

The Rance Tidal Power Plant Model

Christophe Cochet, Malo Lambert
Hydro Engineering Centre, EDF
Le-Bourget-du-Lac, FRANCE
christophe.cochet@edf.fr

Abstract— La Rance tidal power plant (in North Brittany, France) has been operating since 1966 and the plant operations can influence the hydrodynamics of the tidal flow (water levels, current velocities), in the estuary as well as on the downstream (sea) side of the plant: flow rates can evolve quickly and reach up to 15 000m³/s, leading to velocities higher than 5m/s.

To better understand the hydrodynamics linked to the plant operations, a TELEMAC model of the area (estuary + local coastline) was developed: the plant operations are closely represented with a specific subroutine used to call input data from the plant industrial optimization software and to impose flow rate conditions up- and down-stream of the plant; local ADCP measurements were used for calibration.

Several scenarios, based on real operational conditions, were tested and analysed with regards to a list of safety aspects (impact on boats, swimmers/divers, fishermen, or flood risks) and criteria based on water levels and velocities (maximum value, temporal and spatial gradients). The scenarios to be presented are:

- Study of current velocities upstream and downstream of the plant when filling and emptying the estuary, and evaluation of the risk for drifting boats;
- Study of water levels in case of rapid variations of flow rate when stopping the plant, and evaluation of the propagating surge impact.

The workflow includes the use of Blue Kenue for setting up the model, of Matlab for post-processing result files and of Qgis for generating maps and visualising results.

I. INTRODUCTION

La Rance Tidal Power Plant (TPP) is located on the Emerald Coast in North-East Brittany, where tide ranges can reach 13.5 meters. Forming a road bridge between Saint-Malo and Dinard since 1966, the plant has an installed capacity of 240MW and generates around 500GWh annually. At every tide cycle, the estuary (20km long, 180Mm³ in volume) is filled and emptied, with the circulation of around 100 Mm³ of water. Flow rates can reach up to 15 000m³/s, and velocities exceed 5m/s. As the estuary is shared by many users, it is important for EDF, operator of the plant, to guarantee the safety of all users, by either adapting the plant operations or by informing the public of dangerous zones or periods of operation.

After 50 years of operation, the plant is undergoing a major refurbishment program, which includes the renovation

of the complete automatic control system running the plant 24 hours per day, and 7 days a week, along with the tides. This renovation has led to the update of operating phases and transitions between phases.

In order to better understand the hydrodynamic behaviour of the estuary and guarantee the safety of all users, a TELEMAC-2D model was developed, calibrated, and used to test different operation scenarios.

The present article will describe how the plant was modelled and how the tested scenarios lead to better knowledge of the local hydrodynamics.

II. INPUT DATA AND MODEL CONSTRUCTION

Bathymetric data and mesh construction

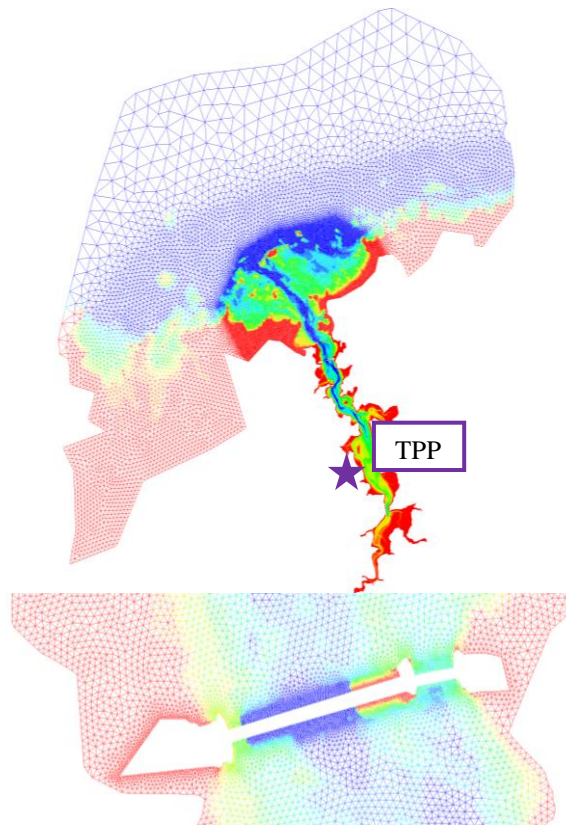


Figure 1. Representation of the model meshed bathymetry and close-up view of the TPP

The bathymetric data used to build the mesh was obtained by a Lidar campaign run in 2011. The mesh has been built using BlueKenue (2D interpolator tool with linear interpolation). The grid size is chosen to be 5 meters in “strategic” areas (near the plant and in the upper (i.e. narrower) part of the estuary) and 20 meters in the rest of the estuary. The mesh grid becomes larger when heading out towards the sea boundary. This leads to a mesh with 201 075 nodes and 393 065 elements (Fig. 1).

Tide data

As sea boundary condition, we use standard TPXO tide conditions ((u,v) velocity components and the water level) from the 2008 European Shelf database, and the corresponding TELEMAC module.

Boundary conditions

The data given on the sea side are the velocity components and the water level so the boundary is of the 5-6-6 type.

The river Rance entering the estuary at Ecluse du Chatelier is modelled with a constant flow rate (boundary 4-5-5). NB: As the flow rate from the river is small in comparison with the flow rate through the plant, a constant value was considered an acceptable assumption.

The plant is modelled as an island in the mesh and its equipment (24 bulb-unit turbines and 6 sluice gates) is represented as weirs (calling modified subroutines LECSNG and CLSING – see §0).

The code was developed using version 6.3 and small adaptations had to be made to the TPXO module in order to make it compatible with the use of weirs. Version 7.1 should have solved this issue although no test was performed.

Workflow

Any required pre-processing (bathymetry correction, plant input data preparation...) is completed using Matlab (MathWorks, R2016b). BlueKenue (CHC) is used for the mesh and boundary files construction. Post-processing of result files is achieved with Matlab scripts before importation and visualisation under Qgis (using Artelia’s PostTelemac Qgis plugin).

Model calibration

ADCP recordings – data from 3 ADCP located close to the plant - were used to calibrate the Strickler coefficient, assumed constant on the domain. The model error (RMSE) obtained was considered satisfying, with values below 10cm for the water levels in the basin and below 20cm/s for the velocities (see Fig. 2 and Tab. 1).

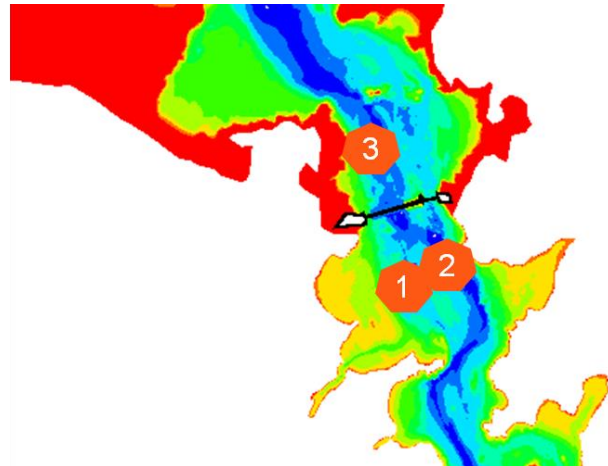


Figure 2. Location of the 3 ADCP used for the model calibration.

RMSE	ADCP 1	ADCP 2	ADCP 3
Water level	0.07 m	0.07 m	0.19 m
Current velocity	0.11 m/s	0.17 m/s	0.18 m/s

TABLE 1. ESTIMATED ROOT MEAN SQUARE ERROR ON WATER LEVELS AND CURRENT VELOCITIES BETWEEN MODEL AND ADCP MEASUREMENTS

III. TECHNICAL ASPECTS FOR MODELLING THE TPP AND ITS SURROUNDINGS.

Plant operations and control

La Rance tidal power plant usually generates electricity when the basin is emptying (although the turbines can also be operated in “reverse” mode); when filling the basin, the turbines are open but not active and the sluice gates are open; once the basin level has reached the sea level, the turbines operate in pump mode to continue filling the basin. This operating mode – pumping at low head – is very efficient as the water pumped will be released later at a much higher head, thus generating much more power.

For the turbines, there are in total 5 operating modes:

- Direct turbinning (generating power with flow from basin to sea),
- Reverse turbinning (generating power with flow from sea to basin),
- Direct pumping (consuming power with flow from sea to basin),
- Direct orifice mode (powerless with flow from basin to sea),
- Reverse orifice mode (powerless with flow from sea to basin).

For the first 3 modes, the flow rate through the turbine is determined by interpolation using the turbines hill chart, the head between basin and sea and the blade incidence, which is computed by the optimisation algorithm and recorded in the predicted operation program (input of LECSNG routine, see hereafter). For the 2 powerless modes, the flow rate is computed using an analytical formula of the form given in (1), where Q is the flow rate, C a discharge coefficient and h_{Basin} and h_{Sea} the water levels.

$$Q = C (h_{Basin} - h_{Sea})^{1/2}. \quad (1)$$

For the sluice gates, the flow rate is controlled by the water levels of the basin and the sea for both flow directions, as well as the downstream water level (sea or basin depending on flow direction); (2) is an example for a flow from sea to basin, where A and B are discharge coefficients.

$$Q = (A \cdot h_{Basin}^{1/4} - B) \cdot (h_{Basin} - h_{Sea})^{1/2}. \quad (2)$$

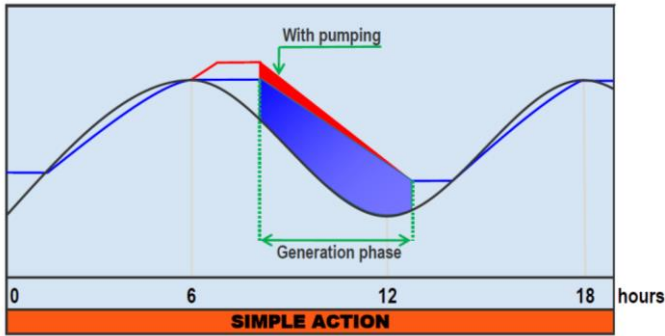


Figure 3. Graphical representation of the water levels (sea in black, basin in blue) during a standard operating cycle

Modelling of plant

LECSNG is a subroutine read only once during the calculation. The subroutine is used to define physically the characteristics of the weirs. In our case, the weirs describe, on one side, the 24 bulb units and, on the other side, the 6 sluice gates. We use a specific input file to define the associated boundary nodes as well as control nodes (i.e. nodes from which the water levels (estuary and sea) are read for the plant control) and some operation parameters (units starting/stopping phase duration, gates opening sequences,...). Also integrated in this subroutine are the turbine performance hill charts (output power and blade incidence as a function of head and flow rate).

A second input file read by LECSNG is the plant operation program, as generated by the optimizing software (based on a 0-D hydraulic model) which is the input data for the plant automated controls system. This file is generated each week depending on predicted tide conditions and plant configuration (number of available units and sluice gates). It gives the operating modes, the number of bulbs and gates available for the operations and the planned water levels.

Using this program file as input allows a very realistic representation of the plant operations.

CLSING uses the data previously loaded by LECSNG in order to simulate the water transfers between the two sides of the weir (turbines and sluice gates). The turbines flow rate is computed using the real turbine hill charts while the sluice gates flow rate is computed based on water levels using an analytical formula (see (2)).

At every time step (30 seconds in the model), the water levels computed by TELEMAC are compared to the

predicted water levels (from the 0-D optimizing software) and the instructions are updated to achieve optimal control of the plant. Depending on the activated mode, the direction, the rate of the flow and the number of units or gates may vary. All these particularities are controlled by CLSING and may change at every time step

The implemented control system allows to test and adjust multiple parameters, including opening sequences of sluice gates or bulb units, in order to control the impact of these parameters on the flow gradient and flow velocities. For example, the gates opening sequence is defined by a 6-element vector indicating time offsets for opening each of the 6 sluice gates. Associated to this vector are the detection of the gates status (closed/opening/open/closing) and the adjustment of the gates flow rate depending on the opening ratio of each gate.

The routines have been coded so that they remain compatible with the parallel computing capabilities offered by TELEMAC-2D.

In particular, the flow rate through the turbines and sluice gates is a function of water levels measured at points located a few hundred meters upstream and downstream of the plant; thus, when the mesh is split into several sub-domains, these nodes may be separated from the plant itself. The solution was to use TELEMAC native functions “*global_to_local_point*”, “*P_DMAX(X)*” and “*P_DMIN(X)*”.

Fig. 4 shows the computation time for a one-week simulation, as a function of the number of processors used: asymptotically, 15 minutes are required to simulate one week.

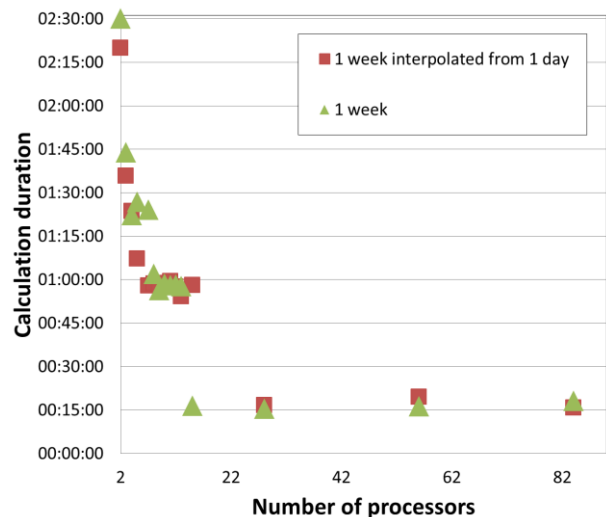


Figure 4. Computation time as a function of the number of processors used

IV. OPERATIONAL SAFETY: GOALS AND RESULTS

One of the goals of the model is to simulate correctly the new operating phases of the power plant in order to evaluate their impact on the local hydrodynamics. We focus on different criteria:

- The instant maximum velocities around the TPP,
- The flow and water level gradients during the transitory phases,
- The drifting time of a non-maneuvring craft.

High current velocities and potential impacts

Some phases can generate high velocity flows around the TPP, with potential impacts to navigators, divers, swimmers, fishermen and moored boats.

The surface velocity was estimated using the model results and a logarithmic law (3) relating the depth-averaged velocity (V_{moy}), the water depth (H) and a bottom friction coefficient (z_0).

$$V(z) = V_{moy} \frac{\ln\left(\frac{z}{z_0}\right)}{\ln\left(\frac{H}{z_0}\right) - 1} \quad (3)$$

Maps of the maximum velocity field for the different phases were generated in Qgis and shared with the plant operator to identify zones/activities potentially at risk.

The example given on Fig. 5 shows that the highest velocities are located inside the protected zone (identified by the safety lines represented in pink). Outside of this zone, the the velocities remain limited and do not exceed the values which were observed before the construction of the plant.

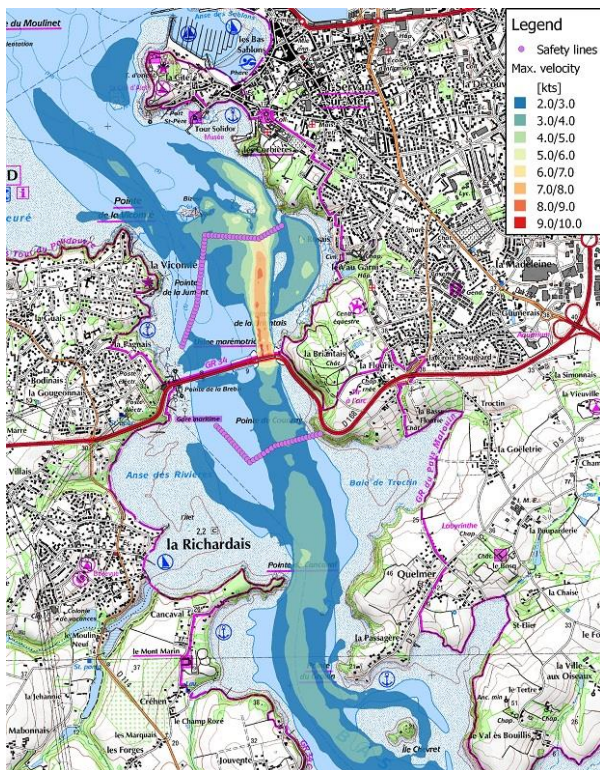


Figure 5. Maximum velocity field when emptying the basin – close up view of the TPP surroundings

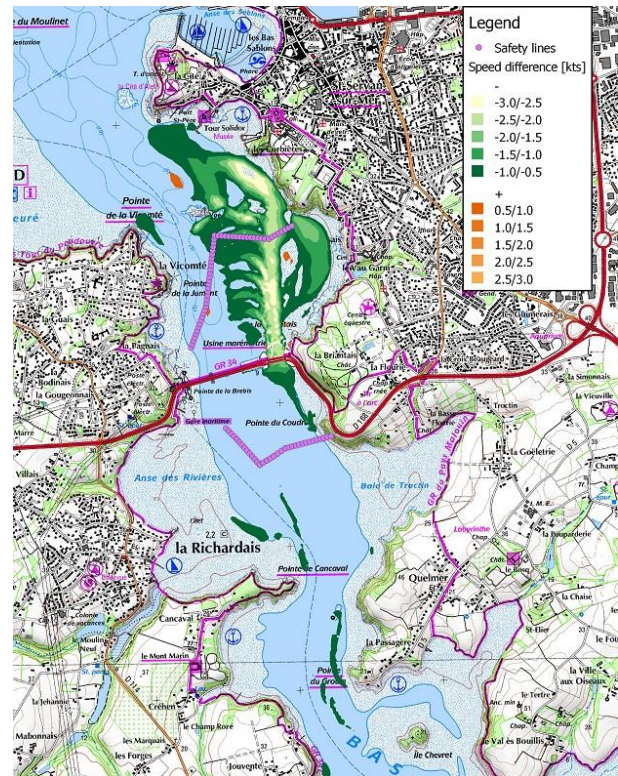


Figure 6. Maximum velocity difference between two sequences: a significant reduction is observed, even outside of the zone protected by the safety line

Rapid turbine stop and potential surge

The stop of the plant turbines in case of emergency can lead to rapid water levels variations, or surge wave, propagating upstream towards the estuary end. These waves are amplified as they progress towards the estuary tail because of the decreasing depth. A 30-cm high wave at the plant can transform into a 150-cm high wave at the Chatelier Lock (end of the estuary); its long period (around 40 min) limits the risks to navigators.

The simulations were compared with good accuracy to real life tests undertaken in the 1960s. Given the complexity of organising such tests, the numerical model is of great help to re-evaluate risk associated to these events.

Influence of sluice gates opening sequences

The opening of the sluice gates – which are 10m high by 15m wide - can lead to high flow gradients. The opening sequence (the gates open successively with a given time offset) can be adjusted to limit the safety risks. Sensitivity studies were realized for the different opening phases/flow directions and for a set of opening sequences by analysing the flow rate, flow rate gradient, basin water level gradient, and velocity field of each scenario. If significant differences can be observed in terms of velocity field (see Fig. 6), it appears that all the sequences tested are compatible with the current regulatory limits (fixed in terms of water level gradient). The results also highlighted the fact that a quicker opening sequence does not always lead to higher maximum velocities: it is sometimes preferable to open all the gates quickly to limit

the head increase (for longer sequences, the basin can fill more slowly than the sea is rising, thus leading to an increasing water level difference).

This study also provided some information on the impact of the sequence choice on the final basin water level (i.e. at the end of the filling phase), which is an indication of how much energy will be extracted during the following turbinng phase.

Drifting times

Navigation on the estuary is subject to caution when the plant is operating as the strong currents can quickly drag a small boat towards the plant. Although there exists a safety line preventing the boats from coming too close to the plant, drifting times – i.e. duration to reach the plant from a given location, for a given operation phase – were evaluated and helped define the right emergency procedures.

A first approach was to use the algae transport module, but its application to a drifting boat was considered unnecessarily complex. Instead we chose the following approach:

- i. compute the trajectory of fluid particles by integrating the local velocity,
- ii. compute the trajectory of a drifting boat (motor or sailing) using the fluid particles trajectory and hydrodynamic and aerodynamic forces (drag) applied on the boat.

These steps were integrated in the post-processing phase and allowed for a more complete analysis: several operating phases and tide conditions were studied without needing to re-run the simulations and the influence of drifting start time(s) and node(s) was tested in order to identify the most critical situations. Fig. 7 represents the “drift duration” map obtained for filling and emptying phases during an average amplitude tide. Wind conditions (constant force and direction) were also included in the parameters; its influence could be studied further by taking into account real conditions and their statistical distributions.

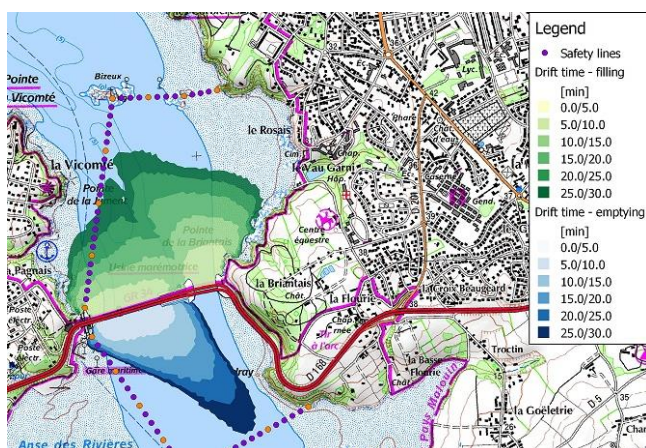


Figure 7. Drifting times for both emptying (blue) and filling (green) phases, for a tide of average amplitude (coefficient 70)

To validate these calculations, a real-life test was undertaken on site: a kayak, equipped with a GPS, was positioned about 700m upstream of the plant during a direct turbinng phase and released in the flow. It was safely recovered after a short drift time. The GPS trajectory was analysed and compared to the theoretical trajectory (from the TELEMAC computation results): Fig. 8 shows the two trajectories and Fig. 9 the estimated drift times. Although the trajectories present slight differences (probably due to local turbulence/wave effects and kayak hull shape), a good correlation – slightly conservative - is observed in terms of “drift time to plant”. More real-life tests are planned to further validate the simulation results.

V. CONCLUSION AND PERSPECTIVES

The model implemented allows a large freedom to simulate virtually any scenario one can imagine. It has already been used to address key safety issues as presented in this article. Further development is now considered, with 2 possible (non-exclusive) perspectives:

- A Sisyphe-coupled model to study the hydro-sediment processes at play in the estuary (work to be coordinated with EDF R&D),
- Implementing and completing a sensitivity analysis on water levels and/or velocities to quantify the model uncertainty (TELEMAC-OpenTurns coupling).

ACKNOWLEDGEMENT

The authors thank Denis Aelbrecht, hydraulics expert at EDF Hydro Engineering Center, for his constant support and wise advice in the studies on tidal power plants, and in particular, on La Rance tidal power plant.

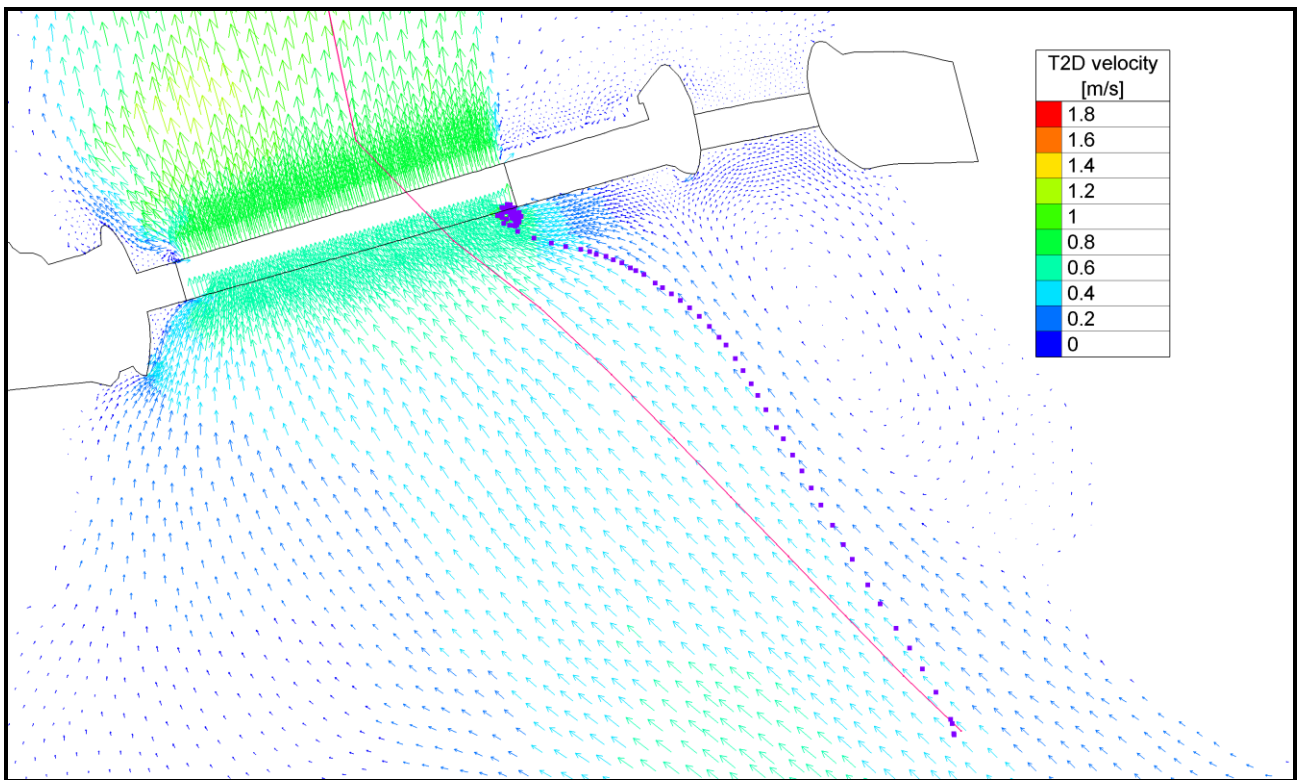


Figure 8. Trajectories of a drifting kayak during the direct turbining phase: real-life test (purple dots) and numerical simulation (magenta line). The flow velocity field is represented as a coloured vector field

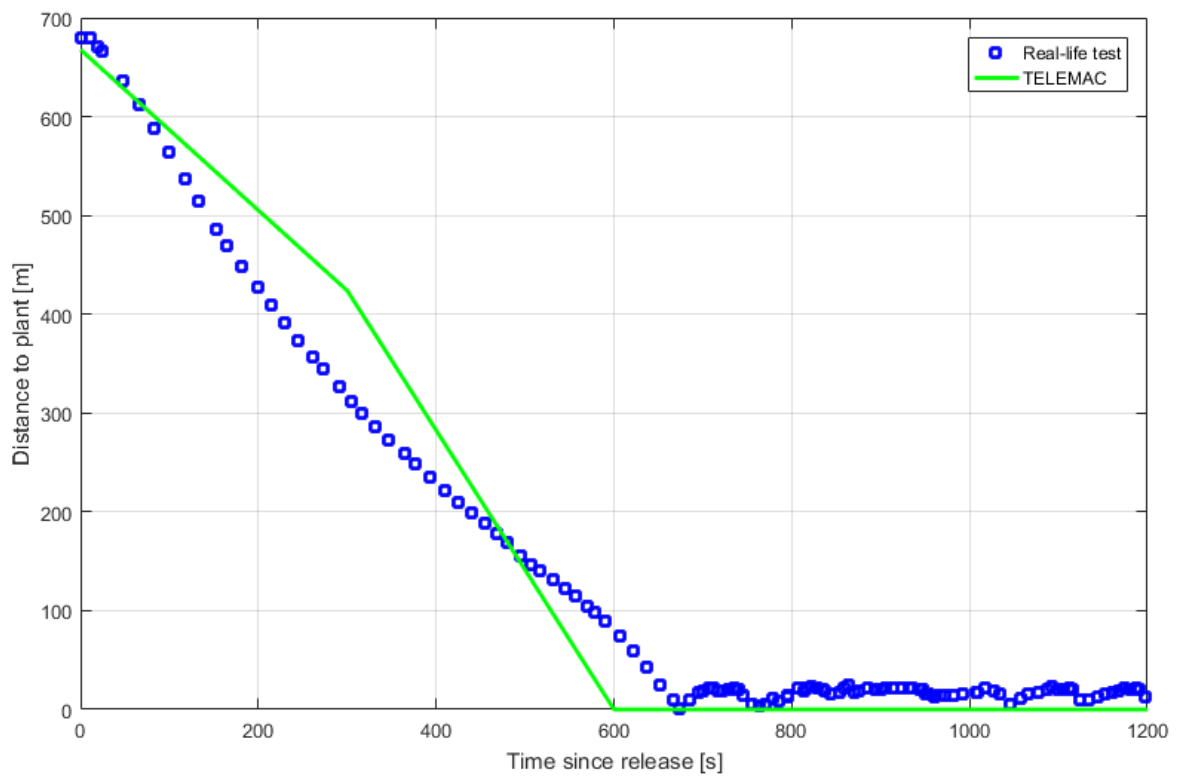


Figure 9. Drift duration of a drifting kayak during the direct turbining phase: real-life test (blue dots) and numerical simulation (green line)