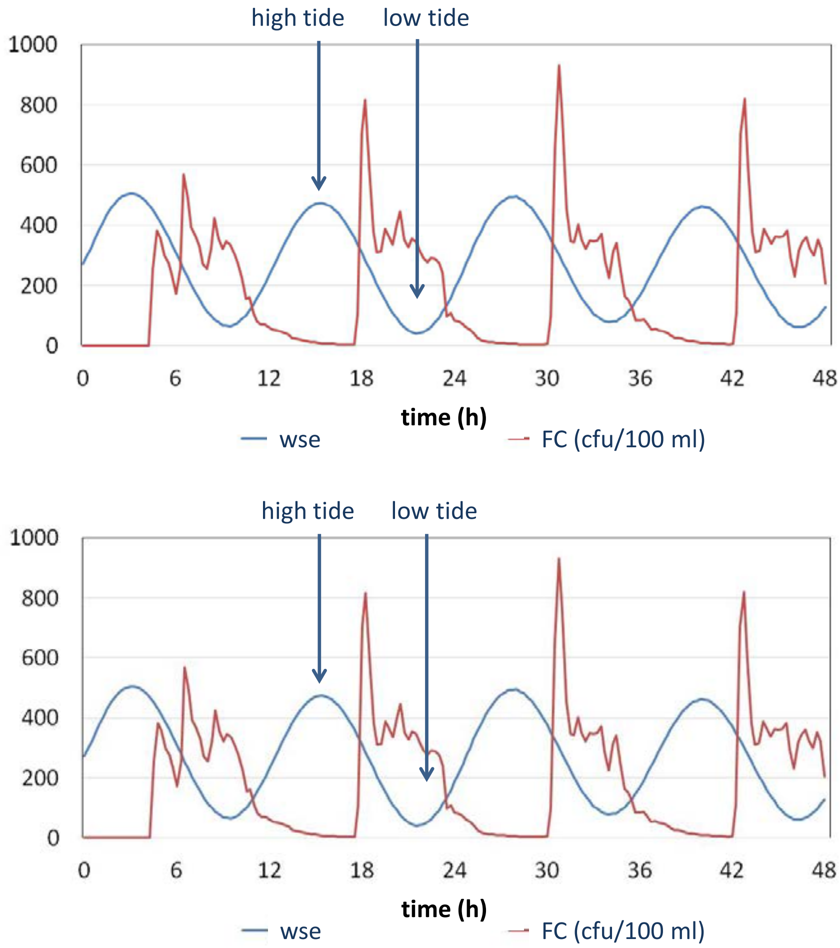



**Figure 144. Coliform concentrations and tidal levels at control point P1 (up) and point P2 (down).**

## 9.5. Conclusions

In this example we have seen how to simulate faecal coliform contamination in Iber. We have used the simplest approach to define the bacterial decay rate, which consists of directly defining a value of T90. The bacterial decay models of Macinin (1978) and Canteras (1995), which depend on meteorological and oceanographic variables, are also implemented in Iber and could be used as an alternative. Based on the results obtained, we can analyse compliance with the regulations regarding faecal coliform concentrations.

## 9.6. References

Bladé, E., Cea, L., Corestein, G., Escolano, E., Puertas, J., Vázquez-Cendón, E., Dolz, J., Coll, A. (2014a). Iber: herramienta de simulación numérica del flujo en ríos. Revista Internacional de Métodos

Numéricos para Cálculo y Diseño en Ingeniería, Volume 30, Issue 1, 2014, Pages 1-10, ISSN 0213-1315, DOI: 10.1016/j.rimni.2012.07.004

Canteras, J. C., Juanes, J. J., Pérez, L. & Koev, K. 1995. Modelling the coliforms inactivation rates in the Cantabrian Sea (Gulf of Biscay) from 'in situ' and laboratory determinations of T90. Water Science and Technology 32 (2), 37–44.

Cea, L., and Bladé, E. (2015). A simple and efficient unstructured finite volume scheme for solving the shallow water equations in overland flow applications. Water Resources Research, 51, 5464-5486. DOI: 10.1002/2014WR01654

Mancini, J.L. 1978. Numerical estimates of coliform mortality rates under various conditions. Journal of Water Pollution Control Federation 50(11), 2477-2484.