MIKE 3 Flow Model FM

Mud Transport Module

User Guide

PLEASE NOTE

COPYRIGHT

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI A/S (hereinafter referred to as "DHI"). For details please refer to your 'DHI Software Licence Agreement'.

LIMITED LIABILITY

The liability of DHI is limited as specified in your DHI Software Licence Agreement:

In no event shall DHI or its representatives (agents and suppliers) be liable for any damages whatsoever including, without limitation, special, indirect, incidental or consequential damages or damages for loss of business profits or savings, business interruption, loss of business information or other pecuniary loss arising in connection with the Agreement, e.g. out of Licensee's use of or the inability to use the Software, even if DHI has been advised of the possibility of such damages.

This limitation shall apply to claims of personal injury to the extent permitted by law. Some jurisdictions do not allow the exclusion or limitation of liability for consequential, special, indirect, incidental damages and, accordingly, some portions of these limitations may not apply.

Notwithstanding the above, DHI's total liability (whether in contract, tort, including negligence, or otherwise) under or in connection with the Agreement shall in aggregate during the term not exceed the lesser of EUR 10.000 or the fees paid by Licensee under the Agreement during the 12 months' period previous to the event giving rise to a claim.

Licensee acknowledge that the liability limitations and exclusions set out in the Agreement reflect the allocation of risk negotiated and agreed by the parties and that DHI would not enter into the Agreement without these limitations and exclusions on its liability. These limitations and exclusions will apply notwithstanding any failure of essential purpose of any limited remedy.

CONTENTS

1 ABOUT THIS GUIDE

1.1 Purpose

The main purpose of this User Guide is to enable you to use, MIKE 3 Flow Model FM, Mud Transport Module, for applications involving the modelling of cohesive sediment transport.

1.2 Assumed User Background

Although the mud transport module has been designed carefully with emphasis on a logical and user-friendly interface, and although the User Guide and Online Help contains modelling procedures and a large amount of reference material, common sense is always needed in any practical application.

In this case, "common sense" means a background in sediment transport problems, which is sufficient for you to be able to check whether the results are reasonable or not. This User Guide is not intended as a substitute for a basic knowledge of the area in which you are working: Mathematical modelling of cohesive sediment transport.

It is assumed that you are familiar with the basic elements of MIKE Zero: File types and file editors, the Plot Composer, the MIKE Zero Toolbox, the Data Viewer and the Mesh Generator. The documentation for these can be found by the MIKE Zero Documentation Index.

1.3 General Editor Layout

The MIKE Zero setup editor consists of three separate panes.

1.3.1 Navigation tree

To the left there is a navigation tree that shows the structure of the model setup file, and is used to navigate through the separate sections of the file. By selecting an item in this tree, the corresponding editor is shown in the central pane of the setup editor.

1.3.2 Editor window

The editor for the selected section is shown in the central pane. The content of this editor is specific for the selected section, and might contain several property pages.

For sections containing spatial data - e.g. sources, boundaries and output - a geographic view showing the location of the relevant items will be available. The current navigation mode is selected in the bottom of this view, it can be

zoomed in, zoomed out or recentered. A context menu is available from which the user can select to show the bathymetry or the mesh and to show the legend. From this context menu it is also possible to navigate to the previous and next zoom extent and to zoom to full extent. If the context menu is opened on an item - e.g. a source - it is also possible to jump to this items editor.

Further options may be available in the context menu depending on the section being edited.

1.3.3   Validation window

The bottom pane of the editor shows possible validation errors, and is dynamically updated to reflect the current status of the setup specifications.

By double-clicking on an error in this window, the editor in which this error occurs will be selected.

1.4   Online Help

The Online Help can be activated in several ways, depending on the user's requirement:

- **F1-key seeking help on a specific activated dialog:**

    To access the help associated with a specific dialog page, press the F1-key on the keyboard after opening the editor and activating the specific property page.

- **Open the Online Help system for browsing manually after a specific help page:**

    Open the Online Help system by selecting "Help Topics" in the main menu bar.

2   INTRODUCTION

2.1   General Description

The Mud Transport (MT) module of MIKE 3 Flow Model FM describes erosion, transport and deposition of mud or sand/mud mixtures under the action of currents and waves.

The MT module is applicable for:

- Mud fractions alone, and
- Sand/mud mixtures

The following processes can be included in the simulation.

- Forcing by waves
- Salt-flocculation
- Detailed description of the settling process
- Layered description of the bed
- Morphological update of the bed

In the MT-module, the settling velocity varies, according to the salinity, if included, and the concentration taking into account flocculation in the water column. Waves, as calculated by MIKE 21 SW for example, may be included. Furthermore, hindered settling and consolidation in the fluid mud and under-consolidated bed are included in the model. Bed erosion can be either non-uniform, i.e. the erosion of soft and partly consolidated bed, or uniform, i.e. the erosion of a dense and consolidated bed. The bed is described as layered and characterised by the density and shear strength.

2.1.1   Application areas

The Mud Transport Module can be applied to the study of engineering problems such as:

- Sediment transport studies for fine cohesive materials or sand/mud mixtures in estuaries and coastal areas in which environmental aspects are involved and degradation of water quality may occur.
- Siltation in harbours, navigational fairways, canals, rivers and reservoirs.
- Dredging studies.

Mud Transport Module FM - © DHI A/S

3 Getting Started

The hydrodynamic basis for the Mud Transport Module must be calculated using the Hydrodynamic Module of the MIKE 3 Flow Model FM modelling system.

If you are not familiar with setting up a hydrodynamic model you should refer to User Guide for the Hydrodynamic Module and the comprehensive step-by-step training guide covering the Hydrodynamic Module of MIKE 3 Flow Model FM. The user guide and the training guide (PDF-format) can be accessed from the MIKE 3 Documentation index:

> MIKE 3 Flow Model, Hydrodynamic Module, User Guide

> MIKE 21 & MIKE 3 Flow Model FM, Hydrodynamic Module, Step-by-Step Training Guide

A comprehensive training guide covering the Mud Transport Module of the MIKE 21 Flow Model FM modelling system is also provided with the DHI Softwave installation.The objective of this training guide is to set up a Mud Transport model for the Grådyb area (in Denmark) from scratch and to calibrate the model to a satisfactory level. The training guide (PDF-format) can be accessed from the MIKE 21 Documentation index:

> MIKE 21 Flow Model FM, Mud Transport Module, Step-by-Step Training Guide

4 Examples

4.1 General

One of the best ways of learning how to use a modelling system such as MIKE 3 Flow Model FM is through practice. Therefore examples are included which you can go through yourself and which you can modify, if you like, in order to see what happens if one or other parameter is changed.

The specification files for the examples are included with the installation of MIKE Zero. A directory is provided for each example. The directory names are as follows:

- Harbour Basin example:
  .\Examples\MIKE_3\FlowModel_FM\MT\HarbourBasin

4.2 Sedimentation in Harbour

4.2.1 Purpose of the example

This simplified example has been chosen to describe a typical case of sedimentation in a harbour.

The harbour is placed at constant water depth and has vertical walls. A small pier is sticking out into the bypassing flow. The computational domain and bathymetry is shown in Figure 4.1.

The pier will cause an eddy to form behind the pier and fine material will settle because of the disturbance of the flow. In this example it is assumed that no waves exist, and that no flow pass the offshore boundary.

To investigate the magnitude of the sedimentation in the harbour the model set-up is simulated for 2.5 days.

4.2.2 Defining the problem

The main condition defining the hydrodynamic problem is:

- An unstructured mesh with 519 nodes and 907 elements is used. The mesh is shown in Figure 4.1.
- The bed level is constant -5 m.
- The flow is running from left to right due to a constant gradient in the water level. There are 3 open boundaries. The water level at the left boundary is 0.10 m and the water level at the right boundary is 0.0 m. The offshore boundary has zero water flux across the boundary.

- The *time step* of 600 seconds is selected and the duration time of the simulation is 60 hours (360 time steps).

- The *horizontal eddy viscosity* type has been chosen to Smagorinsky formulation with a constant value of 0.28.

- The *bed resistance* type has been chosen to Roughness height with a constant value of 0.25 m.

- The length of the *soft start interval* (warm-up period) for the wind has been chosen to 1 hour to aboid chock effects.
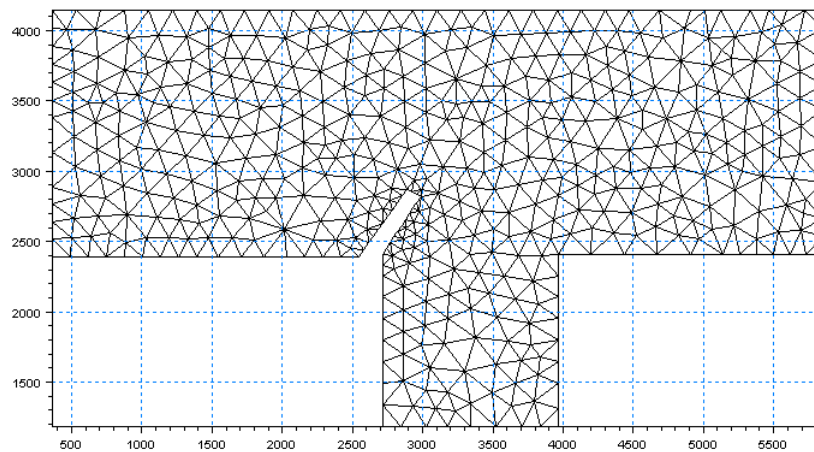
Figure 4.1   Harbour sedimation, model layout and mesh.

The main condition defining the mud transport problem is:

- Mud transport calculations are performed for 2 fractions and 2 layers.

- Flocculation is included with a constant velocity coefficient of 50 m/s for fraction 1 (coarse) and 5 m/s for fraction 2 (fine).

- A Teeter profile is applied with a constant critical shear stress of 0.07 N/m^2 for both fractions.

- Layer 1 is described as a soft mud layer with constant erosion coefficient of $5 \cdot 10^{-5}$ kg/m^2/s and a critical shear stress of 0.1 N/m^2. The power of erosion has a value of 10 and the bed densisty is constant 200 kg/m^3.

- Layer 2 is described as a hard mud layer with constant erosion coefficient of 0.0001 kg/m^2/s and a critical shear stress of 0.2 N/m^2. The power of erosion has a value of 1 and the bed density is constant 400 kg/m^3.

- The bed roughness is described as constant 0.001 m.

- The dispersion is described by the dispersion coefficient formulation with a constant dispersion coefficient of 0.01 m^2/s and 0.015 m^2/s for fraction 1 and 2, respectively.

- During the simulation the concentration of the sediment fractions at the upstream boundary is constant 0.001 kg/m^3 and 0.02 kg/m^3 for fraction 1 and 2, respectively.

- Initially, the surface elevation is zero throughout the model and the concentration of the two fractions are equal to the boundary conditions. The soft mud layer has a thickness of 0.002 m and consist of 10 % coarse material and 90 % fine material.
  The hard mud layer has a thickness of 1.0 m and consist of 80 % coarse material and 20% fine material.

4.2.3   Presenting and evaluating the results

Contour and vector plots of the results from the simulation can be viewed by using the MIKE Zero Data Viewer.

Figure 4.2 shows the resulting current field for the simulation.
The longshore current has a value of about 0.35 m/s. When passing the pier the flow velocity locally exceed 0.5 m/s. The flow is creating an eddy behind the pier that in turn creates a weak circulation current inside the harbour.
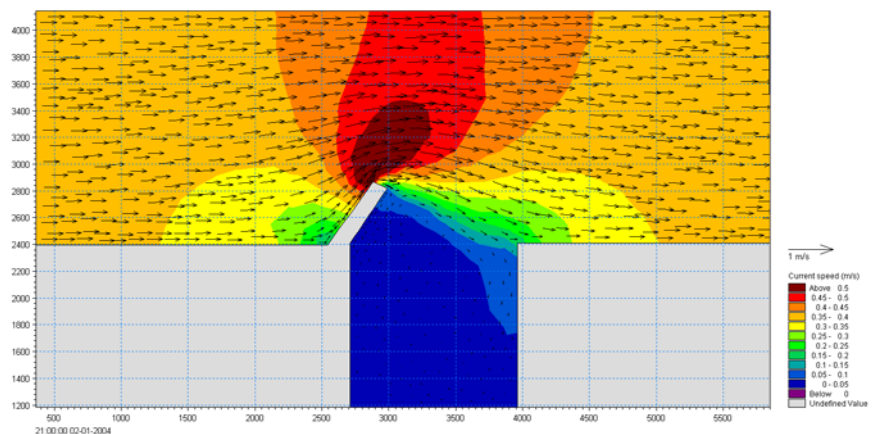
Figure 4.2   Resulting depth averaged current field from simulation

Figure 4.3 show the resulting sedimentation inside and outside the harbour. The bed is eroded outside the harbour at the pier whereas sedimentation occurs in the corner by the pier and inside the harbour. Higher deposition rates appear at the harbour entrance than inside the harbour.

03-01-2004 12:00:00
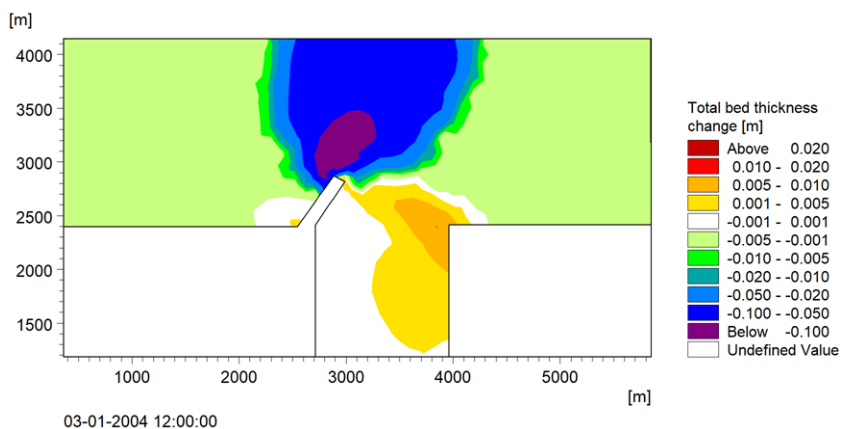
Figure 4.3   Resulting bed level change after 60 hours.

4.2.4   List of data and specification files

The following data files (included in the \MT\HarbourBasin folder) are supplied with MIKE 3 Flow Model MT FM:

File name: HarbourBasin.mesh
Description: Mesh file including the mesh and bathymetry

File name: HarbourBasin.m3fm
Description: MIKE 3 Flow Model FM specification file

5 MUD TRANSPORT MODULE

Sediment transport is dependent on the hydrodynamic conditions. In general, there are two types of sediment transport. Cohesive and non-cohesive. The cohesive is characterized by low settling velocities and long response times for hydrodynamic changes. Therefore, the transport is dominated by the advection of sediment in the water column. For non-cohesive sediments the settling velocities are in general larger and the concentration profile will therefore quickly adjust to changes in hydrodynamics. As a consequence of this a major part of this transport will take place on, or very close to, the bed as bed load.

The mud transport module calculates both the transport of cohesive and non-cohesive sediment based on the flow conditions found in the hydrodynamic calculations and, if included, wave conditions from wave calculations.

5.1 Parameter Selection

Select number of grain size fractions and bed layers in the simulation.

The maximum number of fractions is 8.
The maximum number of layers is 12.

It is recommended to keep the number of layers as low as possible. The number of layers should be sufficient to represent the strength variation in the bed.

5.2 Solution Technique

The simulation time and accuracy can be controlled by specifying the order of the numerical schemes which are used in the numerical calculations. Both the scheme for time integration and for space discretization can be specified. You can select either a lower order scheme (first order) or a higher order scheme. The lower order scheme is faster, but less accurate. For more details on the numerical solution techniques, see the scientific documentation.

The time integration of the shallow water equations and the transport (advection-dispersion) equations is performed using a semi-implicit scheme, where the horizontal terms are treated explicitly and the vertical terms are treated implicitly. Due to the stability restriction using an explicit scheme the time step interval must be selected so that the CFL number is less than 1. A variable time step interval is used in the calculation and it is determined so that the CFL number is less than a critical CFL number in all computational nodes. To control the time step it is also possible for the user to specify a minimum time step and a maximum time step. The time step interval for the transport equa-

tions is synchronized to match the overall time step specified on the Time dialog.

The minimum and maximum time step interval and the critical CFL number is specified in the Solution Technique dialog in the HYDRODYNAMIC MODULE.

5.2.1 Remarks and hints

If the important processes are dominated by convection (flow), then higher order space discretization should be chosen. If they are dominated by diffusion, the lower order space discretization can be sufficiently accurate. In general, the time integration method and space discretization method should be chosen alike.

Choosing the higher order scheme for time integration will increase the computing time by a factor of 2 compared to the lower order scheme. Choosing the higher order scheme for space discretization will increase the computing time by a factor of 1½ to 2. Choosing both as higher order will increase the computing time by a factor of 3-4. However, the higher order scheme will in general produce results that are significantly more accurate than the lower order scheme.

The default value for the critical CFL number is 1, which should secure stability. However, the calculation of the CFL number is only an estimate. Hence, stability problems can occur using this value. In these cases you can reduce the critical CFL number. It must be in the range from 0 to 1. Alternatively, you can reduce the maximum time step interval. Note, that setting the minimum and maximum time step interval equal to the overall time step interval specified on the Time dialog, the time integration will be performed with constant time step. In this case the time step interval should be selected so the the CFL number is smaller than 1.

The total number of time steps in the calculation and the maximum and minimum time interval during the calculation are printed in the log-file for the simulation. The CFL number can be saved in an output file.

The higher order scheme can exhibit under and over shoots in regions with steep gradients. Hence, when the higher order scheme is used in combination with a limitation on the minimum and maximum value of the concentration, mass conservation cannot be guarenteed.

5.3 Water Column Parameters

Water column parameters consist of all processes in the water column.

Here you must specify if the simulation is purely cohesive or if some of the fractions are to be treated as noncohesive fractions (sand). Normally sedi-

ment with a diameter greater than 60 μm are considered non cohesive. Under water column parameters the following sections are included:

- Sand fractions
- Settling
- Deposition
- Viscosity and density

5.3.1 Sand fractions

The Mud Transport module can take transport of non-cohesive sediment (sand) into account. Two sediment transport formulations can be applied

- Suspended load
- Near-bed load and suspended load

When "Suspended load" is selected you must specify the fall velocity for each of the sand fractions.

When "Near-bed load and suspended load" is selected only one fraction can be specified as a sand fraction. You have to specify the path and name of the files containing the sediment transport tables. The tables are ASCII files and the tables are defined by the extension .lon. The program will always check that tables with the same name and extension .crs, .con., .cen, .ufc and .knw exists in the same directory. The tables are generated using the MIKE 21 Toolbox program "Generation of Q3D Sediment Tables"

General description

For the "Suspended load" formulation the suspended transport is taken into account by calculating an equilibrium concentration profile based on the sediment properties and the hydrodynamics.

The bed is assumed to erode as flakes which means that the distribution of fractions within the bed is also the distribution when eroded. This means that the erosion formula used in the MT section controls the maximum erosion of all fractions. After the flakes has been eroded it is assumed that they are destroyed or regrouped by turbulence. Since the sand fractions has no cohesive properties it will be freed by this and behave independently. The model does this by calculating the maximum possible equilibrium concentration for the given sand under the given hydrodynamic properties. If this is above the concentration of the sand fraction, the extra sand will be deposited so that the concentration is the equilibrium concentration.

For the "Near-bed load and suspended load" formulation the sand transport is divided into two contributions, one contribution from the transport close to the bed ($z<z_{wb}$) and one contribution from the transport farther away from the bed ($z>z_{wb}$). Close to the bed the sand transport rates are assumed to be in equi-

librium with the local forcing conditions and the transport rates are found directly by interpolation in sediment transport tables calculated with STP. Further away from the bed than the distance z_{wb} the sand transport is calculated using the advection-dispersion solver with the bed concentration found by interpolation in the concentration table. The Vanoni profile equation is used to calculate the concentration in the center of the bottom element.

For more information see the Scientific Documentation.

Sediment transport tables

The near-bed sediment transport tables have to be generated beforehand by the MIKE 21 Toolbox program "Generation of Q3D Sediment Tables". The model type "Near-bed transport" must be selected. The tables must be generated such that any combination of bathymetry, current, wave and sediment conditions appearing in the simulation are within the range defined in the transport table.

Six tables are required: Two tables with extension .lon and .crs containing the near bed transport of sand in the direction of the current and perpendicular to the direction of the current, a table with extension .con containing the bed concentration, a table with extension .cen containing the distance z_{wb} and finally, the two tables with extension .ufc and .kwn containing the current friction velocity and the apparent bed roughness, respectively.

During the simulation the transport rates are found by linear interpolation in the tables using input parameters from the ongoing simulation. If the table contains only one value for a parameter, this is per default used as the input parameter. Otherwise, if an input parameter is outside the limit of the tables, an exception is made. In most of the cases the minimum or maximum parameter of the table axis is used but for some parameters the program stops. If the maximum limit is exceeded a warning message is issued in the .log-file for the first 200 events.

Table 5.1 describes the possible exceptions for each input parameter for the interpolation in the transport tables.

Table 5.1   Values applied when input parameters are outside table limits.

Parameter	< minimum	> maximum	Comment
V	extrapolation	$(V)_{max}$	Warning
H_{rms}/D	$(H_{rms}/D)_{min}$	$(H_{rms}/D)_{max}$	Warning
H_{rms}	$(H_{rms})_{min}$	$(H_{rms})_{max}$	Warning
T_p	$(T_p)_{min}$	$(T_p)_{max}$	Warning
dD/dx	$(dD/dx)_{min}$	$(dD/dx)_{max}$	Warning
dD/dy	$(dD/dy)_{min}$	$(dD/dy)_{max}$	Warning
S	-	-	Program stops

Table 5.1   Values applied when input parameters are outside table limits.

Parameter	< minimum	> maximum	Comment
d	-	-	Program stops
gamma	-	-	Program stops
V^2/R	p.t. not used	p.t. not used	

Recommended values

The mean settling velocity can be estimated using Stokes law. See *Settling Velocity*.

Remarks and hints

For the "Suspended load" model, no bed load is included. Therefore, if using this model, make sure that only sand for which the main transport can be expected to be suspended transport is included.

Generally, it is recommended to use the "Near-bed load and suspended load" model.

5.3.2   Settling

The settling velocity for mud in suspension can be divided into four categories

- Constant settling velocity
- Flocculation
- Hindered settling
- Fluid mud

If constant settling velocity is chosen the value used under constant settling velocity will be applied.

If flocculation is chosen the choice of including hindered settling is offered as formulated by either Richardson and Zaki (1954) or Winterwerp (1999).

General Description

The settling velocity is dependent on the size of the particles. The settling velocity of a single free particle can be roughly estimated through Stokes law:

$$w_s = \frac{(\rho_s - \rho)}{\rho} \frac{gd^2}{18 \cdot v} \qquad (5.1)$$

in which:

ρ_s: Sediment density (kg/m^3) (Quartz = 2650 kg/m^3)
ρ: Density of water
g: Gravity (9.82 m/s^2)
d: Grain size [m]
ν: Viscosity [m^2/s]
w_s: Settling velocity [m/s]

In case of fine grained cohesive sediment (<0.006 mm), the size of the parti-cles and thereby the settling velocity will depend on the rate of flocculation.

With low concentrations of suspended sediment, the probability for collision between the cohesive particles is low and the settling velocity will be close to the settling velocity for a single grain. With increasing concentration, collision between particles will occur more frequently and the cohesiveness of the par-ticles will result in formation of flocs. This leads to an increase in average par-ticle/floc size and with that an increase in settling velocity.
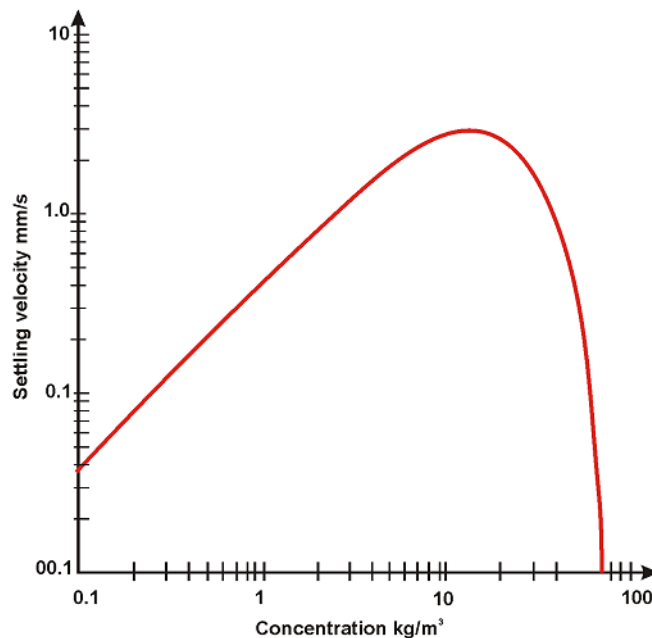

Figure 5.1   Typical settling velocity variation

Many other factors can increase or decrease the floc size. Salinity between 0 and 9 psu will increase flocculation as will high levels of organic material. High levels of turbulence will decrease the floc size due to destruction of flocs.

If sediment concentration increases further, the flocs will eventually interact hydrodynamical so that effectively the flocs during settling cause an upward flow of the liquid they displace and hindered settling occurs, which leads to a reduction in settling velocity.

Further increase in sediment concentration will result in decreasing distance between the flocs, which leads to negligible settling velocity and the mixture will act as fluid mud.

Data

The settling velocity for mud in suspension can be divided into four categories:

- Constant settling velocity
- Flocculation
- Hindered settling
- Fluid mud

If constant settling velocity is chosen, the value used under constant settling velocity will be applied.

If flocculation is chosen, the choice of including hindered settling is offered as formulated by either Richardson and Zaki (1954) or Winterwerp (1999).

Constant settling velocity

Constant settling velocity can be selected if the concentrations are assumed not to influence the settling velocity.

If constant settling velocity is selected, the settling velocity will be kept constant and independent of the concentration of sediment throughout the simulation.

Flocculation

Flocculation is when the concentration of sediment is high enough for the sediment flocs to influence each other's settling velocity. This happens because collisions between flocs will increase floc size leading to higher settling velocities.

Standard flocculation concentrations occur when hindered settling is neglected. The calculation is shown in Figure 5.2.
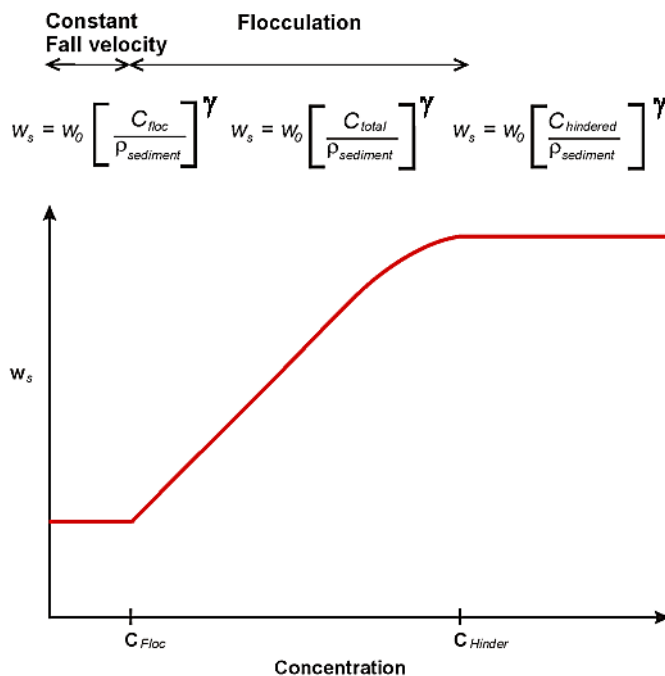
Figure 5.2   Applied concentration profile when flocculation selected

$\rho_{sediment}$	Density of grain material
c_{floc}	Concentration at which flocculation begins
c_{total}	Total concentration of sediment (sum of concentrations of all fractions)
$c_{hindered}$	Concentration at which hindered settling begins
W_s	Settling velocity
W_0	Settling velocity coefficient
γ	Power constant

Hindered settling

Hindered settling is when the concentration of sediment gets high enough for the flocs to influence each other's settling velocity. The concentration gets high enough for the flocs not to fall freely. This results in a lower settling velocity.

When the specified concentration for hindered settling is exceeded hindered settling sets in. The calculation is shown in Figure 5.3.
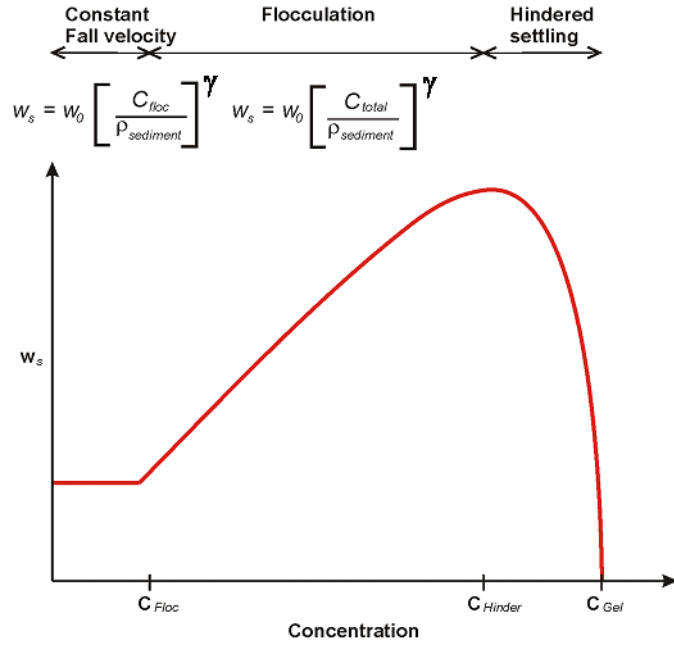
Figure 5.3   Calculation when hindered settling is applied

Two formulations for the settling velocity in this regime are offered.

Formulation by Richardson and Zaki (1954)

For a single mud fraction, the standard Richardson and Zaki formulation is

$$w_s = w_{s,r}\left(1 - \frac{c}{c_{gel}}\right)^{w_{s,n}} \tag{5.2}$$

For multiple mud fractions, the Richardson and Zaki formulation is extended to

$$w_s^i = w_{s,r}^i(1 - \Phi_*)^{w_{s,n}^i} \tag{5.3}$$

where

$$\Phi_* = min(1.0, \Phi) \tag{5.4}$$

$$\Phi = \frac{\sum_i c^i}{c_{gel}} \tag{5.5}$$

In which:

$w_{s,r}$	Settling velocity coefficient
$w_{s,n}$	Power constant for fraction
c_{gel}	Gelling point for sediment

Formulation by Winterwerp (1999)

$$w_s^i = w_{s,r}^i \frac{(1 - \Phi_*)(1 - \Phi_p)}{1 + 2.5\Phi} \tag{5.6}$$

where

$$\Phi_p = \frac{\sum_i c^i}{\rho_s} \tag{5.7}$$

In which ρ_s is the dry density of the sediment.

Fluid mud

Fluid mud is in this model considered as a bottom layer and the settling velocity for this will be treated as consolidation.

Modification of settling velocity due to salinity variation

In fresh/brackish water, the flocculation processes are reduced, which have an impact on the settling velocity. Due to the smaller floc sizes, the settling velocity will be reduced. This is modelled by multiplying the settling velocity with a factor.

$$w_s^i = w_s^i (1 - C_1 e^{S \cdot C_2}) \tag{5.8}$$

where C_1 and C_2 are calibration parameters.

C_1 and C_2 are not shown in the menu and is default set to $C_1 = 0.5$ and $C_2 = -0.33$.

Recommended values

The following values crudely mark the different regimes within the settling process.

Table 5.2   Settling regimes

Sediment stage	Concentration (kg/m^3)	Remarks
Suspension	0-0.01	No flocculation.
Suspension	0.01-10	Flocculation may occur.
Suspension	10-50	Void ratio larger than 6. No effective stresses between grains. Hindered settling begins.

Remarks and hints

For inclusion of the salinity effects on flocculation density variation (function of temperature and salinity or function of salinity) must be included in the Hydrodynamic simulation.

Note that salinity effect on flocculation does not grow when the salinity exceeds 10 PSU. The parameters C_1 and C_2 are by default set to 0.5 and -0.33 and are not shown in the menu. Nor is $W_{s,n}$ and γ. The default values for those are 1.

5.3.3   Deposition

General Description

The deposition of suspended sediment is the transfer of sediment from the water column to the bed. Deposition takes place where the bed shear stress (τ_b) is smaller than the critical shear stress for deposition (τ_{cd}).

The deposition for the i´th mud fraction is described as:

$$D^i = w_s^i c_b^i p_D^i \tag{5.9}$$

where p_D^i is a probability ramp function of deposition, w_s is the fall velocity and c_b is the near bed concentration of fraction i.

The probability ramp function is defined as:

$$\upsilon_D^i = max\left(0, min\left(1, 1 - \frac{\tau_b}{\tau_{cd}^i}\right)\right) \tag{5.10}$$

Concentration profiles

Teeter profile

The mud concentration c_b^i is related to the depth-averaged concentration $\bar{c}^i$ using the Teeter profile:

$$c_b = \bar{c}^i\left(1 + \frac{p_e^i}{1.25 + 4.75p_D^{i\,2.5}}\right) \tag{5.11}$$

where p_e^i is the Peclet number corresponding to the i'th fraction, defined as:

$$p_e^i = 6\frac{w_s^i}{\kappa U_f} \tag{5.12}$$

in which U_f is the friction velocity, and κ is von Karman's constant taken as 0.4.

Rouse profile

The near bed sediment concentration c_b^i is related to the depth-averaged concentration $\bar{c}^i$ using the Rouse profile. Using this the bottom concentration can be defined as:

$$c_b = \frac{\bar{c}^i}{RC} \tag{5.13}$$

In which RC is the relative height of centroid, which is defined as the distance from the sea-bed to the mass centre of the concentration profile divided by the water depth. It is time independent and therefore the concentration profile is considered stationary. The deposition rate can then be calculated as:

$$D = c_b \cdot w_s, \text{ when } \tau_b < \tau_{cd}, \tag{5.14}$$

Data

The critical shear stresses for deposition can be specified as

- Constant (in domain)
- Varying in domain

For the case with varying in domain you have to prepare a data file containing the critical shear stresses before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data.

Recommended values

The critical shear stress for deposition is generally a calibration parameter. The value is generally less than the critical shear stress for erosion. Normal values are in the interval 0-0.1 N/m². Relative height of centroid is generally close to 0.3.

Remarks and hints

If the critical shear stress for deposition is high, more sediment will deposit, and opposite if it is low.

5.3.4 Viscosity and density

If you have chosen other than barotrophic density in the Hydrodynamic Module, you may choose to specify parameters for a possible feedback from the mud transport calculations on the viscosity and density in the hydrodynamic calculations.

You must specify the density of the suspended sediment, and a base and a reference concentration for the feedback on the viscosity.

Remarks and hints

The feedback on the hydrodynamic calculations will only have a significant effect for very high concentrations.

5.4 Bed Parameters

Bed Parameters are defined as the parameters governing the processes going on in the bed. Here you must specify parameters for the erosion, density and bed roughness of each layer.

Under *Bed Parameters* the following sections are included:

- Erosion
- Density of bed layers
- Bed roughness
- Transition between layers

5.4.1 General description

The sediment bed of the Mud transport module consists of one or more bed layers. Each bed layer is defined by the sediment mass contained in the layer and by the dry density and erosion properties of the layer. The sediment mass of a bed layer is comprised by the summation of the mass of each sediment fraction present in the layer. The bed layer masses are considered the state variables of the sediment bed, which means that the model during simulation tracks their evolution in space and time. The dry density and erosion properties, on the other hand, are considered properties of each of the bed layers and are therefore kept constant in time.

The bed layers are perceived as "functional" layers, where each layer is characterised by its dry density and erosion properties, rather than physical layers, whose physical properties will typically vary in time due to consolidation and other processes. In the present bed description, the consolidation process is expressed as a transfer of sediment mass from one bed layer to another.

The bed layers are organised such that the "weakest" layer (typically fluid mud or newly deposited sediment) is defined as the first (uppermost) layer and that the subsequent layers have increasing dry density and strength. Figure 5.4 shows an example of a bed description including two bed layers and the processes affecting it. During a simulation one or more layers may be completely eroded such that the layer becomes 'empty' in some places. The active bed layer at a certain time in a certain place is defined as the first bed layer taken from the top, which is not empty. Erosion will always take place from the active layer. Depositing sediment will, on the other hand, always enter the uppermost bed layer.
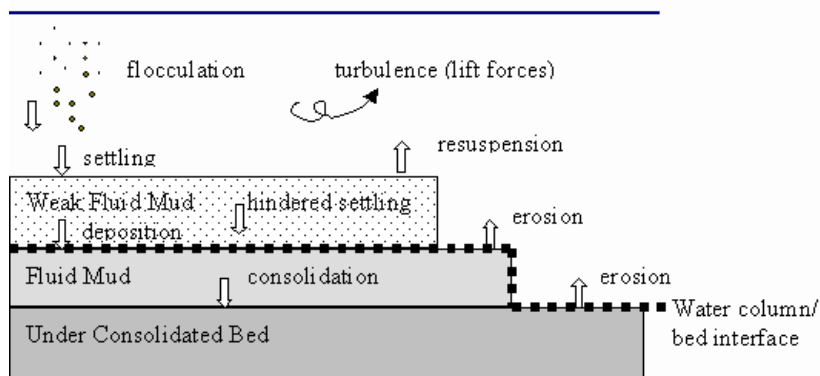

Figure 5.4   Processes included in the Mud Transport module. The bed layers are the layers below the water column-bed interface (dotted line)

The mass of the i'th sediment fraction in the j'th bed layer in a certain horizontal grid point is updated every time step following the expression:

$$m_{i,j}^{new} = m_{i,j}^{old} + (D_i - E_i)\Delta t + (T_{i,j-1} - T_{i,j}), \tag{5.15}$$

where m (kg/m²) is sediment mass, D (kg/m²/s) is a possible deposition (only in the uppermost bed layer), E (kg/m²/s) is a possible erosion (only from the active bed layer), T_i (kg/m²/s) is a possible downward transfer of sediment and Δt (s) is simulation time step.

The thickness of the i'th bed layer is a derived parameter determined as:

$$H_j^{new} = \frac{M_j}{\rho_{d,j}} = \frac{\sum_i m_{i,j}^{new}}{\rho_{d,j}}, \tag{5.16}$$

where H (m) is bed layer thickness, M (kg/m²) is total sediment mass and ρ_d (kg/m³) is dry density.

5.4.2 Erosion

The erosion settings for the bed layer must be specified for each separate layer.

General description

The erosion of a bed layer is the transfer of sediment from the bed to the water column. Erosion takes place from the active bed layer (see *Bed Description*) in areas where the bed shear stress (τ_b) is larger than the critical shear stress for erosion (τ_{ce}). The bed parameters are considered constant for each layer. Therefore erosion is calculated for the active layer. The eroded material is then distributed to the different fractions according to the distribution in the bed.

Data

Erosion description

Hard bed description
For a dense consolidated bed the erosion rate for the j´th layer is described as (Partheniades, 1989).

$$E^j = E_0^j p_E^{j^{E_m}} \tag{5.17}$$

where p_E^j is a probability ramp function of erosion, E_0 is the erosion coefficient and E_m is the power of erosion.

$$p_E^j = \max\left(0, \frac{\tau_b}{\tau_{ce}^j} - 1\right) \tag{5.18}$$

Soft bed description

For a soft, partly consolidated bed the erosion rate for the j'th layer is described as (Parchure & Mehta, 1985):

$$E = E_0^j \exp(\alpha(\tau_b - \tau_{ce}^j)) \tag{5.19}$$

where α is a coefficient.

Critical shear stress for erosion

The criteria for erosion is that the critical shear stress for erosion is exceeded corresponding to the driving forces exceeding the stabilising forces. The critical shear stress for erosion is constant throughout the simulation.

Data format

The erosion parameters must be specified as a **Constant** in time for each bed layer.

The critical shear stresses for erosion for each layer can be specified as:

- Constant (in domain)
- Varying in domain

For the case with varying in domain you have to prepare a data file containing the critical bed shear stresses before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data.

Recommended values

The value E is a proportion factor governing the speed of erosion. For soft bed it is generally between 0.000005 and 0.00002 kg/m²/s. For hard bed it is usually around 0.0001 kg/m²/s.

If the values E_m or α are set, the erosion rates will evolve exponentially. α usually lies between 4 and 26.

For critical shear stress the following typical values are given:

Table 5.3   Critical shear stresses

Mud type	Density [kg/m^3]	Typical τ_{ce} [N/m^2]
Mobile fluid mud	180	0.05-0.1
Partly consolidated mud	450	0.2-0.4
Hard mud	600+	0.6-2

The critical shear stress for erosion can be estimated from the residual yield shear stress in the following way:

$$\tau_{ce} = \sqrt{0.00001}\,\tau_{yj} \qquad \tau_{yj} \leq 0.00015\,N/m^2$$
$$\tau_{ce} = 0.0000025\,\tau_{yj} \qquad \tau_{yj} > 0.00015\,N/m^2$$

(5.20)

In which τ_{yj} is the residual yield shear stress for the layer j.

Remarks and hints

In general, only layers which recently have been relocated are considered soft layers. All other layers are normally considered hard layers.

5.4.3   Density of bed layers

The density of the bed layer must be specified for each layer. The density is specified for the whole layer containing all fractions.

General Description

Different sediment types have different densities depending on their previous geological history, chemical properties, organic content and several other factors.

If the area of interest has large extends, it may be necessary to obtain knowledge of sediment in different locations within the area in order to generate a representative density map of the area for each layer. This can be done from either measurements or through researching the geological history of the area. In the vertical direction the density varies with the degree of compression and with the type of mud. Different geological periods have left soils with different densities. For instance, areas that have been covered by glaciers can have very hard layers, and areas that has been sedimentation areas for a long time can be covered by relatively loose mud. Therefore it is necessary to assess the density and the strength at different depths of the sea-bed in order

to determine the vertical resolution of the bed and the bed densities for each layer.

The bed density is defined as dry density as follows:

$$\rho_d = \frac{\text{Mass of grains}}{\text{Volume of mixture}} \qquad (5.21)$$

Data

The bed density can be specified as:

- Constant (in domain)
- Varying in domain

The bed density is defined as the dry density.

For the case with varying in domain you have to prepare a data file containing the bed density before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data.

Recommended values

The following values can be used as guidelines for the range of the bed density.

Table 5.4   Typical bed densities

Sediment stage	General description	Rheological behaviour	Dry density (kg/m^3)
Freshly deposited (1 day)	Fluff	Mobile fluid mud	50-100
Weakly consolidated (1 week)	Mud	Fluid stationary mud	100-250
Medium consolidated (1 month)		Deforming cohesive bed	250-400
Highly consolidated (1 year)		Stationary cohesive bed	400-550
Stiff mud (10 years)	Stiff clay	Stationary cohesive bed	550-650

Remarks and hints

If only the bulk density (or wet density) is known, the density is determined as:

$$\rho_d = \frac{\rho_s(\rho_b - \rho)}{(\rho_s - \rho)}$$

(5.22)

in which ρ_b is bulk density, ρ_s is grain density and ρ is water density.

5.4.4 Bed roughness

The bed roughness is the resistance against the flow. It is included for calculating the bottom shear stress. The bed roughness depends on the shape of the bed (dunes, ripples, etc.) and the grain size. Despite the dynamic nature of the dunes and ripples, the bed roughness is constant in time since the local bed shape change is considered constant in time on average. The bed roughness is independent of the other bed parameters.

Data

The bed roughness can be specified as:

- Constant (in domain)
- Varying in domain

The bed roughness is defined as the Nikuradse roughness (k_n).

For the case with varying in domain you have to prepare a data file containing the bed roughness before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data.

Recommended values

k_n is generally defined as 2.5 times the diameter of the sediment. However, for fine sediment the bottom shape becomes the dominant factor and the recommended value is around 0.001m.

5.4.5 Transition between layers

Consolidation is included between the layers as a transition rate of sediment between the layers, Teisson (1992).

If transition of mud between layers is included you must specify a calibration parameter related to the transition rate.

Data

The transition can be specified as

- Constant (in domain)
- Varying in domain

The transition unit is e.g. [kg/m²/s].

5.5 Sediment Properties

When a sand fraction is included and the "Near-bed load and suspended load" is selected the calculation requires the sediment characteristics for the area.

The grain size is regarded as a representative grain size for a specific grid point in the modelling area. This means it can vary spatially. In river bends, coarser sediment is often found in deeper parts of the cross-section while finer sediments are found along the inner banks. On coasts with wave impact, coarser sediment is often found on the beach and on shallow water while finer sediment are found on deeper water.

5.5.1 Sediment Properties

The volume of the sand is described by the porosity of the material.

You must specify the mean grain diameter and the grading coefficient, σ_g.

Finally, the density of sand must be specified.

Data

The sediment data, e.g. the mean grain diameter and grading coefficient, can be specified as

- Constant (in domain)
- Varying in domain

For the case with values varying in domain you have to prepare a data file containing the mean grain diameter and grading coefficient before you set up the sand transport simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used piecewise constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data.

5.5.2 Recommended values

This module calculates the sediment transport rates based on theory that has been developed for sand sized material, e.g. grain diameters in the range from 0.06 mm to 2 mm.

Porosity of the bottom sediment is usually in the range of 0.3 to 0.7.

For sand, a typical value of the sediment density is 2650 kg/m^3.

5.6 Morphology

If the morphological changes within the area of interest are expected to be comparable to the water depth in certain areas, then it is necessary to take the morphological impact on the hydrodynamics into consideration. Typical areas where this is necessary are shallow areas where long term effects are being considered or dredging/dumping sites in shallow areas.

5.6.1

General description

The morphological development is included by updating the bathymetry for every time step with the net sedimentation. This ensures a stable evolution of the bed that will not destabilise the hydrodynamic simulation.

$$Z^{n+1} = Z^n + \Delta z^n \tag{5.23}$$

where:

Z^n	Bathymetry level at present timestep
Z^{n+1}	Bathymetry level at next timestep
Δz	Net sedimentation at present timestep
n	Timestep

The morphological update also offers to speed-up the morphological evolution in the following way.

$$Z^{n+1} = Z^n + \Delta z^n \cdot Speedup \tag{5.24}$$

In which *speedup* is a dimension less speed-up factor, defined as integer. The layer thicknesses are updated in the same way. Note that the amount of suspended sediment is not affected by this. Only the bed is affected.

5.6.2 Model definitions

You may choose to exclude the morphological development of the bed by de-activating the feedback on the hydrodynamic and wave calculation and calculations of the suspended sediment. Thus, the calculated values will be based on the initial bed level throughout the entire simulation.

You may speed-up the update of the bed by the Speedup factor. However, specifying a large speedup factor may destabilize the hydrodynamic solution by generation of internal waves during the update.

You can specify the speedup factor in three ways:

- Constant
- Constant in time, varying in domain
- Varying in time and domain

For the cases with values varying in domain you have to prepare a data file containing the speed-up factor before you setup the simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, a piecewise constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data.

In case the speedup factor is defined as constant in time, the first time step in the speedup factor file is applied. In case the speedup factor is defined as varying in time, the data must cover the complete simulation period. The time step of the input data file does not, however, has to be the same as the time step of the simulation. A linear interpolation will be applied if the time steps differ.

5.6.3 Boundary conditions

Boundary conditions must be defined for the near bed transport in addition to the Boundary Conditions given for the suspended sediment.

Initially, the set-up editor scans the mesh file for boundary codes (sections), displays the recognized codes and suggest a default name for each. You can re-name these names to more meaningful names in the Domain dialog (see Boundary names).

Depending on the choice of property page you can get a geographic view or a list view of the boundaries.

The specification of the individual boundary information for each code (section) and each component is made subsequently. From the list view you can go to the dialog for specification by clicking the "Go to.." button.

Boundary specification

You can choose between the following three boundary types:

- Land
- Zero sediment flux gradient
- Zero sediment flux gradient for outflow, zero bed change for inflow

5.6.4 Remarks and hints

The speed-up factor is only applied if the total bed thickness is larger than 0.01 mm.

Specifying a large speed-up factor may destabilise the hydrodynamic solution by generation of internal waves during the update.

5.7 Forcings

The only forcing that is included at this time is waves.

5.7.1 General description

Waves will contribute to increased shear stress in the bottom layer, leading to higher erosion rates and thus higher concentrations of suspended sediment. The shear stress will be calculated using a combined wave-current shear stress formulation.

5.7.2 Waves

You can define the waves in your calculations as:

- No waves
- Wave field

If waves are included you need to specify some additional parameters.

Liquefaction

Liquefaction is a weakening of the sediment due to pore-pressure flocculation caused by the waves.

To include this you must specify a liquefaction factor.

Generally, the inclusion of liquefaction will increase erosion.

Minimum water depth

You must also specify the minimum water depth for waves. This is the water depth below which the pure current solution for the bed shear stress will be used instead of the wave-current solution.

Bed shear stress

It is possible to select which formulation to use when calculating the bed shear stress for combined wave-current action.

The solution can be a parameterised version by Soulsby et. al. (1993), calculating the mean shear stress or the maximum shear stress, respectively.

It is also possible to choose an approach by Fredsøe (1981), where the bed shear stress is found by the maximum value of a pure current solution and a solution that considers a modified bed roughness due to the waves.

Wave field

You can include waves in your calculations as:

- Constant waves

- Time and space varying waves

- Wave database
  (This is only available if wind is included in the simulation)

You must select to specify the wave period as the peak wave period or the mean wave period. In the calculations the ratio between the peak wave period and the mean wave period is assumed to be 1.2106538.

You must specify the Bed shear stress formulation to be used along with the Minimum water depth for which a solution based on currents only is preferred.

For all wave calculations the effect of Liquefaction can be included.

Constant waves

If you choose constant waves, the waves will be sinusoidal with no directional spreading. You must specify the significant wave height, wave period T_z and the angle to true north.

Time and space varying waves

For the case with varying in domain you have to prepare a data file containing the wave properties (significant height, wave period T_z and angle to true north) before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data. The program will interpolate if the time step is not equal to that of the simulation, see Figure 5.5.
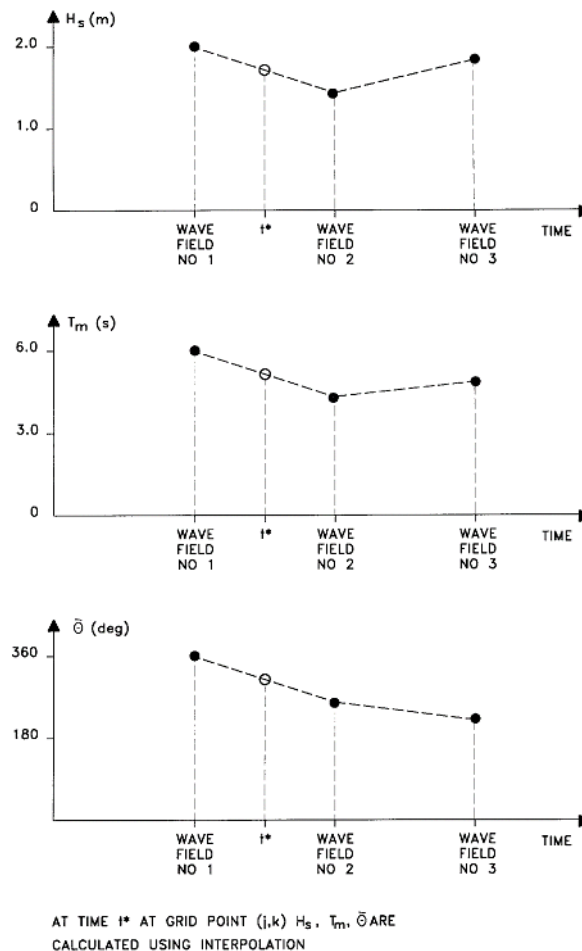
AT TIME t* AT GRID POINT (j,k) H_s, T_m, $\tilde{\Theta}$ ARE
CALCULATED USING INTERPOLATION

Figure 5.5   Interpolation of wave fields in time

Wave database

This option is available if wind is included in the simulation. The wind parameter is set in this HD dialog.

In an area dominated by wind-waves, and where the waves respond quickly to a change in water level, wind speed and direction, it is possible to use a wave database description. The wave database consists of discrete wave fields, which are simulated using combinations of water level, wind speed and direction. It is important to try to minimise the number of wave fields within the database. E.g. if the water level variations are small compared to the water depth, then the water level does not need to be discretized. Or if the wind is mainly blowing from the same direction (s), the wind direction discretization can be minimised.

Interpolation

For every grid point, the local values of water level, wind speed and direction are used to determine the wave parameters (significant wave height (H_s), zero-crossing wave period (T_{02}) and mean wave direction (θ) by interpolating the wave fields within the wave database, see the example in Figure 5.6.
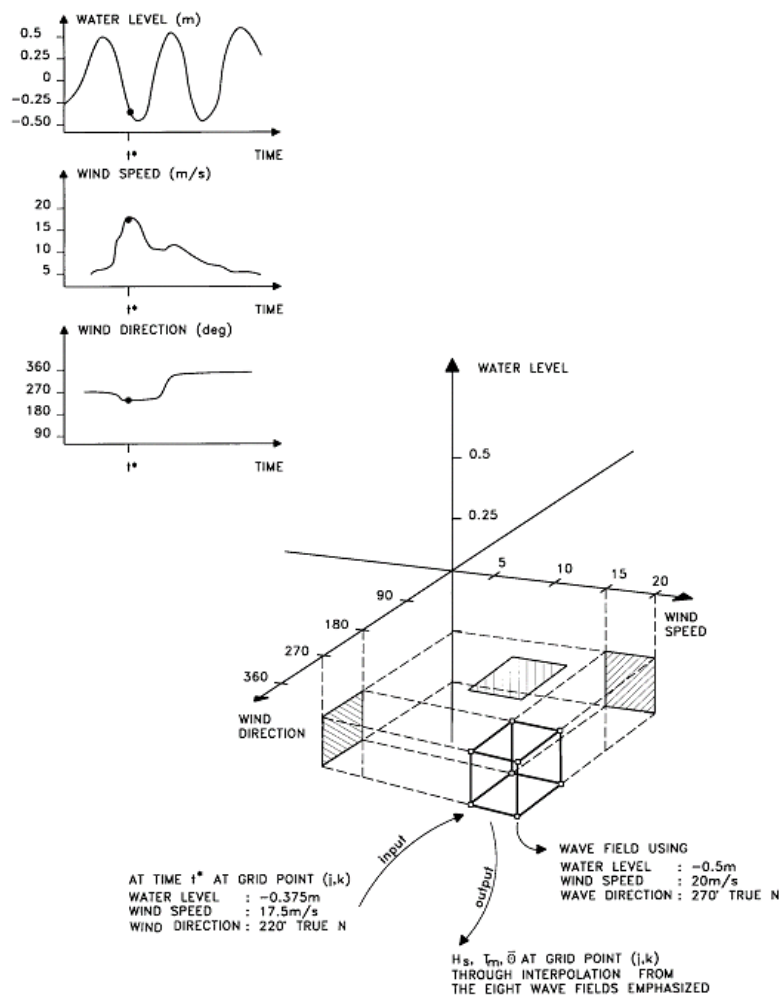

Figure 5.6   Interpolation of wave fields from wave database

5.8   Dredging

Dredging activities contribute to the suspended concentration through spilled material and it contributes to changing the morphology through the dredging itself.

In the main Dredging dialog you can add a new dredger by clicking on the "New dredger" button. By selecting a file in the dredger list and clicking on the "Delete dredger" you can remove this dredger. For each output file you can specify the name of the dredger and whether the dredger should be included or not. The specification of the individual dredgers is made subsequently. You can go to the dialog for specification by clicking on the "Go to ..." button.

If one or more dredgers are included in the simulation, you must specify if you want to update the bed due to the dredging.

5.8.1 Dredger

For each dredger a data file must be specified with the location of the dredger (the two coordinates in the geographical domain), the dredge rate (using dry weight) and the dredge spill (in %).

You must also specify the map projection (Longitude/Latitude, UTM, etc.) in which the horizontal location of the dredger location is given.

Positive dredge rates correspond to dredging. Negative dredge rates correspond to dumping.

Type of spilling

The spilling of material in the water column depends on the type of dredgers. Hence, for 3D simulation the type of spilling must be specified. It can be specified as:

- Spill in the bottom layer
- Spill in the top layer
- Spill uniformed in the bottom and the top layer
- Spill uniformed distributed in the water column

Update dredger

You must specify if you want up update the dredger due to dredging.

If update of the dredger is included, you must also specify the capacity of the dredger, the initial dredged mass on the dredger and the distribution of the initial dredged mass on the fractions included in the simulation. You must specify the distribution of the material in a given fraction in %. This means that the total value must be 100%.

Distribution

The definition of the distribution relates to the update of the dredger. You must specify the distribution of the material for a given fraction in %. This means that the total value must be 100%.

When the update of the dredger is included, the specified distribution is the inital distribution of material on the dredger. The actual distribution on the dredger, taken into account dredger material, is calculated during the simulation and this distribution is used for disposal of material and the related spill. The distribution for dredged material, the related spill and the material placed on the dredger, when update of the dredger is included, is the distribution in bed.

When update of the dredger is excluded, the specified distribution is used for the disposal of material (i.e. negative source) and the related spill.

Data

You have to prepare a data file containing the location of the dredger (the two coordinates in the geographical domain), the dredge rate and the spill (%) before you set up the mud transport simulation. The data file must be a time series data file (dfs0). The data must cover the complete simulation period. The time step of the input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

5.8.2 Remarks and hints

In the log file dredging statistics are written for the simulation.

Note that the actual dredge rate will be limited by the available amount of sediment.

5.9 Disposal

A nearfield/farfield coupled approach is used for modelling sediment disposal from a split barge, hereafter referred to as "Vessel". The nearfield solution is based on an integrated approach independent from the MIKE computational mesh.

In the main Disposal dialog, you can add a new vessel by clicking on the "New vessel" button. By selecting a vessel in the vessel list and clicking on the "Delete vessel" button you can remove this vessel. For each vessel you can specify the name of the vessel, whether the vessel should be included or not and if special disposal information should be written to an output file. On the property page output, it is possible to specify the name of the output file for each vessel. The specification of the individual vessels is made subsequently. You can go to the dialog for specification by clicking on the "Go to ..." button or by selecting a vessel in the vessel list and clicking on the "Edit vessel" button.

You must also specify if stripping should be included or excluded. For more detail on stripping see General description and Scientific documentation. If

stripping is included, you must specify the stripping coefficient for both the convective descent plume and the collapsing surge. The default value of 0.003 is based on typical stripping rates.

5.9.1 Vessel

For each vessel the user must specify the specification for the vessel

- Overall area
- Inner area
- Opening area
- Unloaded draft

This information is used to estimate the emptying process (duration and the insertion momentum) more accurately. In the below figure the different definitions for the vessel area are depicted.
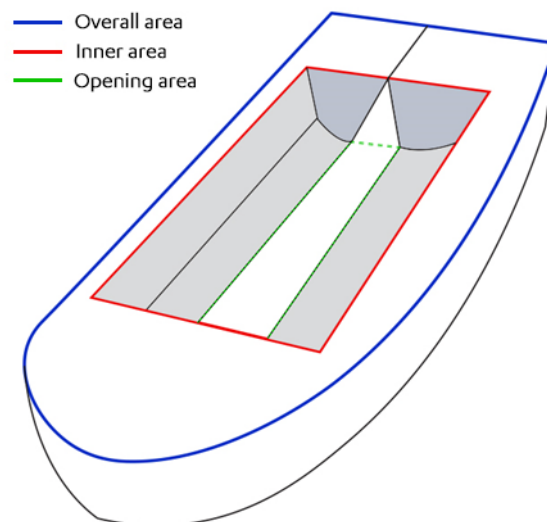

Figure 5.7   Definition of Vessel areal parameters

You must also specify the distribution of the material on the vessel. For each fraction the weights in % must be specified. This means that the total value must be 100%. The distribution can be specified either from a table or from an input an input data file.

For each vessel a data file must be specified. The data file must be a time-series data file (dfs0) containing all the disposal instances of the vessel. It can be either equidistant or non-equidistant calendar axis. The data file must contain the following items:

1. x-coordinate for the disposal site
2. y-coordinate for the disposal site
3. Total load volume
4. Solid content (sediment volumetric concentration)
5. Temperature of the water inside the vessel
6. Salinity of the water inside the vessel

If the distribution is specified from a data file, the data file must also contain the distribution of the material on the vessel for each fraction in % (percentage). It is not necessary for the data file to cover the entire simulation time period. It only requires the disposal instances. The load instances for a vessel should have a minimum time distance of an AD (or Decoupled) time step, otherwise they will be treated as one load, where the volume is summed up and the remaining input parameters for the last load instances is used. You must also specify the map projection (Longitude/Latitude, UTM, etc.) in which the horizontal location of the disposal site (vessel location) is given.

5.9.2   Output

If output is included for a vessel you must specify the name of the output file. The file will contain information of the calculated disposal plume.

The output file contains the following 9 items for the plume:

1. x end position - x-position when the plume becomes passive
2. y end position - y-position when the plume becomes passive
3. z end position - z-position when the plume becomes passive
4. Plume diameter (x-direction)
5. Plume diameter (y-direction)
6. Plume sediment concentration
7. Total mass (suspended) for each fraction
8. Total mass (deposition) for each fraction
9. Stop code - the stop code is an integer corresponding to the stop criteria described in General description.

The suspended mass is the sum of mass for the stripped sediments and the sediment inserted in the water column when the plume become passive. The deposition mass is the sum of the mass for fall-out sediment during the descent phase and the deposition sediment during the collapse phase. For more detail see the General description and Scientific documentation. Note, that the output file will be empty if there is no disposal instance within the simulation period and it will not be possible to open the file.

5.9.3 General description

The behaviour of the disposal material can be separated into three phases: Convective descent, during which the disposal cloud or discharge falls under the influence of gravity; Bottom collapse, occurring when the descending cloud hits the bottom, and Passive Dispersion. The first two are so-called nearfield processes, where the dynamics and energy of the plume controls its behaviour. The last phase is the so-called farfield where the material transport and spreading are determined more by ambient currents and turbulence.

The nearfield phases (convective descent and bottom collapse surge) are calculated by a sub-model using an integrated solution (detailed description in the scientific documentation). Stripped sediments during both convective decent and the collapsed plume surge are inserted into the water column (see Figure 5.8). If there is significant size difference between the sediment fractions in the disposal plume (such as clumps), there is possibility for heavier fractions to fall out of the plume as soon as their individual fall velocity exceeds the plumes downward descend velocity. The fall-out sediments are placed into the first bed layer, with a deposition pattern corresponding to the plume size at that moment (see Figure 5.9). While the collapsing surge is spreading over the seabed, there will be continuous deposition of sediments taking into account the surge velocity and the critical shear stress for deposition of the corresponding fraction. Again sediments are placed into the first bed layer. As the plume enters the passive phase, either in mid water-column or after collapsing over the seabed, the passive plume is inserted into the water column corresponding to the latest location, size, and concentration of the plume. In Figure 5.8 a simple description of the nearfield calculations is shown.
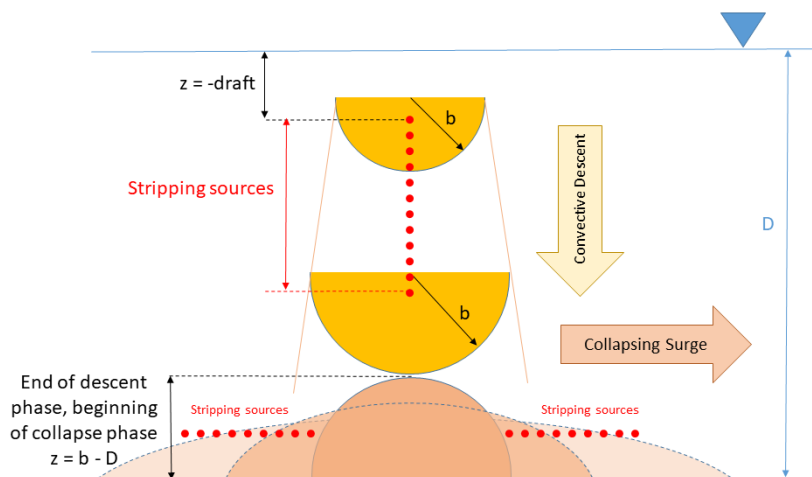

Figure 5.8   Schematic description of the nearfield model definitions

Considerably larger horizontal dimensions of the collapsing surge mean that different parts of the collapsing surge may experience different ambient conditions such as bottom slope, or interactions with land/boundaries. Therefore, the calculation of collapsing surge is done into four (4) quarters and each quarter may behave differently. In Figure 5.9 the four quarters are shown.
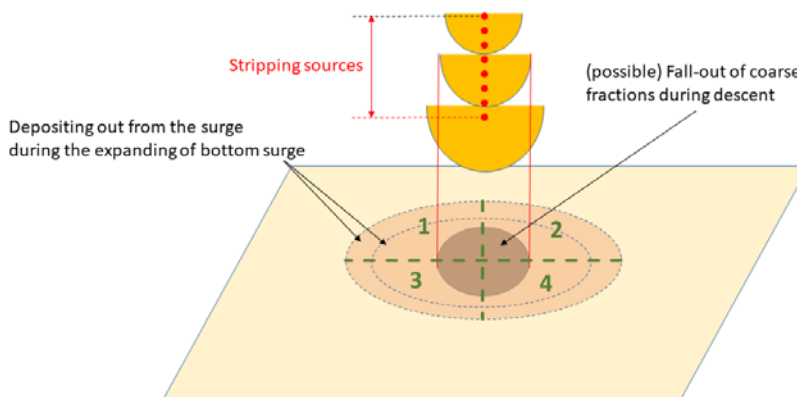

Figure 5.9   Deposition into the seabed may occur due to fall out during convective descent or during the surge spreading

The plume can become passive under different conditions. In the output file, the stop code indicates under which conditions, and in which phase the plume has become passive and released into the water column.

In the descent phase the stop code is a one-digit number and are defined as follows:

1. Disposal volume is zero
2. Initial position of the disposal is outside the domain
3. Initial position of the disposal is on land
4. Plume density becomes equal or smaller than ambient density
5. Plume momentum become smaller than ambient momentum

In the collapse phase the stop code is a four-digit number, where each digit represents the stop code for one of the four quarters. The order of the digits in the stop code follows the same order shown in Figure 5.9. The definition of each digit is defined as follows:

1. Collapsing surge becomes passive due to high mixing and entrainment
2. All sediment volume in the quarter either stripped and/or deposited
3. Collapsing surge quarter momentum becomes smaller than ambient momentum

4. Collapsing surge quarter concentration is less than or equal 80 mg/l

5. Collapsing surge quarter velocity has less than 10% difference with ambient velocity

6. Collapsing surge quarter center of mass outside domain

7. Collapsing surge quarter center of mass on land

5.10 Dispersion

In numerical models the dispersion usually describes transport due to non-resolved processes. In coastal areas it can be transport due to non-resolved turbulence or eddies. Especially in the horizontal directions the effects of non-resolved processes can be significant, in which case the dispersion coefficient formally should depend on the resolution.

In a 3D model it is important to distinguish between horizontal dispersion due to e.g. non-resolved eddies, and vertical dispersion due to e.g. bed generated turbulence. Hence, dispersion in horizontal and vertical directions is specified separately.

The dispersion is specified individually for each fraction.

5.10.1 Dispersion specification

The horizontal dispersion can be formulated in three different ways:

- No dispersion
- Dispersion coefficient formulation
- Scaled eddy viscosity formulation

Selecting the dispersion coefficient formulation you must specify the dispersion coefficient.

Using the scaled eddy viscosity formulation the dispersion coefficient is calculated as the eddy viscosity used in solution of the flow equations multiplied by a scaling factor. For specification of the eddy viscosity see Eddy Viscosity in the manual for the Hydrodynamic module.

Dispersion coefficient
When selecting the Dispersion coefficient option the format of the dispersion coefficient can be specified as:

- Constant (in both time and domain)
- Varying in domain

For the case with dispersion coefficient varying in domain the values are constant in the vertical domain and only varying in the horizontal domain. You

have to prepare a data file containing the dispersion coefficient, before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data.

Scaled eddy viscosity

When selecting Scaled eddy viscosity option the format of the scaling factor can be specified as:

- Constant
- Varying in domain

For the case with values varying in domain you have to prepare a data file containing the scaling factor, before you set up the hydrodynamic simulation. The file must be a 3D unstructured data file (dfsu) or a 3D data grid file (dfs3). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs3-file is used, bilinear interpolation is used to map the data.

5.10.2 Recommended values

When more sophisticated eddy viscosity models are used, such as the Smagorinsky or k-ε models, the scaled eddy formulation should be used.

The scaling factor can be estimated by $1/\sigma_T$, where σ_T is the Prandtl number. To be consistent with the empirical constants for the k-ε turbulence model the value of σ_T should be the same as the value for the Prandtl number specified on the Equation dialog in the TURBULENCE MODULE. The default value here for the Prandtl number is 0.9 corresponding to a scaling factor of 1.1.

The dispersion coefficient is usually one of the key calibration parameters for the Mud Transport Module. It is therefore difficult to device generally applicable values for the dispersion coefficient. However, using Reynolds analogy, the dispersion coefficient can be written as the product of a length scale and a velocity scale. In shallow waters the length scale can often be taken as the water depth, while the velocity scale can be given as a typical current speed.

Values in the order of 1 are usually recommended for the scaling factor for the horizontal plane. For more information, see Rodi (1980).

The scaling factor for the vertical plane is usually much smaller, e.g. 0.00001. As it is a calibration factor this can sometimes even be 0.

5.11 Sources

Point sources of dissolved and/or suspended components are important in many applications as e.g. release of sediments from rivers, intakes and outlets from cooling water or desalination plants.

In the Mud Transport Module the source concentrations of each component in every source point can be specified. The number of sources, their generic names, location and discharge magnitude are specified in the Sources dialog in the HYDRODYNAMIC MODULE.

Depending on the choice of property page you can see a geographic view or a list view of the sources.

The source concentrations are specified individually for each source and each component. From the list view you can go to the dialog for specification by clicking on the "Go to.." button.

5.11.1 Source specification

The type of sources can be specified in two ways:

- Specified concentration
- Excess concentration

The source flux is calculated as $Q_{source} \times C_{source}$, where Q_{source} is the magnitude of the source and C_{source} is the component concentration of the source. The magnitude of the source is specified in the Sources dialog in the HYDRODYNAMIC MODULE.

Selecting the specified concentration option, the source concentration is the specified concentration if the magnitude of the source is positive (water is discharged into the ambient water). The source concentration is the concentration at the source point if the magnitude of the source is negative (water is discharged out of the ambient water). This option is pertinent to e.g. river outlets or other sources where the concentration is independent of the surrounding water.

Selecting the excess concentration option, the source concentration is the sum of the specified excess concentration and concentration at a point in the model if the magnitude of the source is positive (water is discharged into the ambient water). If it is an isolated source, the point is the location of the source. If it is a connected source, the point is the location where water is discharged out of the water. The source concentration is the concentration at the source point if the magnitude of the source is negative (water is discharged out of the ambient water).

Data

The format of the source information can be specified as:

- Constant (in time)
- Varying in time

For the case with source concentration varying in time you have to prepare a data file containing the concentration of the source before you set up the hydrodynamic simulation. The data file must be a time series file (dfs0). The data must cover the complete simulation period. The time step of the input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

5.11.2 Remarks and hints

Point sources are entered into elements, such that the inflowing mass of the component initially is distributed over the element where the source resides. Therefore the concentration seen in the results from the simulation is usually lower than the source concentration.

5.12 Initial Conditions

The initial conditions are the spatial distribution of the component concentration throughout the computational domain at the beginning of the simulation. Initial conditions must always be provided. The initial conditions can be the result from a previous simulation in which case the initial conditions effectively act as a hot start of the concentration field for each component.

The initial conditions are specified individually for each component.

5.12.1 Fraction concentration

Here you must specify the initial concentration for each fraction.

Data

The format of the initial concentration (in component unit) for each fraction can be specified as:

- Constant (in domain)
- Varying in domain

A typical background concentration is 0.01 kg/m^3.

For the case with varying in domain you have to prepare a data file containing the concentration (in component unit) before you set up the simulation. The file must be a 3D unstructured data file (dfsu) or a 3D grid data file (dfs3). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs3-file is used, bilinear interpolation is used to map the data. In case the input data file contains a single time step, this field is used. In case the file contains several time steps, e.g. from the results of a previous simulation, the actual starting time of the simulation is used to interpolate the field in time. Therefore the starting time must be between the start and end time of the file.

5.12.2 Layer thickness

Here you must specify the initial bed thickness for each layer.

Data

The initial bed thickness for each layer can be specified as:

- Constant (in domain)
- Varying in domain

For the case with varying in domain you have to prepare a data file containing the bed thickness before you set up the simulation.

The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2).

The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2 file is used, bilinear interpolation is used to map the data. In case the input data file contains a single time step, this field is used. In case the file contains several time steps, e.g. from the results of a previous simulation, the actual starting time of the simulation is used to interpolate the field in time. Therefore the starting time must be between the start and end time of the file.

5.12.3 Fraction distribution

Here you must specify the content of a given fraction in a given layer in %. This means that for a layer the total value in each point must be 100%. The distribution is given as a constant for each area.

5.13 Boundary Conditions

Initially, the set-up editor scans the mesh file for boundary codes (sections), displays the recognized codes and suggest a default name for each.You can re-name these names to more meaningful names in the Domain dialog (see Boundary names *(p. 30))*.

Depending on the choice of property page you can get a geographic view or a list view of the boundaries.

The specification of the individual boundary information for each code (section) and each component is made subsequently. From the list view you can go to the dialog for specification by clicking the "Go to.." button.

5.13.1 Boundary specification

You can choose between the following three boundary types:

- Land
- Specified values (Dirichlet boundary condition)
- Zero gradient (Neumann boundary condition)

Data

If specified values (Dirichlet boundary condition) is selected the format of the fraction concentration at the boundary can specified as:

- Constant (in time and along boundary)
- Varying in time and constant along boundary
- Varying in time and along boundary

For the case with boundary data varying in time but constant along the boundary you have to prepare a data file containing the fraction concentration before you set up the hydrodynamic simulation. The data file must be a time series file (dfs0). The data must cover the complete simulation period. The time step of the input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. You can choose between different types of interpolation (see Interpolation type).

For the case with boundary data varying both in time and along the boundary you have to prepare a data file containing the fraction concentration before you set up the hydrodynamic simulation. The data file must be a dfs2 file or a 2D dfsu file containing information from a vertical plane. The data must cover the complete simulation period. The time step of the input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. You can choose between different types of time interpolation (see Interpolation type).

Interpolation type

For the two cases with values varying in time two types of time interpolation can be selected:

- Linear

- Piece wise cubic

In the case with values varying along the boundary two methods of mapping from the input data file to the boundary section are available:

- Normal
- Reverse order.

Using normal interpolation the first and last point of the line are mapped to the first and the last node along the boundary section and the intermediate boundary values are found by linear interpolation. Using reverse order interpolation the last and first point of the line are mapped to the first and the last node along the boundary section and the intermediate boundary values is found by linear interpolation.

Soft start interval

You can specify a soft start interval during which boundary values are increased from a specified reference value to the specified boundary value in order to avoid shock waves being generated in the model. The increase can either be linear or follow a sinusoidal curve.

5.14 Outputs

Standard data files with computed results from the simulation can be specified here. Because result files tend to become large, it is normally not possible to save the computed discrete data in the whole area and at all time steps. In practice, sub areas and subsets must be selected.

In the main Outputs dialog you can add a new output file by clicking on the "New output" button. By selecting a file in the Output list and clicking on the "Delete output" you can remove this file. For each output file you can specify the name (title) of the file and whether the output file should be included or not. The specification of the individual output files is made subsequently. You can go to the dialog for specification by clicking on the "Go to .." button. Finally, you can view the results using the relevant MIKE Zero viewing/editing tool by clicking on the "View" button during and after the simulation.

5.14.1 Geographic view

This shows the geographical position of the output data.

5.14.2 Output specification

For each selected output file the field type, the output format, the data type, the treatment of flood and dry, the output file (name, location and file type)

and time step must be specified. Depending on the output format and data type the geographical extend of the output data must also be specified.

Field type

For a 3D simulation both 2D and 3D field parameters can be selected. The mass budget for a domain and the discharge through a cross section can also be selected.

Output format

The possible choice of output format depends on the specified field type.

For 2D field variables the following formats can be selected:

- Point series. Selected field data in geographical defined points.
- Lines series. Selected field data along geographical defined lines.
- Area series. Selected field data in geographical defined areas.

For 3D field variables the following formats can be selected:

- Point series. Selected field data in geographical defined points.
- Lines series. Selected field data along geographical defined lines.
- Vertical plane series. Selected field data along geographical defined vertical plane.
- Volume series. Selected field data in geographical defined areas.

If mass budget is selected for the field type, then you have to specify the domain for which the mass budget should be calculated. The file type will be a dfs0 file.

If discharge is selected for the field type, then you have to specify the cross section through which the discharge should be calculated. The file type will be a dfs0 file.

Data type

You must specify the data type. You can select discrete values or interpolated values. For point series, area series and volume series both options are available. For line series only interpolated values is available. See also Table 5.5.

If "discrete values" is selected for the data type, the values written in the data file are the cell-averaged values.

If "interpolated values" is selected for the data type, the values written in the data file are determined by 2nd order interpolation. The element in which the point is located is determined and the point value is obtained by linear interpolation using the vertex (node) values for the actual element. The vertex val-

ues are calculated from on the cell-averaged values using the pseudo-Laplacian procedure proposed by Holmes and Connell (1989).

Note, that all adjacent elements, including dry elements, are considered in the interpolation calculation.

Output file

A name and location of the output file must be specified along with the file type. The file type depends on the Output format and the Data type as shown in Table 5.5.

Vectors components and tensors components in a 2D dfsu file and horizontal vector components in a 3D dfsu file are given in the projection coordinate system. In a dfs2 file and a dfs3 file the vector and tensor components are given in the local coordinate system.

Table 5.5   List of file types for the output files

Output format	Data type	File type
Point series	Discrete values	dfs0
	Interpolated values	dfs0
Line series	Interpolated values	dfs1
Area series	Discrete values	dfsu, dfs2*
	Interpolated values	dfs2
Vertical plane series	Discrete values	dfsu
	Interpolated values	dfs2
Volume series	Discrete values	dfsu
	Interpolated values	dfs3
Cross-section	Not relevant	dfs0
Domain series	Not relevant	dfs0

* For area series with discrete values selected for the data type the file type dfs2 can only be specified when the Mesh and bathymetry is specified using a bathymetry data file.

Table 5.6   List of tools for viewing, editing and plotting results

File type	Viewing/editing tools	Plotting tools
dfs0	Time Series Editor	Plot Composer
dfs1	Profile Series Editor	Plot Composer Result Viewer
dfs2	Grid Series Editor Data Viewer	Plot Composer Result Viewer Data Viewer
dfs3	Grid Series Editor	Plot Composer Result Viewer
dfsu	Data Viewer Data Manager	Data Viewer Result Viewer Plot Composer (2D)

Flood and dry

For 2D and 3D field parameters the flood and dry can be treated in three different ways:

- Whole area
- Only wet area
- Only real wet area

When selecting the Only wet area option, the output file will contain delete values for land points. The land points are defined as the points where the water depth is less than a drying depth. When selecting the Only real wet area option, the output file will contain delete values for points where the water depth is less than the wetting depth. The drying depth and the wetting depth are specified on the Flood and Dry dialog. If flooding and drying is not included, both the flooding depth and the wetting depth are set to zero.

Time step

The temporal range refers to the time steps specified under Simulation Period in the Time dialog.

If checking the "Use simulation end time" check box, then the simulation end time specified in the Time dialog will automatically be used as last output time step. Hence, the last output time step will be automatically updated if the simulation period is changed.

Point series

You must select the map projection (Long/Lat, UTM-32, etc.) in which you want to specify the horizontal location of the points.

The geographical coordinates of the points are either taken from the dialog or from a file. The file format is an ascii file with four space separated items for each point on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. For 3D field data the third item must be an integer for the Layer number if discrete values are selected and a float (real number) for the z-coordinate if interpolated values are selected. The layers are numbered 1 at the bed and increasing upwards. For 2D field data the third item is unused (but must be specified). The last item (the remaining of the line) is the name specification for each point

Layer number

The layer number selected for discrete values in the point output is defined from the lowest active layer (=1) increasing upwards. In case the mesh is a type sigma mesh the number of active layers in the water column will always be the same in any point in the domain. In case the mesh is a combined sigma-z level mesh the number of active layers may vary in the domain. An example is shown in Figure 5.10.
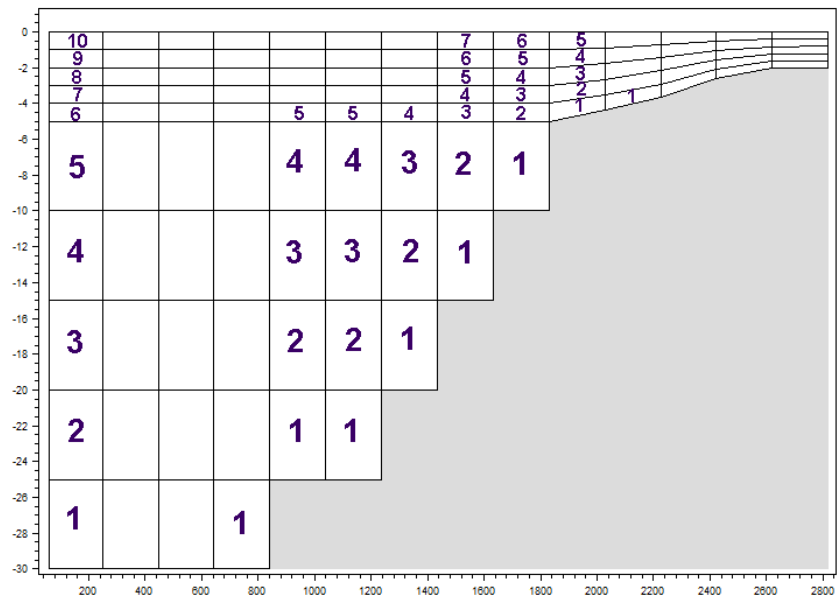

Figure 5.10   Example of layer numbers in point output specification in case of combined sigma-z level mesh.

Line series

You must select the map projection (Long/Lat, UTM-32, etc.) in which you want to specify the horizontal spatial information.

The line is defined by defining a polyline in the horizontal domain. The polyline is given by a number of geo-referenced points. A minimum of two points is required. The polyline is composed of a sequence of line segments. The line segments are straight lines between two successive points. The geographical coordinates are taken from the dialog or from a file. The file format is an ascii file with three space separated items for each of the points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate (Easting and Northing or Longitude and Latitude). For 3D field data the third item must be a float (real number) for the z-coordinate. For 2D field data the third item is unused (but must be specified).

Note: If spherical coordinates (map projection LONG/LAT) is used for a 3D model simulation, the line must be either a horizontal or a vertical line.

Area series

You must select the map projection (Long/Lat, UTM-32, etc.) in which you want to specify the horizontal spatial information.

If discrete values is selected for the data type the discrete field data within a polygon can be selected. The closed region is bounded by a number of line segments. You must specify the coordinates of the vertex points of the polygon. Two successive points are the endpoints of a line that is a side of the polygon. The first and final point is joined by a line segment that closes the polygon. The geographical coordinates of the polygon points are taken from the dialog or from a file. The file format is an ascii file with three space separated items for each of the two points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. The third item is unused (but must be specified).

If interpolated values is selected for the data type the values is calculated on a structured grid (overlay). For each direction you should specify

- The origin
- The grid spacing
- The number of grid points

Furthermore, the orientation of the grid at the origin must be specified. This is defined as the angle between y-axis in the selected projection and the y-axis of the grid measured clockwise.

Vertical plane series

You must select the map projection (Long/Lat, UTM-32, etc.) in which you want to specify the horizontal spatial information.

The vertical plane is defined by defining a polyline in the horizontal domain. The polyline is given by a number of geo-referenced points. A minimum of two points is required. The polyline is composed of a sequence of line segments. The line segments are straight lines between two successive points. The geographical coordinates are taken from the dialog or from a file. The file format is an ascii file with three space separated items for each of the points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate (Easting and Northing or Longitude and Latitude). The third item is not used (but must be specified).

If discrete values is selected for the data type you must also specify the range of layers (first and last Layer number) which should be stored in the output file The intersection between the line segments and the faces of the unstructured mesh is determined and added to the list of points on the polyline. A vertical unstructured mesh is then created from the horizontal polyline points and the vertical discretization for the computational mesh. For each element of the of vertical mesh the discrete value at the centroid is determined as the discrete value in the element of the computational mesh, where the centroid point is located.

If interpolated values is selected for the data type the values is calculated on a structured grid (overlay). You should specify the number of grid points along the polyline. For the vertical direction the minimum and maximum z value and the number of grid points should be specified.

In the dfs2 output file Dimension 1 is the coordinate along the polyline and Dimension 2 is the vertical coordinate (z-direction). The origin in the z-direction in the dfs2 file is determined as the specified maximum z value.

Layer number

The layer number(s) selected for the vertical plane output refer to the vertical discretization specified on the Vertical Mesh page. Here the layers are numbered 1 at the lowest layer and increase upwards. In case of a combined sigma-z level mesh only the active elements are saved in the output.

Volume series

You must select the map projection (Long/Lat, UTM-32, etc.) in which you want to specify the horizontal spatial information.

If discrete values is selected for the data type the discrete field data within a polygon can be selected. The closed region is bounded by a number of line segments. You must specify the coordinates of the vertex points of the polygon. Two successive points are the endpoints of a line that is a side of the polygon. The first and final point is joined by a line segment that closes the

polygon. The geographical coordinates of the polygon points are taken from the dialog or from a file. The file format is an ascii file with three space separated items for each of the two points on separate lines. The three items must be floats (real numbers) for the x-, y- and z-coordinate. You must also specify the range of layers (first and last Layer number) which should be stored in the output file.

If interpolated values is selected for the data type the values is calculated on a structured grid (overlay). For each direction you should specify

- The origin
- The grid spacing
- The number of grid points

Furthermore, the orientation of the grid at the origin must be specified. This is defined as the angle between y-axis in the selected projection and the y-axis of the grid measured clockwise.

Note, the origin, $z0_{dfs3}$, in the z-direction in the dfs3 file is determined as $z0_{dfs3} = z^0 + dz*(nz-1)$, where z^0 is the user specified origin, dz is the grid spacing and nz in the number of grid points.

Layer number

The layer number(s) selected for the volume output refer to the vertical discretization specified on the Vertical mesh page. Here the layers are numbered 1 at the lowest layer and increase upwards. In case of a combined sigma-z level mesh only the elements containing water are saved in the output. An example is shown in Figure 5.11.
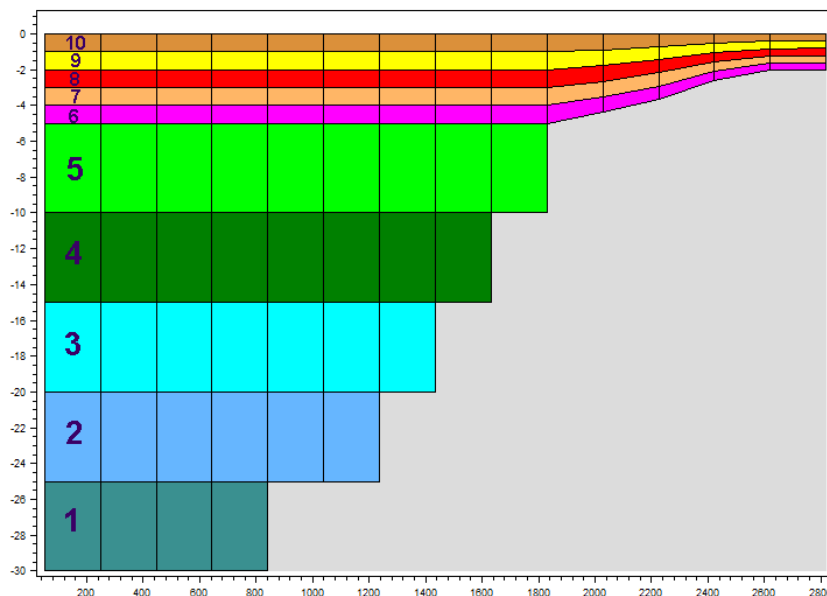
Figure 5.11   Example of layer numbers in volume output specification in case of combined sigma-z level mesh.

Cross section series

The cross section is defined by defining a polyline in the horizontal domain. The polyline is given by a number of geo-referenced points which together make up a polyline. A minimum of two points is required. The polyline is composed of a sequence of line segments. The line segments are straight lines between two successive points. The geographical coordinates are taken from the dialog or from a file. The file format is an ascii file with three space separated items for each of the two points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. The third item is unused (but must be specified). The polyline (cross section) in the numerical calculations is defined as a section of element faces. The face is included in the section when the line between the two element centres of the faces crosses one of the line segments. The faces defining the cross section are listed in the log-file.

You must also select the map projection (Long/Lat, UTM-32, etc.) in which you want to specify the horizontal location of the points.

Domain series

The domain for which mass budget should be calculated is specified as a polygon in the horizontal domain. The closed region is bounded by a number of line segments. You must specify the coordinates of the vertex points of the polygon. Two successive points are the endpoints of a line that is a side of the polygon. The first and final point is joined by a line segment that closes the

polygon. The geographical coordinates of the polygon points are taken from the dialog or from a file. The file format is an ascii file with three space separated items for each of the two points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. The third item is unused (but must be specified).

You must also select the map projection (Long/Lat, UTM-32, etc.) in which you want to specify the horizontal location of the points.

5.14.3 Output items

2D field variables

You can select basic output variables and additional output variables.

The basic variables are

- Bed thickness for each layer
- Bed mass per area for each layer and each fraction
- Near-bed load, x-component
- Near-bed load, y-component

The additional variables are

- Bed shear stress
- Near-bed load, magnitude
- Near-bed load, direction
- Acc. near-bed load, x-component
- Acc. near-bed load, y-component
- Total bed thickness change
- Total bed mass change
- Bed level
- Erosion for each fraction
- Deposition for each fraction
- Net deposition for each fraction
- Net deposition accumulated for each fraction
- Total net deposition accumulated
- Wave height
- Wave period
- Wave direction [deg true North]
- Mean velocity components

The wave period is here the mean wave period. Wave directions are defined positive clockwise from true North (coming from).

The type of bed shear stress (Mean/Max) depends on the choice of Bed shear stress formulation.

Transport loads are output in solid masses and do not include speed-up factor or porosity.

The mean velocity components are the mean velocity components estimated based on the bed shear stresses. This mean velocity is used in the near-bed sediment transport calculation for the sand fraction.

3D field variables

You can select basic output variables and additional output variables.

The basic variables are

- Suspended sediment concentration (SSC) for each fraction

The additional variables are

- Settling velocity for each fraction
- Total Suspended sediment concentration (SSC)
- Velocity components

Mass budget

You can select the mass budget calculation to be included for the flow and for the suspended sediment fractions. For each selected component the following items are included in the output file

- Total area - total volume/mass within polygon

- Wet area - volume/mass in the area within polygon for which the water depth is larger than the drying depth.

- Real wet area - volume/mass in the area within polygon for which the water depth is larger than the wetting depth

- Dry area - volume/mass in the area within polygon for which the water depth is less than the drying depth

- Transport - accumulated volume/mass transported over lateral limits of polygon

- Source - accumulated volume/mass added/removed by sources within polygon

- Process - accumulated volume/mass added/removed by processes within polygon

- Error - accumulated volume/mass error within polygon determined as the difference between the total mass change and the accumulated mass due to transport, sources and processes

The accumulated volume/mass error contains the contribution due to correction of the transported component when the values become larger than the specified maximum value or lower than the specified minimum value. For the water volume the minimum value is 0, while there is no upper limit. For the suspended sediment concentrations the minimum values are 0 while the maximum values are 1010.

Discharge

You can select the discharge calculation to be included for the flow and for the suspended sediment fractions. Each selected component will result in a number of output items.

You can select between two types of output items:

- Basic
- Extended

The basic output items are as follows:

- Discharge - volume/mass flux through the cross section
- Acc. discharge - accumulated volume/mass flux through the cross section

The extended output items that are included in the output file in addition to the basic output items are as follows:

- Positive discharge
- Accumulated positive discharge
- Negative discharge
- Accumulated negative discharge

By definition, discharge is positive for flow towards left when positioned at the first point and looking forward along the cross-section line. The transports are always integrated over the entire water depth.

Note: Discharge output is not possible using the near-bed load and suspended load sand formulation in combination with speedup of the morphology.

INDEX