



**HAL**  
open science

# Simulation for the convective descent phase of dredged-sediment releases in the seawater by a two-fluid model

Duc Hau Nguyen, Sylvain Guillou, Kim Dan Nguyen, Damien Pham van Bang, Julien Chauchat

## ► To cite this version:

Duc Hau Nguyen, Sylvain Guillou, Kim Dan Nguyen, Damien Pham van Bang, Julien Chauchat. Simulation for the convective descent phase of dredged-sediment releases in the seawater by a two-fluid model. THESIS 2011, Apr 2011, Paris, France. pp.1-4. hal-00621135

**HAL Id: hal-00621135**

**<https://hal.science/hal-00621135v1>**

Submitted on 9 Sep 2011

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

## *Simulation for the convective descent phase of dredged-sediment releases in the seawater by a two-fluid model*

**Duc Hau NGUYEN, Sylvain GUILLOU**

Laboratoire Universitaire des Sciences Appliquées de Cherbourg (EA4253), Université de Caen  
Postal address<sup>1</sup>: Site Universitaire de Cherbourg, BP 78, 50130 Cherbourg-Octeville, France  
Tel. +33 (0) 02 33 01 40 32, Fax: +33 (0) 02 33 01 42 01,

e-mail : [sylvain.guillou@unicaen.fr](mailto:sylvain.guillou@unicaen.fr); [duc-hau.nguyen@unicaen.fr](mailto:duc-hau.nguyen@unicaen.fr)

**Kim Dan NGUYEN, Damien PHAM VAN BANG**

Laboratoire d'Hydraulique Saint-Venant (Univ. Paris-Est, Laboratoire commun EDF/R&D-CETMEF-ENPC)  
Postal address: 6 quai Watier, BP 49, F-78401 Chatou Cedex, France

e-mail : [dan.nguyen@saint-venant-lab.fr](mailto:dan.nguyen@saint-venant-lab.fr); [damien.pham-van-bang@saint-venant-lab.fr](mailto:damien.pham-van-bang@saint-venant-lab.fr)

**Julien CHAUCHAT**

Laboratoire d'Écoulements Géophysique et Industriels, CNRS UMR 5519, Université de Grenoble  
Postal address: BP 53, F-38041 Grenoble Cedex 9, France

e-mail : [julien.chauchat@hmg.inpg.fr](mailto:julien.chauchat@hmg.inpg.fr)

*The harbour structures are often located in shallow areas, especially in estuaries. It is then necessary to perform dredging to allow ships accessing to the docks. The dredged sediment is released over a predefined deposit zone. During the release, the sediment cloud, of which the concentration can be initially more than 350g/l, contains a lot of contaminants and then will impact to the environment. The purpose of this work is to numerically study the process of dredged sediment with the help of a two-phase model [4]. The simulations are performed in the configuration of the experiments of Villaret et al. [6].*

### **Key words**

Dredged sediment release, sediment transport, two-phase flow modelling, fine sand

## **I INTRODUCTION**

The process of dredged-sediment disposal generally follows three different steps [1]: (i) convective descent during which the sediment falls under the influence of gravity; (ii) dynamic collapse, occurring when the descending cloud or jet impacts the bottom (iii) passive transport-dispersion, commencing when the sediment transport and spreading are determined more by ambient currents and turbulence than by the dynamics of the disposal operation.

Previous numerical studies have shown the limits of the “*passive scalar*” approach in simulating dredged sediment disposal [2]. This is due to the very high concentration of the sediment cloud. In recent work, Freson et al. [3] has used a bi-specie model, which is based on a degeneration of the two-phase model, and considers the sediment-water mixture. Interesting results were obtained for the falling phase whereas the bottom transport phase remains still problematic.

The purpose of this work is to study the convective descend step using a two-phase model [4]. This model has been applied [5] to the experimental configuration of Villaret et al. [6]. The process of dredged-sediment disposal is studied with different values of the current, the sediment diameter, and the initial sediment concentration.

## **II GOVERNING EQUATIONS**

The two-phase flow model is based on the Eulerian-Eulerian (or two-fluid) description. The model is fully described in [4]. Herein we briefly recall the basic equations with special closures for

---

<sup>1</sup> Corresponding author

the sediment dumping case. The governing equations (1) are written for each phase. The subscript  $k$  could be “ $f$ ” for fluid and “ $s$ ” for solid phase:

$$\left\{ \begin{array}{l} \frac{\partial(\alpha_k)}{\partial t} + \vec{\nabla} \cdot (\alpha_k \vec{u}_k) = 0 \\ \frac{\partial(\alpha_k \vec{u}_k)}{\partial t} + \vec{\nabla} \cdot (\alpha_k \rho_k \vec{u}_k \otimes \vec{u}_k) = \frac{1}{\rho_k} \vec{\nabla} \cdot (\alpha_k (-p_k \bar{I} + \bar{\tau}_k + \bar{\tau}_k^{Re})) + \alpha_k \vec{g} + \frac{1}{\rho_k} \vec{M}_k \end{array} \right. \quad (1)$$

where  $\alpha_k$ ,  $\vec{u}_k$  and  $\rho_k$  stand for the volume fraction, velocity and density of phase  $k$  respectively,  $\vec{g}$  is the acceleration of gravity,  $M_k$  the inter-phase momentum transfer,  $p_k$  the pressure of phase  $k$ ,  $\bar{\tau}_k$  (or  $\bar{\tau}_k^{Re}$ ) the viscous (or Reynolds, respectively.) stress tensor. The sum of volume fractions,  $\alpha_k$ , is obviously equal to 1. In this model, the viscous stress tensor is considered as a function of shear-stress tensors. It is worth noting that viscosity coefficients used for the solid stress are weighted by an amplification factor  $\beta$ , which accounts for the non-Newtonian behaviour of sediment flows. This is depending on the inter-particle distance, which depends itself on the maximum volume fraction  $\alpha_{s,max}$  (close to the packing concentration, whose value is equal to 0.625 for non-cohesive spherical and mono-dispersed solid particles). Different turbulent closures are available in the present version of the code: the reader can find a detailed presentation in [7].

### III STUDY CONFIGURATION AND SET UP

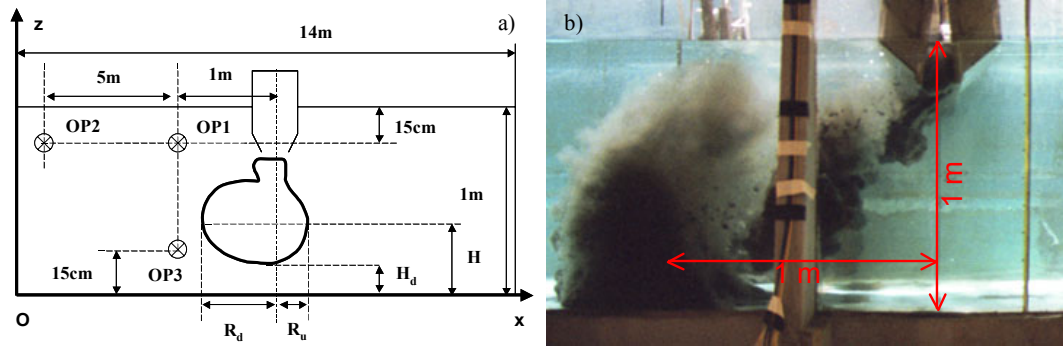


Figure 1: Definition sketch: a) location of optical sensors (OP) for turbidity measurements; b) mud dumping [6]

Villaret et al. [6] realised a laboratory experiment of sand release in a straight channel of 72 m long, of 1.5 m wide and of 1.5 m height. A specifically designed recipient (maximum capacity of 60 liters) is placed at 15 cm below the free-water surface. The ambient current is maintained with the help of a hydraulic pump. The sand-water mixture is initially filled-up in the recipient at a given concentration. Then the bottom of recipient is suddenly opened (Figure 1b) and the measurement (photo-camera and concentration) starts simultaneously. Table 1 gives an index of the different testing conditions.

Table 1. Testing conditions and nomenclature:  $W_{inj}$  is the injection velocity from the recipient,  $D_p$  the sediment particle diameter (90  $\mu\text{m}$ ),  $\rho$  the dry density of the solid (2650  $\text{kg/m}^3$ ),  $C_m$  is the concentration of the mixture (450 g/l),  $V_r$  the volume of dumped material (60 l),  $U_c$  the ambient velocity [6].

Tests	e6	e11	e12	e13	e14	e15	e16	e17	e18	e19	e20
$W_{inj}$ (m/s)	0.6	0.79	0.89	0.79	0.89	0.79	0.89	0.79	0.89	0.79	0.89
$D_p$ ( $\mu\text{m}$ )	90	90	160	90	160	90	160	160	90	160	90
$C_m$ (g/l)	350	450	450	450	450	450	450	450	450	450	450
$V_r$ (l)	45	60	60	60	60	60	60	60	60	60	60
$U_c$ (cm/s)	0	0	0	10	10	15	15	20	20	25	25

#### IV TWO-PHASE MODELING & RESULTS

The 2-D X/Z computation domain extends over 14m in the  $x$ -direction (centred at the release location) and over 1m on the vertical one ( $z$ -ascendant coordinates). A regular mesh of 61x1401 nodes as well as a time step of  $\Delta t=0,001$  s are used. The numerical parameters are set in Table 1. The lateral ends are considered as open boundaries. At the injection location (inlet diameter equal to 10 cm), the fluid and sediment fluxes are imposed following a Poiseuille profile with a given maximum velocity ( $W_{inj}$ ).

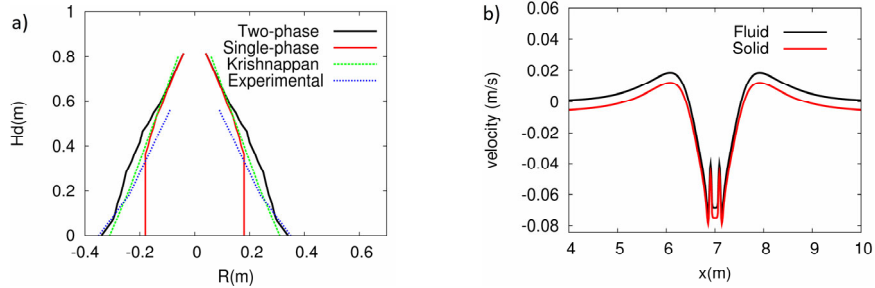


Figure 2: Test 11: a) numerical and experimental radius of the sediment cloud; b) vertical velocity profile at  $z=-0.83$  m

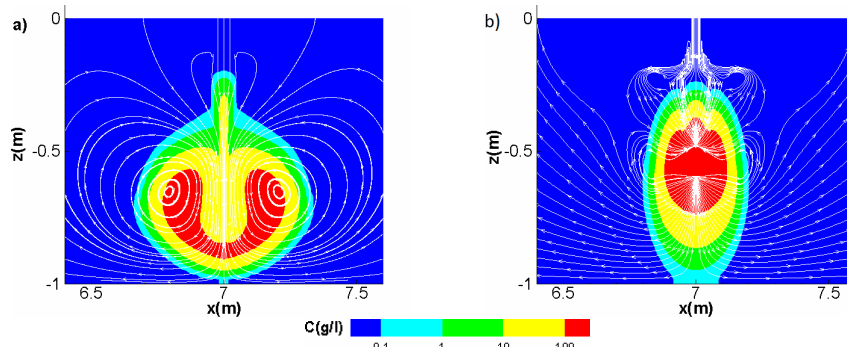


Figure 3: Simulation results for test e11: a) by two-phase approach; b) by single-phase one

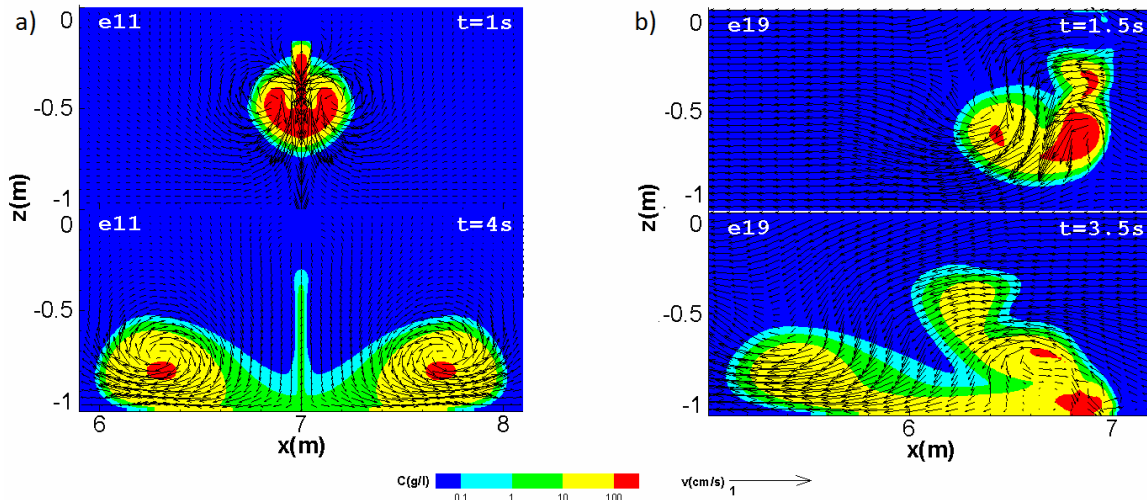


Figure 4: Development of two counter-rotating vortices: a) test e11; b) test e19: solid phase velocity.

Figure 2a presents the time evolution of the maximum radius  $R$  of the sediment cloud obtained by the two-phase model and the single-phase one, compared with the experiment [6] and the analytical solution proposed by Krishnappan [8]. Clearly, the two-phase model results are fairly in agreement with the experiment and the analytical solution, whereas the single-phase model results more and more differ the experimental values during the second half period of convective descent phase. Indeed Figure 2a shows that over this period, the single-phase model gives a radius, which is nearly constant

( $\approx 18$ cm). This can explain by the fact that the development of the sediment cloud in a single-phase model is only engendered by the density gradient, and no solid-fluid interaction is taken in consideration. Yet this interaction that express by the drag force is dominant in such a flow. The longitudinal profiles of vertical velocity plotted in Figure 2b show that the vertical velocity of the solid phase is always larger than the fluid phase one. Figure 3 presents the iso-value contours of sediment concentration as well as streamlines at  $t=1.5$  s obtained from the two-phase model (Figure 3a) and the single-phase one (Figure 3b) for test *e11*. Obviously the two-phase model reproduces two counter-rotating vortices, which allow a full development of the sediment cloud. As the cloud grows, the surrounding water is entering inside the cloud while the iso-value contour is expanding. This illustrates that the horizontal velocity component of fluid- and solid- phase are unequal in the direction and magnitude. No counter-rotating vortices are observed in the single-phase model results. This is why the sediment cloud obtained by the single-phase model is not developed as physically expected. The experimental observations are therefore qualitatively well captured by the two-phase model.

Figure 4 presents the evolution of the sediment cloud as well as solid velocity fields in still water (Figure 4a) and in a water flow of velocity,  $U_c=25$  cm/s (Figure 4b). Note that the conservation of mass in the computation domain is good enough as far as the relative error is around 0.05%.

## V CONCLUSIONS

This study presents numerical simulations of dredged sediment disposal with and without ambient currents by using a two-phase flow model. This modelling well suit to such cases with fluid and solid phase velocities (all the components) differ each other in amplitude and in direction. The numerical results obtained by the model are fairly in agreement with experimental data during the convective descent phase. The formation and propagation of density current are also realistic with experiments. For the parameters remaining inaccessible from the experiment such as the concentration field, the formation of counter-rotating vortices, the influence of ambient current or particle diameter, the numerical model provides an estimate or a representation that is in qualitative accordance with other published data. The actual study deals with non-cohesive sediments. For such a case, no consolidation effect is encountered. To deal with cohesive sediment, the consolidation routine should be activated but more important is the need to account for the flocculation/deflocculation processes that should play a fundamental role in the dynamic of the system.

## REFERENCES

- [1] 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 Development and verification results, Department of the Army, Waterways Experiment Station, Corps of Engineers, Vicksburg, MS 39180-6199.
- [2] Garapon A., Villaret C., Boutin R. (2002) – *3D numerical modelling of sediment disposal*, Proceedings of the Physics of Estuarine. PECS, 13 p.
- [3] Farout-Freson I., Sergent P., Lefrançois E., Dhatt G. (2006) – *Modèle numérique de clapage – phase de chute*, IX<sup>èmes</sup> JNGCGC, Brest, pp 179-186. doi:10.5150/jngcgc.2006.018-F
- [4] Nguyen K.D., Guillou S., Chauchat J., Barbry N. (2009) – *A two-phase numerical model for suspended-sediment transport in estuaries*. Advances in Water Resources, **32**, pp 1187-1196.
- [5] Guillou S., Chauchat J., Pham Van Bang D., Nguyen D.H., Nguyen K.D. (2011) – *Simulation of the dredged sediment's release with a two-phase flow model*. Bulletin of the Permanent International Association of Navigation Congresses, **142**, 25-33, ISSN 0374-1001.
- [6] Villaret C., Claude B., Du Rivau J.D. (1998) – *Etude expérimentale de la dispersion des rejets par clapage*. He-42/98/065/a, LNHE, EDF.
- [7] Chauchat J., Guillou S. (2008) – *On turbulence closures for two-phase sediment-laden flow models*. J. Geophys. Res., **113**, C11017.
- [8] Krishnappan B.G. (1975) – *Dispersion of granular material dumped in deep water*. In Scientific series, Environment Canada, **55**, 113 p.