Numerical modelling of oil spill in inland waters

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/104228

Vorgeschlagene Zitierweise/Suggested citation:

Standardnutzungsbedingungen/Terms of Use:
Die Dokumente in HENRY stehen unter der Creative Commons Lizenz CC BY 4.0, sofern keine abweichenden Nutzungsbedingungen getroffen wurden. Damit ist sowohl die kommerzielle Nutzung als auch das Teilen, die Weiterbearbeitung und Speicherung erlaubt. Das Verwenden und das Bearbeiten stehen unter der Bedingung der Namensnennung. Im Einzelfall kann eine restriktivere Lizenz gelten; dann gelten abweichend von den obigen Nutzungsbedingungen die in der dort genannten Lizenz gewährten Nutzungsrechte.

Documents in HENRY are made available under the Creative Commons License CC BY 4.0, if no other license is applicable. Under CC BY 4.0 commercial use and sharing, remixing, transforming, and building upon the material of the work is permitted. In some cases a different, more restrictive license may apply; if applicable the terms of the restrictive license will be binding.
Numerical modelling of oil spill in inland waters

Cédric GOEURY
Saint-Venant Laboratory for Hydraulics and Waves
Joint research unit EDF R&D CEMEF, Ecole des Ponts
Chatou, France
Cedric-externe.goeyury@edf.fr

François THOUVENEL
VEOLIA, Environment Research and Innovation (VERI)
Rueil Malmaison, France

Jean-Michel HERVOUET1,2
1Saint-Venant Laboratory for Hydraulics and Waves
Joint research unit EDF R&D CEMEF, Ecole des Ponts
2EDF R&D, National Laboratory of Hydraulics and Environment (LNHE)
Chatou, France

Isabelle BAUDIN-BIZIEN
VEOLIA, Environment Research and Innovation (VERI)
Rueil Malmaison, France

Abstract—The European Water Framework Directive together with the requirement to monitor water resources for drinking as well as leisure and industrial purposes, have substantially increased the demand for water-quality evaluation and monitoring systems. The Migr’Hycar research project was initiated to provide decisional tools, and to fulfill operational needs, for risks connected to oil spill drifts in continental waters (rivers, lakes, estuaries).

Within the framework of the Migr’Hycar project, a new numerical oil spill model has been developed by combining Lagrangian and Eulerian methods. This model enables to simulate the main processes that act on the spilled oil: advection, diffusion, evaporation, dissolution, spreading and volatilization. Though generally considered as a minor process, dissolution is important from the point of view of toxicity. The Lagrangian model describes the transport of an oil spill near the free surface.

To model dissolved oil in water, an Eulerian advection-diffusion model is used. The fraction of dissolved oil is represented by a passive Eulerian scalar. This model is able to follow dissolved hydrocarbons in the water column (PAH: Polycyclic Aromatic Hydrocarbons). The Eulerian model is coupled with the Lagrangian model. In parallel with model development, two types of experiments on the behavior of hydrocarbons have been carried out:

- Static chemical laboratory experiments in order to study the kinetic of dissolved petroleum in a beaker.
- Dynamic experiments in artificial river facility. After releasing refined commercial products (fuel and heavy oil) into an artificial channel, the aim of these experiments was to study the drift of the oil spill and the dissolution in the water column.

Static experiments allow a calibration of evaporation, dissolution and volatilization mass transfer coefficients used in the model. Then, the model is validated with the artificial river experiments. Comparisons of numerical results with measured data are presented in this paper.

I. INTRODUCTION

Although in almost half of all instances of contamination the exact cause is never determined, oil spills can be due to human error, accidental or voluntary discharge of cargo residues, domestic or industrial tank overflow, leakage from fuel stations, traffic accidents or fire, amongst other causes.

When faced with hydrocarbon contamination of inland waterways, authorities and other organizations can seldom rely on dedicated decision-making tools to intervene in an effective way.

Whereas considerable management and monitoring resources are rapidly deployed for off- and inshore oil incidents, the more frequent occurrence of continental water pollution is dealt with using relatively modest means. A limited grasp of the nature and magnitude of such events often renders both industry and government powerless in controlling their impact.

The Migr’Hycar research project (www.migrhycar.com) was initiated to provide decisional tools, and fulfill operational needs, for risks connected to oil spill drifts in continental waters (rivers, lakes, estuaries). These tools are meant to be used in the decision-making process after an oil spill pollution and/or as reference tools to study scenarios of potential impacts of pollution on a given site. The Migr’hycar consortium has been organized to closely match project objectives and comprises modelling technology developers (EDF, Saint-Venant Laboratory for Hydraulics, VEOLIA), researchers with long-standing experience of hydrocarbon physicochemical behaviour (Agribusiness laboratory LCA, CEDRE), engineering consultants liaising closely with local and regional authorities (SOGREAH), two water intake operators directly concerned with project-related issues and well experienced in applying protective warning systems (EDF, VEOLIA), and a major player in the oil industry (TOTAL). The consortium has therefore the expertise required to develop a surface-water risk monitoring and prevention system against oil spillage contamination.

In this study, a hybrid two-dimensional trajectory and fate model has been developed to simulate the process of advection, turbulent diffusion, evaporation and dissolution in the water column. The developed model has then been applied to simulate an oil spill in the Gironde estuary and in an artificial river.

In section 2 we briefly describe the conceptual model. In section 3 we deal with the main physical phenomena and show
how they are modelled. Finally our first results and validation cases are described in section 4.

II. CONCEPTUAL MODEL


The hybrid oil spill model we introduce here combines an Eulerian and a Lagrangian approach. It is new due to the fact that the Lagrangian model describes the transport of an oil spill near the surface. The oil slick is represented by a large set of small hydrocarbon particles. Each particle is a mixture of hydrocarbons. That is why in this model particles are represented by component categories (PAH, pseudo-components characterized by distillation curves), and the fate of each component is tracked separately.

Each particle has an area, a mass, its element number, its barycentric coordinates within this element, physical-chemical properties of each component, amongst other properties, associated to it. This model allows the main processes that act on the spilled oil: advection, effect of wind, diffusion, evaporation, dissolution, to be simulated. Though generally considered as a minor process, dissolution is important from the point of view of toxicity. To model PAH dissolution in water, a Eulerian advection-diffusion model is used. The fraction of each dissolved PAH is represented by a passive Eulerian scalar and its quantity directly depends on the dissolved mass of particle PAH component.

III. PHYSICAL PROCESSES

When an oil spill occurs, the slick moves due to advection and diffusion phenomena. At the same time, the mass of the oil slick changes because of evaporation and dissolution (Figure 1). Therefore, the fate and transport oil spill processes described in the following paragraph need to be included in the oil spill model.

Figure 1. Fate and Transport oil surface slick processes.

A. Transport processes

1) Advection

a) In 2D: Free surface velocity evaluated

Telemac-2D solves the depth-averaged free surface flow equations. Therefore, in order to calculate the surface slick displacement, we need to evaluate the surface velocity using the depth-averaged velocity. The hypothesis of logarithmic profile for the vertical velocity has been made in order to estimate the surface velocity. In fact, we calculate the averaged velocity by integrating the logarithmic profile. A function of the mean velocity, the surface velocity and the friction velocity is deduced. We specify the friction velocity at the bottom thanks to the friction coefficient. In this way the following relationship for the surface velocity is obtained:

$$ u(h) = \left( u \right) \left( 1 + \frac{1}{\kappa} \sqrt{\frac{C_f}{2}} \right) $$

where $u(h)$ is the surface velocity, $\langle u \rangle$ is the depth-averaged velocity, $\kappa$ is the Karman constant, $C_f$ is the friction coefficient.

b) Wind effect on the oil spill drift.

We consider a float which moves due to wind and current. A solid body submerged in a fluid which moves with constant velocity is subjected to the following force:

$$ F = \frac{1}{2} \rho S C_d \left| \mathbf{u} \right| $$

where $\mathbf{u}$ is the fluid velocity, $S$ is the surface of the solid, $\rho$ is the density, $C_d$ is the drag coefficient. If the solid moves with velocity $\mathbf{v}$, it is necessary to replace the vector $\mathbf{u}$ by the vector $\mathbf{v} - \mathbf{u}$.

Figure 2. Forces acting on a float.

The float is subjected to wind and current (Figure 2). At steady state, Newton’s second law allows us to write the following relationship:

$$ \rho_a S C_{d,a} \left| \mathbf{v} - \mathbf{u}_a \right| (\mathbf{v} - \mathbf{u}_a) + \rho_w S C_{d,w} \left| \mathbf{v} - \mathbf{u}_w \right| (\mathbf{v} - \mathbf{u}_w) = 0 $$

where $\mathbf{v}$ is the body velocity, $\rho_a$ is the air density, $\rho_w$ is the water density, $C_{d,a}$ is the drag coefficient in the air, $C_{d,w}$ is the drag coefficient in the wind, $S_a$ is the area of the solid in the air, $S_w$ is the surface of the solid in the water, $\mathbf{u}_a$ is the current velocity, $\mathbf{u}_w$ is the wind velocity.

If the float velocity $\mathbf{v}$ is expressed in a basis formed by $\mathbf{u}_a$ and $\mathbf{u}_w$ which are known, a solvable system of equations is obtained. This system has the following solution:
\[ v = \frac{u_x + \beta u_z}{1 + \beta} \quad \beta = \frac{\rho_u S_{v,\infty} C_{d,\infty}}{\rho_u S_{v,\infty} C_{d,\infty}} \quad (4 \& 5) \]

The drag coefficient for a petroleum slick is a function of the area of the slick, so the drag coefficients \( C_{d,\infty} \) and \( C_{d,\infty} \) are equal. We thus obtain:

\[ \beta = \frac{\rho_u}{\rho_u} = 0.036 \quad (6) \]

We can deduce that the oil spill transport induced by wind is 3.6 \% of the wind velocity. A similar result is suggested in [10]. Thus, in light wind without breaking wave conditions, oil spill drift induced by wind is 3.5\% of the wind velocity.

2) Diffusion

A drifting substance, for instance petroleum parcels submerged in a current, will diffuse. This diffusion is mostly induced by the turbulent flow. In order to take this phenomenon into account, a stochastic approach is adopted. The hypothesis of “white noise” is made in order to consider the random displacement of a petroleum parcel in water. This hypothesis allows us to define the particle displacement like a Markov process which means that each particle displacement at each time step is independent of its displacements at previous time steps.

Contaminant dispersion is modelled using one governing equation, namely the Advection-Diffusion equation [3]:

\[ \frac{\partial h C}{\partial t} + \nabla \cdot (h C (u)) = \nabla \cdot \left( \frac{h v}{\sigma_x} \nabla C \right) \quad (7) \]

where \( h \) is the water depth, \( C \) the depth-averaged pollutant concentration, \( \sigma_x \) is the neutral turbulent Schmidt number, \( v_i \) is the turbulent velocity. The turbulent Schmidt number can be set to \( \sigma_x = 0.72 \) [5].

A transformation will be applied to the Advection-Diffusion equation (7) to obtain a Lagrangian equation, according to the following process. The first step in this transformation is to interpret the concentration \( C(X,t) \) as a probability \( P(X,t) \) of finding a particle at a location \( X \) at a time \( t \). Then, using mass conservation, we develop and simplify the previous equation, which leads to

\[ \frac{\partial P}{\partial t} = -\nabla \cdot \left[ \left( u \right) - \frac{1}{h} \frac{h v}{\sigma_x} \nabla P \right] + \frac{v_i}{\sigma_x} \nabla \cdot \nabla P \quad (8) \]

This equation is called the Fokker Planck equation. The main benefit of having rewritten the equation in form is that it is equivalent to the Ito stochastic differential equation [1]:

\[ \mathbf{X}(t + \delta t) = \mathbf{X}(t) + \left[ \mathbf{X}(t) - \frac{1}{h} \frac{h v}{\sigma_x} \mathbf{V} \right] \delta t + \sqrt{\frac{2v_i}{\sigma_x}} \delta \xi(t) \quad (9) \]

where \( \delta t \) is the time step, \( \xi(t) \) is a vector with independent, standardized random components.

B. Weathering processes

1) Spreading

The spreading process is the most important weathering process. In fact, all mass transfer phenomena which occur during an oil spill are influenced by the area of the surface slick. Oil discharged onto a water surface will immediately start to increase its surface area. This slick expansion is controlled by mechanical forces such as gravity, inertia, interfacial tension and viscosity. Fay (1971) [16] developed a three-phase spreading theory. The three phases are:

- 1\textsuperscript{st} phase: gravity (spreading) and inertia (retardation) dominate;
- 2\textsuperscript{nd} phase: gravity and viscous (retardation) forces dominate;
- 3\textsuperscript{rd} phase: surface tension (spreading) and viscous forces dominate.

Some authors neglect the inertial forces which are important only in the first phase of spreading [7]. The spreading phenomenon is described as follows:

\[ S - \arctan(\alpha S) = 4\pi \mu \quad (10) \]

with:

\[ \mu = \frac{\sigma_{ow} - \sigma_{wa}}{K} \quad \alpha = \frac{2}{3V} \sqrt{\frac{2\mu K}{\Delta \rho \sigma}} \quad (11) \]

where \( \sigma_{wa} \) is the water-air surface tension, \( \sigma_{ow} \) is the oil-air surface tension, \( \sigma_{wa} \) is the oil-water surface tension, \( K \) is the friction coefficient at the oil water interface, \( V \) is the volume of oil spilled, \( g \) is the gravity, \( \Delta \) is a parameter which relates the oil and water densities: \( \Delta = (\rho_w - \rho_o)/\rho_o \) with \( \rho_o \) the oil and water density respectively.

Moreover, according to [7], some experiments show that more than 90\% of the surface slick is controlled by gravity. This area is surrounded by a thinner oil slick controlled by surface tension. In this paper, the third phase is neglected according to these observations. With this hypothesis and by considering the friction coefficient \( K = (\rho_w \nu) e \) (where \( e \) is the slick thickness and \( v_o \) is the oil kinematic viscosity), the slick surface evolves according to the following equation:

\[ S = \left( \frac{2\pi V^{1/4} \Delta g}{2 \nu_{o}^{1/4} - 1} \right)^{1/4} \quad (12) \]

2) Mass transfer processes

The mass transfer between two phases is quantified by theories. These theories are based on the hypothesis that the mass transfer resistance is located close to the interface between the two phases. Whitman [15] has suggested one of these theories. In the next sections, all processes are based on the Whitman theory which formulates the mass transfer flux for every mass transfer phenomenon.

a) Evaporation

Evaporation is the most important mass transfer process that oil undergoes after spillage. In a few days, light crude or
Dissolution is an important phenomenon from a toxicological and environmental point of view, although it only accounts for a negligible fraction of the initial mass of oil. Due to their physico-chemical properties, only PAHs are assumed to be dissolved in the water. In fact, the dissolved mass of PAH i in the water column is defined at each node by the following relation:

\[
C_i(t) = S_i X_i + [C_i(t-1) - S_i X_i] \exp(-\alpha \delta t)
\]

where \(\alpha\) is the mass transfer coefficient \(K_{\text{dist}}\) is the mass transfer coefficient, \(A_p\) is the particle area, \(V\) is the node volume. The order of magnitude of the dissolved mass transfer coefficient \(K_{\text{dist}}\) is of several cm/h [11, 4].

Thus, thanks to the relationship which links mass with concentration, the mass loss at time \(t\) for each component \(i\) can be deduced:

\[
\text{mass}_{\text{comp}}(t) - \text{mass}_{\text{comp}}(t-1) = \delta t \left[ -S_i X_i + [C_i(t-1) - S_i X_i] \right] \exp(-\alpha \delta t)
\]

\[
\text{mass}_{\text{dissolved}}(t) = \sum_{j=0}^{n} \text{mass}_{\text{dissolved}}(i)(t) = \sum_{j=0}^{n} \text{mass}_{\text{dissolved}}(i)(t) \]

A problem can occur if \(\text{mass}_{\text{dissolved}}\) is bigger than the mass of the PAH component. In this case, the dissolved mass needs to be multiplied by a coefficient:

\[
\text{coefficient} = \frac{\sum_{j=0}^{n} \text{mass}_{\text{comp}}(i)}{\sum_{j=0}^{n} \text{mass}_{\text{dissolved}}(i)}
\]

Thus, the quantity of tracer at the time step \(t\), at node \(j\), added by dissolution is defined by

In Telemac-2D each variable is defined on every node of the mesh. If we consider a particle \(P\) inside an element (Figure 3), it is important to define the dissolved mass of the particle at each element node.

Figure 3. Particle \(P\) inside its element.

The coefficient \(\alpha\) must be defined at each node \(j\). For this, we define the reduced particle area as

\[
A_r \text{SHP}(j) = A_r(j)
\]

where \(\text{SHP}(j)\) is the barycentric coordinate at the node \(j\) and \(A_r(j)\) is the reduced area at node \(j\).

An area is defined around each mesh node, according to a method defined in [3]. The volume \(V\) is obtained by multiplying the node area by the depth of the node (Figure 4).

Figure 4. Area of mesh node.
c) Volatilization

Oil components dissolved in the water can be volatilized to the atmosphere if they are not covered by the oil surface slick. The volatilization flux is expressed as follows:

\[ F_i = -K_{vol}C_i \]  \hspace{1cm} (22)

where \( F_i \) is mass flux of component \( i \) (Kg/m\(^2\)s), \( C_i \) is the concentration of component \( i \) in the water (Kg/m\(^3\)) and \( K_{vol} \) is the overall volatilization rate coefficient (m/s).

IV. EXPERIMENTAL DEVICES.

A. Kinetic experiments

Within the Migr’HyCar project, a static test campaign was conducted by the Agribusiness laboratory LCA located in Toulouse (France).

1) Experimental protocol

During the kinetic experimentations, eight hydrocarbons were tested. The aim of these experiments is to study the hydrocarbons dissolution phenomenon. In a beaker, the tested hydrocarbons are in contact with water during two days. Then, some water samples are taken at different times. An analysis of each sample allows to define the PAH concentrations and the hydrocarbons kinetic is obtained.

2) Results

These hydrocarbon kinetics are used to calibrate the numerical model. In fact, the mass transfer coefficients (\( K_{vol} \), \( K_{dis} \) and \( K_{evap} \)) are obtained by fitting the numerical results of the kinetic experiments with the experimental data.

B. Artificial river experiments

In addition to previous laboratory tests, an artificial river test campaign was conducted by Veolia Environment Research and Innovation (UBA, Berlin). During the tests, four hydrocarbons were tested: heavy fuel oil, home heating oil, kerosene and SP95E10. The main objective of these experiments is to observe the capacity of the pollutant to dissolve PAHs according to four variable parameters, such as flow velocity, the injection position, the oil volume and the presence of obstacles.

1) Experimental device

The UBA (Umweltbundesamt German Federal Agency for the Environment) has on its site 16 identical systems of artificial rivers with each 100 m in circumference. Among these rivers called FSA (acronym for Flies und StillgewässersimulationsAnlage: simulator rivers and lakes), eight are located outdoors. A water flow is generated in these rivers with a screw pump. A system for continuous measurement of physical parameters river is installed for each river, and there is one weather station.

For the purpose of the project, two rivers were linked together to increase the installation length and sinuosity (Figure 6).

2) Experimental protocol

The release of the hydrocarbon is achieved through a ring on water surface, the pollutant is injected inside it (Figure 7). Then, the ring is removed to allow the pollutant transport.

To observe the evolution of the concentration of dissolved PAHs, a fluorescence probe is used. Every morning a sample called “white” is made to know the initial concentration of PAHs already present in the channel. When the signal (%) is approximately on the peak, a water sample is taken during 30 seconds using an automatic device located with the probe. The samples are then sent to the CEDRE for the analysis of dissolved concentrations of PAHs in samples. For each sample, there is therefore a concentration of total PAHs (ng/L) and a probe signal (%).

3) Results
With the various tests carried out the same day and with the same hydrocarbon, it is possible to draw a calibration curve. This curve will allow to convert the signal probe in % into a total PAH concentration in ng/l (Figure 8). Then, the real concentration as function of time is obtained by subtracting the value of the corresponding blank.

The profiles obtained are used to calculate the mass balance (dissolved mass/injected mass) and validate the numerical model.

V. NUMERICAL RESULTS

A. Spreading process

Some spreading experiment has been presented in [9]. During these experiments, a volume of hydrocarbons was spilled in a rectangular Plexiglass tank. The evolution of the slick surface is followed thanks to a camera which allows to quantify the area increasing of the surface slick. In this paper, the result for the heavy fuel called “#6 fuel” is used. The results is summarized in the following table:

<table>
<thead>
<tr>
<th>Time (min)</th>
<th>Area $f^2$</th>
<th>Mass $M^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0.04</td>
<td>0.0037</td>
</tr>
<tr>
<td>3</td>
<td>0.10</td>
<td>0.0093</td>
</tr>
<tr>
<td>4</td>
<td>0.13</td>
<td>0.0121</td>
</tr>
<tr>
<td>5</td>
<td>0.15</td>
<td>0.0139</td>
</tr>
<tr>
<td>6</td>
<td>0.16</td>
<td>0.0147</td>
</tr>
<tr>
<td>8</td>
<td>0.17</td>
<td>0.0158</td>
</tr>
<tr>
<td>10</td>
<td>0.18</td>
<td>0.0167</td>
</tr>
<tr>
<td>15</td>
<td>0.19</td>
<td>0.0177</td>
</tr>
<tr>
<td>20</td>
<td>0.20</td>
<td>0.0186</td>
</tr>
<tr>
<td>25</td>
<td>0.22</td>
<td>0.0204</td>
</tr>
<tr>
<td>30</td>
<td>0.23</td>
<td>0.0214</td>
</tr>
<tr>
<td>35</td>
<td>0.26</td>
<td>0.0242</td>
</tr>
</tbody>
</table>

A comparison between numerical results and these experimental data can be done, the result is presented in Figure 9.

The numerical result are in good agreement with the experimental data. At the beginning of the spreading process, the model overestimates the spreading of the surface slick. Then, the model superimposes with the experimental results.

B. The artificial river simulation

To evaluate the hybrid oil spill model performance, a simulation is carried out with the artificial river experimental conditions.

1) Mesh characteristics

The mesh to model the artificial river includes only triangular elements. It is composed of about 15000 elements and 7500 nodes. The maximum distance between two nodes is 0.2 m. A detail of the mesh is shown in Figure 10.

2) Simulation results

In this paper, only heavy fuel spill in the artificial river is considered. The following heavy fuel parameters are used:

$\rho$ (at 20°C) = 950 Kg/m$^3$

$\nu_o$ (at 20°C) = 4465 cSt

The turbulent viscosity $\nu_t$ used in the transport model of the surface slick is calculated thanks to a depth-averaged $k$–$\varepsilon$ turbulence model.

Figure 9. Spreading evolution of heavy fuel (“#6 fuel”).
In Figures 11 and 12, the black dots represent the oil surface slick whereas the scalar tracer represents the dissolution phenomenon in the water column.

![Figure 12. Oil spill simulation in the artificial river curve, without obstacles (top) and with obstacle (bottom).](image)

The oil surface slick movement is well modelled, as shown in Figure 12. In fact, on the right part of the figure, the movement of oil particles in the surface slick tail is induced by the turbulence phenomenon.

For what concerns the dissolved petroleum in the water column, the first case is an heavy fuel spill which occurs in the first artificial river straight line close to the screw pump. The flow velocity is imposed to 0.1 m/s and there is no obstacle in the channel. The numerical and experimental concentration peak appears at the same time as show in Figure 13.

![Figure 13. Concentration evolution in the water column (case 1).](image)

So, we can deduce that the transport is well modelled. Moreover, the dissolved hydrocarbons concentration in the water column has the same order of magnitude and compares well with experiment. More precisely, the numerical model seems to overestimate the maximum of dissolved concentration and to underestimate the dispersion phenomenon.

Two other cases have been carried out to confirm the previous observations. These two cases concern an oil spill which occurs in the first artificial river curve, with obstacles in the channel.

The river flow is imposed to 0.1 m/s in the second case (Figure 14, left), and 0.2 m/s in the third case (Figure 14, right). The previous observations are confirmed with these cases. However, the numerical concentration peaks for each case arrive earlier in comparison to the experimental concentration peak. This phenomenon can be explained by the outdoor conditions which cannot be modelled, such as gusts of wind.

![Figure 14. Concentration evolution in the water column: case 2 (top) and case 3 (bottom).](image)

To conclude, these results show the model capacity to follow the dissolved PAHs evolution in the water column according to the release position, the oil spill volume and the presence of obstacles. Nevertheless, a complementary study is necessary in order to exemplify the capability of the numerical model to deal with different petroleum products.

### C. Gironde estuary simulation

Some water intake operators and a nuclear power plant are located on the coast of Gironde estuary. An oil spill can have a strong impact on the management of these industries. So, it is important to be able to model accurately an oil spill which would occur in the estuary.
Figure 15. Simulation result of heavy fuel spill in Gironde estuary.

In the simulation presented on Figure 15, a hypothetical heavy fuel spill is considered to occur with an initial oil volume of 1 m$^3$. After one day, the shape of the slick is shown on Figure 15. Even if the surface slick is not stranded, some heavy fuel reached the estuary coast. This phenomenon can have a strong impact for intake operators. Thus, it is important to follow up the dissolved oil in the water column and the surface slick for operational management of risks.

VI. CONCLUSION

The hybrid model has been developed to simulate oil spill in continental waters. This model predicts the movement of an oil slick, the fate processes and the dissolution concentration of oil in the water column. This oil slick model has been coupled with the Telemac hydrodynamic model.

Verifications of the model were carried out by comparing numerical and experimental results. In fact, static experiments have allowed to calibrate the numerical model use for the validation. Then, the calibrated model has been tested by reproducing dynamic artificial river experiments. The obtained results for heavy fuel spill are close to the experimental data. However, even if the numerical results are promising, a complementary study is necessary in order to validate the fate oil spill model for other petroleum products.

Application of the model to a case of oil spill in Gironde estuary exemplifies the capability of the model to deal with different phenomena.

ACKNOWLEDGEMENT

The Migr’HyCar research project is supported by the French Research Agency ANR as a result of the PRECODD 2008 Call for proposals. The PRECODD Eco-technology and Sustainable Development Programme is a research scheme aimed at supporting emerging techniques, procedures and concepts that can help control the environmental impact of industrial and urban activity.

REFERENCES