TOWARDS IMPROVED PREDICTION OF DREDGING PLUMES: NUMERICAL AND PHYSICAL MODELLING

by

Boudewijn Decrop[1] and Marc Sas[2]

ABSTRACT

Construction and maintenance of ports and waterways involves dredging activities in many cases. Dredging projects require assessment and mitigation of a number of environmental impacts. Some of the potential impacts are related to turbidity plumes resulting from hydraulic and mechanical dredging processes bringing sediment into suspension.

In the recent past, environmental awareness and by consequence environmental legislation has become stronger. As a result, dredging contractors and dredging consultancy have been faced with the challenge to implement better control mechanisms for environmental management purposes. More specifically, turbidity plumes have been monitored closely in the past to follow up on their fate. Numerical simulations allow for real-time forecasting of the fate of the turbidity plumes in the near future. By means of a well-calibrated tidal flow model, planned dredging activities can be implemented as sediment sources in the numerical flow models. In this way, the plume dispersion due to interaction of the tidal flows and the timing of activities spilling sediments can be predicted up to a week ahead.

In the past, large-scale numerical flow models have been applied, and covered the wider areas around the project site that can potentially be affected by the works. Overflow losses from Trailer Suction Hopper Dredgers (TSHD) are one of the main sediment spills during the execution of dredging projects. In the past, near-field sediment distributions from overflow spills have been determined using simplified laws and crude estimates of losses.

In the presented work, efforts have been made to improve the accuracy of plume simulations by performing highly-detailed computational fluid dynamics (CFD) simulations of the flows of the water-sediment-air mixture around the ship hull and its interaction with the propellers. These detailed simulations have several benefits, such as assessment of overflow design and insights in the three-dimensional distribution of sediments near the dredger, but are too time-consuming to be used in operational forecasting of turbidity plumes. In the work presented in this paper, the CFD results have been applied to develop a parameterized model, significantly faster compared to the CFD simulations, but more accurate compared to the previous generation of near-field models.

The coupling of this new generation of near-field spill models with the far-field (large scale) flow models allows for a significant increase in accuracy of turbidity forecasting. In this way, using forecasting models, dredge and disposal productions can be optimised while complying with turbidity levels imposed in the environmental criteria.

The presented paper has been submitted before to the jury of the De Paepe-Willems Award 2017 and went on to receive the first prize, handed over to the first author at the PIANC AGA in Cairns, Australia on June 19th, 2017.

[1] International Marine and Dredging Consultants (IMDC), Belgium, Boudewijn.Decrop@imdc.be

[2] International Marine and Dredging Consultants (IMDC), Belgium, Marc.Sas@imdc.be

1. INTRODUCTION

Construction and maintenance of ports and waterways involves dredging activities in many cases. Dredging projects require assessment and mitigation of a number of environmental impacts. One of these impacts is increased turbidity and sedimentation due to dredging and disposal activities bringing sediments in suspension. Reduction of light penetration, increased sedimentation and increased suspended sediment concentrations can have potentially adverse effects on sensitive habitats (eg. coral & seagrasss ecosystems) or nearby human activities (eg. aquaculture, industrial/drinking water intakes). The extent of the impacts will depend on the quantity, frequency and duration of dredging, adopted methodology, site-specific conditions (wind, wave and current fields, grain-size distribution and water depth), proximity to sensitive sites and tolerance of living organisms to altered turbidity conditions (PIANC, 2010).

Increased awareness has instigated stricter environmental legislation related to these activities. Project environmental permits often stipulate project-specific regulations, which can entail strict turbidity thresholds for these activities. Operational turbidity management in these projects is warranted, as exceeding turbidity thresholds can trigger corrective measures, increased monitoring efforts, relocation of dredge activity, a decrease in or –worst case - a cease of dredging and dredge spoil placement activities.

The modelling tools presented in this paper add to the development of a system in which real-time predictions of the plume behaviour can be achieved. The operational planning of dredge operations for the next few days can be implemented in a forecast model environment, In case a breach of turbidity thresholds is predicted, the operational planning can be revised or altered to avoid breaches.

Trailer Suction Hopper Dredgers (TSHD's) often deploy an overflow system through which excess sea water is skimmed from the hopper (Figure 1). The released water contains a varying concentration of fine sediment material, of which a fraction can form turbidity plumes. In the presence of currents, these plumes are advected over longer distances. The plumes can affect environmentally sensitive areas throughout coastal, river or offshore systems (Bray, 2008).

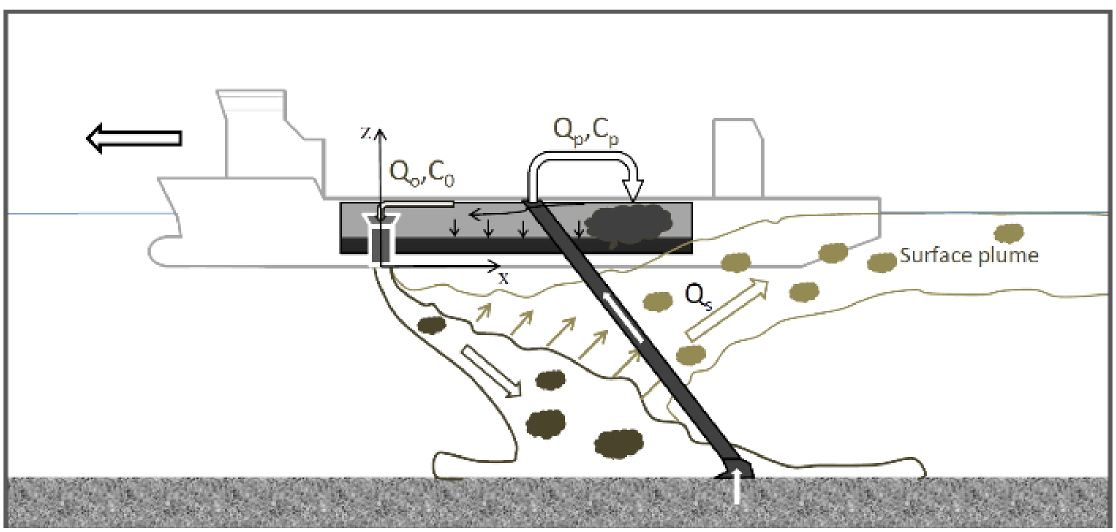

Figure 1. Sketch of a Trailer Suction Hopper Dredger, pumping a water-sediment mixture (mass concentration C_p) towards the hopper, at volume discharge Q_p. The sediment mixture settles while flowing towards the overflow, where a lighter mixture with sediment concentration C0 is released, with discharge Q_0.

Today, these environmental impacts are assessed using predictive simulations based on large-scale modelling of dredging scenarios. Since large-scale modelling of (tidal) currents in estuaries and regional seas is usually executed using a simplified, hydrostatic form of the Navier-Stokes equations (Saint-Venant equations), it is useful to divide dredging plumes in two parts. The first part is the near-field section of the plume, near the exit of the overflow shaft. This part is called the dynamic plume since it is still under influence of the ship and the excess density of the water-sediment-air mixture. The excess density of the plume can be expressed as

$$\Delta\rho=\rho_m-\rho_\infty=c(1-\rho_\infty/\rho_s)-\phi_a(\rho_\infty-\rho_a) \quad (1)$$

where ρ_∞ is the mass density of the sea water, ρ_m the mass density of the sediment-water mixture in the plume, c is the sediment mass concentration, ρ_s the mass density of the sediment material, ϕ_a is the volume fraction of air bubbles and ρ_a is the mass concentration of air bubbles.

The second part of the plume starts where the released mixture is diluted to the point where the plume bulk density ρ_m is no longer significantly higher compared to the surrounding sea water. This part is called the passive plume since it is passively advected with currents in the sea waters, with settling of flocs and benthic aggregates as main process (e.g. Smith and Friedrichs, 2011). Large-scale modelling tools are capable to solve the passive part of the plume. However, in the modelling of dredging plumes the dynamic part of the plume is still a missing link between the sediment discharge at the overflow exit and the passive part of the plume. Today this gap is bridged by roughly estimating the sediment flux to the passive plumes, as a fixed percentage of fines in the production. It has been shown that this percentage can vary widely, even within one loading cycle (Decrop *et al.*, 2014; de Wit *et al.*, 2014).

In the past, the near-field modelling done for the transformation of the bulk overflow outflow to a vertical distribution of the sediment behind the vessel has also been executed with integral models representing the integrated Navier-Stokes equations for a buoyant jet in crossflow (Fischer, 1979; Jirka, 2006). Such models were implemented by e.g. Spearman (2011). However, this type of model cannot incorporate the formation of a surface plume due to the complex flow pattern around the vessel and due to air bubbles. Instead, a mere constant factor can be applied.

In this paper, an overview is given of the tools developed by the author in the recent past, more specifically a highly detailed 3D CFD model and a fast parameter model. Today's application of the developed tools is discussed, leading to improvement of turbidity assessment in planning phase and in operational phase. The presented research was executed by IMDC with additional funding from IWT in Belgium. Special thanks go to professors De Mulder (Hydraulics Laboratory, Ghent University) and Toorman (Hydraulics Laboratory, KULeuven) for supervising the research.

2. NEAR-FIELD CFD MODEL

A 3D numerical simulation model has been developed in the Ansys Fluent™ environment. The aim of this Computational Fluid Dynamics (CFD) model is to represent accurately the flow patterns of the water-sediment-air mixture in the direct vicinity of a hopper dredger while trailing.

In the following sections, a short overview of the model development is given, as well as results of the analysis of influencing factors on plume dispersion.

2.1 Set-up and validation

The 3D CFD model solves for the velocity vectors of three phases: water, sediments and air bubbles. The following approach was chosen to handle the three phases. First, the momentum and continuity equations for a mixture of water and sediments are solved. Then, the relative velocity (slip velocity) of the sediment compared to the water velocity is determined by including the effects of settling and drag (Manninen *et al.*, 1996) and including the effect of turbulent diffusion using a drift flux term. This method is allowed when the expected slip velocity is low. The slip velocity of air bubbles in water has a much higher range compared to the sediments in the overflow plume, which are mainly fines. Therefore, a different approach was used for the air bubble dispersion. It was solved using a Lagrangian approach in which the acceleration vector is determined from a force balance consisting of drag, gravity, virtual mass and pressure gradient (Decrop *et al.*, 2014).

The input of momentum and swirl due to the propellers is modelled using an actuator disk approach. In this approach, the axial and tangential velocity components - as a function of the radial distance from the hub - are imposed by implementing a pressure jump over a circular surface in the model grid.

The actual geometry of an existing TSHD is embedded in the model grid. The model domain is discretised using an unstructured grid. This allows for an accurate representation of the complex geometry of the ship. Refined grid cell layers are included to resolve the velocity profiles at the wall boundary layers at ship hull and in the overflow shaft (Figure 2). Additional grid refinements were included in regions of plume presence, strong gradients, strain near the bow, propeller flow and near the sea bed. The surface mesh at the hull was refined in zones of strong curvature to represent the shape in an optimal way.

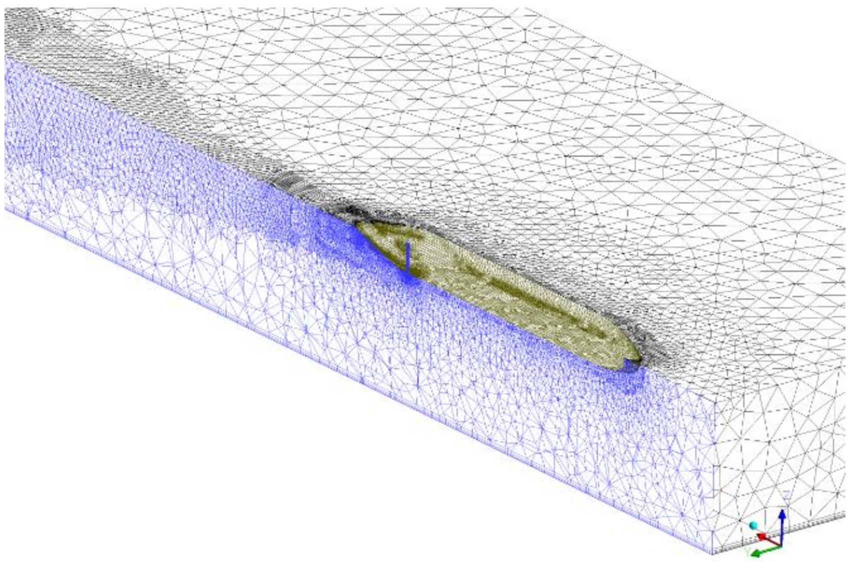

Figure 2. Example of a model grid, tailored to a specific plume case for computational efficiency. Grid is sliced along the ship axis. Ship hull grid in brown, water surface grid in black, subsurface grid along slice in blue.

The turbulent flow field was solved using the Large-Eddy Simulation (LES) technique, in which the larger turbulent vortices are explicitly resolved on the grid. Amongst other reasons, this is needed to include the interactions between an individual vortex and the local sediment gradients near the edges of the plume.

The CFD model was validated in a number of steps. Two major steps were taken: validation of a laboratory-scale model (Decrop *et al.*, 2015a) and validation of a full-scale model. A thorough validation of a laboratory-scale CFD model was executed based on measurements taken in a physical model. The CFD model results were compared to highly-detailed measurements of sediment concentration, turbulent sediment fluxes, mean flow velocity components (*U, V, W)* turbulent velocity fluctuations and, finally, the Reynolds stress

After the detailed validation at laboratory scale, the CFD model was converted to a full-scale model, including a realistic vessel geometry, propellers and overflow shaft (Figure 3).

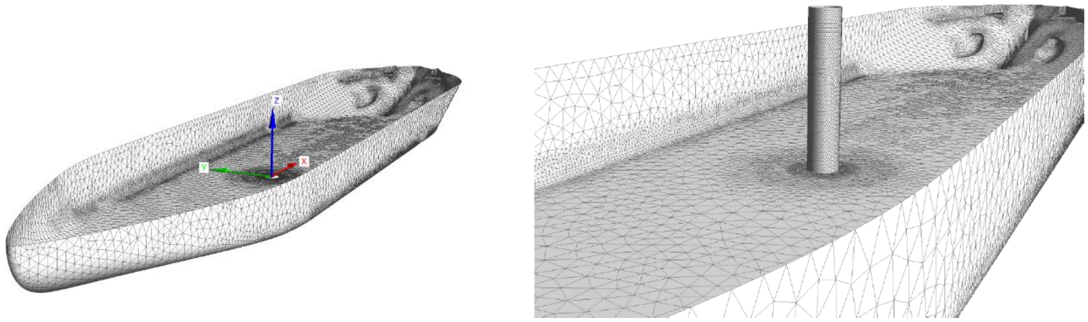

Figure 3. Details of the surface mesh at the TSHD hull and overflow shaft walls.

A measurement campaign was set up to measure a real-life overflow dredging plume in the field (Decrop and Sas, 2014). Detailed vertical profiles of sediment concentration were recorded throughout the full water column, from surface to 2 cm above the sea bed. In this way, also the near-bed highly-concentrated mud layers could be monitored. Also, sediment concentration observations were conducted closer to the surface and along the complete plume using Optical Backscatter instruments towed behind the survey boat. Finally, Acoustic Doppler Current Profiler (ADCP) measurements have been taking, allowing for the visualisation of the concentration levels along a vertical slice of the plume.

Subsequently, CFD simulations were set up representing identical ambient conditions as observed in the field. The results of the simulations were compared with the field data, as a validation exercise for the full-scale CFD model. Suspended sediment concentration measurements in the surface plume matched well with the CFD model (Figure 4, upper panel). Also, observations of the deeper parts of the overflow plumes corresponded well with the CFD model (Figure 4, lower panel).

Two examples of results of the CFD model are shown in Figure 5. In this example, the head-current is relatively strong, leading to a significant surface plume (with the potential to travel long distances).

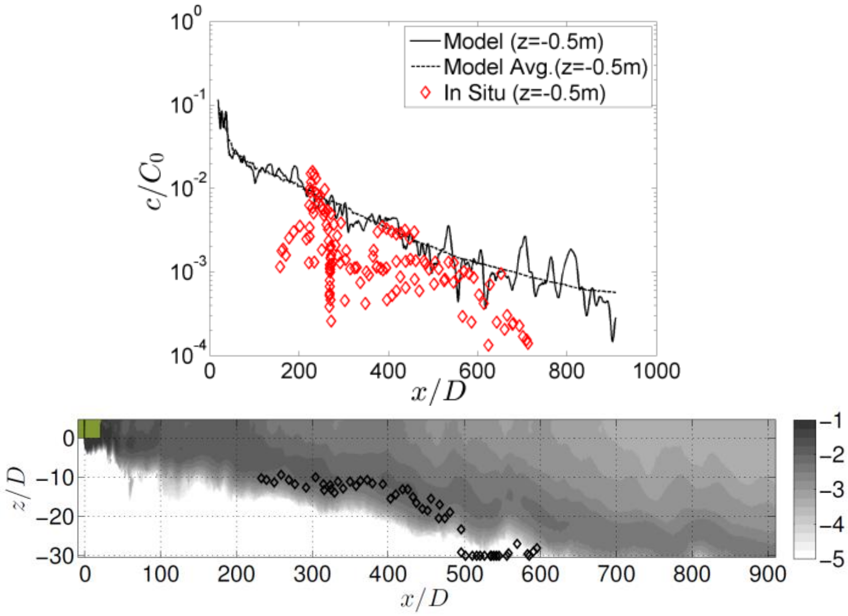

Figure 4. Upper panel: range of observed sediment concentration *c/C₀* (red markers) compared to the centerline (maximum) *c/C₀* values from the CFD model. Lower panel: Vertical slice of CFD sediment concentration log(*c/C₀*) (along the ship axis, in grayscale), compared to the lower edge of the plume as observed in the field (black diamonds).

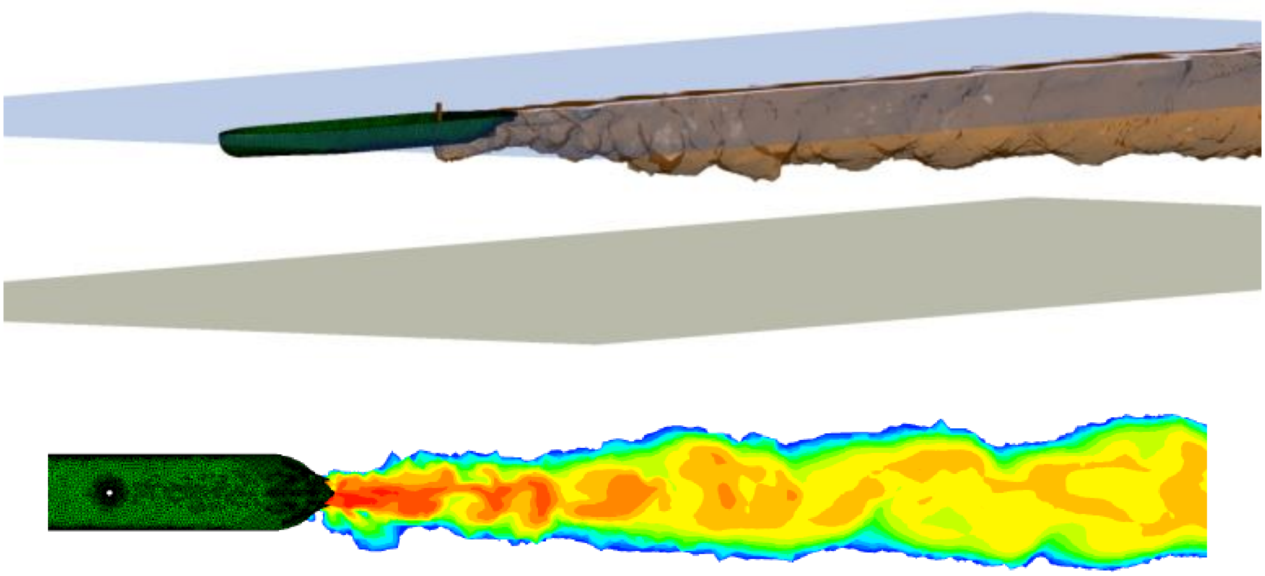

Figure 5. Examples of the turbulent sediment plumes computed using the CFD model. The isosurface connecting the locations at which $c/C_0=10^{-4}$ is shown in the top panel. The lower panel gives a top view of the computed sediment concentration at the surface.

2.2 Sensitivity of plume dispersion to boundary conditions

The main goal of near-field CFD modelling is to compute the vertical and horizontal distributions of the overflow behind the ship, for far-field model input. However, the full three-dimensional fields of flow velocity and concentrations of sediment and air bubbles are also available for analysis. For example, the influence of a number of operational and environmental parameters has been studied by comparing two simulations in which only one parameter has been changed. These insights have in later stages been used for the development of simplified models.

2.2.1 Influence of dredging speed

First, two simulations are carried out in which a water-sediment mixture is released with a varying average velocity (Figure 6). It is shown that the stronger relative flow velocity induces an increase of the surface plume sediment concentration with a factor 10.

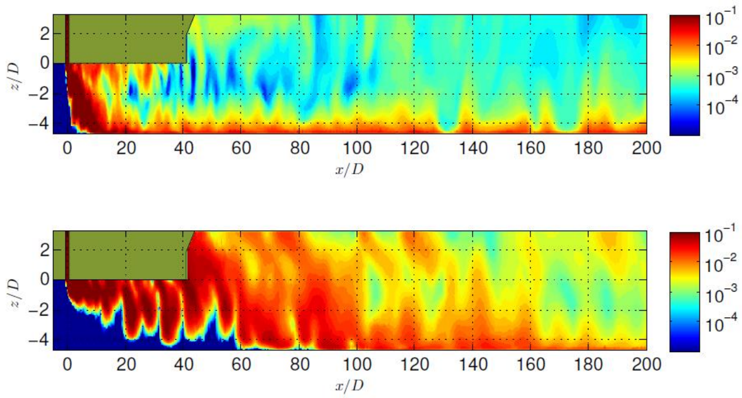

Figure 6. CFD results (relative concentration *c/C₀*) with low dredging speed (top panel) and with high dredging speed and/or head current (lower panel).

2.2.2 Influence of mixture density

Secondly, simulations are carried out in which the flow velocity relative to the ship, *U0*, is kept constant at 1 m/s, while the other parameters are equal to previous case. The overflow sediment concentration C_0 is equal to 10 g/l in one simulation (Figure 7, top panel) and to 150 g/l in the other (Figure 7, lower panel). It can be seen very clearly that the fraction of the released sediments going to a surface plume (surface value of c/C_0) is much higher (up to factor 100) when the overflow mixture is light. The mixture does not have sufficient excess density to descend to the sea bed.

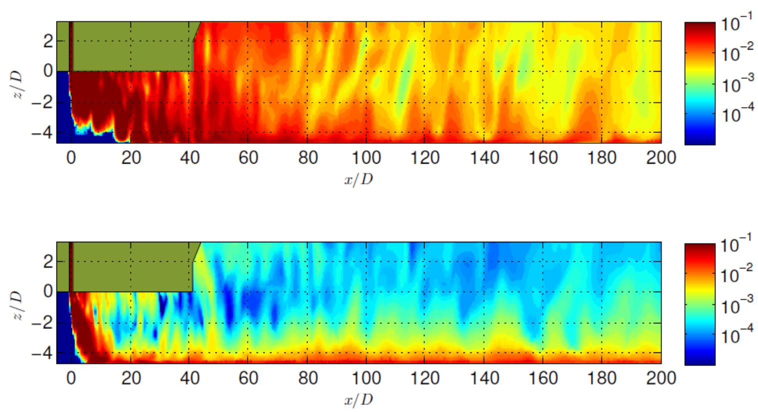

Figure 7. CFD results (relative concentration c/C_0) with low C_0 (10 g/l; top panel) and with high C_0 (150 g/l; lower panel).

2.2.3 Influence of air bubbles

The influence of the entrainment of air bubbles into the overflow shaft was also investigated. More specifically, the influence of the so-called green valve was investigated. The green valve is designed to reduce the turbidity in the water column by choking the flow, reducing the number of air bubbles in the overflow and by consequence reducing the uplifting effect thereof. It is usually assumed that the green valve has an effect under all circumstances. It was shown using the CFD model that the effectiveness of the green valve is largely dependent on the ambient conditions and overflow mixture properties. See Decrop *et al.* (2015b).

3. NEAR-FIELD PARAMETER MODEL

3.1 Introduction

For the study of the behaviour of specific plume cases, or for gaining insights in the effects of operational aspects on the plume behaviour, a CFD model is very valuable. In some phases of a dredging project, however, the long simulation times associated with it are not always acceptable. In the operational project phase, real-time plume predictions are needed to assess the timing and location of dredging in the day-by-day planning of works. At this stage, the long simulation times of the CFD model are prohibitive.

The large-scale simulation of the far-field plumes is generally executed with a shallow-water equations-based hydrodynamic flow model with a sediment transport equation and a source term for the overflow releases. The source term which has to be supplied to the large-scale model needs a vertical distribution. A parameterised model has been designed to perform this task. Essentially, this model is a trade-off between calculation speed and accuracy. It is less accurate compared to a CFD model, but much faster and therefore applicable in cases where the CFD model is not possible, e.g. real-time forecasting simulations.

3.2 Model Set-up

First, the different length scales and fluxes need to be condensed into non-dimensional numbers. This makes the parameterisation of the vertical flux profiles more generic.

In Figure 8, the different scales are sketched. The water depth H is the sum of the TSHD draft H_d and keel clearance H_k. The distance between the overflow and the stern is denoted as L_o.

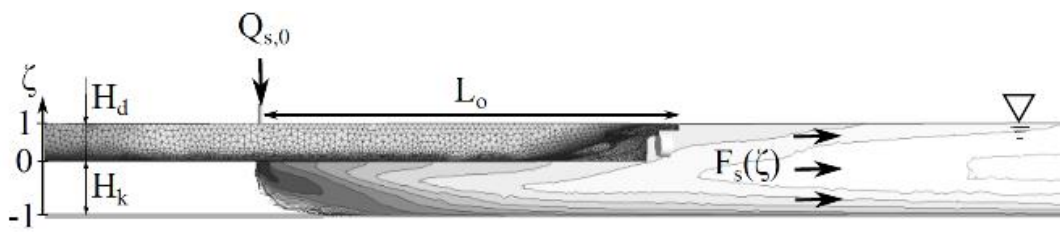

Figure 8. Definition sketch of the parameter model and variables used therein.

The vertical coordinate z can now be scaled to a dimensionless coordinate ζ, equal to -1 at the sea bed, to 0 at the keel and to 1 at the water surface. This transformation allows to make use of Chebychev polynomials for the parameterisation of the shape of the vertical profile of sediment flux.

From a large number of CFD simulations, representing the full range of realistic boundary conditions, results are extracted. These results are used to determine vertical profiles of sediment flux, used as data set to fit the parameters in the parameter model. The profiles are determined as follows. The time-averaged sediment flux *f* in the sediment plume (in kg/(s.m)) is defined:

$$f(x,y,\zeta) = C(x,y,\zeta)U(x,y,\zeta) \tag{2}$$

with C and U the time-averaged sediment concentration and flow velocity.

The flux q_s is integrated over the width of the plume, determined as:

$$q_s = \int_{-B/2}^{B/2} f(x,y,\zeta)dy \tag{3}$$

Where *B* is the width of the plume. At this point we have a sediment flux in kg/s in the plume at every location along x and per dimensionless unit of height (ζ is non-dimensional). A distance x_p needs to be defined at which the vertical profile of q_s is evaluated. A fixed distance is defined at which the CFD model output is evaluated and by consequence at which the parameter model is valid. The distance x_p was chosen at $2.5L_s$, with L_s the vessel length. At this distance from the vessel, the parameter model output is valid for implementation in a far-field model.

The vertical profile of the flux that will be parameterised is non-dimensionalised and defined by:

$$F_s(\zeta) = q_s(x_p,\zeta)/Q_{s,0} \tag{4}$$

where $Q_{s,0}=C_0Q_0$ is the sediment outflow from the overflow, C_0 is the overflow sediment mass concentration, Q_0 is the volume discharge through the overflow.

For each CFD result in the data set, the profile $F_s(\zeta)$ is determined at x_p=2.5*L*.

The next step is to parameterise the shape of the vertical profiles of F_s. The parameters describing the shape of the profiles will then be linked through a multivariate regression to the boundary conditions such as current velocity, sailing speed, et cetera. Depending on the ambient conditions and overflow jet exit conditions, two distinct types of plumes can be distinguished: the near-bed density current and the seabed-detached plume. The shape of the vertical flux profile of both types of plumes is clearly different (Figure 9).

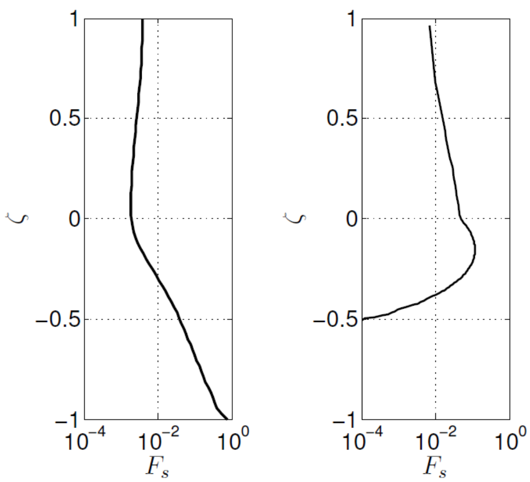

Figure 9: Vertical profile of F_s for two types of plumes: Near-bed density current type (left panel) and the seabed-detached plume (right panel).

In order to select which type of profile will occur, a preliminary estimate of the vertical position of the plume centerline at $x=x_p$ is required. For this purpose, the Lagrangian model for the trajectory of buoyant jets of (Lee and Cheung, 1990) and (Lee and Chu, 2003) is used as a starting point, with corrections based on regression analysis using the CFD results. In case the preliminary plume centreline is at $\zeta <$

-0.75, the plume is close to the sea bed and considered of density current type. If $\zeta > -0.75$, the plume is defined as type seabed-detached.

The shape of these profiles was then parameterised using either a Chebychev polynomial (density current) or a piecewise-linear (seabed-detached) approach.

The first type, the density current type, usually has a relatively smooth profile, and can be approximated using Chebychev polynomials, see e.g. (Lopez, 2001). In this method, a weighted sum of polynomials with order zero to n is considered (eq. 5). The coefficients ψ_i in the weighted sum are fitted to each case in the data set of CFD model-based plumes. Here, polynomials with n=3 provided sufficient capability of following the shape of the profiles:

$$F_s(\zeta) \;=\; \sum_{i=0}^{n} \psi_i \, T_i(\zeta) \tag{5}$$

where Ti are the Chebychev polynomials of the first kind and ψ_i are n+1 coefficients.

For the second type of plume, the seabed-detached plume, a step-wise parameterisation of the flux profile is proposed. The reason for the different parameterisation is the fact that this type of profile is often less smooth, with a sharp edge at the position of the bottom of the plume where the sediment concentration goes to zero rapidly. Fitting using Chebychev polynomials induces wiggles due to the sharp edge. In Figure 9, the step-wise parameterisation (during model training) for both types is shown.

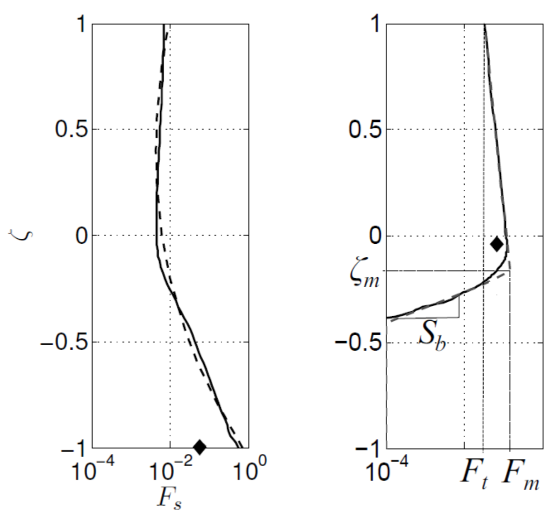

Figure 10: Vertical profile of F_s for two types of plumes. CFD results in full lines, parameterisations in dashed lines, preliminary plume centerline position in black diamond. Near-bed density current type with Chebychev parameterization (left panel). Right panel shows the 'detached' plume type with step-wise parameterisation (dashed line), defined by (co-) ordinates ζ_m, F_t and F_m and the slope S_b.

This gives a total of four parameters to fit to the data set, for both the Chebychev (ψ_i) and stepwise linear approach (ζ_m, S_b, F_t and F_m). These parameters are for that purpose defined as linear functions of the physical quantities of influence such as the solid discharge ($Q_{s,0}$), vessel draught (H_d), keel clearance (H_k), distance of overflow to stern (L_0), the ratio of outflow-to-crossflow velocity (λ) and the densimetric Froude number F_Δ:

$$F_\Delta = \frac{W_0}{g\, D \frac{\Delta\rho}{\rho_\infty}} \tag{6}$$

Where W_0 is the overflow exit velocity, D is the diameter and $\Delta\rho$ as defined in eq. (1).

Using multivariate linear regression, the relationship between the physical boundary conditions and the profile shape parameters was established, by fitting to a large set of CFD results. Here, this process is called model training.

3.3 Model Training

A training data set of 50 CFD simulations was used to relate the parameters to the different boundary conditions of the plume. These boundary conditions consist of dimensionless combinations F_Δ, λ, H_d/L_o, etc. The ranges of these conditions covered by the training data set determines the validity of the model, in this case: 1.2<F_Δ<14.2, 0.5<λ<4, 0.07<H_d/L_o<0.26 and 1<H_k/D<30.4.

The Chebychev coefficients ψ_i (eq. 5) were found to depend mainly on F_∞ (=F_Δ / λ) and the ratio H_k/D. For each coefficient, a multivariate regression is fitted with these two dependent variables. The training data set cases are used for finding $\beta_{c,i}$ (3 x 4=12 coefficients):

$$\psi_i = \beta_{c,i,0} + \beta_{c,i,1} F_\infty + \beta_{c,i,2} \left(\frac{H_k}{D}\right)_m + \epsilon_{i,m} \tag{7}$$

where i=0,...,3 is the number of the Chebychev coefficients, m=1,...,M, with M the number of CFD simulations in the data set, $\beta_{c,i,j}$ are the coefficients to fit and $\epsilon_{i,m}$ are error terms.

The parameters for the step-wise profile of the seabed-detached plumes were found to be best represented as a function of the following near-field plume conditions: F_∞, the ratio H_d/L_o and the ratio H_k/D. For each parameter, a multivariate regression is fitted with these three dependent variables. The training data set cases are used for finding β_d (4x4=16 coefficients):

$$(F_t, F_m, \zeta_m, S_m) = \beta_{d,0} + \beta_{d,1} F_{\infty,m} + \beta_{d,2} \left(\frac{H_d}{L_o}\right)_m + \beta_{d,3} \left(\frac{H_k}{D}\right)_m + \epsilon_m \tag{8}$$

where, m=1,...,M, with M the number of CFD simulations (with seabed-detached plume) in the data set, $\beta_{d,j}$, j=1,2,3, are the coefficients to fit for each of the profile parameters (F_t, F_m, ζ_m, S_m). ϵ_m are error terms.

More details on the mathematical description and parameter settings of the parameter model can be found in Decrop (2015).

3.4 Validation

A second set of CFD simulations not used for the fitting of the coefficients of the multivariate regression was then applied as a validation data set. It turns out that the vertical profile of sediment flux in an overflow plume can be predicted reasonably well for most standard cases by this parameterised model.

Out of a total of 40 validation cases, 75% had a coefficient of determination (R^2) of 0.7 or higher. Two examples of comparison between CFD model results and parameter model results are shown in Figure 11. The parameter model is valid for a large range of boundary conditions and can after this validation exercise be used for generating source terms in a large-scale model, including the interaction between currents, vessel operation and plume behaviour.

3.5 Application in far-field plume models

The near-field sediment flux profiles generated by the parameter model can thus now be imposed as source terms in a large-scale plume dispersion model, in a coupled way. Effectively, an online coupling was established between the near-field parameter model and the TELEMAC code (EDF R&D, 2013) for solving large-scale hydrodynamics and sediment transport: the parameter model provides realistic sediment distributions for the sediment sources in the far-field model in TELEMAC, while the TELEMAC hydrodynamic solver provides water depth (i.e. ship keel clearance H_k), flow velocity and direction (changing in time) as an input for the near-field parameter model (Figure 12). The user provides the static variables to the parameter model at the start of the simulation: overflow discharge Q_0, diameter D and sediment concentration C_0, vessel length L_s and overflow position L_0. Vessel speed and course can be provided as an a priori-defined time series.

In this way, the sediment spill and its vertical distribution is fully adapted to the ambient conditions at any time step in the model.

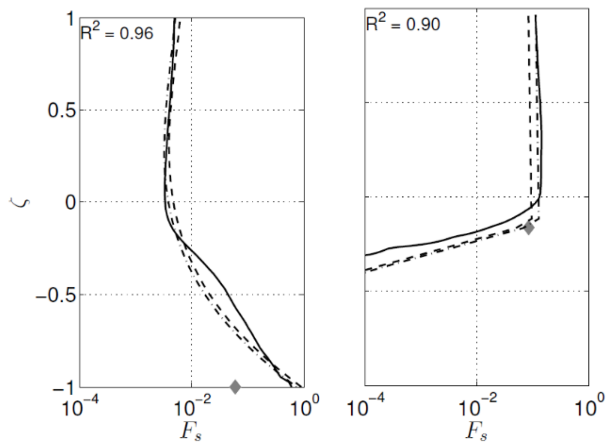

Figure 11. Two validation cases for the parameter model. Full lines represent vertical profiles of sediment flux from the CFD model, dashed lines are the results of the parameter model (before and after a corrector step ensuring the sediment flux continuity is respected).

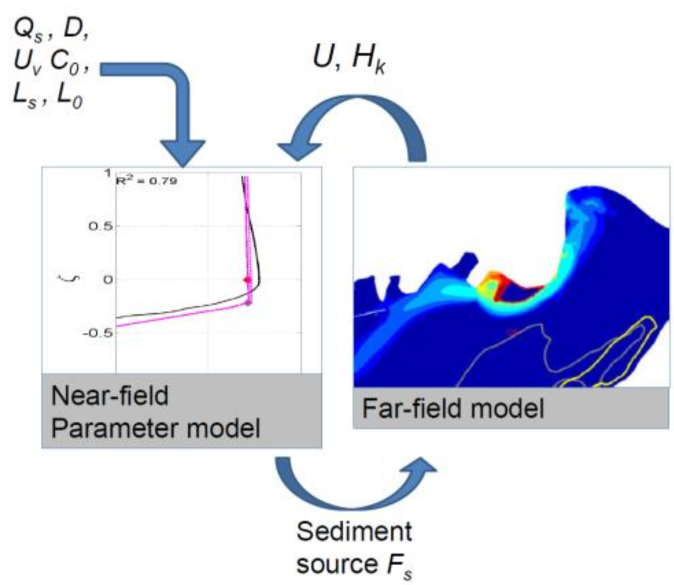

Figure 12. Schematic representation of the online coupling between near-field parameter model and far-field hydrodynamics/plume dispersion in TELEMAC.

3.5.1 Dredging scenarios

In a tender phase or during the phase of planning of the dredging works, predictions of the plume behaviour are calculated in a number of alternatives, to assess predicted compliance with environmental criteria. For example it can be decided which part of a dredging zone should be dredged during neap tide and which part during spring tide conditions.

4. DISCUSSION AND CONCLUSION

A highly-detailed CFD simulation model of near-field overflow has been developed. It was used to gain insight in the highly complex flows of water-sediment-air mixtures behind overflowing TSHD's. Further, it was used in the development of a faster parameter model of near-field overflow plume dispersion.

The sediment flux profiles generated by the newly developed parameter model can now be imposed as source terms in a large-scale plume dispersion model, where the parameter model inputs are coupled with the large-scale flow properties. In this way, the fraction of released sediments moving to the large passive plume is determined every time step. This is a significant improvement over the rather arbitrarily chosen constant value used in the past.

Currently, the grey-box model is applied by consulting engineers at IMDC during environmental impact assessment of port development and maintenance, both in scenario analysis and in a real-time plume forecasting system. Operational analysis of undesired sedimentation of the underwater work areas in between construction phases is another example of application.

In the future, the same approach - CFD simulations of the detailed processes and parameter model fitting based upon it - can be applied to other types of sediment spills from dredging and disposal activities related to port and navigation channel construction.

References

Bray R.N. (2008). "Environmental aspects of dredging. CRC Press".

de Wit L., van Rhee C. and Talmon A. (2014). "Influence of important near field processes on the source term of suspended sediments from a dredging plume caused by a trailing suction hopper dredger: the effect of dredging speed, propeller, overflow location and pulsing". *Environ. Fluid Mech.*, 1–26.

Decrop B. (2015). "Numerical and Experimental Modelling of Near-Field Overflow Dredging Plumes", PhD Thesis. Ghent University.

Decrop B., De Mulder T., Toorman E. and Sas M. (2015a). "Large-eddy simulations of turbidity plumes in crossflow". *Eur. J. Mech. BFluids*, 53, 68–84, doi: http://dx.doi.org/10.1016/j.euromechflu.2015.03.013.

Decrop B., De Mulder T., Toorman E. and Sas M. (2015b). "Numerical Simulation of Near-Field Dredging Plumes: Efficiency of an Environmental Valve". *J. Environ. Eng.*, 141(12).

Decrop B. and Sas M. (2014). "Challenges in the acoustic measurements of dredging plumes". Particles in Europe Conference, Esbjerg, Denmark 2014.

Decrop B., Sas M., De Mulder T. and Toorman E. (2014). "Large-Eddy Simulations of a Sediment-Laden Buoyant Jet Resulting from Dredgers Using Overflow". Int. Conf. Hydrosci. Eng.

EDF R&D (2013). "Telemac Modelling System, 3D hydrodynamics, Operating manual, Release 6.2".

Fischer H.B. (1979). "Mixing in inland and coastal waters". Academic Press.

Jirka G.H. (2006). "Integral model for turbulent buoyant jets in unbounded stratified flows Part 2: plane jet dynamics resulting from multiport diffuser jets". *Environ. Fluid Mech.*, 6(1), 43–100.

Lee J.H. and Cheung V. (1990). "Generalized Lagrangian model for buoyant jets in current". *J. Environ. Eng.*, 116(6), 1085–1106.

Lee J.H.W. and Chu V.H. (2003). "Turbulent jets and plumes: A Lagrangian approach". Springer.

Lopez R.J. (2001). "Advanced engineering mathematics". Addison-Wesley.

Manninen M., Taivassalo V. and Kallio S. (1996). "On the mixture model for multiphase flow". Technical Research Centre of Finland Finland. VTT Publications, 288.

Smith S.J. and Friedrichs C.T. (2011). "Size and settling velocities of cohesive flocs and suspended sediment aggregates in a trailing suction hopper dredge plume". *Cont. Shelf Res.*, 31(10), S50–S63.

Spearman J., De Heer A., Aarninkhof S. and Van Koningsveld M. (2011). "Validation of the TASS system for predicting the environmental effects of trailing suction hopper dredgers". *Terra Aqua*, (125).

NOMENCLATURE

Symbol [Unit]	**Definition**
c [kg/m³]	Sediment concentration
C_0 , C_p [kg/m³]	Sediment concentration at resp. overflow and at dredging pump
D [m]	Overflow diameter
F_s [kg/m² s]	Sediment flux in the plume
F_t [kg/m² s]	Turbulent sediment flux
H_k [m]	Keel clearance
L_0 [m]	Distance from overflow to stern
L_s [m]	Ship length
Q_0 [m³/s]	Volume discharge at overflow
Q_p [m³/s]	Volume discharge at dredging pump
Q_s [m³/s]	Volume discharge in the plume
U, V, W [m/s]	Flow velocity components
U_v [m/s]	Vessel speed
U_0 [m/s]	Ambient flow velocity
W_0 [m/s]	Mean overflow exit velocity
x [m]	Horizontal distance from dredger
z [m]	Vertical distance from keel
φ_a [-]	Air volume concentration
$\varphi_{a,0}$ [-]	Air volume concentration at overflow exit
ρ_a , ρ_m , ρ_s , ρ_∞ [kg/m³]	Mass density of resp. air, plume mixture, sediment grains and ambient waters
ζ [-]	Normalised vertical coordinate