

Definition of mixing zones in rivers

Alonso J. Rodríguez Benítez¹ · Andrés García Gómez¹ · César Álvarez Díaz¹

Received: 15 October 2014 / Accepted: 20 July 2015 / Published online: 28 July 2015
© Springer Science+Business Media Dordrecht 2015

Abstract The adequate definition of the mixing zone generated by the discharge of an effluent is of great importance, as it serves as support for the environmental authorities on the decision-making about the authorization of the discharge. The evolution of the mixing zone of an effluent is affected by different kind of phenomena as temporal variations on the hydrodynamic conditions, spatial variations in the geomorphology and bathymetry of the receiving water, etc. The correct definition of the mixing zone should take into account these factors, for which the use of mathematical modelling is needed. The turbulent hydrodynamic processes in the near field of the discharge and in the far field occur at different spatial and temporal scales. The mathematical model needs to be able of simulating the hydrodynamic and transport processes on both fields. The present paper proposes a methodology to be followed when the prediction of the extents of the mixing zone generated by an effluent discharged into a river is needed. The methodology consists of the obtaining of the necessary hydrodynamic and pollutant concentration data on the effluent and on the receiving water; the building of the adequate calculation meshes for the modelling; the calibration and validation of the model; and finally the definition of the critical conditions for the prediction of the behaviour of the mixing zone. An example of application of the proposed methodology is shown for a real case, the discharge of the WWTP of Casar de Periedo town in the Saja River, Cantabria, Spain, for which field data have been measured. The prediction of the extents of the mixing zone for this case was made using a hydrodynamic two-dimensional depth-averaged long wave model jointly with a transport model. In order to simulate the near field and the far field jointly, an Embedded mesh system was built. For the Embedded mesh system, it was needed the establishment of conditions for the information exchange between the meshes.

✉ Alonso J. Rodríguez Benítez
alonsorodriguez4@hotmail.com

¹ Environmental Hydraulics Institute “IH Cantabria”, Universidad de Cantabria, Parque Científico y Tecnológico de Cantabria, C/Isabel Torres, No 15, 39011 Santander, Spain

Keywords Mixing zone · Rivers · Transport modelling · Near field · Embedded mesh

1 Introduction

Most of human activities use water in their processes, and therefore, these activities generate effluents which, after suffering diverse treatment processes, are discharged into natural water bodies. Depending on the source of the effluent, it may have components of different characteristics, chemical, biological, toxic, etc. [1–3], in general harmful to the receiving water body when they exceed certain concentrations [4, 5]. When the discharge comes into contact with the water of the receiving body it starts a dilution phenomenon due to different factors, such as environmental flows, momentum introduced by the effluent, difference in density and temperature between the effluent and the environmental waters, etc. This dilution increases as the effluent progresses within the receiving water body, reducing the concentration of the pollutants introduced. At certain distance from the discharge point, the concentrations of the pollutants reach environmentally acceptable values. The area between the discharge and the place where the contaminants introduced achieve permissible concentrations is known as mixing zone.

When there is a need for the discharge of an effluent into a natural water body, environmental authorities are required to decide whether to give permission or not depending on if the water quality conditions of the receiving waters can be preserved. The extension of the mixing zone generated by the effluent greatly influences on this decision making, because inside of it the maximum acceptable concentration of pollutants established by the regulations are exceeded [6–8]. Therefore, the environmental authorities should require the proper definition of the mixing zones.

If a discharge is performed into an estuary or into the sea, the mass of water in the receiving body can be considered as infinitely greater than that introduced by the effluent and therefore, it is easy to imagine that dilution produced will continue indefinitely. By contrast, if a discharge is performed into a river, it is necessary to take into account that the mass available for the mixing is limited by the flow regimes of the river. For this reason, in the case of rivers exists the complete mixing concept, which defines the state in which the effluent have been mixed up with the entire mass of water available in the river, and therefore it has reached the maximum possible dilution, until meeting another river flow incorporation [9, 10].

There are many factors of different characteristics influencing the development of the mixing zone produced by an effluent; therefore the definition of the mixing zone is a complex process. Some mixing processes can occur due to the energy introduced by the discharge, or due to environmental flows. There are phenomena that occur in the vicinity of the effluent discharge point and other downstream occurring in sensitive areas of the receiving body, etc. Fetterolf [11] concludes that there is no substitute for the estimation of the extents of the mixing zone for each application case, in relation to the particular physical, chemical and biological characteristics of each ecosystem. Among these factors, the most influential are: type of receiving body (usually river, estuary or coastal area), temporal variations on the hydrodynamic conditions of the receiving body, spatial variations in the geomorphology of the receiving body, variations in the bathymetry, temporal variations on the effluent discharge conditions (flow, pollutant concentration), interference with environmentally sensitive areas, discharge method, density and temperature

difference between the effluent and the receiving water, pycnoclines presence, disposition of the effluent discharge with respect to the receiving water, interaction with other effluents in the receiving water, characteristics of the pollutants introduced; and environmentally admissible concentration of pollutant substances. The hydrodynamic conditions on a natural water body vary in function of climatic and seasonal aspects. Depending on the type of water body, these variations have different characteristics. In estuaries and coastal zones, the main hydrodynamic phenomena are induced by waves and tides. In rivers, flow regimes have the main influence. Lakes and lagoons are the most particular cases as depending on their affluent streams, and on their area and depth conditions, the hydrodynamic can be dominated by the flow currents or by the wind. In some cases evaporation processes can produce critical situations on the water quality conditions. The spatial variations in the geomorphology and in the bathymetry of the water body affects the streams and the hydrodynamic turbulent processes [12]. For example, the presence of curves in a river generates circulation phenomena in a section transverse to the flow [13]. The presence of high bathymetry gradients can produce significant vertical accelerations [14]. The density difference between the effluent and the receiving water can generate buoyancy effects, when the effluent is lighter than the receiving water, and sinking effects, when it is heavier [15]. In some cases, the receiving water body can present density gradients known as pycnoclines which can generate internal oscillations in the evolution of an effluent [16, 17]. The disposition of the effluent discharge with respect to the receiving water can influence the initial behavior of the effluent. When the discharge is made through marine outfall, significant vertical velocities are produced [18]. When the discharge is made through a lateral canal into a receiving water with a stream, like a river, the effluent tends to advance attached to the edge of the river. This phenomena is known as Coanda effect [19]. The interaction with other effluents can increase the concentration of certain pollutants introduced. In some cases the receiving water can have a certain background concentration of the pollutant under study produced by effluents discharged upstream. The pollutants introduced by the effluent can suffer decay phenomena depending on their nature, produced by different elements that are present in the receiving water body as sunlight or temperature. These pollutants are known as non—conservative, in contrast with the conservative pollutants, which are not affected by the decay processes [20, 21].

Even more, it is necessary to have in mind the difference in scales of the mixing process. The mixing of an effluent is a turbulent process that can be divided in an initial dilution process, which occurs in the so called near field; and in a secondary dilution, which occurs in the so called far field [22]. In the initial dilution two mechanisms may prevail:

- If the effluent discharge velocity is high, the dilution occurs principally by the high momentum introduced by the effluent, in which case the discharge is called jet.
- If effluent discharge velocity is low, the dilution occurs principally by the density difference existing between the effluent and the receiving water body (caused by the difference in salinity and temperature), in which case the discharge is called plume.

This process continues as the effluent progresses within the receiving water body, until reaching a state in which the causes of the initial dilution disappear, in other words, when the momentum introduced by the effluent is reduced to the point that is lower than the one present in the ambient waters, and/or where the effluent has been diluted to the point where there is not appreciable difference in its density with respect to the ambient waters. From this region onward the mixing continues by the effect of the currents of the receiving water body. In this stage, the role of the hydrodynamics of the ambient waters is more significant.

This zone corresponds to the so called far field, in which the secondary dilution occurs. The dilution rate in this area is lower than that presented in the near field [23]. Therefore, to analyze the mixing process properly it is necessary to take into account the characteristics of both the near field and the far field. However, turbulent processes in near field and far field are different. Processes in the near field are more intense and occur in a spatial and temporal scale much lower than processes in the far field. A common practice for the analysis of the evolution of the mixing zone considering only the phenomena occurring in the far field is to assume an initial dilution coefficient. Thus, in the analysis, the concentration of the pollutants introduced by the effluent into the receiving waters is reduced by this coefficient, which represents the phenomena of turbulent mixing and exchange of salinity and temperature that occur in the near field, in order to perform then only the analysis of the mixing in the far field [24, 25]. The problem with this practice is that the models that define the initial dilution coefficient are simplified and usually stationary, unable to fully reproduce the complexity and variability of the processes occurring in the near field, introducing errors in the calculation [26].

In summary, due to the large number of parameters that influence the evolution of the mixing zone of an effluent, for its analysis it would not be sufficient to use stationary simplistic models. The correct definition of the mixing zones should include all of the temporal and spatial variations that occur in the reality. Currently the best analysis tool for achieving this goal is the use of mathematical models. Moreover, the mathematical model used should be able to take into account the phenomena of both, near field and far field, and their interaction in the transition from one area to another. However, this requires a special effort in the calculation process due to the difference in spatial and temporal scales between both fields.

In the present paper a methodology is proposed to predict the extents of the mixing zone of an effluent in a river. To perform this methodology it is necessary in first place the correctly description of the stretch of the river under study, and the obtaining of adequate hydrodynamic and pollutant concentration data, of both the discharge of the effluent and the stretch of the river. Based on the available data, appropriate calculation meshes can be built, in an adequate mathematical model able to simulate hydrodynamic and transport processes. This model needs to be calibrated and validated. Finally when this is achieved, the mathematical model can be used to predict the behaviour of the mixing zone of the effluent. For this process it is necessary to model the evolution of the effluent under critical river flow conditions established from statistical analysis.

In order to illustrate an example of application of the methodology proposed in the present paper, the experience on determining the evolution of the mixing zone of a real case of effluent discharged into a river is shown. The real case of effluent analysed is the discharge of the Casar de Periedo WWTP (*Waste Water Treatment Plant*) in Saja River (North of Spain). It was used a two-dimensional long wave model developed by the Universidad de Cantabria. This model solves the RANS equations [27]. It is able to reproduce adequately the currents generated by the tide in coastal zones and estuaries [28], as well as the river hydraulics [29]. A transport model capable of simulating the evolution of pollutants within the flow established by the hydrodynamic model was also used. These models are able to perform simulations under spatial and temporal variable conditions. It was necessary to implement an Embedded mesh system (and the adequate information exchange conditions) in order to analyse jointly the processes of the near field and the far field.

2 Methodology for the analysis of the extent of the mixing zone of an effluent in a river

The calculation of the extent of the mixing zone of an effluent discharged in a river must be achieved following the steps listed below:

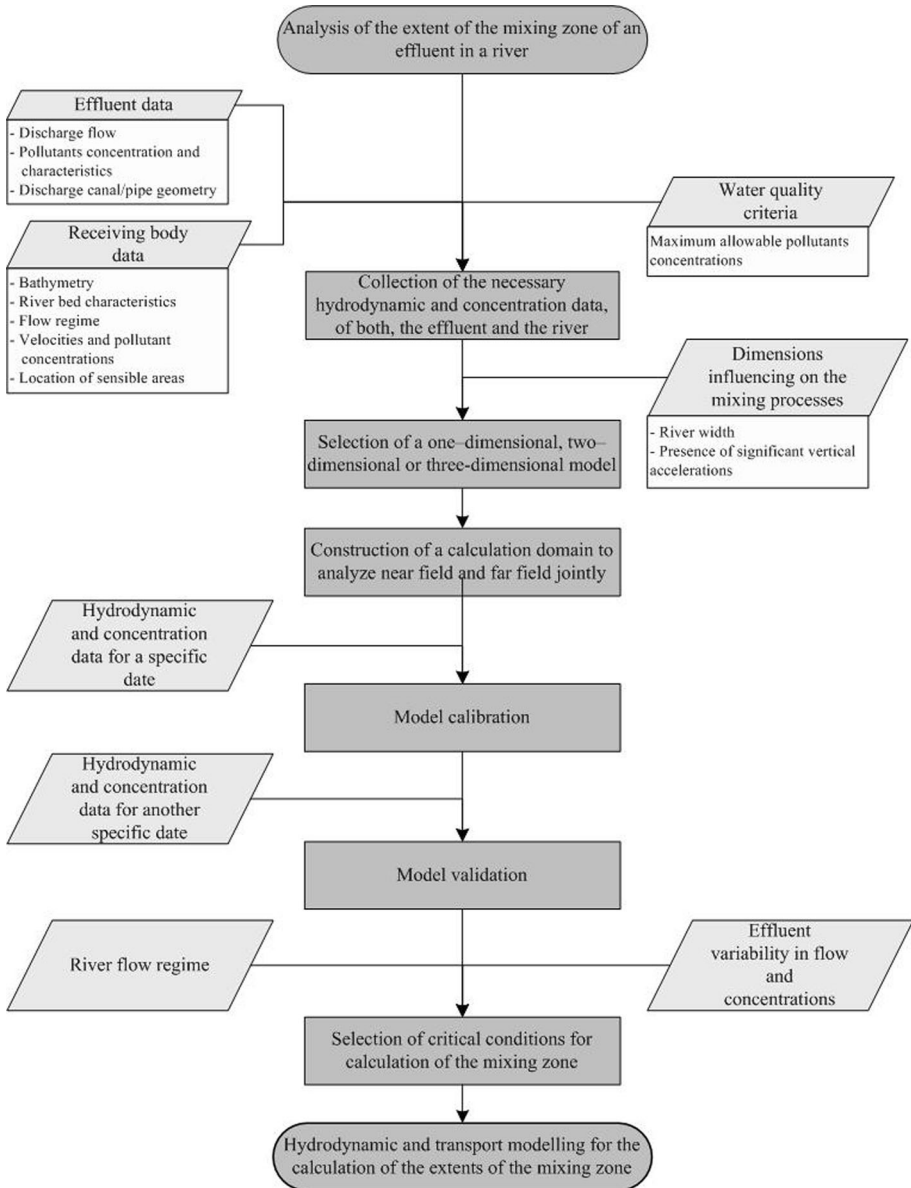


Fig. 1 Methodology for the analysis of the extent of the mixing zone of an effluent in a river

1. Collection of necessary data: Hydrodynamic data and concentration of pollutants, of both the discharge of the effluent and the stretch of the river. Maximum allowable pollutants concentrations.
2. Selection of a one—dimension, a two—dimension, or a three—dimension model according to the dimensions affecting the mixing processes.
3. Construction of a calculation domain able to analyze jointly the near field of the effluent as well as the far field.
4. Calibrate the parameters of the hydrodynamic and transport models, basing on field data measured in a specific date.
5. Validate the parameters calibrated for the models, basing on field data measured in a different specific date.
6. Selection of the critical conditions for the calculation of the mixing zone, basing on the river flow regime and on the effluent variability in flow and concentration.
7. Hydrodynamic and transport modelling for the calculation of the extents of the mixing zone, basing on the maximum allowable pollutants concentrations.

The main aspects of these steps are explained next. The Fig. 1 shows the procedure of this methodology.

2.1 Collection of necessary data

For the analysis of the extent of a determined case of mixing zone, specific data is needed regarding to the effluent, the receiving water body and the applicable normative respect to the maximum allowable pollutants concentrations.

2.1.1 Effluent data

The main data regarding to the effluent are the discharge flow and the introduced pollutant concentration and characteristics. About the discharge flow and pollutant concentration it is important to know its temporal variability, which can be associated with the nature of the activity that generates the effluent (urban wastewaters, industrial effluents, etc.). Normally the entities that generate the effluents exert proper control in order to obtain and maintain the discharge authorizations; therefore in most cases it may be possible to assume mean representative values for the discharge flow and pollutant concentration. However, there may be situations in which pollutant concentration peaks occur, such as failures or temporary disturbances on industrial processes, where interesting analysis cases are generated.

About the characteristics of the introduced pollutant, it is interesting to know if it can be considered conservative; otherwise, the decay phenomena that suffer the pollutant in the receiving water should be included in the transport model. Furthermore, it is necessary to have in mind that in the zone of interest there may be more than one effluent discharge. The mathematical model to be used should be capable of simulate the hydrodynamic of a water body with various flow contributions. Similarly, the effluents can contain more than one kind of pollutants considered of importance from the environmental point of view. Usually, the evolution of each pollutant is modelled independently. For this practice, the use of a modelling system that simulates first the hydrodynamics and then the transport processes represents computational cost savings, because the results of a hydrodynamic model run can serve to different executions of the transport model with different pollutants.

The discharge geometry is also an important data. The spatial resolution in the modelling mesh should be established in order to make possible the representation of the effluent discharge, and the processes occurring in the near field.

2.1.2 Receiving water body data

The main data with respect to the receiving water body are the bathymetry and the bed characteristics for the whole area of interest; and the flow characteristics, including the boundary conditions. The bathymetry used in the analysis should adequately describe the stretch of the river in the computational domain, in such manner that it allows analyzing the influence of curves, slope changes, tidal flats, etc. The most important characteristic of the river bed is the average size of the particles composing it, as it regulates the bottom friction, which is a very influential factor in the mathematical modelling.

Mathematical models require the specification of the river flow and the free surface level in the computational boundaries. The flow of a river has a high variability according to several different parameters (mainly the precipitation). For this reason it is necessary to perform the mixing zone analysis using statistical techniques. The ideal situation would be to have access to a flow regime obtained from daily mean flow data for a significant period of years. In case of not having access to statistical data to obtain a flow regime, the river flow can be also obtained from a hydrologic analysis of the surface runoff taking into account the basin contributing to the study area.

From the river flow regime it can be extracted the critical cases for analyzing the evolution of the mixing zone; however, as will be discussed later, before using the mathematical model for this purpose, the turbulence and diffusion parameters should be calibrated and validated. For these processes, the ideal situation would be to have data for a specific date in order to calibrate the model, and data for a different specific date in order to validate the model. These data should not only be taken in the boundary but also in several control points located along the computational domain. Both, the hydrodynamic model and the transport model needs to be calibrated and validated, therefore these data should consist on velocities and pollutant concentrations.

About the receiving water body it is also important to know if in the zone affected by the effluent there are areas that may be sensitive to the pollutant introduced. The proximity of the resulting mixing zone to environmentally protected areas can influence significantly on the decision about the discharge authorization.

2.1.3 Maximum allowable pollutants concentrations

In order to preserve good water quality conditions on the receiving body, the authorities of each country establish maximum limits to the concentration of each type of pollutant with a recognized potential of environmental damage. For the analysis it is necessary to know the maximum allowable concentrations of each of the pollutants introduced by the effluent, as they define the extents of the mixing zone.

2.2 Selection of the appropriate model

The mathematical modelling can be performed on a one-dimensional, two-dimensional or three-dimensional frame, depending on the complexity of the variables affecting the analysis. The more complex models increase the computational cost and the complexity of

the input data and of the results interpretation, so it is important to choose a frame that fits each case.

For narrow rivers, where transverse mixing processes are negligible, it is possible to use a one-dimensional model. These models work describing several cross sections along the river. On the other hand, when the river is wide enough so transverse mixing processes are significant, or when there are stream irregularities such as curves, widening or narrowing, bifurcations, etc. that may influence the evolution of the mixing zone, the analysis should be performed with a two-dimensional model, vertically averaged.

In cases where vertical accelerations are significant, three-dimensional analysis is required. This situation can be presented on water bodies with density or temperature gradients. Normally these gradients are introduced by the same discharge to analyze, producing buoyancy or sinking effects. Discharges made from the bed of the river with vertical upward direction, such as marine outfalls, also require a three-dimensional analysis.

2.3 Construction of the calculation domain

As it has been discussed before, to analyze the mixing process properly it is necessary to take into account the characteristics of the near field and the far field jointly. The hydrodynamic and transport processes in each field occur at different spatial and temporal scales. The spatial discretization of the far field of a river can have values from 1 m to tens of meters in function of the dimensions of the stretch of the river under study. The canals or pipes used for the discharge of an effluent can have a width of the order of tens of centimetres to meters. The calculation domain to be built for the analysis of the mixing zone should be able to simulate the processes with both spatial discretizations.

Several techniques can be used in order to obtain an appropriate calculation domain. One possible technique is to use variable spatial discretization in the model. This technique has the inconvenience that the time step used should be related to the finer spatial resolution, which increases the computational time if we take into account that for the far field such a restrictive time step is not needed. Another technique consists on using an Embedded mesh system by constructing two different calculation meshes, one representing the far field, covering the entire stretch under analysis; and other one representing the near field of the effluent discharge with smaller spatial and time resolution. This technique is more complicated to implement because it needs the establishment of conditions for information exchange between the meshes, in order to make sure that the hydrodynamic and transport processes occurring in one mesh affects the other, but has the advantage that it saves computational time as for each field an appropriate time step can be used.

2.4 Calibration and validation

One of the main concerns on the use of a hydrodynamic mathematical model is the turbulence representation. There are several closure models to represent the turbulence processes; all of them need the establishment of constant values established for each case of study. Furthermore, for the use of transport models it is necessary the establishment of values for the diffusion coefficient. The ideal procedure to establish these parameters is to perform a calibration process, for which it is necessary to have data obtained from field measurements for a specific date. Calibration process is based on the comparison of the modelling results with the field data, therefore the field data should include hydrodynamic and transport values (velocities and concentrations) for different control points located

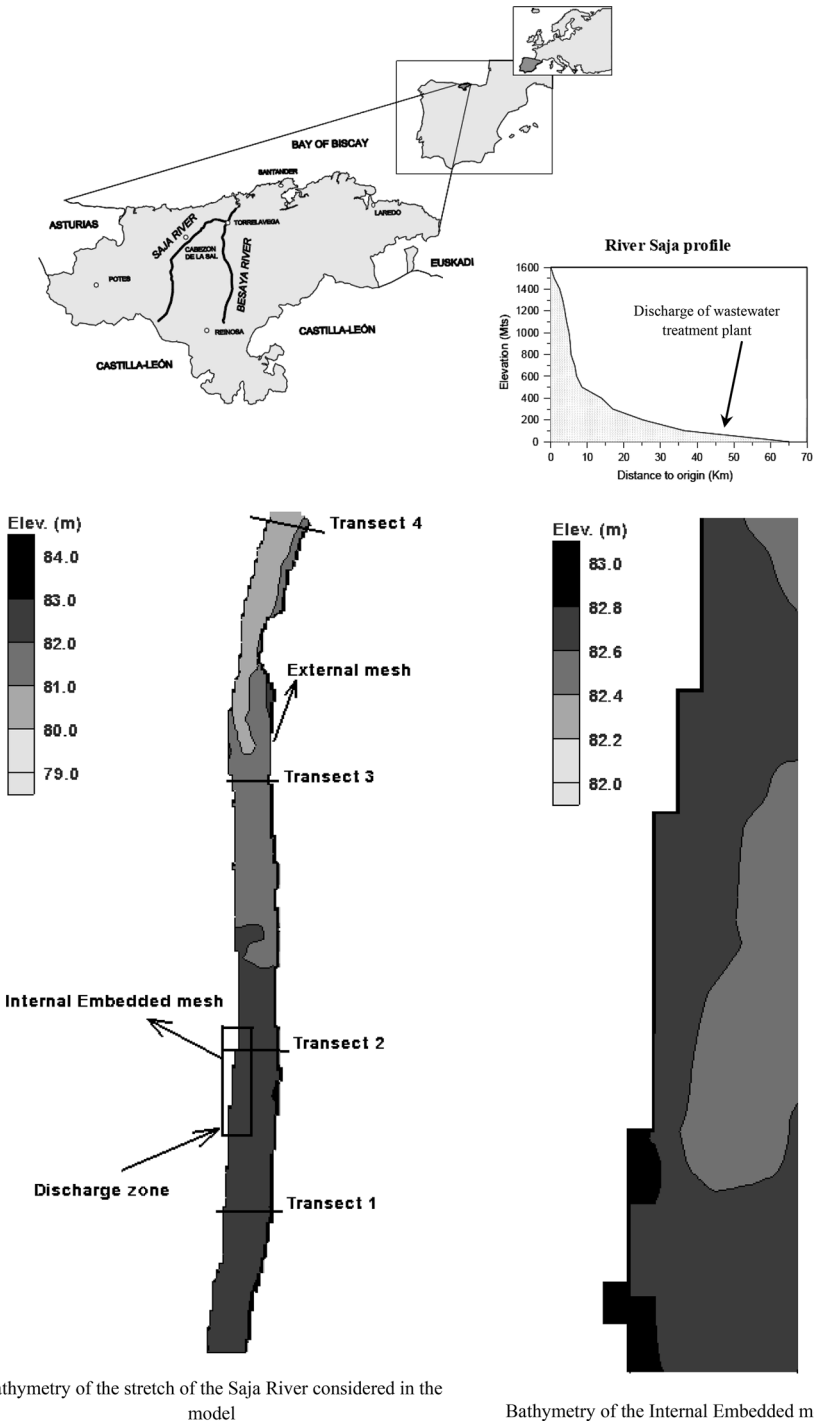


Fig. 2 Area of study, location of measurement transects and External and internal Embedded meshes

along the stretch of the river as well as for the boundaries. The performance of the model using the parameters obtained from the calibration has to be validated, performing another modelling using different field data measured on a different specific date. If field data are not available to perform the calibration and validation processes, recommended values obtained from empirical studies can be used. Finally, it should be noted that if an Embedded mesh system is used, different values for the calibration parameters should be established for the different meshes.

2.5 Selection of critical calculation cases

Once the model is calibrated and validated, it can be used to predict the extents of the mixing zone. Model simulations should be performed for critical conditions of river flow and effluent discharge (flow and concentration) in order to evaluate the maximum possible impact of the introduced pollutants on the environment. The selection of the critical river flow should be based on the statistical analysis of the river flow regime, as discussed above. Usually, critical conditions are related to lower river flow values. With high river flow values the extension of the mixing zone is smaller due to the velocities and turbulent phenomena associated to high flows are higher and there exists more ambient water available to the dilution processes. The critical conditions for the effluent are associated to high pollutant concentrations expected on the discharge, in combination with low discharge flows. Depending on the effluent discharge regime, it may be necessary to analyse more than one discharge conditions in order to include the possible combinations of flow and discharge.

2.6 Calculation of the extents of the mixing zone

Finally, the hydrodynamic and transport mathematical modelling is performed for the critical cases chosen. The extents of the mixing zone of the effluent for each case analysed is obtained by locating the area in the stretch of the river where concentration results reach the maximum allowable values. The predicted extent of the mixing zone adopted for environmental analysis is the maximum obtained from the modellings of the different cases.

3 Application of the proposed methodology to the Casar de Periedo WWTP discharge

In order to show an example of the application of the methodology proposed in the present study for the characterization of the mixing zone of an effluent, it is analysed the real case of the discharge of the WWTP of Casar de Periedo, in the Saja River (north Spain). Casar de Periedo is a small village in the Administrative Division of Cantabria. The design equivalent population for the wastewater treatment plant is 20.000 inhabitants. Trough the village area crosses a stretch of the Saja River. The Saja River has a basin of 440 km², and a total length of 67 km. It discharges in the Ría de San Martín de la Arena. Figure 2 show the location of the area of study.

The discharge of the WWTP of Casar de Periedo is made in an edge of the river with a cross section of around 30 m width. Field surveys were carried out to obtain the

bathymetry of the study area [30]. The bathymetrical information measured covered a stretch of the river 150 m upstream and 380 m downstream of the discharge point. Bathymetry of the stretch considered in the model is shown in Fig. 2.

The discharge of the WWTP is made with a mean flow of 0.138 m³/s. The disposal of the effluent is made using a channel with a width of approximately 3.5 m and an average discharge velocity, u_0 , of 0.19 m/s. The hydrodynamic data and chloride concentration in the river were measured in field surveys performed in spring and summer seasons. For the day when measurements were made in spring season, the average flow recorded for the Saja River was 12.5 m³/s, while for the day when measurements were made in summer season the flow registered was 2.97 m³/s. For the days when spring and summer field measurements were made, no significant variation was observed on the river average flow, therefore, the mathematical modelling can be performed under stationary conditions, in order to reduce computational costs. The free surface levels in the downstream boundary of the stretch are taken from a weir, which permits to know the free surface level in the boundary for each river flow under analysis. The bottom friction has been taken variable over the calculation domain in order to take into account the influence of the water depth on this parameter. For this purpose, values of Chezy coefficients between 20 and 30 were used.

For the measurement of velocities in the Saja River stretch under study, four different transects were selected. The first was located 50 m upstream the discharge while the remaining three were placed 50, 220 and 400 m downstream the WWTP discharge. The

Table 1 Chloride concentration in the WWTP effluent

Spring survey	
Time (h)	Chloride concentration (mg/l)
9	20
12	34
15	49
18	78
21	64
24	73
27	88
30	61
Summer survey	
Time (h)	Chloride concentration (mg/l)
9.5	139
12	38
15	42
18	28
21	41
24	44
27	56
30	44

localization of these transects can be seen in Fig. 2. The velocities measured in these transects are shown in Fig. 6 for spring data and in Fig. 12 for summer data, for spring flow and summer flow respectively.

Chloride concentrations in the discharge channel and in the transects set in the river were measured during a period of 30 h, every 3 h. Additionally, chloride data were taken in a transect coincident with the channel mouth. For the measurements in each one of the transects, data were taken near the bank and in the center of the river's cross section. Table 1 shows chloride discharge concentrations in the WWTP effluent, while chloride data collected in the river are shown in Table 2.

3.1 Overview of the two-dimensional long wave model used for present case

The two-dimensional long wave model used solves the depth-averaged continuity and momentum equations using a finite differences scheme. The model considers

Table 2 Chloride concentrations measured in the Saja River

Spring survey								
Transect	Chloride concentration (mg/l)							
	1 (50 m upstream)		2 (50 m downstream)		3 (220 m downstream)		4 (400 m downstream)	
Time (h)	Center	Bank	Center	Bank	Center	Bank	Center	Bank
9	8.4	9.3	8.9	20	7.9	21	7.8	9
12	8.2	7.1	7.7	26	7.7	22	6.9	20
15	10.7	10.5	10.4	38	11.6	27	12.4	25
18	13.4	12.8	10.9	32	11.4	23	11.9	22
21	10.6	9.8	11.9	36	11.8	20	11.4	16
24	11	11.7	11.8	38	12.1	30	10.9	24
27	10.4	11.9	10.2	56	10.4	37	10.7	26
30	10.4	10.8	9.9	35	9.7	20	9.8	21
Summer survey								
Transect	Chloride concentration (mg/l)							
	1 (50 m upstream)		2 (50 m downstream)		3 (220 m downstream)		4 (400 m downstream)	
Time (h)	Center	Bank	Center	Bank	Center	Bank	Center	Bank
9.5	33.8	21.8	47.7	55	16.9	55.9	11.6	15.9
12	29	18.1	26	32	21	27	21	31
15	31	32	28	43	34	43	30	31
18	15.4	12.6	11.7	27	15.1	24	15.7	15.9
21	18.6	18.9	21	36	25	40	21	22
24	26	25	26	38	25	42	26	29
27	21	23	26	50	25	38	21	25
30	26	26	27	41	24	39	24	26

incompressible flow, hydrostatic pressure and the Boussinesq approximation [14]. The model is capable of simulate the behaviour of a hydrodynamic flow under the action of tide, wind, density variations (salinity and temperature), fluvial flows and Coriolis accelerations. Model governing equations are expressed as [29, 31]:

$$\frac{\partial uh}{\partial x} + \frac{\partial vh}{\partial y} + \frac{\partial h}{\partial t} = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = fv - g \frac{\partial z_1}{\partial x} + \frac{1}{\rho h} [\tau_{xz(z_1)} - \tau_{xz(z_0)}] + v_e \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right] \tag{2}$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -fu - g \frac{\partial z_1}{\partial y} + \frac{1}{\rho h} [\tau_{yz(z_1)} - \tau_{yz(z_0)}] + v_e \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right] \tag{3}$$

where z_0 is the water bed level with respect to an established reference level, z_1 is the water free surface level with respect to the reference level, $h = z_1 - z_0$ is the total water depth (m), g is the gravitational acceleration (m/s^2), u and v are the vertical integrated velocities in x and y directions respectively, ρ is the average water density, f is the Coriolis parameter, $\tau_{xz(z_1)}$, $\tau_{yz(z_1)}$ are the free surface tangential stress, $\tau_{xz(z_0)}$, $\tau_{yz(z_0)}$ are the water bed tangential stress and v_e represents the eddy viscosity for the turbulence closure. Figure 3 shows the reference system used.

The hydrodynamic model uses the Alternating Direction Implicit technique, ADI [32], to solve Eqs. 1 to 3. Turbulence model used for the closure of the Eqs. 1 to 3 is based on the 0 equation eddy viscosity model.

The hydrodynamic field calculated by this mathematical model can be used as input of a transport model to analyze the behaviour of a substance introduced into the water body, by solving the transport equation in the following form [33, 34]:

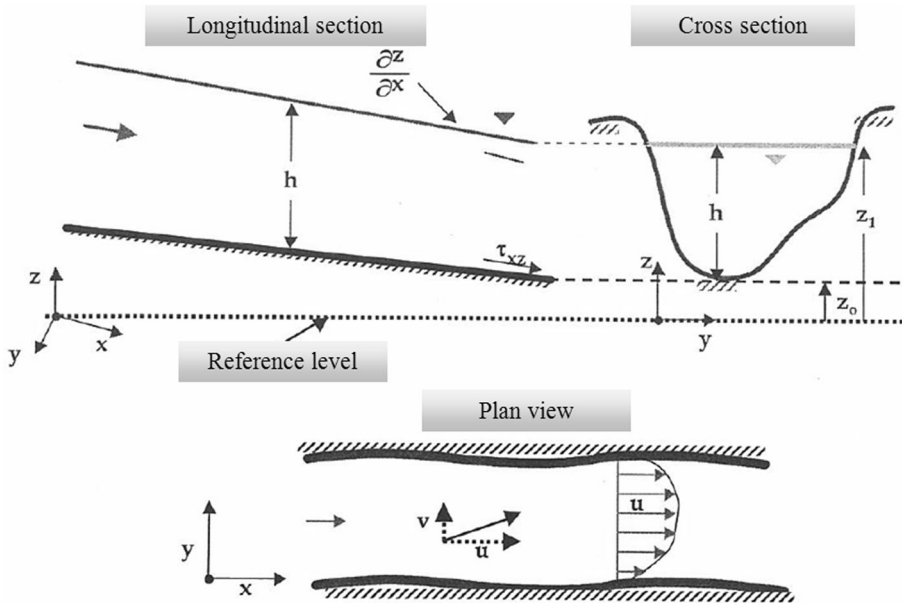


Fig. 3 Reference system for the mathematical models

$$\frac{\partial hC}{\partial t} + \frac{\partial uhC}{\partial x} + \frac{\partial vhC}{\partial y} = \frac{\partial}{\partial x} \left(hD_x \frac{\partial C}{\partial x} \right) + \frac{\partial}{\partial y} \left(hD_y \frac{\partial C}{\partial y} \right) + FH \quad (4)$$

where C is the substance concentration; D_x and D_y are the diffusion coefficients in the x and y direction respectively; and F is a term representing the sink or source of the substance. In the present study the contaminants analyzed are conservative, so the term F is neglected. The transport equation is solved with a finite difference explicit upwind algorithm for the advection processes, and with a centered algorithm for the diffusion processes.

3.2 Embedded mesh system

For the modelling of the Saja River stretch under study, a cell size $\Delta x = \Delta y = 2$ m can be established. Normally, the time step, Δt , should be set to satisfy the Courant condition, as shown in Eq. 5, where h is a characteristic depth, which can be set around 0.2 m for the bathymetry considered. This results in considering a hydrodynamic time step Δt not bigger than 2 s.

$$\Delta t < \sqrt{\frac{\Delta x^2 + \Delta y^2}{g \times h}} \quad (5)$$

The WWTP discharge channel has a width of 3.5 m, thus the cell size should be lesser than this value in order to properly simulate the hydrodynamic of the area surrounding the jet discharge. To get a good representation of the discharge channel in the model, the cell size can be established in $\Delta x = \Delta y = 0.5$ m. By applying the Courant condition to this cell size, a hydrodynamic time step $\Delta t < 0.49$ s is obtained. This cell size and hydrodynamic time step are too restrictive considering the stretch of the river modelled, since it supposes a huge number of cells and a high computational time.

The scale difference between the values of cell size and time step needed for the area surrounding the discharge and the values needed for the entire stretch considered, makes it necessary to use an Embedded mesh system for this modelling. So, an External mesh and an Embedded mesh were built to perform the modelling. The External mesh represents the total stretch considered in the present study, having $\Delta x_E = \Delta y_E = 2$ m and $\Delta t_E = 2$ s. The Embedded mesh represents the surroundings of the discharge point, with an area of 16 m \times 70 m, and is set to have $\Delta x_I = \Delta y_I = 0.5$ m and $\Delta t_I = 0.2$ s. The Internal Embedded mesh constructed for the modelling and its relative location inside the External mesh are shown in Fig. 2.

Using of an Embedded mesh system for modelling implies the needing of defining adequate conditions to exchange hydrodynamic information between the two meshes. In order to achieve the level of detail desired in the discharge simulation, the Embedded mesh (where the discharge is made) needs to receive as boundary conditions the currents and free surface level from the External mesh, while External mesh needs to receive a feedback of the hydraulic process occurring inside the Embedded mesh. The details on the information exchange between these two meshes are presented next.

3.2.1 General considerations for the use of the embedded mesh system

Inside the Internal mesh the same governing equations of the External mesh are solved. Different values for the hydrodynamic data are defined for the Internal mesh (z_I , u_I , v_I , Δt_I ,

Δx_I and Δy_I) and for the External mesh ($z_E, u_E, v_E, \Delta t_E, \Delta x_E$ and Δy_E). The execution method of the Embedded mesh system consists in performing a calculation cycle of the External mesh, followed by a number, N_{CI} , of calculation cycles of the Internal mesh. The N_{CI} value is determined by the relationship between the time steps of both meshes in the following form:

$$N_{CI} = \frac{\Delta t_E}{\Delta t_I} \tag{6}$$

For the case in study, $N_{CI} = 2 \text{ s}/0.2 \text{ s} = 10$ calculation cycles, in other words, after a calculation cycle of the External mesh is made, 10 cycles of the Internal mesh are performed until complete a simulation time $\Delta t_I N_{CI} = \Delta t_E$, after which another calculation cycle of the External mesh is made and so on.

In order to ease the comprehension of the calculation process of the Embedded mesh system, the Internal mesh has been located so that their boundaries, Γ_I , match the edges of External mesh’s cells. To make this situation possible Δx_I has been established as sub-multiple of Δx_E . Figure 4 shows the scheme of this arrangement. On this way, each cell of the External mesh spatially corresponds to a certain number of cells of the Internal mesh, N_{CC} , expressed as follows:

$$N_{CC} = \left(\frac{\Delta x_E}{\Delta x_I} \right)^2 \tag{7}$$

Similarly, in the boundaries of Γ of the Internal mesh, the edges of every cell of the External mesh spatially corresponds to the edges of a certain number of cells of the Internal mesh, $N_{CC\Gamma}$, defined as:

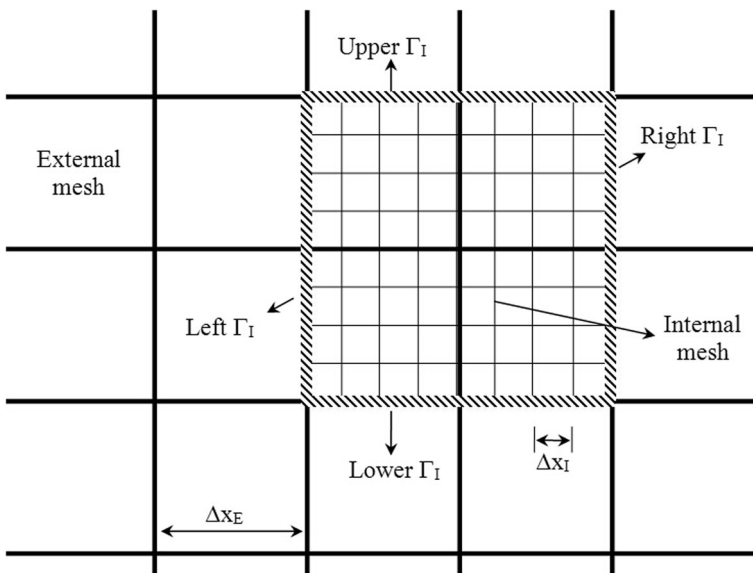


Fig. 4 Discretization scheme used for the Embedded mesh system

$$N_{CCF} = \frac{\Delta x_E}{\Delta x_I} \tag{8}$$

In our application case, $N_{CC} = (2 \text{ m}/0.5 \text{ m})^2 = 16$, which means that every cell of the External mesh spatially corresponding with the Internal mesh, matches 16 cells of the Internal mesh. Additionally $N_{CCF} = 1 \text{ m}/0.2 \text{ m} = 4$, meaning that every cell of the External mesh spatially corresponding with one of the boundaries of the Internal mesh, Γ_I , matches 4 of the cells of the Internal mesh in that particular boundary.

Calibration parameters, as the eddy viscosity for the turbulence model and the diffusion coefficient for the transport model, can be established with different values for the External mesh and for the Internal mesh, depending on the modelling requirements. The flow-chart on Fig. 5 shows the coupling procedure used for the modelling with the Embedded mesh system.

3.2.2 Use of Embedded mesh for the hydraulic model

The boundary condition for the Internal mesh used for the hydrodynamic model consists in imposing directly in the cells of the Internal mesh’s boundary, Γ_I , the velocity of the cell spatially corresponding with the External mesh as follows:

$$\Phi_{I\Gamma_i}^{t+1} = \Phi_{E\Gamma_i}^t \tag{9}$$

where Γ_I is the boundary of the Internal mesh under analysis, Φ_I is the velocity of the Internal mesh perpendicular to the boundary (u_I or v_I), Φ_E is the velocity of the External

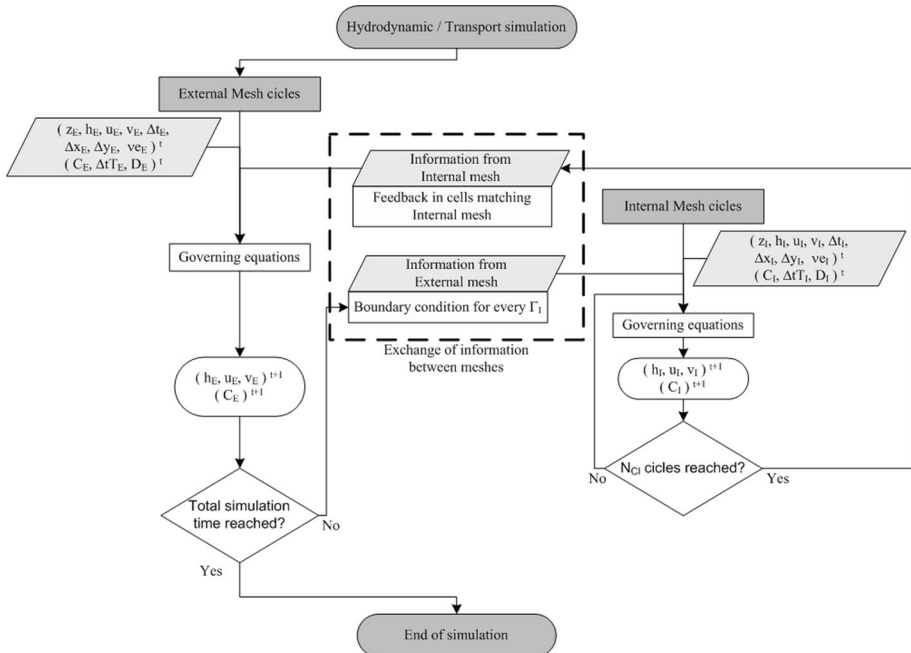


Fig. 5 Coupling procedure used for the modelling with the Embedded mesh system

mesh perpendicular to the boundary (u_E or u_E) and i is each one of the N_{CCF} cells of the Internal mesh spatially corresponding to the cell of the External mesh j .

The feedback condition from Internal mesh to External mesh is expressed as follows:

$$\Phi_{E_{i,j}}^t = \frac{\sum_1^{N_{CC}} \Phi_{I_{m,n}}^t}{N_{CC}}; \quad \eta_{E_{i,j}}^t = \frac{\sum_1^{N_{CC}} \eta_{I_{m,n}}^t}{N_{CC}} \tag{10}$$

where η_E is the free surface water level of the External mesh, η_I is the free surface water level of the Internal mesh; and m,n is every cell in the Internal mesh spatially corresponding to the cell i,j of the External mesh. Under this feedback condition, for every cell in the External mesh spatially corresponding to the Internal mesh, the average of the hydrodynamic information (currents and level) of the cells in the Internal mesh matching each cell of the External mesh is imposed.

3.2.3 Use of Embedded mesh for the transport model

As well as for the hydraulic model, the use of Embedded mesh for the transport model requires to ensure the adequate exchange of information of pollutant concentrations between the External and the Internal meshes.

The boundary condition for the Internal mesh consists in performing a mass balance in the following way [35, 36]:

$$C_{I_{\Gamma,i}}^{t+1} = \frac{C_{I_{\Gamma,i}}^t \times Vol_{C_{I_{\Gamma,i}}} + C_{\Delta Vol} \times \Delta Vol}{Vol_{C_{I_{\Gamma,i}}} + \Delta Vol} \tag{11}$$

where C is the pollutant concentration, Vol_C is the volume of the cell in the boundary, ΔVol is the volume of water that either enters or exits the Embedded mesh trough the boundary, and $C_{\Delta Vol}$ is the concentration of pollutant that either enters or exits the Embedded mesh with ΔVol . The values of ΔVol and $C_{\Delta Vol}$ are established as follows:

$$\Delta Vol = \Phi_{E_{\Gamma,i}}^t \times h_{I_{\Gamma,i}} \times \Delta x_I \times \Delta t_I \tag{12}$$

$$C_{\Delta Vol} = C_{E_{\Gamma,i}}^t \tag{13}$$

where C_E is the pollutant concentration in the cell of the External mesh corresponding to the boundary of the Embedded mesh Γ_I , $h_{I_{\Gamma,i}}$ is the depth in the cell i of the Embedded mesh in the boundary Γ_I , and Δt_I is the temporal discretization for the transport model for the Embedded mesh.

For the feedback condition from the Embedded mesh to the External mesh, it was established a simple imposing of pollutant concentration. This consists in establishing in every cell of the External mesh the sum of the masses of the pollutant in the corresponding cells of the Embedded mesh in the following way:

$$C_{E_{i,j}}^t = \frac{\sum_1^{N_{CC}} m_{I_{m,n}}^t}{Vol_{CE_{i,j}}^t} \tag{14}$$

where $m_{I_{\times m,n}}$ is the total pollutant mass of the N_{CC} cells m, n of the Embedded mesh that spatially corresponds to the External mesh i, j ; and $Vol_{CE_{i,j}}$ is the volume of the cell i, j of the External mesh.

3.3 Calibration of the model—spring flow

Once the Embedded mesh system is built, it is needed the establishing of the proper values for the calibration parameters in order to obtain a suitable model to representation of the mixing zone of the effluent. These parameters are the eddy viscosity values for the turbulence representation in the hydrodynamic model, and the diffusion coefficient for the transport model. For calibration purposes, it have been modelled the conditions presented by the river when the field measurements were made in spring.

3.3.1 Hydrodynamic model

As commented before, for the turbulence representation in the hydrodynamic model a 0 equation eddy viscosity turbulence model is used. As turbulence processes in the near field and in the far field occur at different magnitude scales, different eddy viscosity values should be used for the External mesh, ve_E , and for the Embedded mesh, ve_I , in a similar way as different values of spatial and time discretization are used for both meshes.

Several hydrodynamic modellings were performed with the embedded mesh system, using different values of ve_E and ve_I , in order to calibrate these parameters to obtain an adequate representation of the mean river flow for the days when spring field measurements were made. Good modelling results in concordance with the field data were obtained for ve_E value of $0.1 \text{ m}^2/\text{s}$, with an ve_I value of $0.05 \text{ m}^2/\text{s}$. Figure 6 shows for the External

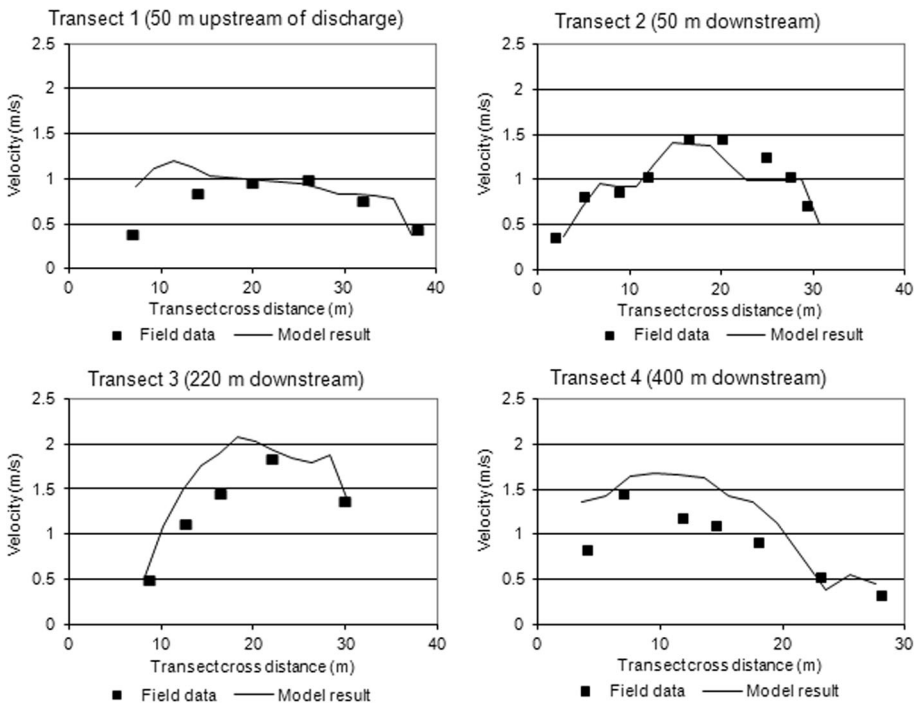


Fig. 6 Comparison of modelled velocity profiles versus field data measured in transects of the Saja River. Calibration with spring flow

mesh, the comparison between the velocity profiles obtained from the numerical simulation against field data measured in the river transects. The velocities comparison for the Embedded mesh is shown in Fig. 7 for the transect number 2, which it is the only one located within the mesh. The modelled velocity fields for External mesh and Internal Embedded mesh are shown in Fig. 8.

3.3.2 Transport model

As well as for the hydrodynamic model, different diffusion coefficients should be used for the External mesh, D_E , and for the Embedded mesh, D_I , and for the is the temporal discretization (time step for the External mesh, $\Delta t T_E$, and for the Embedded mesh, $\Delta t T_I$), as mixing processes in the near field and in the far field occur at different magnitude scales.

The transport modelling was performed taking the results from the hydrodynamic model as input data. The evolution of the chloride discharged by the WWTP was simulated during the time period when the field measurements were taken (30 h), for spring flow. After several simulations, good concordance between modelling results and measured concentrations was obtained using for the External mesh $\Delta t T_E = 0.2$ s, and $D_E = 0.05$ m²/s, with $\Delta t T_I = 0.05$ s, and $D_I = 0.01$ m²/s for the Embedded mesh.

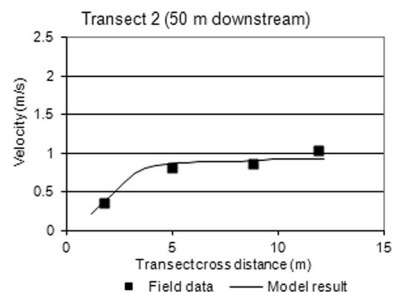
Figure 9 shows the comparison between the chloride concentrations measured in the sampling stations with model results, for the External mesh. The comparison of measured data and modelled concentrations for the Embedded mesh is shown in Fig. 10 at the bank of transect 2, since it is the only one included in the Embedded mesh area.

Figure 11 shows the concentration fields obtained from the modelling, for External and Embedded mesh, after 27 h simulated. It can be observed that the pollutant introduced in the Embedded mesh is dispersed adequately in the External mesh. As it was expected, the chloride mass advances through the river attached to the left bank, where there are found chloride concentration values higher than the ones found on the center of the river. For the concentration field of the External mesh in the zone corresponding to the Embedded mesh, good concordance is observed with the concentration field of the Embedded mesh.

3.4 Validation of the model—summer flow

With the calibration parameters adequately obtained, the validation of the model to represent the mixing zone can be achieved, by performing a modelling with different conditions in order to contrast its results. In present study, for validation purposes, modellings have been made to simulate the conditions presented by the river when the field measurements were made in summer.

Fig. 7 Comparison of modelled velocities versus field data along transect 2 in the Internal Embedded mesh. Calibration with spring flow



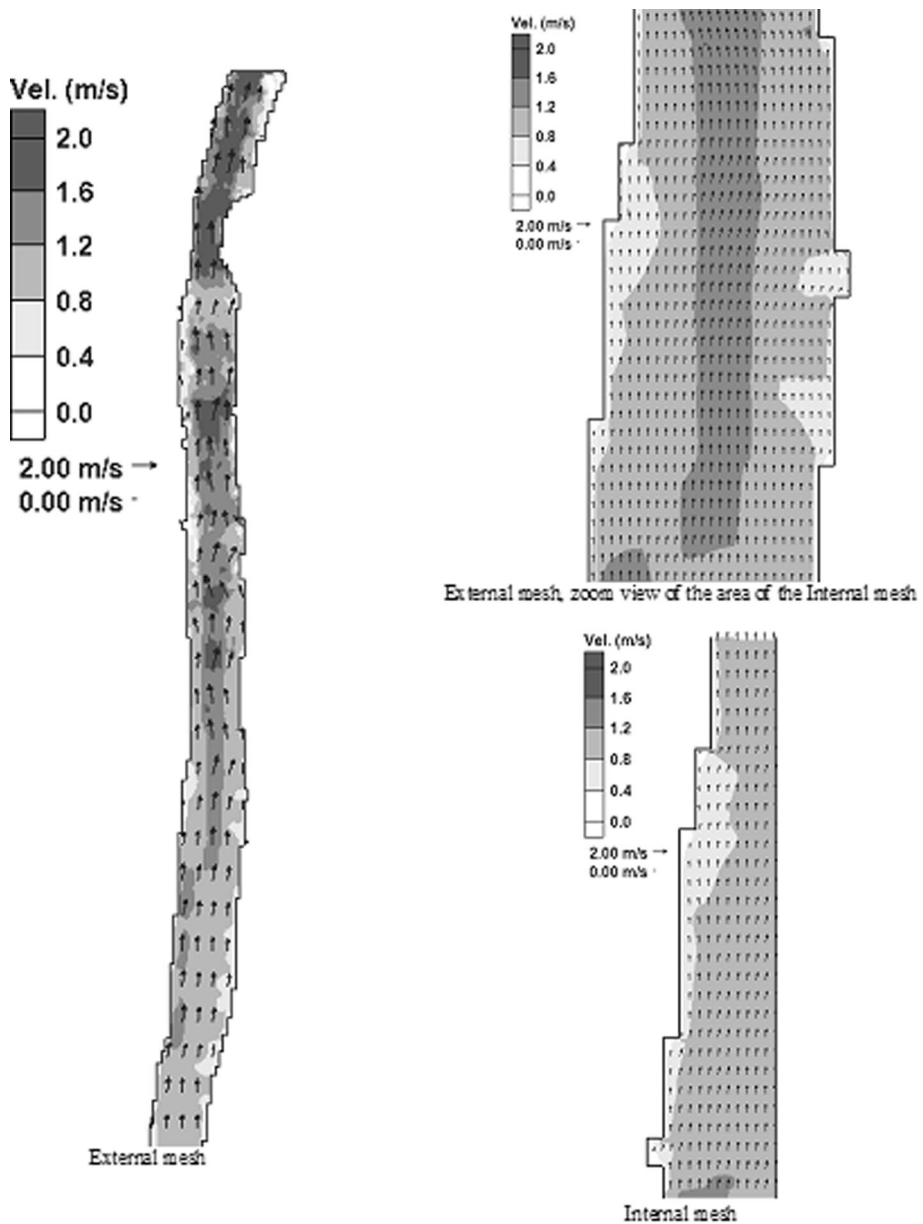


Fig. 8 Velocity fields obtained from the hydraulic modelling. Calibration with spring flow

Figures 12 and 13 show the results of the velocity profiles in the transects established obtained from the hydrodynamic modelling, compared with the field data. The modelled velocity fields for External mesh and Internal Embedded mesh are shown in Fig. 14. Figures 15 and 16 show the time variation of the chloride concentration obtained from the transport model compared with the field data. Finally, Fig. 17 shows the concentration

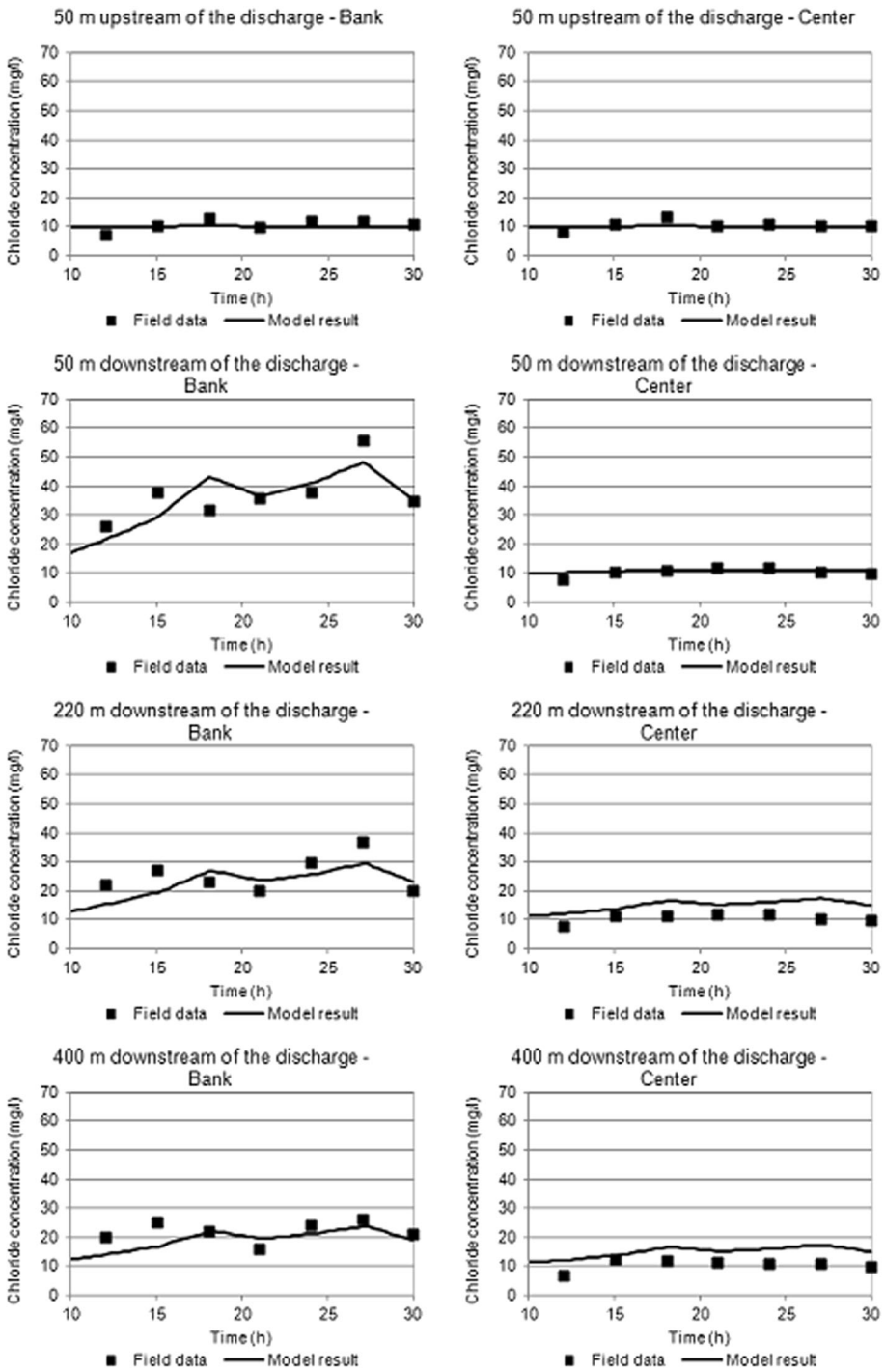
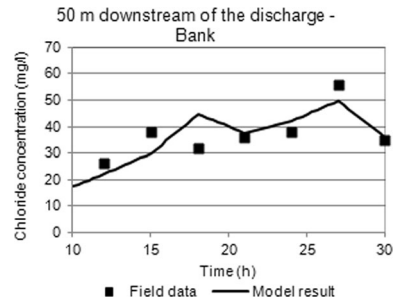


Fig. 9 Temporal evolution of chloride concentration. Model results versus field data. External mesh. Calibration with spring flow

Fig. 10 Temporal evolution of chloride concentration. Model results versus field data. Embedded mesh (concentrations at the bank of transect 2). Calibration with spring flow



fields obtained from the modelling, for External and Embedded mesh, after 27 h simulated. Good agreement between the results obtained from the hydrodynamic and transport models and the field data is observed. Therefore, the model is considered validated to the representation of the mixing zone of the effluent.

3.5 Analysis of the suitability of the model used

In order to determine how much suitable is the Embedded mesh system used on present paper to perform the modelling of the mixing zone, an analysis is made next consisting of three issues: In first place, the experience of using the embedded mesh system with different grid sizes is exposed. In second place, it is presented an analysis using only the External mesh for the modelling. Finally, a comparison with the use of a well known and widely used discharge model system (Cormix) is made.

3.5.1 Analysis with different grid sizes for the embedded mesh system

In general terms, the main advantage of specifying smaller grid sizes consist of allowing more detail on the analysis of the turbulent hydrodynamic processes as well as the better representation of the topography of the stretch of the river, leading to obtain more accurate modelling results, within the capabilities of every specific model. The great disadvantage is the increase of the runtime of the model due to both, the greater number of cells that needs to be created, which implies more calculation effort; and the smaller time discretization that is needed to fulfil stability conditions with smaller cell sizes, which implies more calculation cycles. On the other hand, specifying bigger grid sizes decreases the runtime of the model, but the lesser accuracy can be expected for the results.

In the present paper, the cell size for the Internal Embedded mesh was established in order to have the discharge channel width represented by several cells ($3.5 \text{ m}/0.5 \text{ m} = 7$ cells representing the channel width) as interest is given on properly simulate the hydrodynamic of the area surrounding the jet discharge. Cell size established for External mesh allows a good representation of the cross sections where data field were measured. In order to analyze the behaviour of the model under different cells size combinations, additional tests have been made with the following configurations (always using squared cells where $\Delta x = \Delta y$):

- Configurations decreasing cell sizes:

$$\Delta x_E = 1 \text{ m} \quad \Delta x_I = 0.5 \text{ m}$$

$$\Delta x_E = 2 \text{ m} \quad \Delta x_I = 0.25 \text{ m}$$

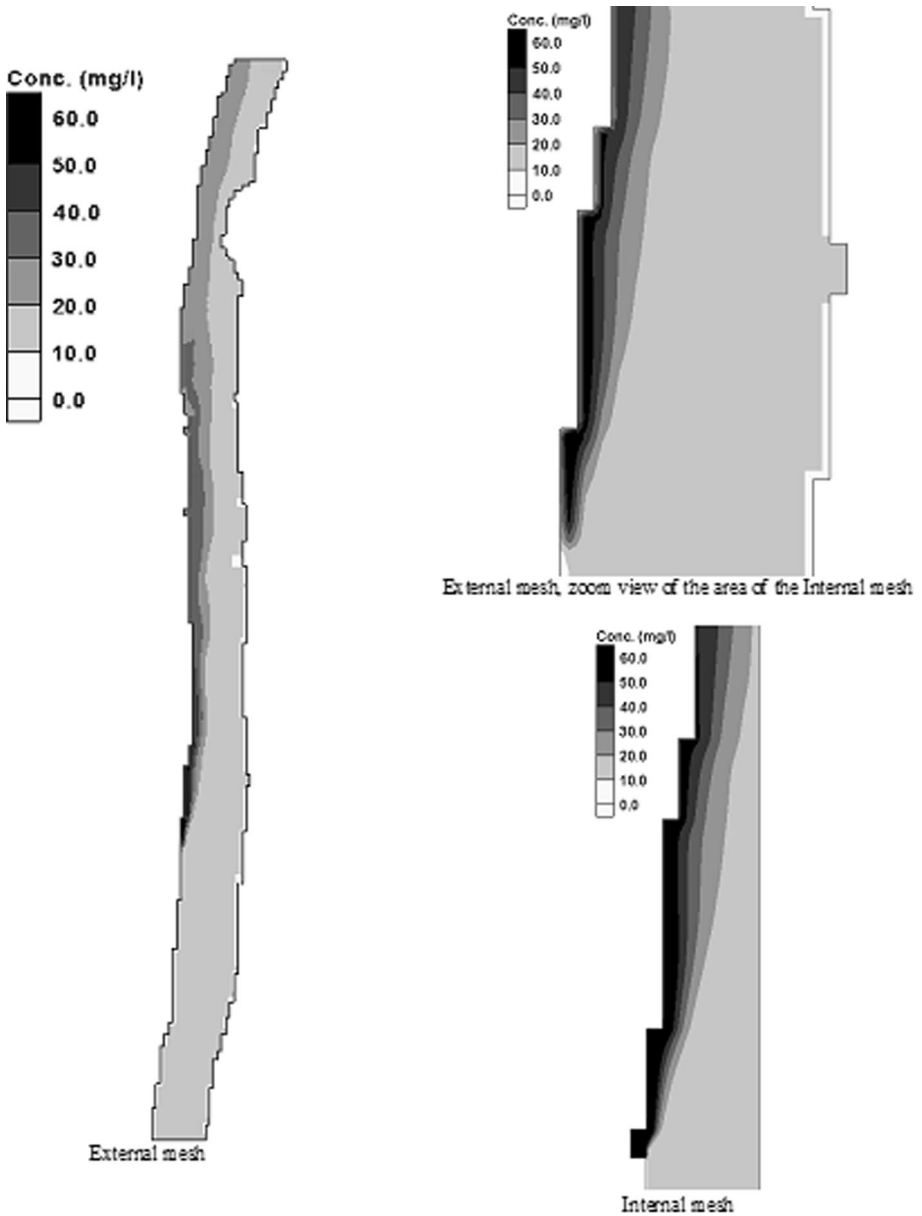


Fig. 11 Pollutant concentration fields obtained from the transport modelling after 27 h simulated. Calibration with spring flow

– Configurations increasing cell sizes:

$$\Delta x_E = 2 \text{ m } \Delta x_I = 1 \text{ m}$$

$$\Delta x_E = 4 \text{ m } \Delta x_I = 0.5 \text{ m}$$

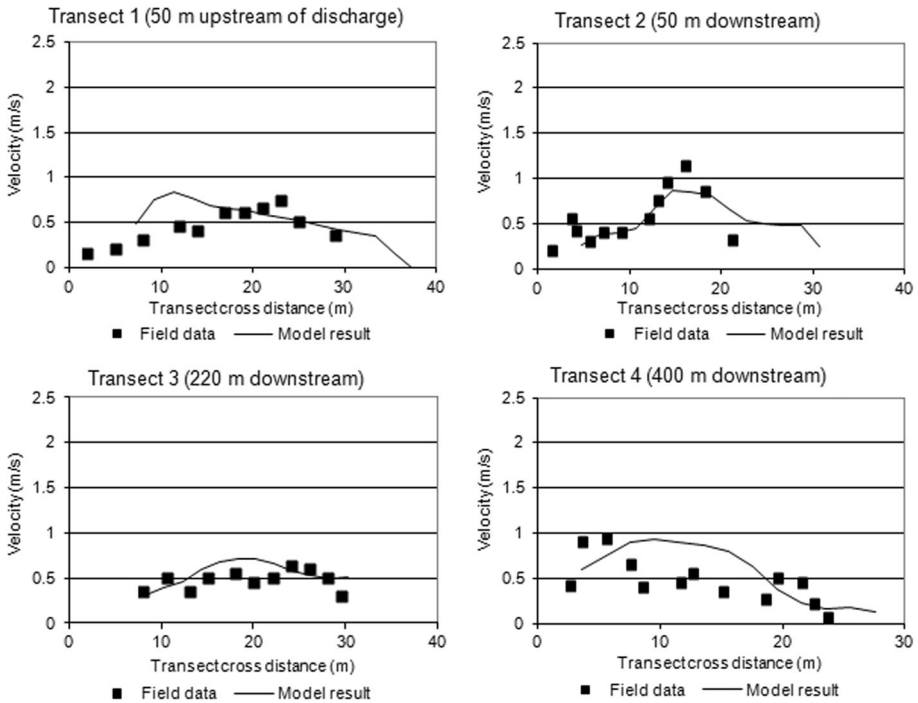
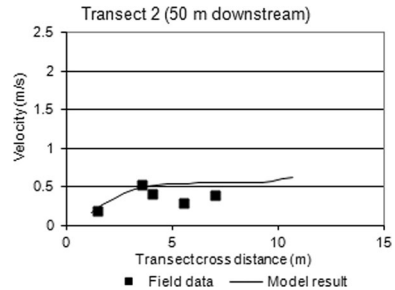


Fig. 12 Comparison of modelled velocity profiles versus field data measured in transects of the Saja River. Validation with summer flow

Fig. 13 Comparison of modelled velocities versus field data along transect 2 in the Internal Embedded mesh. Validation with summer flow



Spring flow conditions were used on the tests. Figure 18 compares for these configurations, and for the configuration previously established, $\Delta x_E = 2 \text{ m}$ $\Delta x_I = 0.5 \text{ m}$, the mean relative errors (from comparing the model results with the field data) and the run time increase (over the runtime for $\Delta x_E = 2 \text{ m}$ $\Delta x_I = 0.5 \text{ m}$) on the hydrodynamic model resulting from the tests. These mean relative errors were calculated for the velocities obtained from the Internal mesh at the transect 2. As shown in the figure, for present application, the changes on the cell size of the Internal Embedded mesh has greater influence on the error as well as on the runtime of the model than the changes on the cell

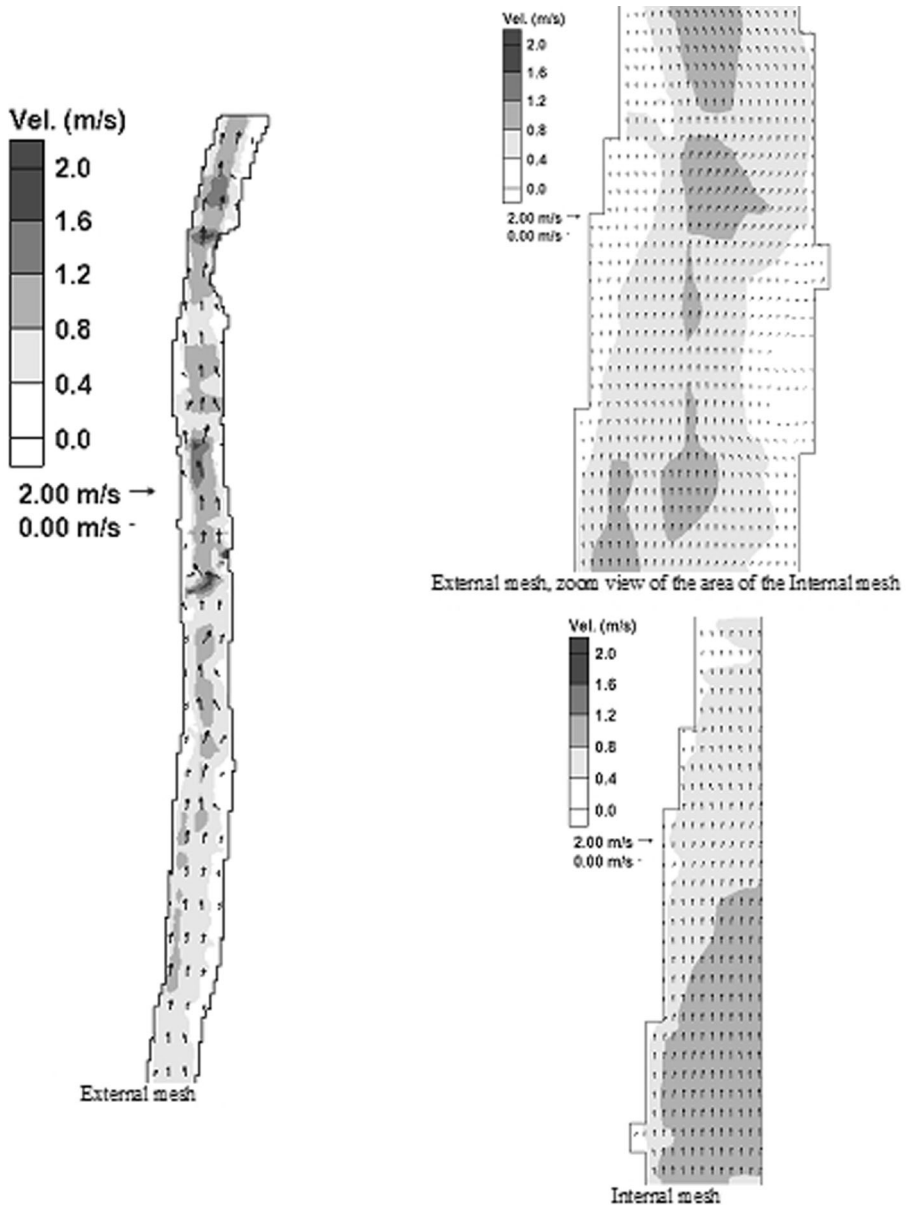


Fig. 14 Velocity fields obtained from the hydraulic modelling. Validation with summer flow

size of the External mesh. For the configurations decreasing cell sizes, it can be observed that the reduction on the error obtained from adopting smaller cell sizes is not significant. Additionally, for the case $\Delta x_E = 2 \text{ m}$ $\Delta x_I = 0.25 \text{ m}$, it is produced a very high run time increase. For the configurations increasing cell sizes it can be observed that the reduction on the runtime of the model is not significant enough to justify the rise in the error. From

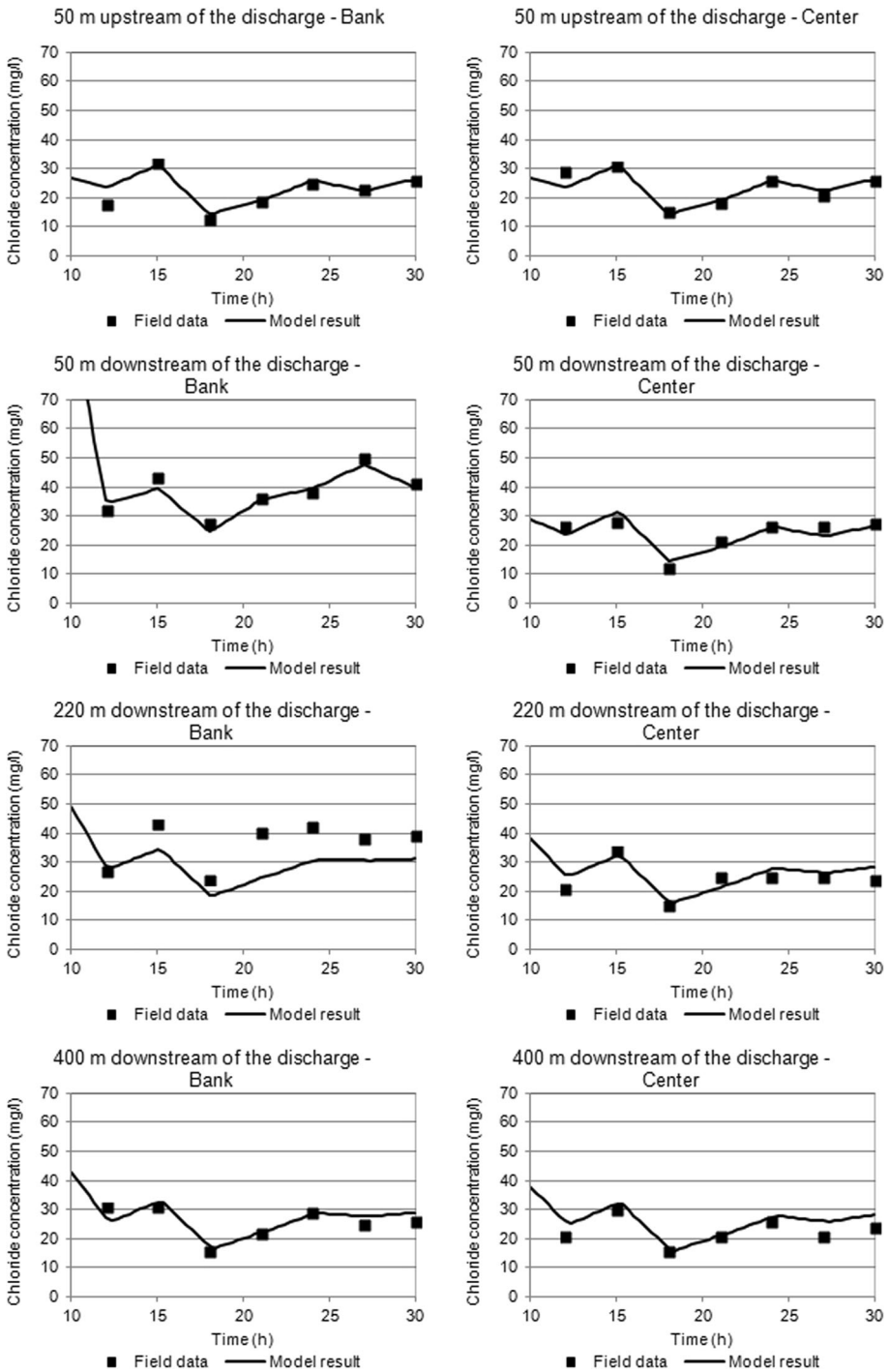
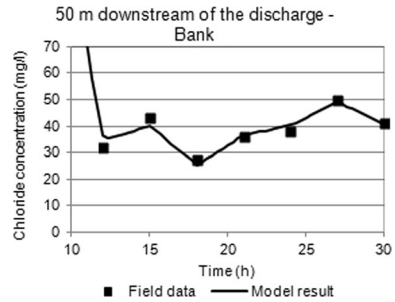


Fig. 15 Temporal evolution of chloride concentration. Model results versus field data. External mesh. Validation with summer flow

Fig. 16 Temporal evolution of chloride concentration. Model results versus field data. Embedded mesh (concentrations at the bank of transect 2). Validation with summer flow



this, the configuration previously established ($\Delta x_E = 2 \text{ m}$ $\Delta x_I = 0.5 \text{ m}$) is considered adequate to the present case.

3.5.2 Analysis using only the External mesh

Embedded mesh system have been used in present paper in order to analyze the processes that occurs at different scales with different meshes. This supposes additional effort in the creating of the algorithm, in the building of the meshes and in the post processing of the results. In order to determine if this effort is worthwhile, tests have been made using only the External mesh. Tests were made using spring flow conditions. In first place, it have been established the discharge of the effluent on the External mesh initially built, with $\Delta x_E = 2 \text{ m}$. This test was expected to require less runtime, but to give less accurate results. In second place, test was made having the External mesh with the cell size initially established for the Internal mesh, $\Delta x_E = 0.5 \text{ m}$. As this cell size worked well for the Internal mesh in simulating the processes occurring on the surrounding areas of the discharge, it is expected from this test to obtain results comparable to the results using the Embedded mesh system, or even more accurate, due to the use of a single mesh eliminates the errors that may be introduced at the information exchange processes between the meshes. Although, as uses a fine cell size for all of the mesh, including the far field away from the discharge point, it is expected that this test requires a greater runtime.

Figure 19 shows the error obtained and the runtime required for these test using only External mesh, compared with the experience of using the Embedded mesh system. For the case using only the External mesh with $\Delta x_E = 2 \text{ m}$ it was found an error much bigger than using the Embedded mesh system. The runtime needed was lesser but it does not justify the error obtained. For the case of the External mesh with fine cell size, $\Delta x_E = 0.5 \text{ m}$, the error found was comparable with the one obtained using the Embedded mesh system, nevertheless it was needed an extremely high amount of runtime, which makes the use of the Embedded mesh system the most suitable option.

3.5.3 Comparison with discharge analytical solution model system (Cormix)

The Cormix is a discharge prediction software, able to predict the dilution and the trajectory of an effluent. The model is commonly used and have been widely documented [37, 38]. Basing on length scale concepts the model classifies the flow within a data base of nearly 80 cases. The model uses a different analytical model for every flow case. Therefore, by using Cormix the results of the model applied on the present paper can be compared with analytical solutions.

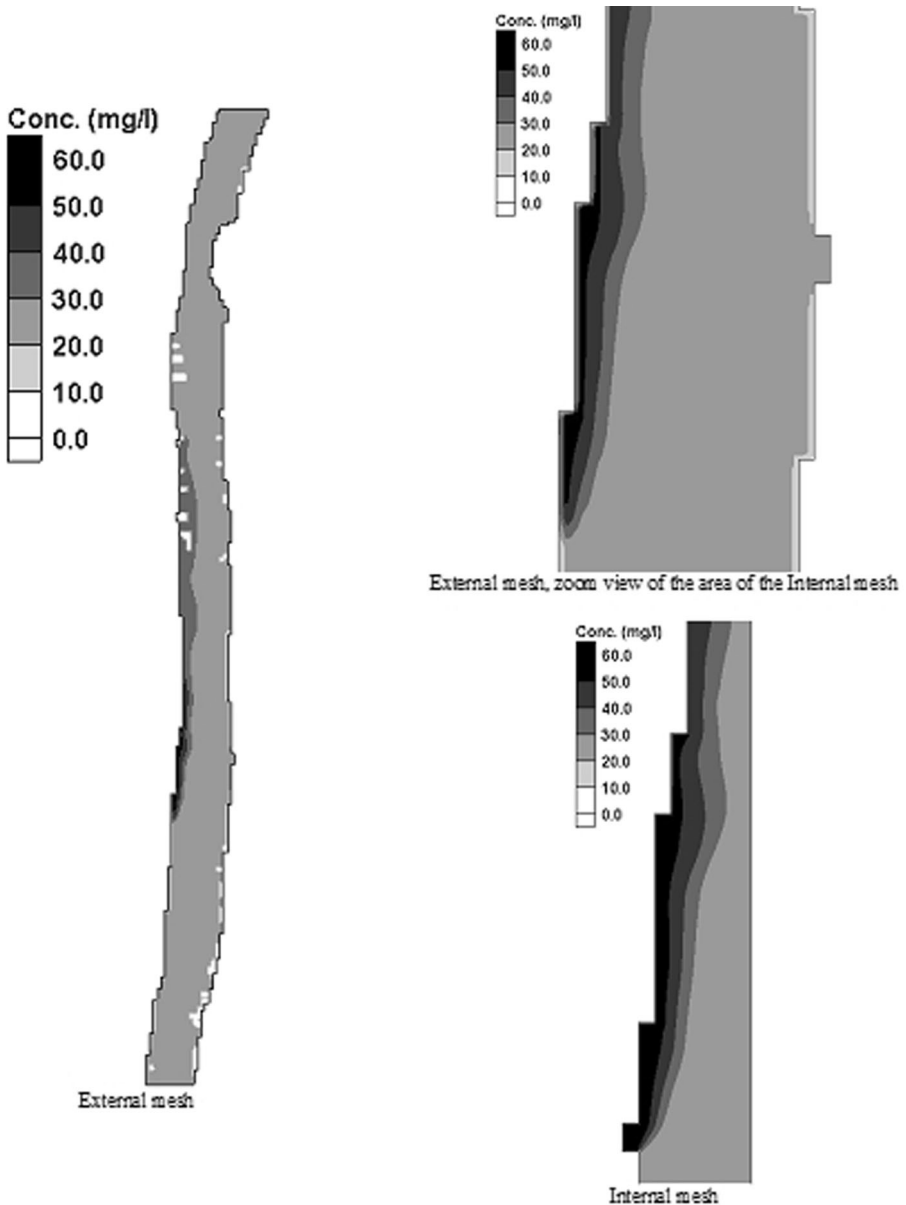


Fig. 17 Pollutant concentration fields obtained from the transport modelling after 27 h simulated. Validation with summer flow

A limitation of Cormix for application in present case is that the variability of the stretch of the river can not be specified. The receiving water body is represented with a constant bottom slope and a constant width. Also, the Cormix is a stationary model, which means that several simulations were needed to be made in order to represent the variation of the concentration on the discharge. The different simulations were set so each one represents

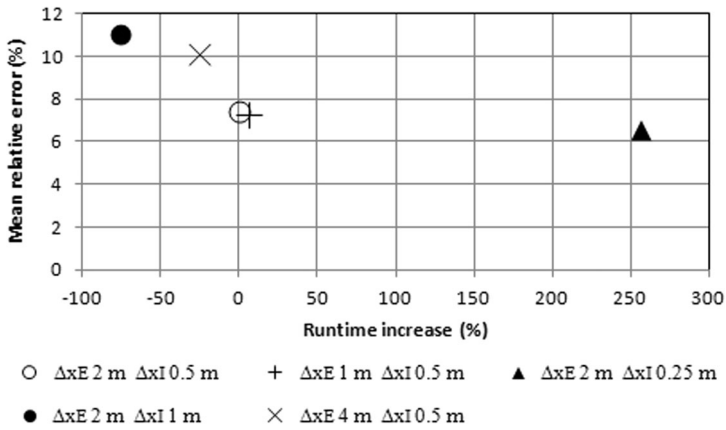


Fig. 18 Comparison of modelling with different cell sizes configurations

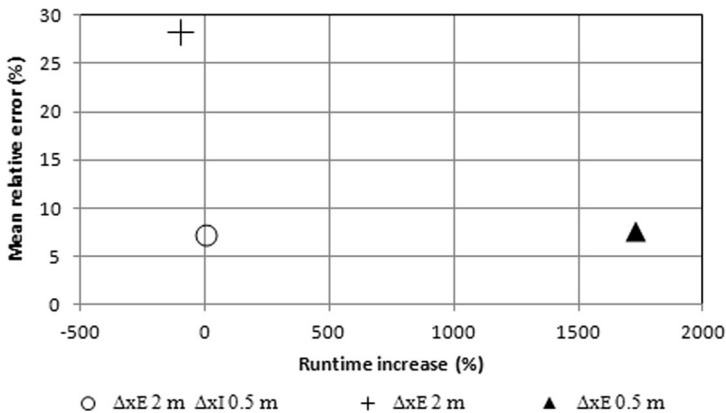


Fig. 19 Comparison of modelling using only the External mesh with the use of the Embedded mesh system

an hour when data field were taken. Modelling with Cormix shows that the flow is mainly a “wall jet/plume” which means that it advances with the river flow downstream attached to the bank, according to what it was seen on the calibration process of the modelling with the Embedded mesh system. Figure 20 shows the results of the modellings for the different hours using Cormix. The figure also shows the data measured on the field, as well as the results obtained from modelling with the Embedded mesh system, for the points taken on the bank. Compared with field data, figures show better results for the Embedded mesh system than for the Cormix, due to the capabilities of the two-dimensional long wave model to represent the flow complex variability, introduced in present case mainly by bathymetry gradients and soft river banks plan irregularities.

3.6 Characterization of the mixing zone of the effluent

With the mathematical model calibrated and validated for the simulation of the evolution of the specific discharge under study, the mixing zone of the effluent can now be

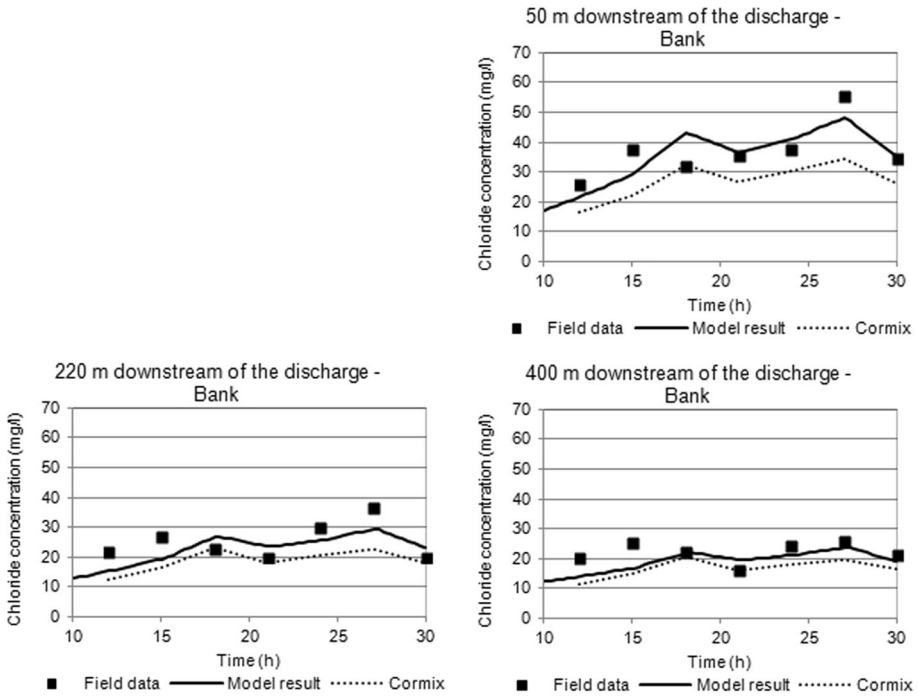


Fig. 20 Comparison between results of analytical solution (Cormix model) with the use of the Embedded mesh system

characterized. The main goal of this characterization is to predict the extents of the mixing zone of a pollutant introduced to the receiving water by the effluent, in such way that it can serve as basis for environmental authorities on the decision-making about conceding its authorization.

For this purpose, an effluent is assumed containing a conservative pollutant whose maximum concentration allowed in the river water by the regulations is 0.05 mg/l, and that the discharge concentration of this pollutant in the effluent is 1 mg/l. The discharge flow is assumed as 0.138 m³/s coinciding with the mean flow registered in the WWTP. No pollutant source is considered upstream of the discharge, so background pollutant concentration in the receiving water is assumed 0 mg/l.

The flow of the river has an important role in the mixing zone developing, and therefore the analysis should take into account its statistical variation. The flow regime of the Saja River is obtained from the document “Study of water resources of the rivers of the northern

Table 3 Daily mean flow in Saja River, for the area of the Casar de Periedo WWTP discharge

Non-exceedance probability (%)	Daily mean flow (m ³ /s)
10	1.240
30	2.618
50	4.653
70	10.360
90	23.631

watershed of Cantabria. Saja y Besaya River Basin” (*Estudio de los recursos hídricos de los ríos de la vertiente norte de Cantabria. Cuenca del Río Saja y Besaya*) presented by the Environmental Hydraulics Institute of the Cantabria University in 2008. From this document it has been extracted the daily mean flow shown in Table 3, related to its non-exceedance probability. As the critical extensions of the mixing zone are presented for low river flow values, the analysis includes the flows associated to a non-exceedance probability of 10 and 30 % (hereafter, Q10 and Q30 %). Also, the flow of a 50 % non-exceedance probability (from now on, Q50 %) is analysed in order to characterize the average conditions of the mixing zone.

The embedded mesh system built, with the parameters established in the calibration and validation process, was used to model the effluent discharge evolution under these river flows conditions. The characteristics of the effluent (flow and pollutant concentration) are considered stationary and therefore, the modellings were performed during the necessary time until stable conditions were achieved. The resulting pollutant concentrations were extracted on 6 points of the Embedded mesh and on 12 points of the External mesh, all of them located close to the river bank, where higher concentrations were found during the calibration and validation process, in order to evaluate the longitudinal extent of the mixing zones obtained. In the Embedded mesh these points were located at 5, 10, 20, 30, 45 and 60 m downstream from the effluent discharge point. In the External mesh these points were located from 80 m to 300 m downstream of the discharge point, spaced every 20 m.

Figure 21 shows the variation of the pollutant concentration with the distance downstream the discharge point, for the different river flows analysed. It can be observed that for a given distance, as the river discharge rises the pollutant concentration diminishes, resulting in higher dilution. Taking into account the allowable pollutant concentration limit, assumed as 0.05 mg/l, from these results it can be deduced that the maximum longitudinal extension of the mixing zone is 121.67 m for Q10 %, 105.35 m for Q30 % and 80.45 m for Q50 %.

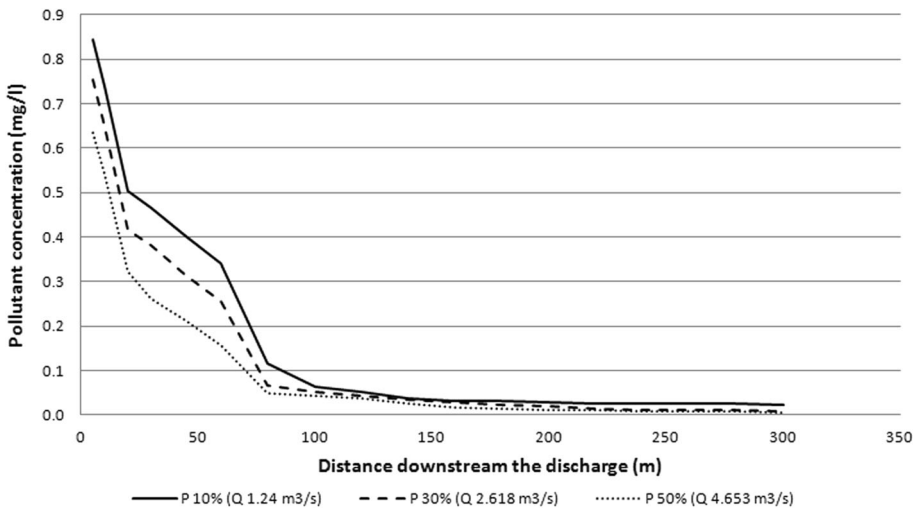
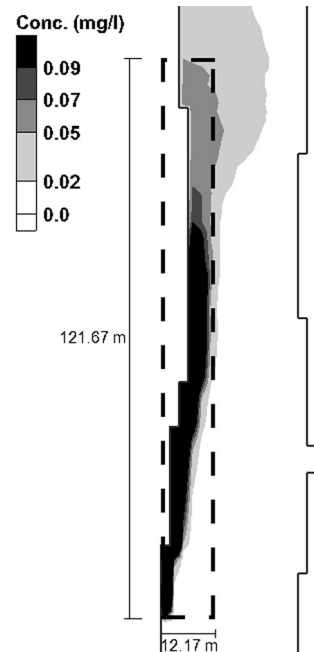


Fig. 21 Variation of the pollutant concentration with the distance to the discharge point for the modelled flows

Table 4 Length of the mixing zone in the transverse direction for each section

Distance downstream the discharge point (m)	Distance to allowable concentration limit (m)			Mixing zone width (m) ^a		
	Q10 %	Q30 %	Q50 %	Q10 %	Q30 %	Q50 %
5	3.38	3.09	2.87	3.38	3.09	2.87
10	4.52	3.74	3.01	4.52	3.74	3.01
20	4.76	4.32	3.56	6.76	6.32	5.56
30	5.49	4.91	4.13	7.49	6.91	6.13
40	5.97	5.10	4.29	9.97	9.10	8.29
60	6.17	5.20	4.39	12.17	11.20	10.39
80	5.55	3.55	1.76	11.55	9.55	7.76
100	3.57	2.36	–	9.57	8.36	–
120	1.82	–	–	5.82	–	–

^a Mixing zone width referred to the transversal distance to the effluent discharge point

Fig. 22 Mixing zone calculated for the discharge. Q10 %

Once longitudinal extensions have been obtained, the widths of the mixing zones resulting from the model were measured in different cross sections of the stretch of the river under study. For this purpose, there were extracted from each cross section the transversal distances at which the pollutant concentration is diluted to a value lower than the assumed allowable limit. Table 4 shows the distances downstream the discharge point at which these cross sections were located, and the mixing zone width found for each one of the river flow analysed. As the river bank geometry is variable, the width of the mixing

zone is referred to the transversal distance to the effluent discharge point. The maximum mixing zone width resulting from the modellings are 12.17 m for Q10 %, 11.2 m for Q30 % and 10.39 m for Q50 %.

These longitudinal and transversal extensions define the predicted extents of the mixing zone of the effluent for each river flow considered. The maximum extension of the mixing zone was found for the Q10 %. This result was expected, since this presents the most critical river flow conditions. Figure 22 shows the calculated mixing zone for the Q10 %.

4 Conclusions

The present paper proposes a methodology that can be followed when the prediction of the extents of the mixing zone produced by an effluent discharged on a river is needed. The prediction of the extents of the mixing zone is helpful for the environmental authorities as it serves as basis for decision-making on determining if a pollutant discharge on a natural water body can be accepted.

Mathematical modelling is needed for the proposed methodology in order to make possible the analysis of the influence of many different factors on the evolution of the mixing zone, such as spatial variations in the geomorphology and in the bathymetry of the receiving body, temporal variations on discharge conditions (flow and pollutant concentrations), characteristics of the pollutants introduced, etc. The hydrodynamic turbulent mixing processes of the near field and the far field occur at different spatial and time scales; therefore, the methodology indicates that the mathematical model needs to be able of modelling both fields jointly.

The proposed methodology establishes the need to obtain field data and statistical data. Field data is needed for the calibration and validation processes of the model. Statistical data is needed for determining the critical cases for the prediction of the extents of the mixing zone.

The application process of the proposed methodology is illustrated for a real effluent case, discharged on a natural river. The characterization of the mixing zone was achieved by using a two-dimensional long wave model and an advection–dispersion model. In order to analyse the near field and the far field jointly it was implemented an Embedded mesh system. An External mesh was built to represent the total stretch considered, and an Embedded mesh was built to represent the surroundings of the discharge point. The use of the Embedded mesh system generated the necessity of establishing conditions for the information exchange between the Embedded mesh and the External mesh, for the hydrodynamic model as well as for the transport model.

The calibration parameter of the hydrodynamic model was the eddy viscosity value for the turbulence representation. For the transport model the calibration parameter was the diffusion coefficient. These parameters were calibrated by comparing the modelling results with hydrodynamic and concentration data measured in the field for a specific date in spring season. As scales of the processes are different on the near field and on the far field, the calibration parameters received different values for the Embedded mesh and for the External mesh. After this, the mathematical model was validated by comparing the modelling results, using the calibration parameters obtained, with field data measured for a specific date in summer season. The evolution of the pollutant in the receiving water presented the expected pattern according to the Coanda effect, as it advanced downstream attached to the left river bank.

For academic purposes, the suitability of the Embedded mesh system used for the modelling have been analysed. The analysis was performed by modelling with different grid sizes, for the External mesh as well as for the Internal mesh. Also, the use of only the External mesh with different grid sizes was tested. Finally, the model results were compared with analytical solutions with the help of the Cormix model. From the analysis with different cell sizes it results that a coarser spatial discretization diminishes the accuracy of the results. A finer spatial discretization produces results with similar accuracy, but requires more runtime for the modelling. In a similar way, the analysis of the modelling using only the External mesh have shown that for obtaining results that represent adequately the data measured on the field it is needed a spatial resolution of the order of the one used for the Embedded mesh, leading to very high increase of the runtime. Finally, the comparison with results from Cormix model, which are intended to offer a comparison to analytical solutions, it have been highlighted the benefits of using a model that includes the geometrical variability of the receiving water body. As result of the different analysis made, it have been concluded that the model presented on present paper is adequate for simulate the behaviour of the discharge in the river.

Once the mathematical model was calibrated and validated, it could be used to predict the evolution of the mixing zone of a pollutant introduced to the river. For this process it was necessary the evaluation of the influence of the statistical hydrodynamic conditions on the river during the useful life of the effluent. For this reason, the analysis of the extension of the mixing zone was performed using the flow regime in the river, extracted from a local statistical data base. Applying the methodology proposed in the present paper to this specific effluent, the extension of the mixing zone was obtained for three different river flows taken as critical cases, according to their non-exceedance probability. As it was expected, the maximum extension of the mixing zone was found for the lowest river flow condition analyzed.

Acknowledgments Alonso J. Rodríguez Benítez is an Agencia Española de Cooperación Internacional MAEC-AECI Scholarship Beneficiary.

References

1. Hall T, Fisher R, Rodgers J, Minshall G, Landis W, Kovacs T, Firth B, Dube M, Deardoff T, Borton D (2008) A long-term, multitrophic level study to assess pulp and paper mill effluent effects on aquatic communities in four US receiving waters: background and status. *Integrated Environmental Assessment and Management* 5(2):189–198
2. Kumar A, Tewary B, Banerjee M, Ahmad M (2010) A simplified approach for removal of suspended coal fines from black water discharge of mining and its allied industries. *Journal of Mines Metals Fuels* 58(11–12):346–348
3. David A, Tournoud M, Perrin J, Rosain D, Rodier C, Salles C, Bancon-Montigny C, Picot B (2012) Spatial and temporal trends in water quality in a Mediterranean temporary river impacted by sewage effluents. *Environmental Monitoring Assessment* 185(3):2517–2534
4. Salomons W (1995) Environmental impact of metals derived from mining activities: processes, predictions, prevention. *Environmental Journal of Geochemical Exploration* 25(1):5–23
5. Cailleaud K, Michalec F, Forget-Leray J, Budzinski H, Hwang J, Schmitt F, Souissi S (2011) Changes in the swimming behavior of *Eurytemora affinis* (Copepoda, Calanoida) in response to a sub-lethal exposure to nonylphenols. *Aquat Toxicol* 102(3–4):228–231
6. Lung W (1995) Mixing-zone modeling for toxic waste-load allocations. *J Environ Eng* 121(11):839–842
7. Alvarez C, Revilla JA, García A, Juanes JA, Medina R (2004) Design and operation of an industrial high-density waste water disposal system: The Usgo submarine outfall (Spain). 3rd International

- conference on marine waste water disposal and marine environment. 1st International Exhibition on Materials Equipment and Services for Coastal WWTP Outfalls and Sealines, Catania
8. Xu J, Lee J, Yin K, Liu H, Harrison P (2011) Environmental response to sewage treatment strategies: Hong Kong's experience in long term water quality monitoring. *Mar Pollut Bull* 62(11):2275–2287
 9. Biron P, Ramamurthy A, Han S (2004) Three-dimensional numerical modeling of mixing at river confluences. *Journal of Hydraulic Engineering* 130(3):243–253
 10. Laraque A, Loup G, Filizola M (2009) Mixing processes in the Amazon River at the confluences of the Negro and Solimões Rivers, Encontro das Águas, Manaus, Brazil. *Hydrol Process* 23(22):3131–3140
 11. Fetterolf C (1973) Mixing zone concepts. Biological methods for the assessment of water quality. American Society for Testing and Materials, Philadelphia, pp 31–45
 12. Boxall J, Guymr I (2007) Longitudinal mixing in meandering channels: new experimental data set and verification of a predictive technique. *Water Res* 41(2):341–354
 13. Martin J (1997) Ingeniería fluvial. Ed. Universidad Politécnica de Cataluña, Barcelona
 14. Castanedo S (2000) Desarrollo de un modelo hidrodinámico tridimensional para el estudio de la propagación de ondas largas en estuarios y zonas someras. Tesis doctoral. Universidad de Cantabria
 15. Einav R, Harussi K, Perry D (2002) The footprint of the desalination processes on the environment. *Desalination* 152(1–3):141–154
 16. Honji H, Matsunaga N, Sugihara Y, Sakai K (1995) Experimental observation of internal symmetric solitary waves in a two-layer fluid. *Fluid Dyn Res* 15(2):89–102
 17. Chernykh G, Zudin A (2005) Linear and nonlinear numerical models of local density perturbation dynamics in a stable stratified medium. *Russian Journal of Numerical Analysis Mathematical Modelling* 20(6):513–534
 18. Jirka G (2008) Improved discharge configurations for brine effluents from desalination plants. *Journal of Hydraulic Engineering* 134(1):116–120
 19. Lalli F, Bruschi A, Lama R, Liberti L, Mandrone S, Pesarino V (2010) Coanda effect in coastal flows. *Coast Eng* 57(3):278–289
 20. Ki S, Hwang J, Kang J, Kim J (2009) An analytical model for non-conservative pollutants mixing in the surf zone. *Water Science Technology* 59(11):2117–2124
 21. Sinton L, Hall C, Lynch P, Davies-Colley J (2001) Sunlight inactivation of fecal indicator bacteria and bacteriophages from waste stabilization pond effluent in fresh and saline waters. *Appl Environ Microbiol* 68(3):1122–1131
 22. Schnurbusch S (2000) A mixing zone guidance document prepared for the Oregon Department of Environmental Quality. Master of Environmental Management Thesis, Portland State University
 23. Jirka G, Bleninger T, Burrows R, Larsen T (2004) Environmental quality standards in the EC-Water Framework Directive: consequences for water pollution control for point sources. European Water Management Online. European Water Association (EWA). http://www.ewaonline.de/journal/2004_01h.pdf
 24. Proni J, Huang H, Dammann W (1994) Initial dilution of southeast Florida ocean outfalls. *Journal of Hydraulic Engineering* 120(12):1409–1425
 25. Huang H, Fergen R, Proni J, Tsai J (1998) Initial dilution equations for buoyancy-dominated jets in current. *Journal of Hydraulic Engineering* 124(1):105–108
 26. Chin D (1985) Outfall dilution: the role of a far-field model. *J Environ Eng* 111(4):473–486
 27. García A, Juanes JA, Álvarez C, Revilla JA, Medina R (2010) Assessment of the response of a shallow macrotidal estuary to changes in hydrological and wastewater inputs through numerical modeling. *Ecol Model* 221:1194–1208
 28. Bárcena J, García A, García J, Álvarez C, Revilla J (2012) Surface analysis of free surface and velocity to changes in river flow and tidal amplitude on a shallow mesotidal estuary: an application in Suances Estuary (Northern Spain). *J Hydrol* 420–421:301–318
 29. Alvarez C, Juanes J, Revilla J, García A (2006) A two dimensional hydrodynamic model for the study of instream flows in rivers. 7th International conference on hydroinformatics, HIC 2006, Nice, France
 30. Pérez E (2010) Methodological approach for the establishment of the mixing zone of the discharge of the wastewater treatment plant of Casar de Periedo in the Saja River. M.Sc. in environmental management of hydraulic systems. Universidad de Cantabria, Santander, p 67
 31. Soriano T (2002) Modelado matemático de la evolución de contaminantes en sistemas fluviales. Tesis doctoral. Universidad de Cantabria
 32. Koutitas C (1983) Elements of computational hydraulics. Pentech Press Limited, Londres
 33. García A (2004) Desarrollo de un modelo tridimensional para la determinación del transporte de sustancias en estuarios y zonas someras. Tesis doctoral, Universidad de Cantabria

34. Sámano ML, Bárcena JF, García A, Gómez AG, Álvarez C, Revilla JA (2012) Flushing time as a descriptor for heavily modified water bodies classification and management: application to the Huelva Harbour. *J Environ Manage* 107:37–44
35. Graf WH, Altinakar MS (1998) *Fluvial hydraulics. Flow and transport processes in channels of simple geometry*. Wiley and Sons, Chichester
36. Chin D (2006) *Water quality engineering in natural systems*. Wiley-Interscience, Wiley and Sons Inc, Hoboken
37. Doneker R, Jirka G (1991) Expert systems for mixing-zone analysis and design of pollutant discharges. *Journal of Water Resources Planning and Management* 117(6):679–697
38. Doneker R, Jirka G (2002) Boundary schematization in regulatory mixing zone analysis. *Journal of Water Resources Planning Management* 128:46–56