

3D Simulations of Dredged Sediment Disposal Plumes Over a Realistic Bathymetry Using a Two-Phase Model

MARINE 2021

Foteini Kyrousi^{1,*} and Boudewijn Decrop²

¹ International Marine and Dredging Consultants (IMDC) Van Immerseelstraat 66, B-2018 Antwerp, Belgium. E-mail: foteini.kyrousi@imdc.be, web page: <http://www.imdc.be>

² International Marine and Dredging Consultants (IMDC) Van Immerseelstraat 66, B-2018 Antwerp, Belgium. Email: boudewijn.decrop@imdc.be, web page: <http://www.imdc.be>

* Corresponding author: Foteini Kyrousi, foteini.kyrousi@imdc.be

ABSTRACT

The vertical propagation of a dredged spoil release and the propagation of the gravity current due to the sediment cloud collapse at the bottom is investigated in the present study. A two-phase numerical model, able to calculate the amount of the detrained material as function of the conditions under which it is disposed, is developed in Fluent. Initially, a 2D case was validated versus experimental and numerical data available in literature. The validation includes the falling time of the sediment cloud, the of the evolution of the cloud bottom as well as the time evolution of the current front. Further, a 3D two-phase model is presented that simulates the complete process of sediment release over a realistic bathymetry and during realistic flow conditions. The outcome of the 3D model shows how the material is distributed on the river bed, during and immediately after the release of sediment by a hopper dredger.

Keywords: dredged sediment disposal; two-phase model; realistic bathymetry

1. INTRODUCTION

Dredging operations performed in navigation channels and harbors produce a significant quantity of sediment every year. The dredged material is then discharged into disposal sites in the land, the sea, estuaries or rivers. Disposal of dredged material into open waters is a complex task and may rise important environmental and economic concerns. Thus, the ability to determine the propagation of the material after the disposal is of prime interest.

It has been shown that the disposal of dredged material can be divided in three stages [5]: (a) a convective descend during which the material is falling under the influence of gravity (b) the dynamic collapse, occurring when the descending cloud impacts the bottom and (c) passive transport-dispersion where the material transport and spreading are determined mainly by ambient currents and turbulence. Various studies have been conducted on the mechanisms of the disposal of dredged material placed in open waters [1,3,5,7,8,11,12].

During the descent phase, the surrounding water is entrained onto the sediment cloud and as a result part of the dredged material is separated from the cloud and remains in the upper part of the water column [11]. After the collapse of the cloud at the bottom, a gravity current propagates radially, until sufficient energy is dissipated and the material begins to settle. The part of the sediment that remains into suspension and the one carried away from the disposal area by the horizontal gravity currents is referred to as “lost” material.

The objective of this work is to study, through 3D numerical simulations, the different stages of the disposal process at real flow conditions and investigate the influence of bathymetry on the propagations of the gravity current.

Previous studies [7] have shown that a two-phase approach is able to represent the convective descent of dredged sediment release compare to the single-phase models. Thus, in the framework of this work an Eulerian two-phase model embedded in Fluent is used. To validate the developed numerical model the experimental data obtained from a large physical model testing facility were used [1,12] as well as similar numerical studies available in literature [7].

The text is organized as follow: Section 2 illustrates briefly the set of the governing equations of the numerical model. Sec. 3 present the 2D test case used for the validation of the numerical model. The results obtained by the real case scenario are discussed in Sec.4 followed by a brief summary and the main conclusions is Sec. 5.

2. MODEL DESCRIPTION

In the Eulerian two-phase model in Fluent the water and the sediment phase are considered as interpenetrating continua. This description incorporates the concept of volume fractions, which represents the space occupied by each phase. In this approach, each phase is simulated separately, whereas the coupling between the two phases is accomplished via momentum transfer.

2.1 Governing equations

In this work the fluid motion was simulated using the 3-D RANS equations with a k- ϵ mixture turbulence closure model. The conservation equations for each case are presented below [6]:

$$\frac{\partial}{\partial t}(a_q \rho_q) + \nabla \cdot (a_q \rho_q \vec{v}_q) = \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) \quad (1)$$

$$\frac{\partial}{\partial t}(a_q \rho_q \vec{v}_q) + \nabla \cdot (a_q \rho_q \vec{v}_q \vec{v}_q) = -a_q \nabla p + \nabla \cdot \bar{\tau}_q + a_q \rho_q \vec{g} + \sum_{p=1}^n (\vec{R}_{pq} + \dot{m}_{pq} \vec{v}_{pq} - \dot{m}_{qp} \vec{v}_{qp}) + (\vec{F}_q + \vec{F}_{lift,q} + \vec{F}_{wl,q} + \vec{F}_{vm,q} + \vec{F}_{td,q}) \quad (2)$$

where \vec{v}_q is the velocity of phase q, \dot{m}_{pq} characterizes the mass transfer from the pth to the qth phase and \dot{m}_{qp} characterizes the mass transfer from phase q to p. \vec{F}_q is an external body force, $\vec{F}_{lift,q}$ is a lift force, $\vec{F}_{wl,q}$ is a wall lubrication force, $\vec{F}_{vm,q}$ is a virtual mass force and $\vec{F}_{td,q}$ is a turbulent dispersion force. \vec{R}_{pq} is an interaction force between phases and p is the pressure shared by all phases. $\bar{\tau}_q$ is the qth phase stress-strain tensor defined as following:

$$\bar{\tau}_q = a_q \mu_q (\nabla \vec{v}_q + \nabla \vec{v}_q^T) + a_q \left(\lambda_q - \frac{2}{3} \mu_q \right) \nabla \cdot \vec{v}_q \bar{I} \quad (3)$$

here μ_q and λ_q are the shear and the bulk viscosity of the phase q. For the sediment phase where the granular flow approach was considered the shear and bulk viscosities are obtained by applying the kinetic theory. More specifically, the sediment shear viscosity is equal to:

$$\mu_s = \mu_{s,col} + \mu_{s,kin} + \mu_{s,fr} \quad (4)$$

where $\mu_{s,col}$ is the collisional viscosity as in [3,9], $\mu_{s,kin}$ is the kinetic viscosity as in [3] and $\mu_{s,fr}$ is the frictional viscosity as in [8].

2.2 Turbulence model

The turbulence model used in the present study is the k- ε mixture turbulent model available in ANSYS FluentTM [6]. The k and ε equations describing this model are as follows:

$$\frac{\partial}{\partial t}(\rho_m k) + \nabla \cdot (\rho_m \vec{u}_m k) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_m \varepsilon \quad (5)$$

$$\frac{\partial}{\partial t}(\rho_m \varepsilon) + \nabla \cdot (\rho_m \vec{u}_m \varepsilon) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_\varepsilon} \nabla \varepsilon \right) + \frac{\varepsilon}{k} (C_{1\varepsilon} G_{k,m} - C_{2\varepsilon} \rho_m \varepsilon) \quad (6)$$

where the mixture density and velocity, ρ_m and \vec{u}_m , are computed from:

$$\rho_m = \sum_{i=1}^N a_i \rho_i \quad (7)$$

$$\vec{u}_m = \frac{\sum_{i=1}^N a_i \rho_i \vec{u}_i}{\sum_{i=1}^N a_i \rho_i} \quad (8)$$

The turbulent viscosity, $\mu_{t,m}$, is computed from

$$\mu_{t,m} = \rho_m C_\mu \frac{k^2}{\varepsilon} \quad (9)$$

And the production of turbulent kinetic energy, $G_{k,m}$, is computed from

$$G_{k,m} = \mu_{t,m} (\nabla \vec{u}_m + (\nabla \vec{u}_m)^T) : \nabla \vec{u}_m \quad (10)$$

$C_{1\varepsilon}$, $C_{2\varepsilon}$ and C_μ are constants.

2.3 Numerical technique

Due to the unsteady nature of the problem investigated in this work, transient simulations have been carried out using ANSYS FluentTM solver [6]. For the time integration the first order implicit method was adopted. The momentum equation was discretized using a second-order finite volume technique, whereas QUICK scheme was applied for the volume fraction. The SIMPLE algorithm was used to solve the pressure-velocity coupling.

3. VALIDATION CASE

3.1 Experimental set-up

For the validation of the numerical model developed on the framework of this work, data obtained from the physical model described in [1,12] is used. The experimental set up consists of a straight channel 72m long, 1.5m wide and 1.5m maximum height (see Figure 1). The sediment were released by a specifically designed recipient (maximum capacity 60L) which was placed 17cm below the water surface. During the test all the three stages of the disposal process were recorded. The opening of the recipient was synchronized with a

camera that take pictures every 0.5s. Concentration measurements at specific locations were also obtained using optical transducers.

In the physical test different sediment mixtures were used (pure sand, sandy mud and pure mud) and different flow conditions. In this work only pure sand mixtures are considered and zero cross flow. The parameters of the physical test that is used for the validation are presented in Table 1.

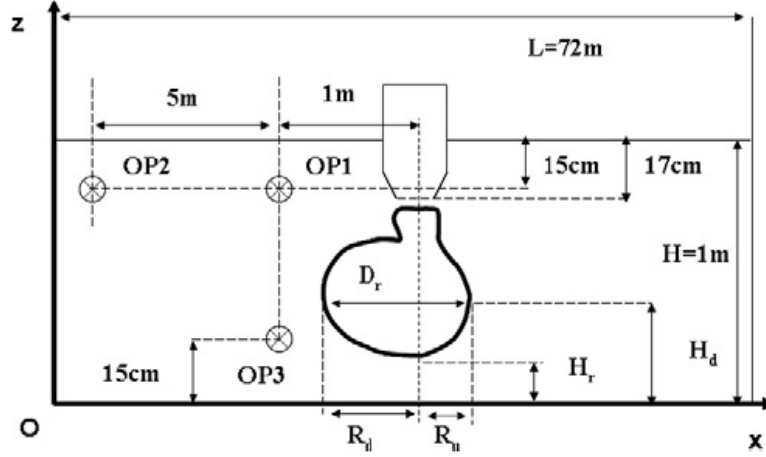


Figure 1. Schematic representation of the physical model set-up [1].

Table 1. Testing condition: H is the water depth, W_0 is the injection velocity from the recipient, D_p the sediment particle diameter, ρ the dry density of the solid, C_m is the concentration of the mixture and V_d the volume of dumped material.

Test	H [m]	W_0 [m/s]	D_p [μm]	ρ [kg/m^3]	C_m [g/L]	V_d [L]
e12	1	0.89	160	2650	450	60

3.2 Numerical model

A two dimensional (2D) computational domain used to simulate the sediment release is 14m long and 1 high. As in the experiments the recipient is 10cm long and is located 17cm below the free surface. The spatial resolution of the grid is 1.5cm x 1.5cm. The time step is taken equal to 10^{-3} sec.

To avoid reflections, a pressure outlet boundary condition is imposed at the lateral boundaries of the domain. The bottom of the domain is modeled as no-slip wall whereas a free slip boundary condition is applied at the free surface, since no surface motion was detected during the experiments. The flow exiting the recipient assumes a velocity inlet a Poiseuille-type velocity profile as proposed in [7]:

$$W_{inj} = W_0 \left(1 - \frac{x^2}{R_{inj}^2} \right) \quad (10)$$

where R_{inj} is the half of the recipient bottom opening and x is the distance from the vertical axis of the recipient.

3.3 Results

The contour plots of sediment concentration obtain from the numerical model along with the flow velocity vectors are presented in Figure 2, for 4 different time instants. The numerical and the experimental results were

compared based on (1) the falling time of the sediment cloud (time lapse between the release of the sediment and the moment when the cloud touches the bottom).

Table 2 presents the falling time of the sediment cloud for both the physical test and the numerical model. It should be mentioned that an uncertainty is observed on the experimental data. This may be explained by the rapidity of the phenomena (<1.5 s). Under this conditions, the numerical and the experimental results are in good agreement.

Further validation of the numerical model is done based on similar numerical studies found in literature [7]. In this case the results were compared based on (1) the time evolution of the sediment cloud bottom and (2) the time evolution of the current front. The time evolution of the sediment cloud bottom for both studies is presented in Figure 3.

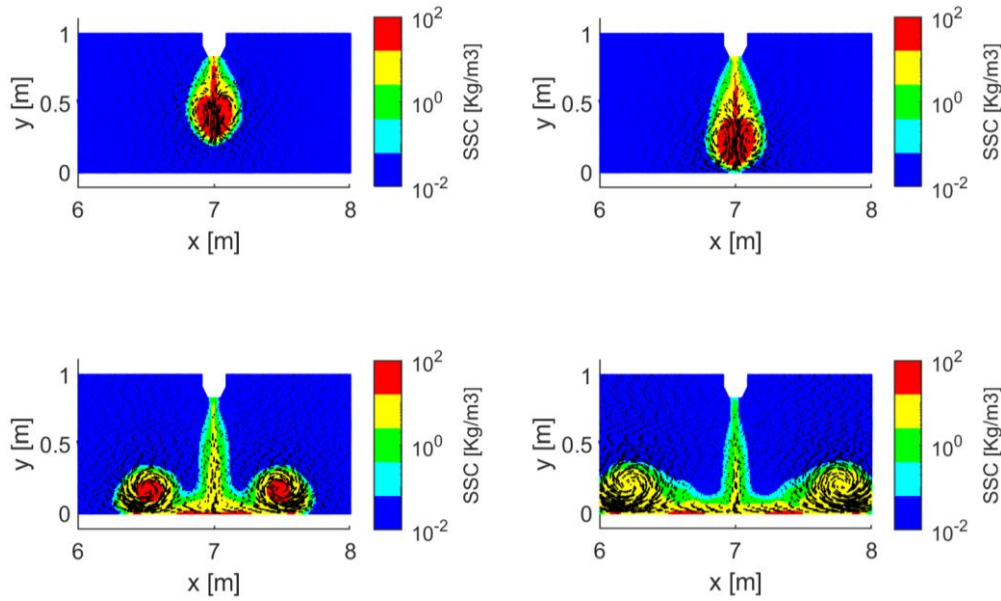


Figure 2. Contour plots of the sediment concentration and vector plot of flow velocity at time instants (a) $t = 1$ sec, (b) $t = 1.3$ sec, (c) $t = 2.6$ sec and (d) $t = 4.0$ sec.

Table 2. Comparison of the falling time of the sediment cloud

Falling time [sec]	
Experimental	Numerical
0.72 (0.22-1.22)	1.3

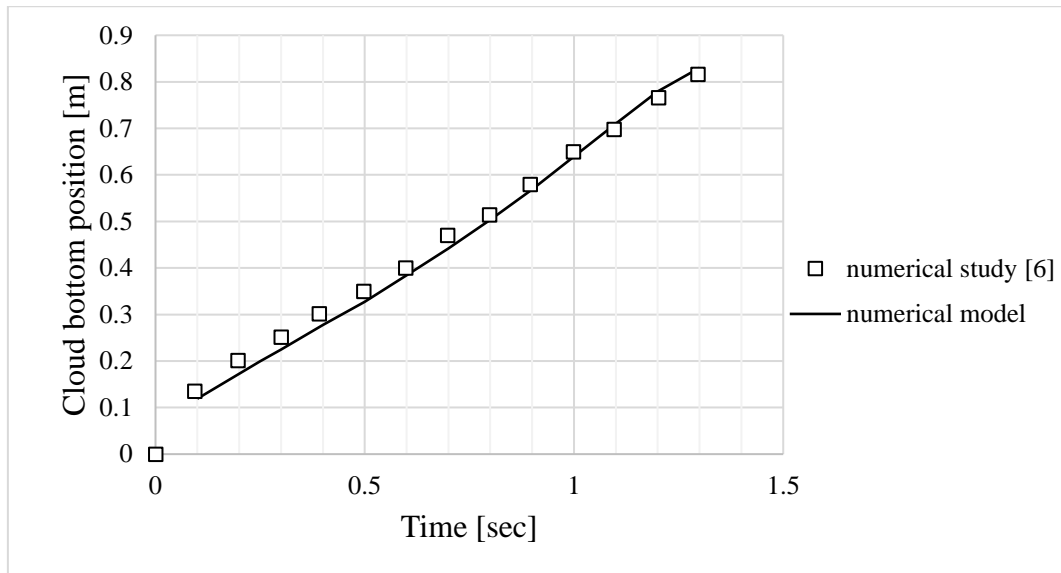


Figure 3. Comparison of the time evolution of the sediment cloud bottom between the present numerical study and available literature data [6].

After the sediment cloud collapse at the bottom it is converted to a horizontal gravity current. The time evolution of the front of the current for both numerical studies is presented in Figure 4. A concentration threshold equal to 0.5 kg/m³ is used to determine the front of the current. The propagation of the current is symmetric with respect the vertical axis of the recipient and as expected at this initial stage the $x_f \sim t$ relation is linear. The aforementioned results indicate a very good agreement between the present numerical model and the literature data.

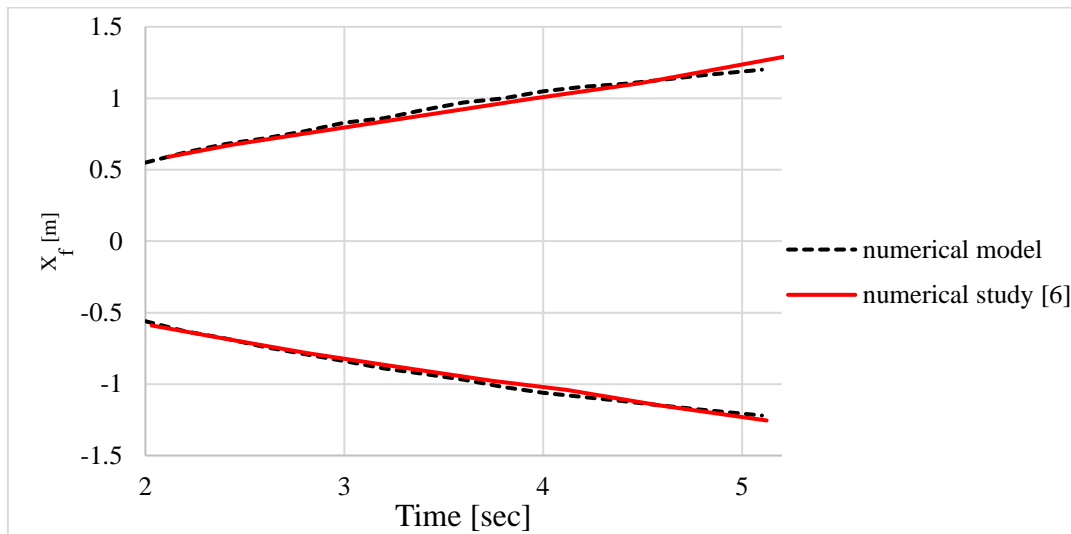


Figure 4. Comparison of the time evolution of the front of the current between the present numerical study and available literature data [6].

4. SEDIMENT DISPOSAL IN SCHELDT ESTUARY

After the model was validated, it was applied to simulate sand disposal at the Scheldt Estuary. The layout of the model is presented in Figure 5. In this case, sediment release from the bottom dredging vessel is simulated considering real flow conditions and bathymetric data.

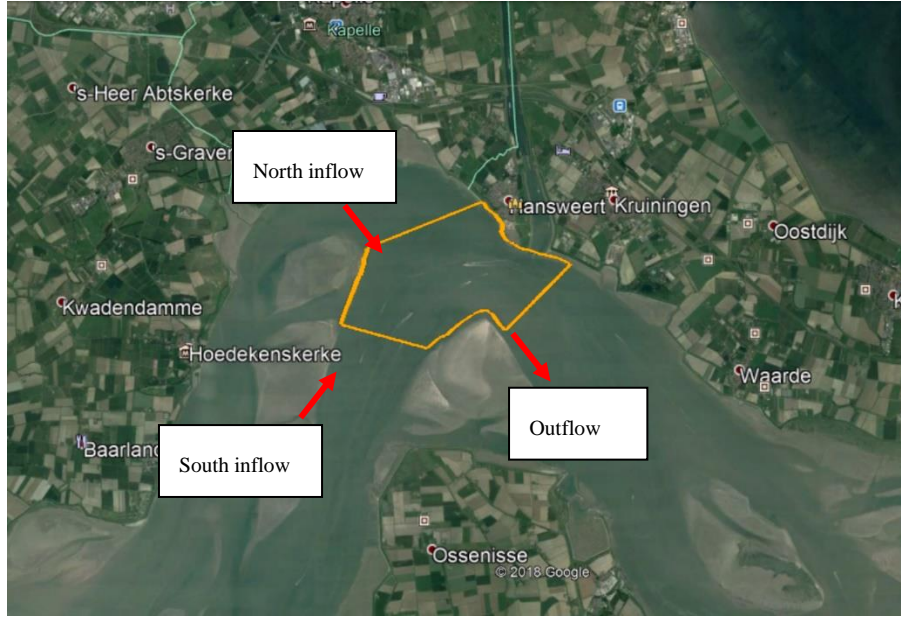


Figure 5. Time evolution of the front of the current for both experiments and numerical model.

4.1 Model set-up

The geometry consists of the vessel and a surrounding liquid flow domain. The vessel is assumed to be static inside the model domain and has a total length of 137 m. The bottom of the domain is created based on bathymetric data of Scheldt river for year 2016 (see Figure 6).

The generated numerical grid consists of polyhedral elements along with an inflation layer close to the bed, for resolving the near bed velocity gradients. The mean edge length of the mesh equals 7.5m. For the better representation of the flow around the vessel and the sediment distribution the grid has been refined at the immediate vicinity of the vessel. The cell size at the hull of the vessel is equal to 0.6m whereas at the doors of the vessel is equal to 0.4m.

The disposal of dredged material is carried out in an already developed velocity field that corresponds to max flood conditions. The flow field is generated by means of prescribed velocity profile at the upstream boundaries of the model. The prescribed velocity is obtained by a continental shelf model (iCSM) developed in TELEMAC 2D at IMDC [2]. At the downstream boundary of the model a pressure outlet boundary condition is applied.

To simulate the sediment release, at the bottom door of the hopper a constant velocity is applied for both water and sediment phases. This velocity is set in order to match the total hopper volume released in a predefined disposal time. In the same zone a the initial sediment volume fraction is set. At the water surface a free slip boundary condition is applied, whereas the bottom, the side walls of the domain and the hull of the hopper are defined as no slip boundaries.

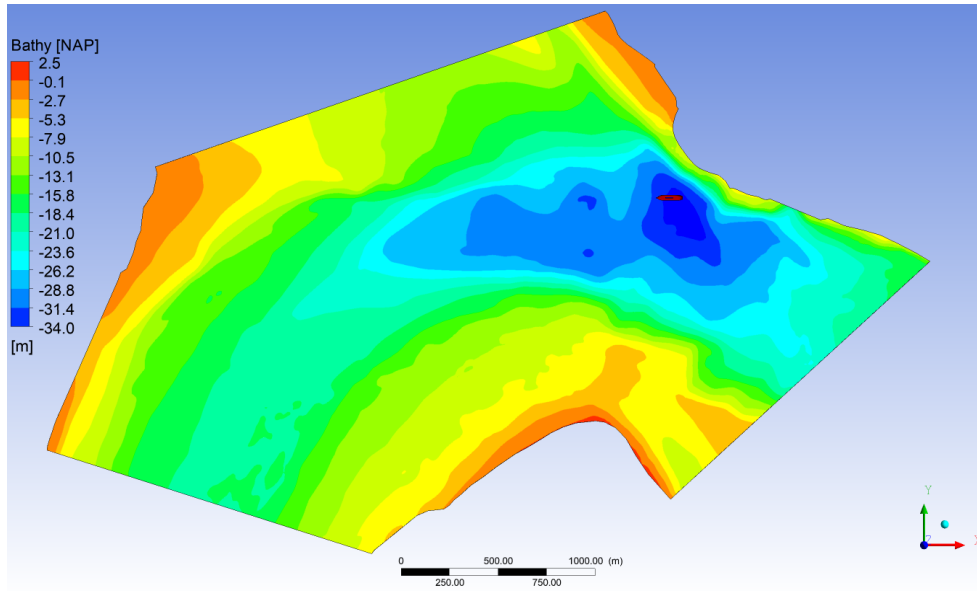


Figure 6. Contour plot of the bathymetric data used for the numerical model.

The parameters used in the current simulation are presented in Table 3. These data are based on the Pallieter vessel, commonly used in Scheldt river for dredging operations. The overall duration of the simulations is equal to 380 sec. This period is divided in two parts. The disposal period with duration 261 seconds, and the period after the closing of the doors of the vessel with duration 119 seconds. The second part is important in order to study the collapse of the cloud on the bed.

Table 3. Model parameters: U_{inj} is the injection velocity of the sediment mixture from the hopper, ϕ_{init} is the initial sediment volume fraction, D_p is the sediment diameter and ρ_s is the density of the solids.

Hopper volume with flushing water [m ³]	Door length [m]	Door width [m]	Mass of dry solids [tons]	Valve duration [sec]	U_{inj} [m/s]	ϕ_{init} [-]	D_p [mm]	ρ_s [kg/m ³]
5200	39.02	4.82	6890	261	0.106	0.5	0.2	2650

4.3 Initial velocity field

As mentioned above to reproduce the ambient flow conditions velocity profiles that corresponds to the max flood conditions were obtained by the iCSM model . The velocity field obtained by the CFD simulations is compared with the velocity filed of the large scale model in Figure 7. The results are in good agreement. Slight differences may observed due to the different approaches used for solving the flow.

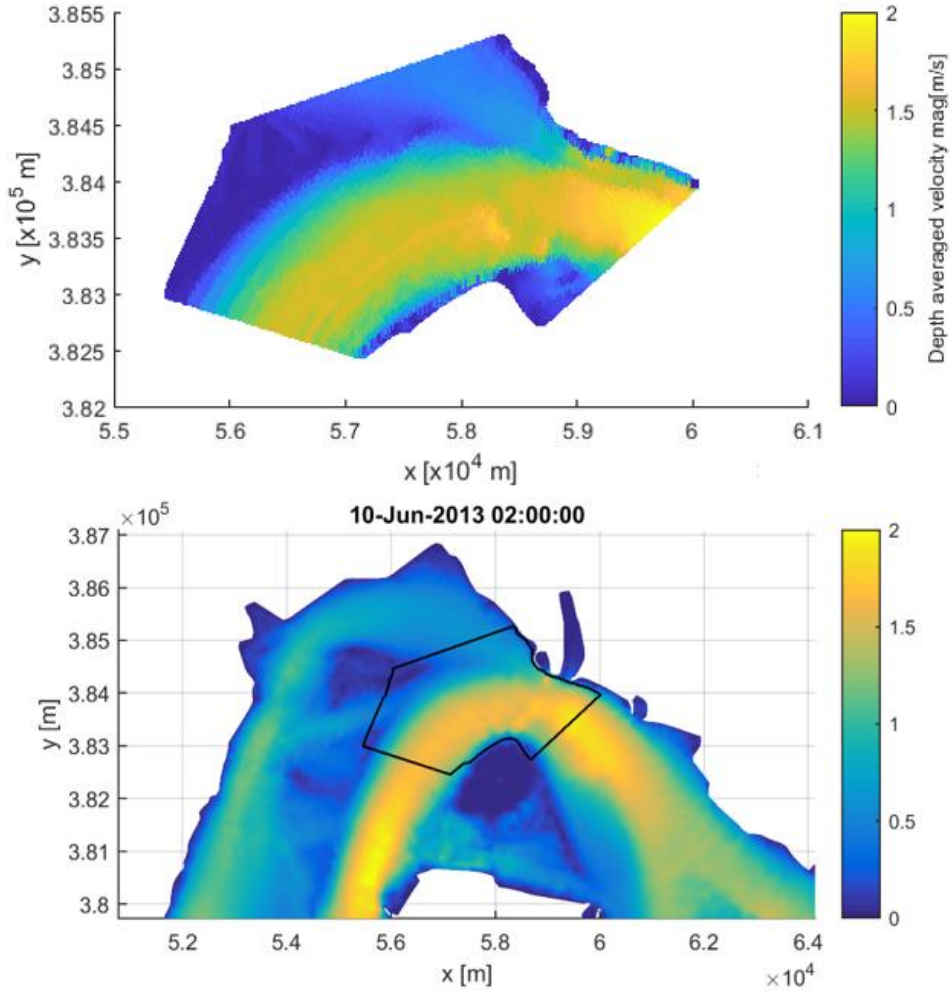


Figure 7. Comparison of the depth averaged velocity field, between CFD model and large-scale validated Telemac 3D Scheldt model. The black line indicates the outline of the CFD domain.

4.4 Results

The contour plots of the sediment concentration at different time instants, are presented in Figure 8. The obtained results clearly show the influence of the ambient flow and the bottom slope on the propagation of the gravity current. The current slows down as it propagates away from the vessel and the negative slope of the bottom is becoming steeper. For this reason at the right side sediment is deposited closer to the vessel. Moreover, it can be observed that the shape of the deposition pattern is not symmetric but follows the direction of the flow. At the end of the simulation the diameter of the deposited area is around 450 m at the direction of the flow and 300m at the direction perpendicular to the flow.

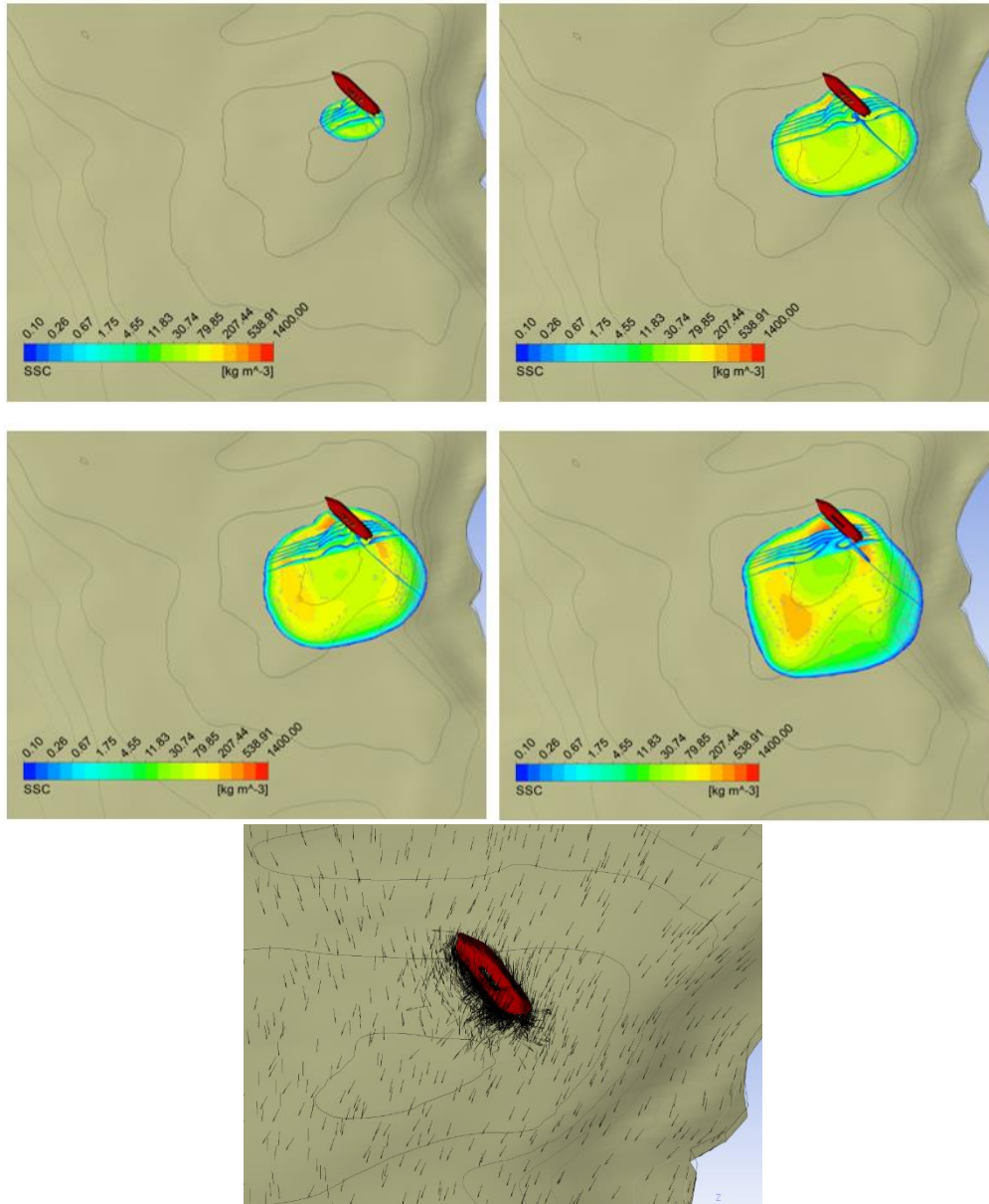


Figure 8. Contour plots of the suspended sediment concentration at different time instants. a) $t=50$ s b) $t=150$ s, c) $t=261$ s (closing of the valves) and d) $t=380$ s (end of the simulations). In the last figure the surface flow direction is indicated. In the last figure the flow direction is indicated.

5. CONCLUSIONS

In this work a 3D two phase model has been developed to investigate sand disposal in tidal estuaries. This model has been initially validated based on experimental data and similar numerical studies available in literature. Good agreement has been obtained in terms of falling time of the sediment cloud as well as time evolution of the cloud bottom and the front of the current.

In the second part, the numerical results of sediment release from a hopper at real flow conditions and over realistic bathymetric data are discussed. The suspended sediment distribution during and immediately after the disposal has been studied to assess the sediment spreading.

The results of the numerical simulations show that bathymetry plays a primary role on the propagation of the horizontal density current. Due to the negative slope of the bed along a trajectory from the disposal zone in the deepest part of the river towards less deep areas, the density current slows down. This means that sediments are deposited closer to the vessel.

The background velocity also has an effect on the sediment distribution. It was observed that the direction of the ambient flow may have a secondary effect on the shape of the deposition area. In case no ambient flow is considered the gravity current propagates radially under the vessel. However, in this case the deposition area has lost its symmetry and follows the flow.

Future work shall focus in validating the numerical model against field measurements of sand plumes, where emphasis is made on the shape of the plume and depositional area after release. Different disposal scenarios may be then studied, using different flow conditions and bathymetry.

REFERENCES

- Chu, K., Breugem, W.A., Wolf, T. and Decrop, B. (2020). Improvement of a Continental Shelf Model of the North Sea. *In Online proceedings of the papers submitted to the 2020 TELEMAC-MASCARET User Conference October*, pp. 146-152.
- Gensheimer, R.J., Adams, E.E. and Law, A.W. (2013). Dynamics of particle clouds in ambient currents with application to open-water sediment disposal. *Journal of Hydraulic Engineering*, 139(2), pp.114-123.
- Gidaspow, D., Bezburuah, R., and Ding, J., (1992). Hydrodynamics of Circulating Fluidized Beds, Kinetic Theory Approach. In Fluidization VII, *Proceedings of the 7th Engineering Foundation Conference on Fluidization*, 75-82.
- Johnson, B.H. and Fong, M.T., (1995). Development and verification of numerical models for predicting the initial fate of dredged material disposed in open water. Report 2. Theoretical developments and verification results. *Army engineer waterways experiment station Vicksburg MS*.
- ANSYS Inc, (2017). ANSYS Fluent Theory Guide. 275 Technology Drive Canonsburg, PA 15317.
- Nguyen, D.H., Levy, F., Van Bang, D.P., Guillou, S., Nguyen, K.D. and Chauchat, J., (2012). Simulation of dredged sediment releases into homogeneous water using a two-phase model. *Advances in Water Resources*, 48, pp.102-112.
- Ruggaber, G.J., (2000). Dynamics of particle clouds related to open-water sediment disposal. *PhD diss.*, Massachusetts Institute of Technology.
- Schaeffer, D.G., (1987). Instability in the Evolution Equations Describing Incompressible Granular Flow. *J.Diff. Eq.* 66. 19–50
- Syamlal, M., Rogers W., and O'Brien, T.J., (1993). MFX Documentation: Volume1, Theory Guide. National Technical Information Service, Springfield, VA. DOE/METC-9411004, NTIS/DE9400087
- Truitt, C.L., (1988). Dredged material behavior during open-water disposal. *Journal of Coastal Research*, pp.489-497.
- Villaret, C., Claude, B. and Du Rivau, J.D., (1998). Etude expérimentale de la dispersion des rejets par clapage. He-42/98/065/a, LNHE, EDF.