



Multiphase CFD modeling of nearfield fate of sediment plumes

Saremi, Sina; Hjelmager Jensen, Jacob

Published in:
Book of Proceedings

Publication date:
2014

Document Version
Publisher's PDF, also known as Version of record

[Link back to DTU Orbit](#)

Citation (APA):
Saremi, S., & Hjelmager Jensen, J. (2014). Multiphase CFD modeling of nearfield fate of sediment plumes. In *Book of Proceedings IAH*R.

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

MULTIPHASE CFD MODELING OF NEARFIELD FATE OF SEDIMENT PLUMES

SINA SAREMI⁽¹⁾, JACOB HJELMAGER JENSEN⁽²⁾

⁽¹⁾ Technical University of Denmark, Kgs. Lyngby, Denmark,
remi@mek.dtu.dk

⁽²⁾ Technical University of Denmark, Kgs. Lyngby, Denmark,
jahj@mek.dtu.dk

Abstract

Disposal of dredged material and the overflow discharge during the dredging activities is a matter of concern due to the potential risks imposed by the plumes on surrounding marine environment. This gives rise to accurately prediction of the fate of the sediment plumes released in ambient waters. The two-phase mixture solution based on the drift-flux method is evaluated for 3D simulation of material disposal and overflow discharge from the hoppers. The model takes into account the hindrance and resistance mechanisms in the mixture and is capable of describing the flow details within the plumes and gives excellent results when compared to experimental data.

Keywords: Overflow; Disposal; Dredging; Plume; Multiphase;

1. Introduction

Sediment releases in open waters is a common consequence of different offshore and coastal projects including dredging works, land reclamation, coastline extension and contaminated sediment disposal. Disposal of dredged material and the overflow discharge during the dredging activities is a matter of concern due to the potential risks imposed by high concentration plumes on the surrounding marine environment. The plume can often increase the local turbidity levels and cause burial of biological habitats. The ability to model and predict the behavior of released material through overflow from hoppers or during sediment disposal is required as part of EIA (environmental impact assessment) process and also to obtain higher efficiency in projects engaging material disposal such as land reclamation.

The bulk release of negative buoyant particles into water undergoes three main stages, which are i) convective descent, ii) collapse and iii) long term dispersion. The reason for such division is the difference in governing mechanisms at each stage. During the initial stage, the plume, which also is referred as "cap" or "parent cloud", settles as an almost bulk entity due to the (negative) buoyancy, expanding initially and entraining the surrounding ambient fluid by small eddies due to density differences and shear acting along its perimeter. Density gradients between the plume and the surrounding fluid constitute the main drive in its downward movement. Hindrance mechanisms due to high concentrations inside the plume are active as well, which in case of a wide grain size distribution stripping or separation may occur during the falling stage. The second stage is when the plume reaches the bottom and collapses. Depending on the magnitude of density gradients when the plume collapses, it can be followed

by a spreading density current at the bottom. Afterwards, as inertial mechanisms of disposal weaken out, the plume enters the final stage of dispersion transport, where particles are mostly driven by ambient currents, turbulent eddies and hindered settling mechanisms.

Numerical methods, both in Eulerian and Lagrangian approach, have been used to model and resolve most features of descending plumes. Eulerian solutions have been more popular due to their less computational cost. Many authors applied the so called Boussinesq's approximation (or the passive scalar approach) in solving the flow equations and transport of material in particle clouds (Jiang, 1997 and Li, 1997). This approach disregards the volume occupancy of sediments which has considerable effects through the fulfillment of the continuity equation in high concentrations. As a result, it typically fails to capture and resolve detailed aspects of the plume evolution such as the creation of vortex rings, meandering behavior of the plume during the descent and the characteristic double peak observed in the concentration. These shortcomings compromise the use of two-phase models in plumes modeling. Nguyen et al. (2012) used the two phase numerical model developed by Drew and Lahey (1993). It solves the mass and momentum conservation equations for each phase, which requires sophisticated closures for interphase momentum transfer terms. They illustrated some of the shortcomings of models using the passive approach. However, their model failed to resolve the instabilities produced by the entrainment mechanisms during the descending stage, which resulted in less accurate representation of the plume evolution, particularly the dilution. Ishii and Hibiki (2006) presented the concept of Drift-flux method in the modeling of two phase flows. The method considers the mixture as a whole, solving one continuity and momentum equation based on the mixture center of mass, plus the continuity equation for dispersing phase. This relatively simpler and computationally cheaper approach replaces the complexities stemmed from resolving the interphase interactions by constitutive equations based on kinematic relations between the phases.

The aim of this paper is to validate the applicability and the relevance of drift-flux mixture method in simulating the near field dilution of sediment mixtures from overflow and material disposal in dredging activities. Measured data from the overflow discharge experiment presented by Winterwerp (2002) and disposal test carried out by Burel and Garapon (2002) have been used for validation.

The model equations and solution techniques are described briefly in Section 2. The experimental data are described in Section 3 and in Section 4 the simulation results are presented. Finally in Section 5 the conclusion and discussion on the model performance and applicability are presented.

2. Model description

The two phase system of sediment and water is solved based on the drift-flux theory (Ishii and Hibiki, 2006). It considers the mixture as a whole, and solves the Navier-Stokes equations at its center of mass, or in other words, the properties of the mixture are weight (mass) averaged quantities of its constituents. This is due to the fact that the quantities such as volume, momentum and energy are additive set functions of mass. Regarding the important role of entraining eddies and substantial effects of density gradients and entrainment mechanisms on production and dissipation of turbulence around and inside the sediment plume (Pedersen,

1980), Large Eddy Simulation (LES) method has been used, including sub-grid scale (SGS) models to count for unresolved sub-scale eddies. In this study the model considers one type of sediment and the properties of the dispersing phase are based on the mean grain size and density of the sediments.

2.1 Governing equations

The definition of mixture density and the weighted averaged velocity of the mixture center of mass are defined in Eq. [1] and Eq. [2] where subscripts m, s, and w stand for mixture, sediment and water respectively.

$$\rho_m = \alpha_s \rho_s + (1 - \alpha_s) \rho_w \quad [1]$$

$$U_m = \frac{\alpha_s \rho_s U_s + (1 - \alpha_s) \rho_w U_w}{\rho_m} \quad [2]$$

where α , ρ and U represent volumetric concentration, density and velocity respectively. The conservation of mass and momentum of the mixture are solved through Eq. [3] and Eq. [4], followed by the continuity equation for the sediment phase (Eq. [5]).

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m U_m) = 0 \quad [3]$$

$$\begin{aligned} \frac{\partial \rho_m U_m}{\partial t} + \nabla \cdot (\rho_m U_m U_m) = & -\nabla P_m + \nabla \cdot (\tau + \tau^{SGS}) + \rho_m g \\ & -\nabla \cdot ((1 - \alpha_s) \rho_w U_{w/m} U_{w/m}) \\ & -\nabla \cdot (\alpha_s \rho_s U_{s/m} U_{s/m}) \end{aligned} \quad [4]$$

$$\frac{\partial \alpha_s}{\partial t} + \nabla \cdot (\alpha_s U_m) = -\nabla \cdot (\alpha_s U_{s/m}) + \nabla^2 (\alpha_s v_{SGS}) \quad [5]$$

The total pressure in the mixture, P_m , has been considered to be continuous, and the terms $U_{s/m}$ and $U_{w/m}$, as depicted in Figure 1, are the relative velocities of sediment and water to the mixture respectively.

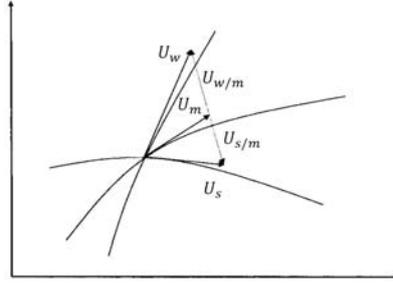


Figure 1. Vector presentation of the velocities in the mixture

The last two terms on the right hand side of Eq. [4], are the momentum diffusion terms. They take into account the momentum transfer inside the mixture due to the relative motion of each phase to the mixture.

$$U_{s/m} = U_s - U_m \quad [6]$$

$$U_{w/m} = U_w - U_m \quad [7]$$

The concept of apparent viscosity of mixtures due to presence of rigid particles was introduced by Einstein (1911) for dilute mixtures, which later was extended to the cases with higher concentrations by several authors. In this study, the viscosity of the mixture ν_m is a function of concentration based on Eq.[8] given by Roscoe (1952).

$$\nu_m = \nu_w(1 - \alpha_s)^{-2.5} \quad [8]$$

The relative velocity of the sediments to the mixture is determined based on their fall velocity. A single particle in clear water has a fall velocity due to the balance between the drag and buoyancy forces acting on it. The presence of other particles will induce hindrance effects which substantially reduces their settling rate at high concentrations (Richardson and Zaki, 1954). The so called slip velocity is defined as the relative velocity of each particle in the mixture to the surrounding fluid. The slip velocity comprises all the hindrance effects acting on particles in a mixture except the hindrance due to the return flow caused by downward displacement of particles. The latter is taken into account automatically when solving the continuity equation for the mixture.

$$U_s - U_w = V_{slip} = W_0(1 - \alpha_s)^{b-1} \quad [9]$$

where W_0 is the particles fall velocity in clear water and b is an empirical value, function of particles Reynolds number (Garside and Al-dibouni, 1977).

$$\frac{5.1 - b}{b - 2.7} = 0.1Re^{0.9} \quad [10]$$

After some algebra, and based on the continuity, the relative velocities of sediment and water to the mixture, $U_{s/m}$ and $U_{w/m}$ are determined as a function of particles slip velocity and concentration.

$$U_{s/m} = (1 - \alpha_s)V_{slip} \quad [11]$$

$$U_{w/m} = -\alpha_s V_{slip} \quad [12]$$

The large eddy simulation method (LES) resolves the large energetic eddies and sub grid scales are modeled with one equation turbulence kinetic energy model. The further damping of SGS scales in regions with high concentration has been taken into account by including the variable mixture density in SGS model equations.

2.2 Numerical technique

The system of equations 1, 2 and 3 is solved by finite volume method on staggered grids using the open source C++ libraries OpenFOAM. The PISO pressure-velocity coupling algorithm is used, where the pressure is solved by the multi-grid technique. The time integrations are solved by the implicit first order Euler scheme and the convective terms are handled by the Gauss method with second order linear interpolation scheme. Application of LES method requires a fine computational mesh in order to have its best performance. The 3D domain is discretized into hexahedral cells graded towards the center of computational domain, where the cell sizes are in the orders of millimeters.

3. Experimental data

The model evaluation is done through simulating two different laboratory test cases. The sediment dumping test done by Burel and Garapon (2002) and the hopper overflow modeling experiment carried out by Boot (2000) in the Laboratory for Fluid mechanics of Delft University of Technology are chosen. The bases for choosing these two cases are both the relevance of their tests to this work and the clear specifications of their testing facilities and measured data.

Burel and Garapon carried out their experiment in a flume of dimensions 80 m long by 1.5 m wide and 1 m high. A mixture of fine sand and water with 200 g/l concentration was released upon ambient water from a cylindrical container having diameter of 172 mm, and height of 200 mm. The sand grains had density of 2650 kg/m³ and diameter of 90 μm. The measured quantities were the cloud diameter and the vertical distance from the bottom of the cloud to the bed, based on threshold concentration of 0.5 g/l.

The hopper overflow experiments presented by Winterwerp (2002) were carried out using a model ship, with a mounted overflow pipe in its hull, in a straight flume with 40 m length, 2m width and 0.33m depth. The model ship had 0.13 m draught and a mixture of Kaolinite and water with concentration of 1033g/l was discharged from a mixing tank into the pipe with constant rate (Figure 2).

The solid material had density of 2650 kg/m³ and grain size diameter of 1-10 μm. The discharge of the overflow mixture through the 25 mm wide overflow shaft at the bottom of model ship was studied, and the radius of descending plume was measured.

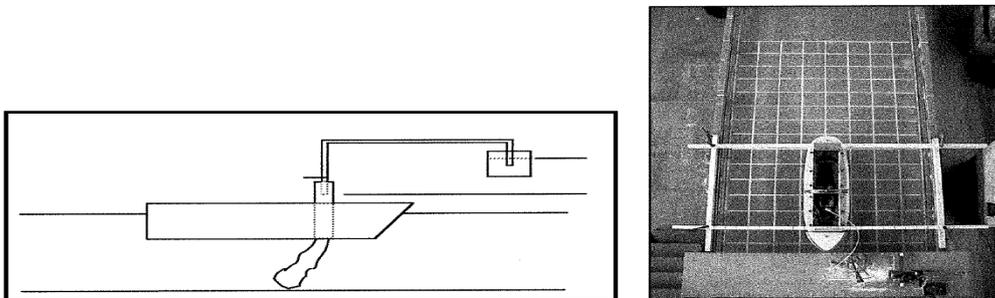


Figure 2. Left: Schematic drawing of the experimental setup, Right: Top view of the model ship in the flume (Boot, 2000)

4. Simulation and results

Numerical simulation of the above described experiments has been performed in a three dimensional domain in Cartesian system with hexahedral grid cells. The free surface is modeled by the slip condition, and the longitudinal sides of the flumes have been modeled as an open (Neumann) boundary condition. The bottom, front and back walls of the flumes are modeled as walls with no flux conditions. Below the results from the CFD model are presented and compared to the measured data.

4.1 Sediment disposal test (Burel and Garapon, 2002)

The disposal of the sediment mixture has been modeled by instantaneous release of the mixture located within a cylindrical wall. The releasing mechanism used in the experiments, as mentioned by the authors as well, had disturbing effects on the falling material. This has been partially compensated by 20% reduction of the cylinder diameter in the simulations. The diameter of the plume during the three stages of conductive descent, collapse and the horizontal spreading has been compared to the experimental data based on the 0.5 g/l concentration threshold (Figure 4).

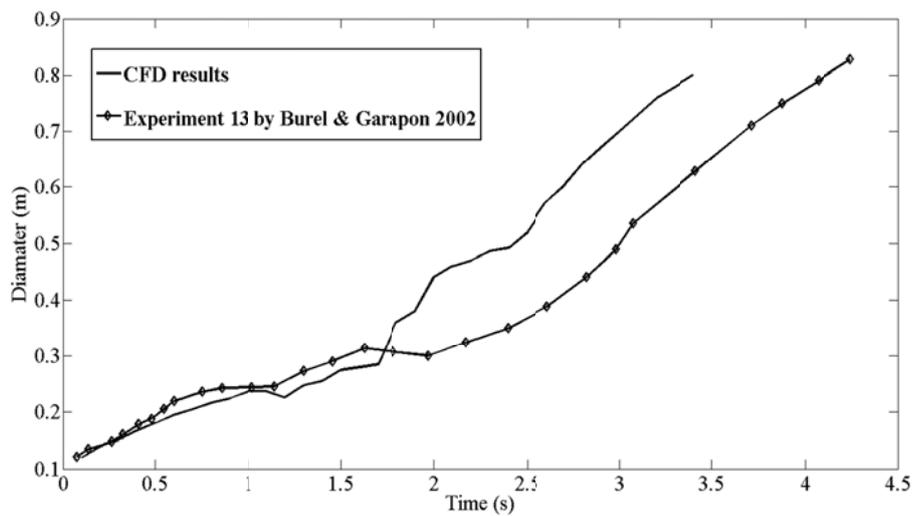


Figure 3. Cloud diameter evolution

The calculated cloud diameters by the CFD model agree well with the experimental data during the beginning stages of falling. The model predicts more rapid spreading of the bottom cloud in later time steps. The reason for higher velocities in the model could be that in the model the disturbance effects at the opening of the container have not been considered. Simulation of the onset of dropping instance is another crucial factor in sediment disposal modeling which has huge impact on the results. Simple release of mixture material inside the computational domain does not exactly represent the physics of the dropping moment. Representing the release period as an inlet discharge (Nguyen et al., 2012) on the other hand, requires empirical assumptions and try and errors to determine the right values for inlet velocities, which again the choice of velocity magnitude and profile at the inlet has significant

impacts on the results. Figure 4 is a qualitative comparison of the plume shape between a photo from the experiments and a snapshot from the CFD model showing the volume counters of concentrations down to the threshold limit of 0.5 g/l.

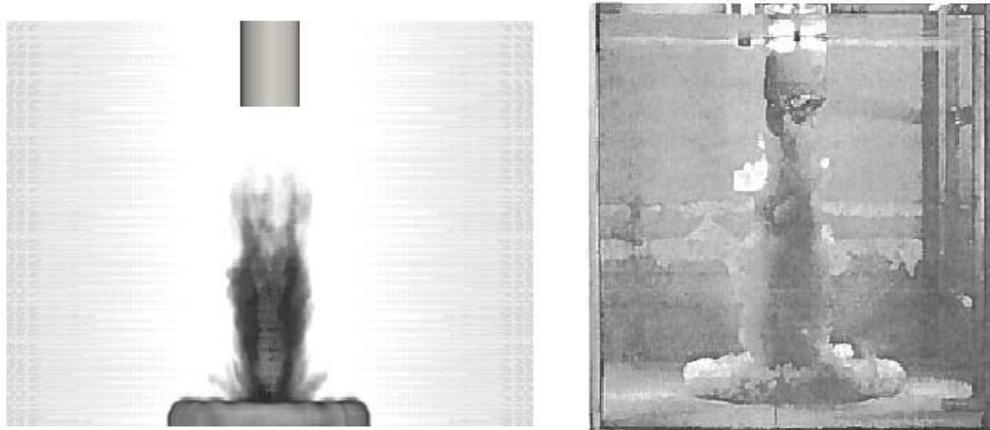


Figure 4. Left: Concentration volume contours from the CFD simulation , Right: photo from the experiment (Burel & Garapon, 2002)

4.2 Overflow discharge test (Winterwerp, 2002)

The constant discharge of the overflow material from the hopper has been simulated as an influx of the mixture into the computational domain from a circular inlet at the top, representing the overflow pipe at the bottom of the model ship. The overflow discharge in calm water condition, with inflow velocity of 0.2 m/s has been simulated by the CFD model. The measured radial expansion of the plume has been compared with CFD results in Figure 5. Here the CFD model underestimates the initial expansion rate. One reason could be that the initial plunging of the overflow material from the overflow shaft into the ambient water has not been considered in the model, as well as the longer time it takes for numerical model to accelerate the flow from rest. Figure 6 is a qualitative comparison of the plume shape between a photo from the experiments and a snapshot from the CFD model showing the volume concentrations. The ability of the numerical model to capture the detailed flow structures inside the plume, such as meandering and creation of concentration fingerings are clearly visible.

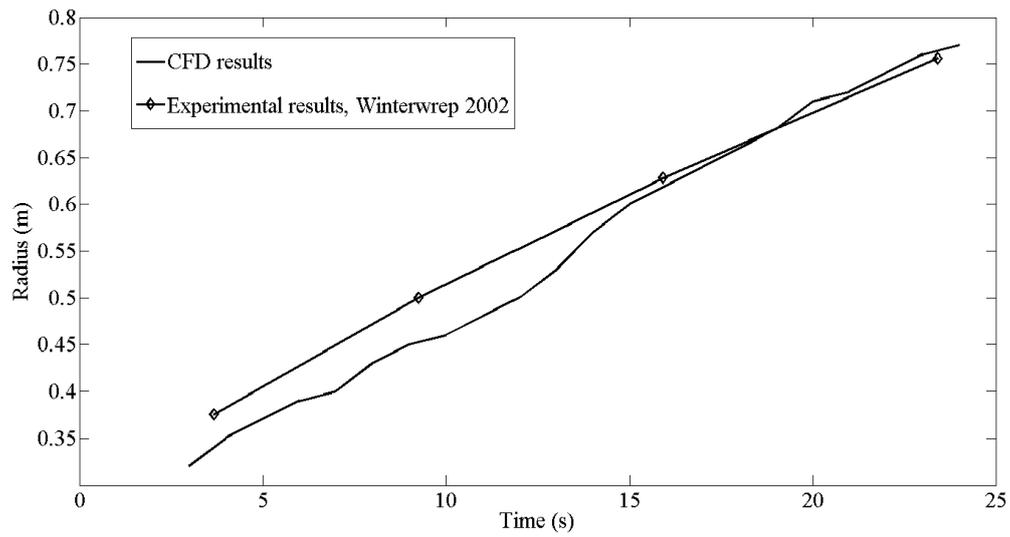


Figure 5. Radial evolution of the bottom cloud

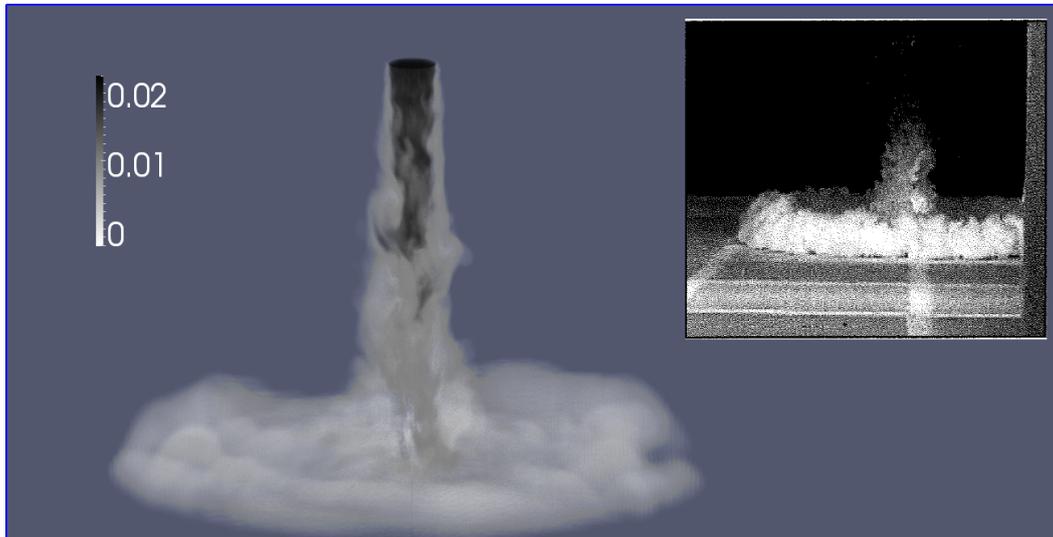


Figure 6. Left: Concentrations from the CFD simulation, Right: photo from the experiment (Boot, 2000)

5. Conclusions and discussion

The performance of the drift-flux method on simulating the near field fate of particle clouds has been evaluated in this study. The model is capable of capturing the flow details and processes as creation of vortex rings and meandering behaviors of the descending plume. It is less demanding in terms of computational resources by considering the two phases as one mixture, and eliminating the sophisticated interphase interactions. This advantage becomes more substantial when modeling multi sized mixture of particles where the coupled interaction between the particles can be handled by kinematic relations inside the mixture. However, this model, same as other two-phase Eulerian models, deals with the solid part of the mixture as another heavier fluid with some additional resisting characteristics stemming from collisional and frictional behavior of the particles. This approach though does a good job as long as the particles are fine enough.

This model will be used to further investigate the behavior of the overflow discharges from the hoppers and sediment disposals depending on the conditions of the receiving medium. Whereas local currents and waves, the propeller of the dredger ship and the air bubbles entraining the plume from overflow shafts have considerable effects on the fate of released sediments.

Acknowledgments

The work has been supported by the Danish Ministry of Science, Technology and Enovation through the GTS-grant "Marine Structures of the Future".

References

- Boot M. 2000. 'Near-field verspreiding van het overvloeiverlies van een sleepopperzuiger', Technische Universiteit Delft, Netherlands.
- Burel D. and Garapon A. 2002. '3D numerical modelling of dredged material descent', *Proc. of 28th International Conference on Coastal Engineering, Cardiff, Wales*, vol. 3, pp.3143-3154.
- Drew D.A., Lahey R.T. 1993. 'Analytical modelling of multiphase flow' in Roco M.C. editor 'Particulate two-phase flow', Butterworth-Heimann Series in Chemical Engineering, USA.
- Einstein A. 1911. , *Ann. Phys., Lpz.*, vol. 34, p.591.
- Garside J. and Al-Dibouni M.R. 1977. 'Velocity-voidage relationships for fluidization and sedimentation in solid-liquid systems', *Ind. Eng. Chem., Process Des. Dev.*, vol. 16, pp.206-214.
- Ishii M. and Hibiki T. 2006. 'Thermo-fluid dynamics of two-phase flow', Springer, United States of America. ISBN-10: 0-387-28321-8.
- Jiang Q. and Kunisu H. 1997. 'Numerical modelling of the settling processes of dredged material disposed in open waters', *Proc. of 7th International Offshore and Polar Engineering Conference, Honolulu, USA*, vol. 1, pp.510-515.
- Li C.W. 1997. 'Convection of particle thermals', *Journal of Hydraulic Research*, vol. 35, pp.363-376.

- Nguyen D.H., Levy F., Van Bang D.P., Guillou S., Nguyen K.D. and Chauchat J. 2012. 'Simulation of dredged sediment releases into homogeneous water using two-phase model', *Advances in Water Resources*, vol. 48, pp.102-112.
- Pedersen F.B. 1980. 'A monograph on turbulent entrainment and friction in two-layer stratified flow', Series paper 25, Institute of Hydrodynamics and Hydraulic Engineering, Technical University of Denmark, Lyngby.
- Richardson J.F. and Zaki W.N. 1954. , *Trans. Inst. Chem. Eng.*, vol. 32.
- Roscoe R. 1952. 'The viscosity of suspensions of rigid spheres', *British Journal of Applied Physics*, vol. 3, pp.267-269.
- Winterwerp J.C. 2002. 'Nearfield behaviour of dredging spill in shallow water', *Journal of Waterway, Port, Coastal and Ocean Engineering*, vol. 128, pp.96-98.