Ocean Dynamics (2017) 67:137–146
DOI 10.1007/s10236-016-1017-0

A parameter model for dredge plume sediment source terms

Boudewijn Decrop[1] · **Tom De Mulder**[2] · **Erik Toorman**[3] · **Marc Sas**[1]

Received: 3 December 2015 / Accepted: 4 November 2016 / Published online: 19 November 2016

Abstract The presented model allows for fast simulations of the near-field behaviour of overflow dredging plumes. Overflow dredging plumes occur when dredging vessels employ a dropshaft release system to discharge the excess sea water, which is pumped into the trailing suction hopper dredger (TSHD) along with the dredged sediments. The fine sediment fraction in the loaded water-sediment mixture does not fully settle before it reaches the overflow shaft. By consequence, the released water contains a fine sediment fraction of time-varying concentration. The sediment grain size is in the range of clays, silt and fine sand; the sediment concentration varies roughly between 10 and 200 g/l in most cases, peaking at even higher value with short duration. In order to assess the environmental impact of the increased turbidity caused by this release, plume dispersion predictions are often carried out. These predictions are usually executed with a large-scale model covering a complete coastal zone, bay, or estuary. A source term of fine sediments is implemented in the hydrodynamic model to simulate the fine sediment dispersion. The large-scale model mesh resolution and governing equations, however, do not allow to simulate the near-field plume behaviour in the vicinity of the ship hull and propellers. Moreover, in the near-field, these plumes are under influence of buoyancy forces and air bubbles. The initial distribution of sediments is therefore unknown and has to be based on crude assumptions at present. The initial (vertical) distribution of the sediment source is indeed of great influence on the final far-field plume dispersion results. In order to study this near-field behaviour, a highly-detailed computationally fluid dynamics (CFD) model was developed. This model contains a realistic geometry of a dredging vessel, buoyancy effects, air bubbles and propeller action, and was validated earlier by comparing with field measurements. A CFD model requires significant simulation times, which is not available in all situations. For example, to allow correct representation of overflow plume dispersion in a real-time forecasting model, a fast assessment of the near-field behaviour is needed. For this reason, a semi-analytical parameter model has been developed that reproduces the near-field sediment dispersion obtained with the CFD model in a relatively accurate way. In this paper, this so-called grey-box model is presented.

Keywords CFD · Dredging · Turbidity · Plumes

This article is part of the Topical Collection on the *13th International Conference on Cohesive Sediment Transport in Leuven, Belgium 7-11 September 2015*

Responsible Editor: Joris Vanlede

✉ Boudewijn Decrop
bdc@imdc.be

[1] Coastal and Estuarine Section, IMDC, Van Immerseelstraat 66, B-2018, Antwerp, Belgium

[2] Hydraulics Laboratory, Ghent University, Department of Civil Engineering, Sint-Pietersnieuwstraat 41, B-9000, Ghent, Belgium

[3] Hydraulics Laboratory, KU Leuven, Department of Civil Engineering, Kasteelpark Arenberg 40, B-3001, Heverlee, Belgium

1 Introduction

In recent times, environmental impact assessments of dredging works have been subject to extensive regulation. Methods for the prediction of adverse effects of marine works are permanently being improved. Dredging using trailing suction hopper dredgers (TSHD) requires discharging of excess water from the vessel's hopper into the sea for optimal efficiency of the operation. The return flow is discharged through a vertical shaft with exit below the dredger's hull, which is commonly referred to as the overflow shaft. The released material usually contains a fraction of fine sediments for which the residence time in the hopper was insufficient to settle and thus stays in suspension until the overflow shaft is reached. Part of the released material might descend to the sea bed as a density current, another part might be dispersed to form a surface plume. The latter part is bound to stay in the water column for a longer period in case the sediment concentration is too low for a buoyancy-driven descent to the sea bed. The turbidity plume generated in this way can be advected by tidal or circulation currents and potentially reaches environmentally sensitive areas.

In the feasibility phase of dredging projects, the fate of these fine sediment plumes needs to be predicted in order to assess the need for mitigation measures. In the operational project phase, real-time plume predictions are needed to assess the timing and location of the works planned for the coming days. The large-scale simulation of the plumes is generally executed with a shallow-water hydrodynamic flow model fitted with a source term for the overflow mixture. The source term to be supplied to the large-scale model is the fraction of the released sediment flux reaching the water column. The determination of this (vertically distributed) source term can be done using time-consuming process-based computational fluid dynamics (CFD) models, or by means of a parameterised prediction model. The development of the parameter model structure, and the fitting of its parameter by means of a large data set of process-based model output is presented in this paper.

1.1 The trailing suction hopper dredger

The TSHD is a self-propelled, seagoing vessel and is widely applied worldwide. While trailing a drag head across the sea bed, a sediment-water mixture is pumped through a suction pipe and into the hopper with discharge Q_p and sediment mass concentration C_p (Fig. 1). Whereas the coarser sediment particles settle in the hopper, the finest sediment fractions can stay in suspension and flow overboard with the excess water through a vertical dropshaft, the overflow. The volume discharge through the overflow is denoted Q_0 and the sediment mass concentration in the mixture as C_0.

The exit of the overflow shaft is usually mounted flush with the keel of the vessel's hull. As a consequence, a negatively buoyant plume of water, air bubbles and fine sediment particles are released vertically below the vessel. Due to the sailing speed of the vessel and/or the ambient currents, the generated plumes are subject to a crossflow.

Part of the sediment plume can be stripped off the main density current by means of air bubbles, propellers and crossflow, see, e.g. Decrop et al. (2012), de Wit et al. (2014), Decrop et al. (2014), Decrop et al. (2015), and Saremi and Jensen (2014). This fraction of the total sediment discharge, Q_s, is subsequently moved to a surface plume. These surface plumes are often visible on aerial photography.

To enable proper assessment of the environmental impact of the plumes, it is important to predict its fate and dispersion in the water column, see, e.g. Bray (2008).

1.2 Overflow dredging plumes

In their spatial evolution from overflow shaft to dissipation, overflow plumes can be divided in two zones. The zone closest to the release is subject to complex interactions between water, sediment, air bubbles, propeller jets and the bulk buoyancy of the plume. This zone is called the near field. In this zone, the plume is usually a dynamic plume, which is defined as a plume under influence of an increased bulk density compared to the surrounding sea water.

After a certain distance behind the TSHD, propeller- and ship-induced mixing have decayed, air bubbles have left the water column and the sediment concentration has reduced to levels at which the bulk density is very close the sea water density. This zone is called the far field. Here, the plume is no longer dynamic, but is referred to as a passive plume. It is this type of plume which can travel over long distances with the sea current.

Breugem et al. (2009), Spearman et al. (2011) and Decrop and Sas (2014) describe water sampling inside the overflow shaft. It is found that the sediment concentration of the water-sediment mixture released by the overflow ranges between 5 and 200 g/l, or even higher during short periods near the end of the loading phase. Sediment concentrations in the far-field plume have been monitored in the past (Newell et al. 1999; Hitchcock and Bell 2004; Black and Parry 1999; Breugem et al. 2009; Smith and Friedrichs 2011). Sediment concentrations found in the plume range from 10 up to 5000 mg/l, the latter value found in resulting near-bed density currents. The length of plumes of individual TSHD's is found to be between between 300 m and 2.5 km. Much larger plumes may form when continuous capital dredging is performed at the same location. Settling velocity of the sediment ranges from less than 0.1 mm/s in microflocs to 5 mm/s for bed aggregates surviving the passage through the TSHD (Smith and Friedrichs 2011).

1.3 Environmental aspects of dredging with overflow

The turbidity generated by dredging with overflow potentially leads to adverse environmental impacts (Bray 2008).

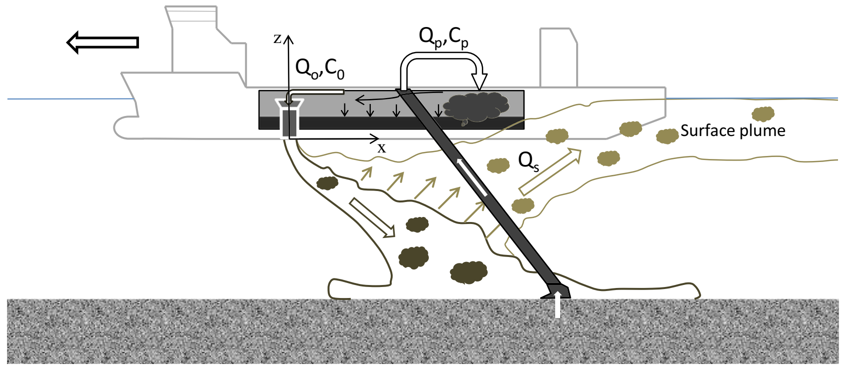

Fig. 1 Sketch of a trailing suction hopper dredger with emphasis on the path of the water-sediment mixture. The sediments move from the sea bed through the suction pipe into the hopper and a fraction is discharged through the overflow, back into the sea

When a surface plume is formed with relatively low sediment concentration, and thus low excess density, it will descend to the sea bed very slowly under influence of the sediment settling velocity. In this case, the surface plume can travel with tidal currents over distances of a few kilometers (Hitchcock and Bell 2004; Breugem et al. 2009). The reduced light penetration or sediment depositions can induce adverse effects to aquatic life such as coral reefs, hunting fish, sea grass patches and benthic organisms.

To prevent plumes to reach these valuable areas, an extensive turbidity monitoring programme is usually deployed during dredging projects. Alarm levels are defined above which turbidity is not allowed to rise. In case these levels are reached, the planning of works need to be revised or ultimately, the dredging works have to be suspended. To avoid such events, the dredging plumes caused by the operational vessels have to be forecasted by model simulations. In the past, a so-called environmental valve has been used in case severe adverse effects are likely. This valve chokes the flow in the overflow shaft, thereby avoiding the plunging jet and subsequent air bubble entrainment. The efficiency of the valve under different circumstances was analysed in Decrop et al. (2015).

1.4 Present-day numerical modelling of overflow dredging plumes

In order to assess the potential of plumes to reach sensitive areas in the phase of tendering, predictions of tidal currents and plume dispersion are needed. Also in the operational phase of dredging projects, real-time forecasting of plume dispersion is an advantage in avoiding suspension of works due to turbidity threshold violations.

To predict the path and concentration of turbidity plumes, numerical models are the only option. Large-scale hydrodynamic models are set up and calibrated to simulate the temporal evolution of tidal currents. When it is planned that a given dredging vessel with a given production will be working at a given location, the sediment source terms can be imposed in the numerical flow model. Based on the production rates and the percentage of fine sediment in the sea bed, a sediment flux through the overflow can be estimated (van Rhee 2002; Jensen and Saremi 2014).

The released sediments are subsequently dispersed through a complex flow pattern influenced by density gradients, air bubbles, propeller mixing and the flow around the TSHD. A fraction of the sediment might reach the sea bed immediately and spread as a near-bed density current. The solution of these detailed processes is not feasible in a large-scale model stretching over a distance of typically 10 to 100 km. There are two main reasons: a number of assumptions in the equations and relatively large grid cells. Both aspects are needed to make predictive calculations feasible for such large timescales and spatial scales. At present, the bulk effect of all the complex near-field processes has to be condensed in one parameter: the fraction of the released sediments that is brought in suspension in the water column (Becker et al. 2015). The vertical distribution of this fraction throughout the water column is also unknown. These factors form the largest uncertainty in defining sediment source terms for overflow plume simulations.

In this work, a parameterised model is developed to determine the vertical distribution of the sediment flux in the overflow plume, based on a data set of highly-detailed, three-dimensional CFD simulations of the complex flows of the water-sediment-air mixture from overflow shaft to far field. The parameter model presented in this paper is different from the similar model of de Wit et al. (2014), in the sense that it can be coupled with a far-field plume dispersion model for overflow spills generated by a wide range of sizes of TSHD's and under a wide range of continuously varying ambient forcings. Also, important geometrical variables such as overflow shaft diameter, the distance between overflow and stern and vessel draught are taken into account.

2 A grey-box parameter model

As mentioned before, the source term which has to be supplied to the large-scale model is the fraction of the overflown sediments that is released into the water column, preferably as a vertical profile. The determination of this source term can be done using the process-based CFD models, or by means of a faster parameterised prediction model. A parameterised model will therefore have to be a trade-off between speed and accuracy. It will be less accurate compared to a CFD model, but will be applicable in cases where the CFD model is not possible, e.g. real-time plume forecasting simulations.

The parameter model will be composed of theoretical solutions as well as on multivariate regression analysis, thefeore the term 'grey-box model' is used.

The parameters in the model will be fitted by means of a large data set of CFD model output, based on 75 CFD model simulations. In the current paper, all situations considered include entrained air in the dropshaft of the overflow. The effect of a choking valve (so-called green valve) is described in Decrop et al. (2015) and is not yet included in the currently described model. The CFD model has been developed by the authors and has been validated against laboratory experiments in Decrop et al. (2015) and against field measurements in Decrop et al. (2014). An example of the sediment plume calculated by the CFD model can be found in Fig. 2. Sediment concentrations at the overflow between 5 and 250 g/l have been considered, corresponding to an excess density ranging from 3 to 155 kg/m^3. The model is based on sailing hopper dredgers using overflow. The parameter model is not valid for anchored barges or hoppers serving as a barge during cutter production. A separate investigation is being conducted on how to append the model with these situations.

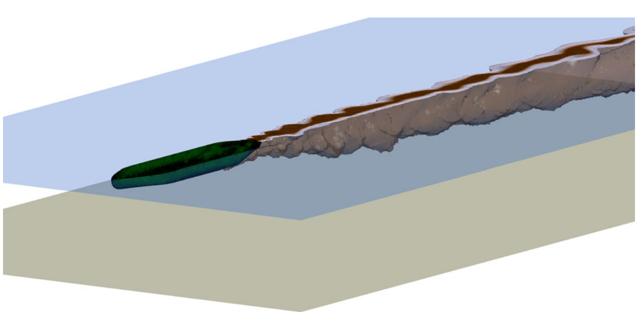

Fig. 2 Example of a result of the 3D CFD model for the dynamics of the water-sediment-air mixture released by a TSHD. Highly-concentrated sub-surface plume is indicated by the brown isosurface, the sediment concentration at the water surface is indicated in brown to white contours. The sea bed is indicated by a brown plane, the water surface by a transparent blue plane (Decrop 2015)

The development of the parameter model structure, the fitting of its parameters and the assessment of its quality are presented in the following sections.

3 Formulation and principles

Before the formulation of a parameterised model can start, 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.

At this point, the influence of a valve choking the flow to reduce air entrainment is not taken into account. All plume data generated using the CFD model is based on simulations with an air volume concentration of 7% in the overflow shaft.

In Fig. 3, 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. 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:

$$\zeta = \left(\frac{z}{H_d}\right)\mathcal{H}(z) + \left(\frac{z}{H_k}\right)\mathcal{H}(-z) \tag{1}$$

where $\mathcal{H}$ is the Heaviside step function.

The time-averaged sediment flux f in the sediment plume (in kg/s/m) is defined based on the CFD results:

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

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

The laterally integrated flux q_s is determined as:

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

where B_w 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 ζ. A distance x_p needs to be defined at which the vertical profile of q_s is evaluated. At this distance from the vessel, the parameter model

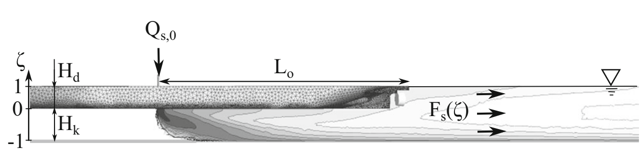

Fig. 3 Sketch of the different length scales and sediment fluxes. As for the Carthesian coordinate system in Fig. 1, the origin is located at the overflow shaft exit and is moving with the vessel

output is valid for implementation in a far-field model. Different options were considered, either with a fixed or with a variable distance from the vessel:

- Fixed distance behind vessel,
- Distance as function of buoyancy threshold of the plume, so that sediment profiles are always evaluated when the plumes no longer have significant negative buoyancy,
- Distance dependent on return to standard Rouse profile for natural sediment transport.

The two latter options have been disregarded because situations would occur in which the required distance for evaluation of the plume would fall outside the domain of the CFD model. Finally, the first option was chosen, in which 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.

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

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

where $Q_{s,0} = C_0 Q_0$ 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=$2.5L_s$ behind the dredger.

The next step is to parameterise the resulting profiles. The parameters describing the shape of the profiles will then be linked through a multivariate regression to the input parameters. Depending on the ambient conditions and overflow jet exit conditions, two distinct types of plumes can be distinguished: the near-bed density current with surface plume and the sea bed-detached plume. The shape of the vertical flux profile of both types of plumes is obviously different (Fig. 4). Both types can be characterised with a surface flux at the top of the water column, F_t=$F_s(\zeta = 1)$, and a ζ-level at which $\mathrm{d}F_s/\mathrm{d}\zeta$ changes rapidly (ζ=-0.1 and -0.2 in Fig. 4a and b respectively.

In order to select which type of profile will occur, an 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 start. The outflow velocity W_0, pipe diameter D, sediment concentration C_0 and the crossflow velocity U_0 (the vector difference of tidal flow velocity and ship speed) are used to determine the trajectory of the buoyant jet, and the centerline height ζ_c at x_p=$2.5L_s$. This solution is an estimate of the trajectory without TSHD-specific influence factors, see the diamond

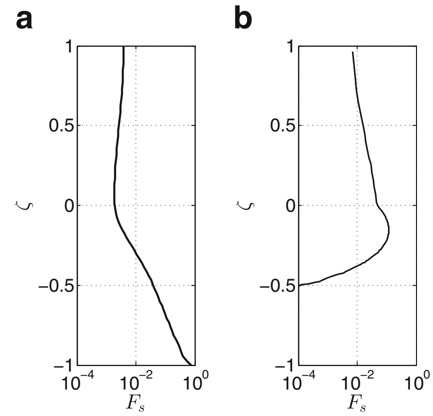

Fig. 4 Vertical profile of F_s for two types of plumes: **a** Near-bed density current type and **b** the detached plume

markers in Fig. 5. In general, the ship and air bubbles have the tendency to move the sediment plume higher than what would be expected from a plain buoyant jet.

The plume centre at x_p=$2.5L_s$ can be extracted from the LES model results (Decrop et al. 2014) and can be compared with the Lagrangian model. The uplifting effect of the vessel is related to the boundary conditions F_Δ and λ.

$$F_\Delta = \frac{W_0}{\sqrt{gD\Delta\rho/\rho_w}} \quad ; \quad \lambda = \frac{W_0}{U_0} \tag{5}$$

Where $\Delta\rho$ is the difference in mass density between the overflow mixture (without air) and the sea water, which has density ρ_w.

A multivariate linear model fit is executed. The independent variables are the initial plume centre height ζ_c, the crossflow-based Froude number $F_\infty = F_\Delta\lambda$ and the ratio H_d/L_o. The dependent variable is the plume height based on the LES model results, $\zeta_{c,LES}$ (Table 1) .

$$\zeta_{c,LES,i} = \beta_0 + \beta_1\, \zeta_{c,i} + \beta_2\, F_{\infty,i} + \beta_3\, (H_d/L_o)_i + \varepsilon_i \tag{6}$$

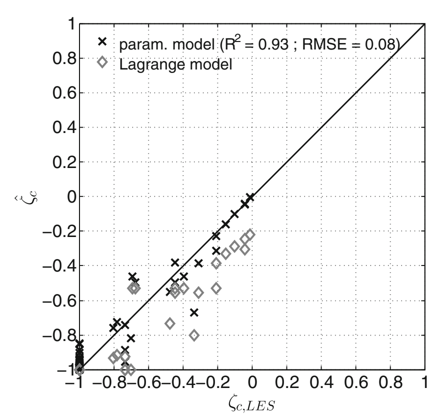

Fig. 5 Relation between Lagrangian estimates (*diamonds*) for a simple buoyant jet and corresponding LES results for TSHD plumes ($\zeta_{c,LES,i}$). Improvement of the estimates by applying a general linear regression is shown by *black crosses*

Table 1 Coefficients β_0, ..., β_3 in Eq. 6

	$i = 0$	$i = 1$	$i = 2$	$i = 3$
β_i	−0.27	0.039	1.02	0.86

where i is the number of observations, β_0, β_1, β_2 and β_3 are the coefficients to fit and ε_i are error terms.

After solving for the coefficients β_0 to β_3 by a least-squares approach, the estimated plume height $\hat{\zeta_c}$ (at x_p=2.5L_s) can be determined. The resulting estimated plume heights $\hat{\zeta_c}$ can be compared with the 'true' plume heights from the LES model in a scatter plot. The root-mean-squared-error and coefficient of determination R^2 can be determined (Fig. 5).

Figure 5 shows that the ζ_c-estimates can be used as a good indication for the height of the plume center at x_p. When $\hat{\zeta_c} < -0.75$, the plume can be considered of type 'near-bed density current', when $\hat{\zeta_c} \geq -0.75$, of type 'bottom-detached'. For both types, the model follows a different approach in terms of the parameterisation of the shape of the vertical profile of F_s.

The density current type 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. The coefficients 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:

$$\widehat{F_s(\zeta)} = \sum_{i=0}^{n} \psi_i \; T_i(\zeta) \tag{7}$$

where T_i are the Chebychev polynomials and ψ_i are $n + 1$ coefficients fitted to the data set.

The so-called Chebychev polynomials of the first kind can be found by the recurrence relation:

$$T_0(\zeta) = 1 \tag{8}$$

$$T_1(\zeta) = \zeta \tag{9}$$

$$T_{n+1}(\zeta) = 2\zeta T_n(\zeta) - T_{n-1}(\zeta) \tag{10}$$

The Chebychev polynomials are defined in the range [-1,1], for which reason the transformation from z to ζ is particularly practical.

An example of a Chebychev parameterisation of a density current type plume is shown in Fig. 6a.

The second type of plume flux profile is the seabed-detached plume. For this type, a step wise parameterisation of the flux profile is proposed. The reason for the different parameterisation is that the 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 Fig. 6b, the step-wise parameterisation (dashed line) is shown. It is defined by the points (F_t,1), (F_m, ζ_m) and the slope S_b. This gives a total of four parameters to fit to the data set, as for the Chebychev approach.

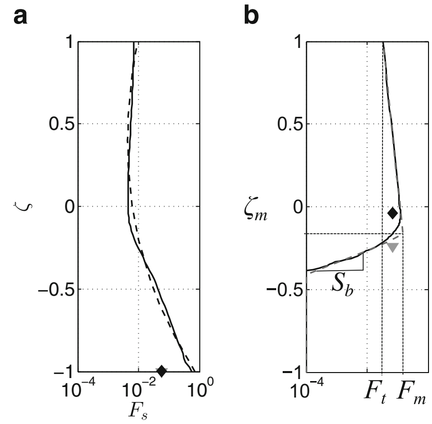

Fig. 6 Vertical profile of F_s for two types of plumes. **a** Near-bed density current type with Chebychev parameterisation (*dashed line*), with initial (*grey triangle*) and corrected plume center position. Both initial and corrected estimate of the plume center is located at the seabed. **b** The detached plume type with step-wise parameterisation (*dashed line*), defined by points (F_t,1), (F_m, ζ_m) and slope S_b

4 Model training

A training data set of 50 CFD simulations was used to relate the coefficients ψ_i (for the density current type) or F_t, F_m, ζ_m and S_b (for the detached plumes) to the different boundary conditions of the plume. These boundary conditions consist of F_Δ, λ, H_d/L_o, etc. The ranges of these conditions covered by the training data set are shown in Fig. 7. The model is therefore valid for 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. 7) were found to depend mainly on F_∞ 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):

$$\psi_i = \beta_{c,i,0} + \beta_{c,i,1} \; F_{\infty,m} + \beta_{c,i,2} \; (H_k/D)_m + \varepsilon_{i,m} \tag{11}$$

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, $\beta_{c,0}$, $\beta_{c,1}$ and $\beta_{c,2}$ are the coefficients to fit and $\varepsilon_{i,m}$ are error terms. After fitting to the data from the CFD model, the coefficients in Table 2 are obtained.

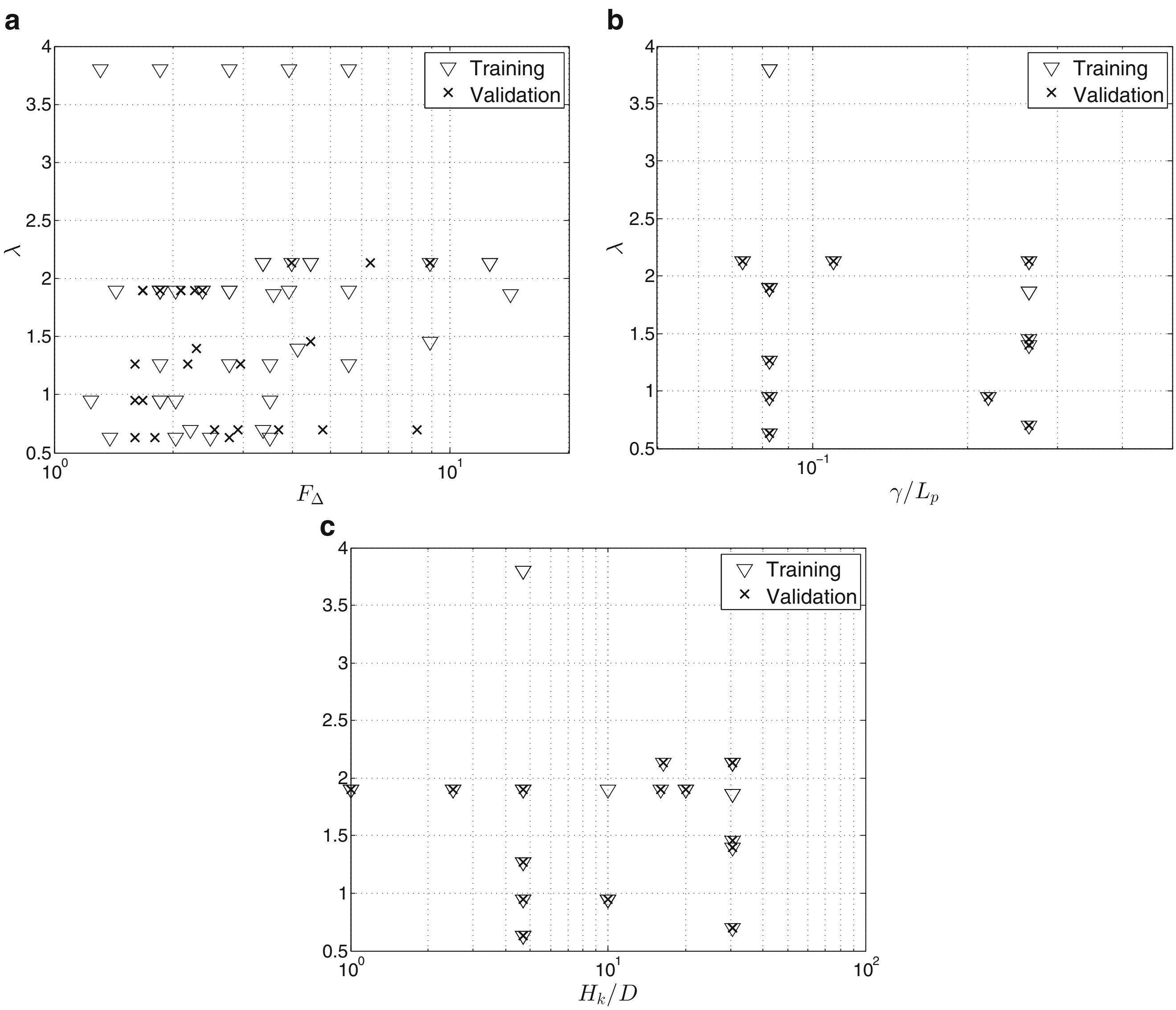

Fig. 7 Properties of the plumes in the data sets for training the parameter model and for validating the parameter model. For all cases, the initial air volume concentration was equal to 7%

The parameters for the step-wise profile of the seabed-detached plumes were found to be best represented as a function of the following 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.

Table 2 Coefficients β_c

	$i=0$	$i=1$	$i=2$	$i=3$
$\beta_{c,i,0}$	1.82	1.04	0.53	0.075
$\beta_{c,i,1}$	0.20	0.12	−0.072	0.029
$\beta_{c,i,2}$	−0.025	0.010	0.016	−0.016

The training data set cases are used for finding β_d (4x4=16 coefficients):

$$(F_t, F_m, \zeta_m, S_m) = \beta_{d,0}+\beta_{d,1}F_{\infty,m}+\beta_{d,2}(H_d/L_o)_m+\beta_{d,3}(H_k/D)_m+\varepsilon_m \quad (12)$$

where, m=1,...,M, with M the number of CFD simulations (with seabed-detached plume) in the data set, $\beta_{d,0}$, $\beta_{d,1}$, $\beta_{d,2}$ and $\beta_{d,3}$ are the coefficients to fit for each profile parameter (F_t, F_m, ζ_m, S_m). ε_m are error terms. After fitting to the data from the CFD model, the coefficients in Table 3 are obtained.

When all coefficients β have been fitted to the training data set, parameter model predictions can be compared with the original profile (from CFD) and with the parameterised CFD-profile. In Fig. 8, the dashed grey lines indicate the

Table 3 Coefficients β_d

	F_t	F_m	ζ_m	S_m
$\beta_{d,0}$	−0.099	−0.024	−1.123	26.058
$\beta_{d,1}$	0.026	0.026	0.060	−1.312
$\beta_{d,2}$	0.030	0.280	0.927	16.716
$\beta_{d,3}$	0.001	−0.003	0.011	−0.315

predictions from the parameter model. At this point, the vertical integral of $F_s(\zeta)$ should be equal to one to maintain the original sediment flux after application of the parameter model. A corrector step is therefore added in which a (small) correction factor is multiplied by F_s. The factor is determined by the inverse of the integrated $F_s(\zeta)$, so that for the corrected profile the vertical integral of $F_s(\zeta)$ becomes equal to one. In Fig. 8, the full grey lines show the corrected flux profile predictions.

5 Model validation

5.1 Against CFD runs

A dataset of 25 CFD runs were not used in the training of the parameter model. This data set is used to validate the performance of the simple profile prediction model. In Fig. 9, a number of examples are given of the predictions of the parameter model against CFD solutions. In Fig. 9a, c, the model correctly identified the plume as of type 'density current', whereas in Fig. 9b the type 'seabed-detached' was correctly identified. In most cases, the typical profile shapes are found. In some cases, very specific profile shapes of the deeper part of the plume were found in the CFD model. In these cases, the exact shape is not reproduced by the parameter model, given the limited number of parameters

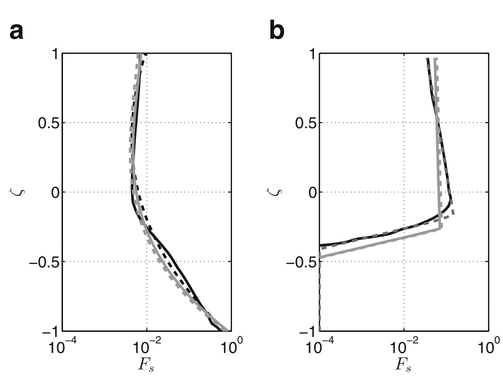

Fig. 8 Vertical profile of F_s for two types of plumes, as in Fig. 6. Here, the parameter model predictions are added to the figure in grey (*dashed line*, first step) and *full grey line* (after corrector step)

(Fig. 9c). Nevertheless, the sediment flux near the surface is reproduced well.

A general overview of the performance of the parameter model, both for the training data set and for the validation data set, is shown (Fig. 10). The fraction of cases is shown for which the coefficient of determination (R^2) has a certain value. Also the cumulative values are shown. It is shown that for the validation cases, 75% of the predicted profiles had a R^2 value of more than 0.7. In about 18% of the cases, the predictions were less accurate with $R^2 \leq 0.5$, while only one in twelve predictions showed a R^2 of 0.4 or lower. Taking into account the simple semi-analytical setup of the parameter model, the overall performance is good.

5.2 Multiple overflow superposition

In many of the largest TSHD's, multiple overflows are mounted. They are not necessarily active at the same time, but the situation of multiple overflow plumes is possible. Therefore, a check is performed on how well the CFD model results compare with superimposed parameter model plumes. The profiles are obtained from the parameter model for each overflow plume separately. Afterwards, the values of F_s of both plume profiles are simply added together.

Two cases have been simulated in the CFD model in which two overflows are active. In these cases, the overflow concentration for both overflows was the same, while the discharge was equally distributed. In this case, the following boundary conditions were imposed: H = 40 m, U_0 = 1.5 m/s, W_0 = 3.2 m/s, C_0 = 20 g/l and D = 1.1 m. The latter three conditions are valid for both overflows. In Fig. 11a, the result is shown after adding both parameter model results together and comparing with the CFD simulation. The plume with L_o = 20 m is of type 'seabed-detached' (grey dashed line), while the plume with L_o = 72 m is of type 'density current'. When both are added together, the shape is similar to the CFD result of the multiple plume. The surface plume, however, is overestimated. This can be explained by the shielding of the plume closest to the stern. Due to the wake of the plume closest to the bow, the other plume experiences less crossflow. Therefore, the L_o = 20-m plume generates a lower added sediment concentration compared to the single-plume situation.

The second case equally consists of two overflows at the symmetry plane, but with larger diameter, D = 2 m. Also, H = 26 m, W_0 = 1.9 m/s, U_0 = 2 m/s and C_0 = 90 g/l. L_o = 30 m for the rear end plume, while L_o = 80 m for the front end overflow plume. In this case, both separate plumes are of type 'density current'. This type of plume is not dependent on the overflow position in the parameter model. Therefore, both parameter model profiles are equal (grey line). The superimposed plume profile is thus simply equal to twice the F_s values from the individual plumes.

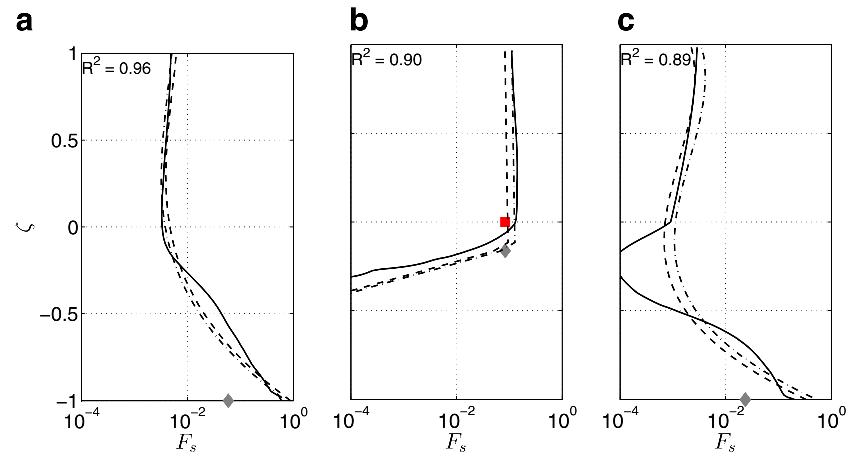

Fig. 9 Examples of F_s-profiless for parameter model validation cases. *Black line* indicates full CFD solutions, *dot-dashed* (*dashed*) *line* shows the uncorrected (corrected) parameter model prediction. Preliminary determinations of plume center levels are shown in *diamond* and *square markers*

In this case, this seems to correspond quite well with the CFD result of the multiple plumes. In this case, the surface plumes formation is more dominated by air bubbles than by crossflow. Both plumes are therefore less influenced by each other, and hence can be superimposed with good result.

Superimposing multiple plumes from a simplified parameter model seems to be allowed in some cases. In other cases deviations from the CFD model results seem to occur. Further investigation is needed to clarify under which circumstances multiple plumes can be superimposed and under which circumstances corrections are needed to the simple addition of plumes.

6 Application of the parameter model

Though less accurate than the CFD model, the parameter model is much faster (order of magnitude of seconds). The parameter model predicts the sediment flux in the overflow plume still with reasonable accuracy. It is therefore well suited to be applied in situations where no sufficient time is available for CFD simulations. These situations include far-field simulations of overflow turbidity, with moving TSHD simulations or during real-time forecasting. In such situations, the tidal flow velocity U and ship keel clearance H_k can vary, and thus varying boundary conditions for the near-field plume simulations are imported from the far-field model, while the plume-related sediment distribution is fed back to the far-field model. This can only be achieved when a simple prediction module is coupled to the large-scale tidal flow model. This type of online coupling is currently being implemented. The principle of this online coupling between the near-field parameter model and the far-field tidal flow model is shown in Fig. 12.

The vertical profile of sediment flux generated by the parameter model can be directly implemented in a 3D tidal

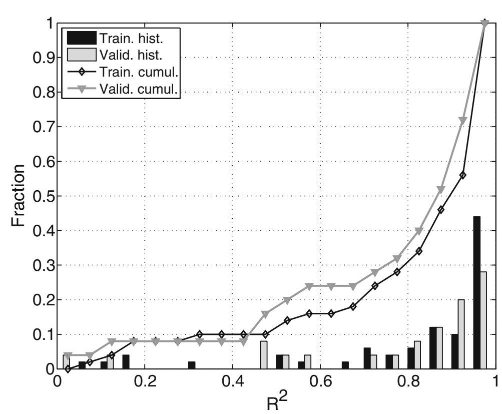

Fig. 10 Statistics of the parameter model performance, for the training data set and for the validation data set. Histograms are shown for each class of R^2 values. The lines with *diamond* markers are drawn for the cumulative fraction of cases with R^2-value lower than or equal to the value on the x-axis

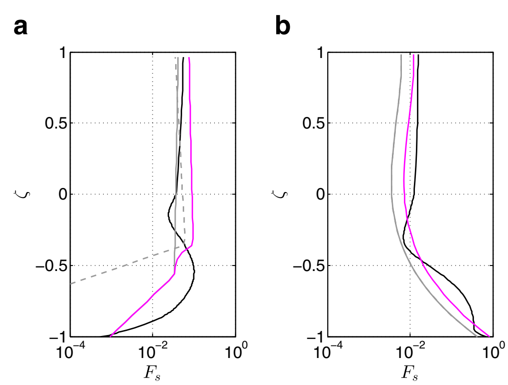

Fig. 11 Parameter model predictions of multiple overflow plumes. **a** Two plumes for which H = 40 m, U_0 = 1.5 m/s, W_0 = 3.2 m/s, C_0 = 20 g/l and D = 1.1 m. One plume originated at L_o = 20 m, the other at L_o = 72 m. **b** Two plumes for which H = 26 m, W_0 = 1.9 m/s, U_0 = 2 m/s, C_0 = 90 g/l and D = 2 m. L_o = 30 m for one plume and 80 m for the second. *Grey lines* indicate individual plume results, of which the magenta line is the sum. The *black line* shows the CFD model result

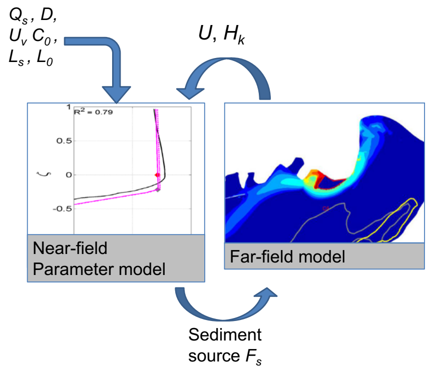

Fig. 12 Principle of the real-time application of the parameter model with a far-field tidal circulation model. Online coupling is set up between the near-field and far-field models.

flow model. In order to represent the near-field sources well in a far-field model, the horizontal grid resolution should be in line width the scale of the spills. A TSHD dredger overflow plume typically has a width of the order of 100 m at $x_p = 2.5L_s$. Grid resolutions of that order or smaller are suited to represent the source term computed by the parameter model. Much larger grid cells would induce artificial diffusion. The vertical grid resolution should be capable of resolving the vertical variation of the computed source term. Depending on the water depth and the shape of the computed profiles, a minimum of six to ten layers is recommended. Sediment source fluxes should be carefully integrated over the thickness of far-field model layers, to ensure conservation of sediment mass.

7 Conclusions

A parameter model for near-field dredging plumes has been developed based on fitting with a large set of CFD simulations. The parameter model uses a combination of analytical solutions and empirical relationships. The parameters in the empirical parts of the parameter model were fitted based on CFD model results for a wide range of boundary conditions. Validation against a separate set of CFD results resulted in a good accuracy of the parameter model, taking into account its simplicity and speed. For 75 % of the validation cases, the coefficient of determination was higher than 0.7. Within the range of conditions for which the parameters have been fitted, the model is valid. In some cases, however, the model is not valid. For example an anchored TSHD in a cross-flow, or in the case of salinity stratification in deep water. In these cases, CFD model results still need to be generated to estimate the overflow plume turbidity.

The developed parameter model is being coupled to the code of commonly used models for far-field tidal flow and sediment transport. The time-dependent flow velocity and keel clearance is fed from the far-field model into the parameter model. In turn, the parameter model returns the overflow plume sediment distribution to the far-field tidal flow and sediment dispersion model.

References

Decrop B, De Mulder T, Troch P, Toorman E, Sas M (2012). In: Coastlab 2012, Proceedings, pp 573–582
de Wit L, van Rhee C, Talmon A (2014) Env Fluid Mech:1–26
Decrop B, Sas M, De Mulder T, Toorman E (2014). In: International Conference on Hydroscience and Engineering, pp 737–744
Decrop B, De Mulder T, Toorman E, Sas M (2015) J Environ Eng 141(12):04015042
Saremi S, Jensen JH (2014) Coast Eng Proc 1(34). posters
Bray RN (2008) Environmental aspects of dredging. CRC Press
Breugem W, Bollen M, Sas M, Vandenbroeck J (2009). in *CEDA Dredging Days 2009 - Dredging Tools for the Future Rotterdam*
Spearman J, De Heer A, Aarninkhof S, Van Koningsveld M (2011). Terra et Aqua (125)
Decrop B, Sas M (2014). In: Particles in Europe 2014, pp 35–41
Newell R, Hitchcock D, Seiderer L (1999) Mar Pollut Bull 38(9):809
Hitchcock D, Bell S (2004) J Coast Res:101–114
Black KP, Parry GD (1999) Can J Fish Aquat Sci 56(12):2271
Smith S, Friedrichs C (2011) Cont Shelf Res 31(10):S50
van Rhee C (2002) On the sedimentation process in a trailing suction hopper dredger. Delft University of Technology. Ph.D. thesis
Jensen JH, Saremi S (2014) J Waterw Port Coast Ocean Eng 140(6)
Becker J, van Eekelen E, van Wiechen J, de Lange W, Damsma T, Smolders T, van Koningsveld M (2015) J Environ Manag 149:282
de Wit L, Talmon A, Van Rhee C (2014) Mar Pollut Bull 88(1):47
Decrop B, Mulder TD, Toorman E, Sas M (2015) Eur J Mech - B/Fluids 53(0):68
Decrop B (2015) Numerical and experimental modelling of near-field overflow plumes. Ghent University, KULeuven. Ph.D. thesis
Lee JH, Cheung V (1990) J Environ Eng 116(6):1085
Lee J, Chu V (2003) Turbulent jets and plumes: A Lagrangian approach. Springer
Lopez RJ (2001) Advanced engineering mathematics. Addison-Wesley. vol. 1158