MIKE 3 Flow Model FM

Sand 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, Sand Transport Module, for applications involving the modelling of non-cohesive sediment transport.

1.2 Assumed User Background

Although the sand 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 non-cohesive sediment transport.

It is assumed that you are familiar with the basic elements of MIKE 3: 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 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 item's 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 it 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 Sand Transport module of MIKE 3 Flow Model FM describes erosion, transport and deposition of sand under the action of currents and waves or pure current.

The sand transport calculations in a 3D model setup are carried out using a mean horizontal velocity component. The sand transport calculations are thus not truely 3-dimensional, but based on a representative current velocity. By using a more detailed 3D hydrodynamic model this current velocity can be derived from either the depth-integrated velocity or from the hydrodynamic conditions near the bed.

2.1.1 Application areas

The Sand Transport Module can be applied to the study of engineering problems such as sediment transport studies for non-cohesive sediment.

3 GETTING STARTED

The hydrodynamic basis for the Sand 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 FM, Hydrodynamic Module, User Guide

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

Two comprehensive training guides covering the Sand Transport Module of the MIKE 21 Flow Model FM modelling system are also available. The first guide is for an application using the pure current formulation and the second for an application using the wave-current formulation. The objective of these training guides is to set up a sand transport model from scratch and to calibrate the model to a satisfactory level. The training guides (PDF-format) can be accessed from the MIKE 21 Documentation Index:

MIKE 21 Flow Model FM, Sand Transport Module, Step-by-Step Training Guide: River Application

MIKE 21 Flow Model FM, Sand Transport Module, Step-by-Step Training Guide: Coastal Application

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:

- U_flume example:
  .\Examples\MIKE_3\FlowModel_FM\ST\U_flume

- Circulation flow example:
  .\Examples\MIKE_3\FlowModel_FM\ST\Basin

4.2 U-Shaped Flume

4.2.1 Purpose of the example

This simplified example has been chosen to describe the influence of using a 3-dimensional flow field to describe the morphological development of a river bend.

The model is a U-shaped flume, initially with a constant slope and depth.The computational domain and bathymetry is shown in Figure 4.1.
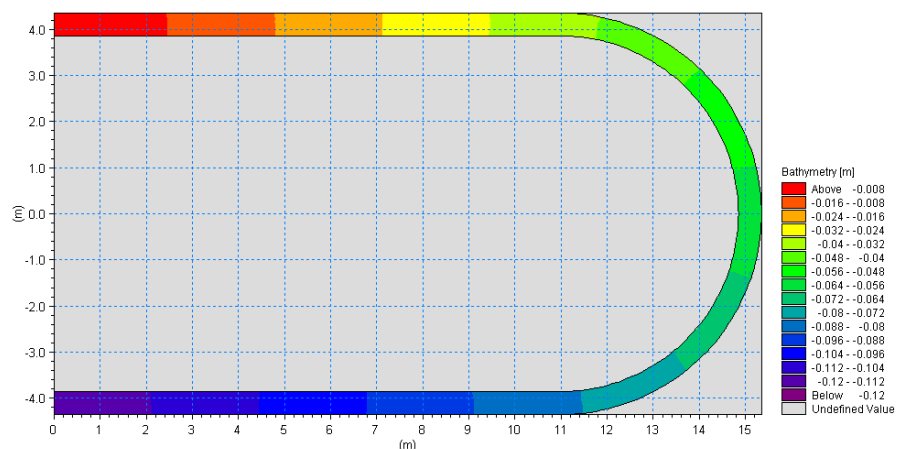
Figure 4.1   Computational domain and bathymetry

The problem is to investigate the morphological development of the bathymetry in the flume bending for uniform discharge conditions.

The test conditions are:

- The flume is approximately 25 m long, has a constant slope of 0.35% and a constant width of 0.5 m.

- The sediment in the flume is uniform with a constant mean grain diameter of 0.088 mm and a constant layer thickness.

- A constant discharge of 0.0057 m^3/s inserted into the model.

4.2.2   Defining the problem

The main condition defining the hydrodynamic problem is:

- An unstructured mesh with 1441 elements and 960 nodes is used. The mesh is shown in Figure 4.2.

- A total of 6 equidistant layers is applied in the vertical.

- An overall time step of 1 second is selected and the duration of the simulation is 40 minutes (2400 time steps).

- The eddy viscosity has been omitted.

- The bed resistance type is defined by a roughness height of 0.05 m

- Initially, the surface elevation is set to 0.01 m for the entire flume.

- The upstream boundary is defined by a constant discharge of 0.0057 m^3/s. The downstream boundary is defined by a constant surface elevation of -0.07066 m. A softstart of 180 s is used for both boundaries.

The main condition defining the sand transport problem is:

- The model type is selected as pure current with a non equilibrium transport condition. The transport theory is selected to Engelund and Hansen for both bed load and suspended load. The bed load factor and suspended load factor is set to 0.8 and 0.2, respectively.

- The mean horizontal velocity component calculated by the depth averated current.

- The sediment transport calculations start at timestep 300, which is 5 minutes after the start of the hydrodynamic calculations. The time step factor is 1, which correspond to the sediment transport calculations being carried out every overall time step, i.e. every second.

- The porosity is set to 0.35 and the relative density to default 2.65.

- The bed resistance type for the sediment transport calculations is derived from the HD calculations.

- The fraction concentration is initially set to equilibrium conditions. Equilibrium conditions are also set as the boundary conditions throughout the simulation.

- The morphological calculations start at timestep 600, which is 10 minutes after the start of the hydrodynamic calculations.

- The morphological update is including feedback on the hydrodynamic conditions. The speedup factor is defined as 1.
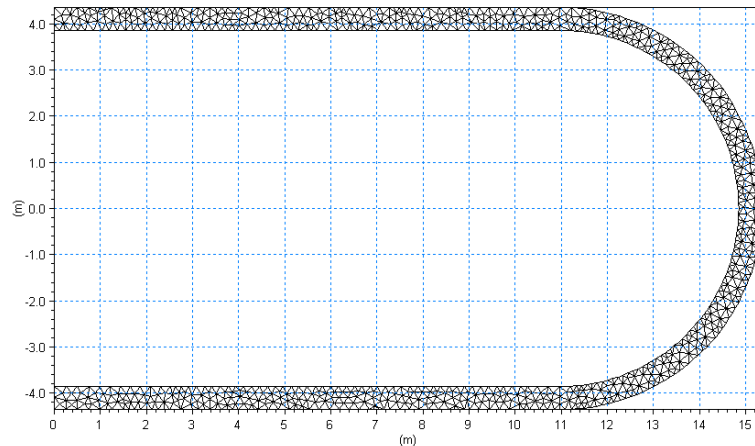
Figure 4.2   Computational mesh

4.2.3   Presenting and evaluating the results

A plot of the initial surface elevation and current field is shown in Figure 4.3. The figure show that the waterlevel is elevating towards the outer part of the bending.
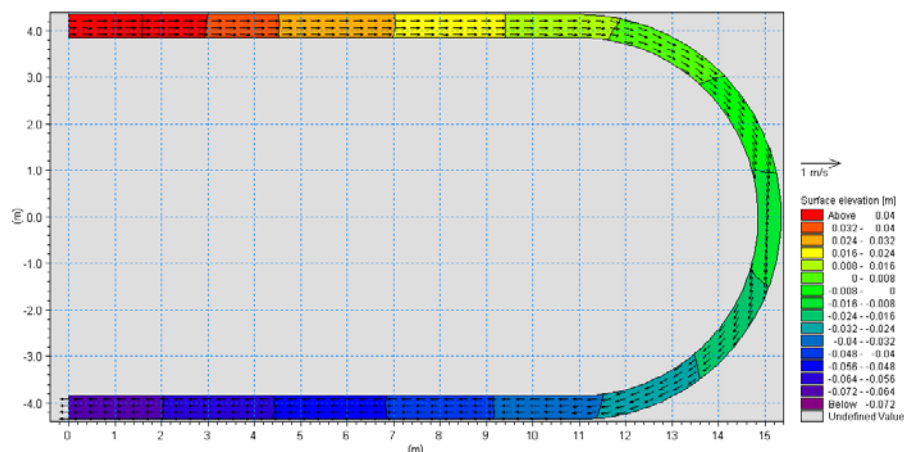
Figure 4.3   Contour plot of water surface, Vector plot of depth-averaged flow field.

The calculation of sand transport in MIKE 3 Flow Model FM is based on the findings of the 3-dimensional flow field. The direction of the mean velocity component is found by the mean current direction, corrected for the deviation angle because of secondary flow phenomena (e.g. helical flow). In a case like a river bend helical movements are very important and cause direction of the sand transport to differ from the mean direction of the flow.

A plot of the bed level change is shown in Figure 4.4.

0:40:00 01-01-2004 Time Step 40 of 40.
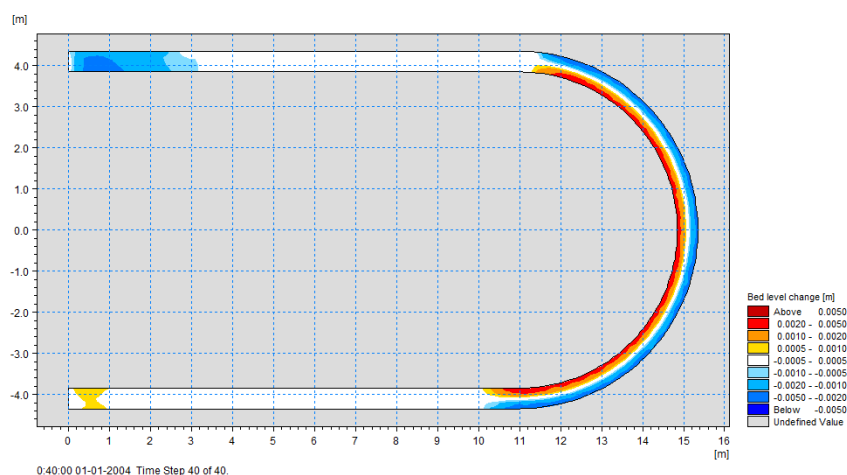
Figure 4.4   Contour plot of bed level change after 30 minutes.

The bed level change found by the 3-dimensional flow field show behavior that is observed in nature for meandering rivers: Erosion at the outer part of the bending and accumulation of material at the inner part of the bending.

Note that in this example the vertical velocity profile calculated by MIKE 3 Flow Model FM resembles a log-distributed profile, see Figure 4.5.

Despite the fact that the two forcing options, depth-integrated 3D flow field or bottom stress, respectively, may give different sand transport rates, the overall sand transport pattern will be similar for the two because of the log-distributed profile and giving the same sand transport gradients. This implies that the resulting bed level change will be very alike as shown in Figure 4.6.
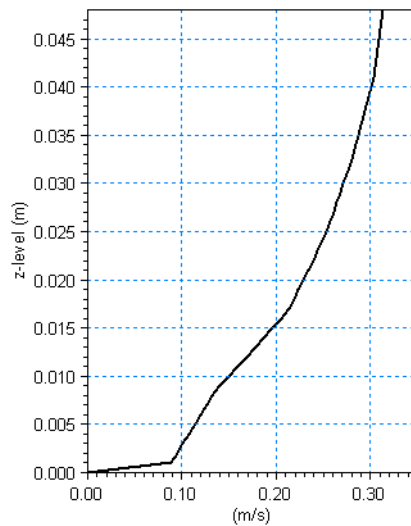
Figure 4.5   Vertical velocity profile in position (15.1 , 0) in domain
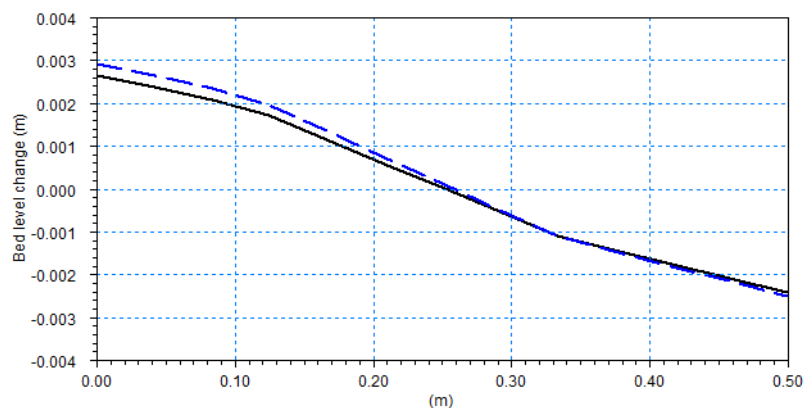
Figure 4.6   Resulting bed level change after 30 minutes for cross-section in the middle of the flume (x = 14.85 m to 15.35 m, y = 0 m).
Solid line: Forcing option as Depth-averaged 3D velocity field
Stippled line: Forcing option as Bottom stress

4.2.4 List of data and specification files

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

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

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

4.3 Sand Transport in Circulation Current

4.3.1 Purpose of the example

This example has been chosen as a fairly simple one to show the effect of choosing one method of forcing to the other under certain circumstances.

The problem is to determine the sand transport in a closed basin, where a circulation current exists due to wind set-up. The simulation of the circulation current is further described and compared to an analytical solution in the User Guide for the MIKE 3 Flow Model FM, Hydrodynamic Module. The test conditions are:

- The basin is rectangular in shape with a length of 1250 m in the east/west direction, and a length of 100 m in the north/south direction. The basin has a uniform depth of 10 m.
- The sediment in the basin is uniform with a constant mean grain diameter of 0.06 mm and a constant layer thickness.
- A westerly wind of 35 m/s is blowing.

4.3.2 Defining the problem

The main conditions defining the hydrodynamic problem is:

- An unstructured mesh with 174 elements and 122 nodes is used. The mesh has a minimum and maximum edge length of 50 m and 50 m, respectively. The mesh is shown in Figure 4.7. In the vertical domain 5 equidistant layers are used.
- The overall *time step* is selected as 10 seconds and the duration time of the simulation is 80 minutes (480 time steps).
- The *horizontal eddy viscosity* type has been chosen to the Smagorinsky formulation and a constant value of 0.28.
- The *vertical eddy viscosity* type has been chosen to the k-ε formulation.

- The *bed resistance* type has been chosen to roughness height and a constant value of 0.05 m is applied.

- The wind is specified as constant with wind speed 35 m/s and wind direction 270°N.

- The length of *soft start interval* (warm-up period) for the wind has been chosen to 20 minutes (1200 seconds) to avoid chock effects.

- A constant *wind friction coefficient* of 0.002425 is applied.

The main condition defining the sand transport problem is:

- The model type is selected as pure current with a non equilibrium transport condition. The transport theory is selected to Engelund and Fredsøe for both bed load and suspended load. The bed load factor and suspended load factor is set to 1.0 in both cases.

- The mean horizontal velocity component is calculated by the depth-averaged velocity field in simulation 1 but by the bottom stress value in simulation 2.

- The sediment transport calculations start at timestep 180, which is 30 minutes after the start of the hydrodynamic calculations. The time step factor is 12, which correspond to the sediment transport calculations being carried out every other minute.

- The porosity is set to 0.4 and the relative density to default 2.65.

- The bed resistance type for the sediment transport calculations is defined as Manning number with a constant value of 100 $m^{1/3}$/s.

- The fraction concentration is initially set to equilibrium conditions.
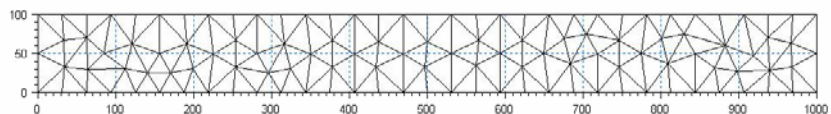
Figure 4.7   Computational mesh for the enclosed basin

4.3.3   Presenting and evaluating the results

Figure 4.8 shows the velocity profile at a vertical line in the centre of the basin (from (500, 50, -9.5) to (500, 50, -0.5)).
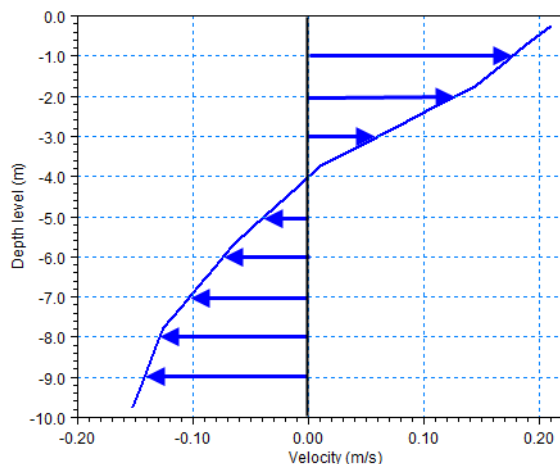
Figure 4.8   Vertical velocity profile in position (500, 50) in domain

The circulation current will cause the mean velocity component for the sand transport calculations to differ for the two forcing options as shown in Figure 4.9.
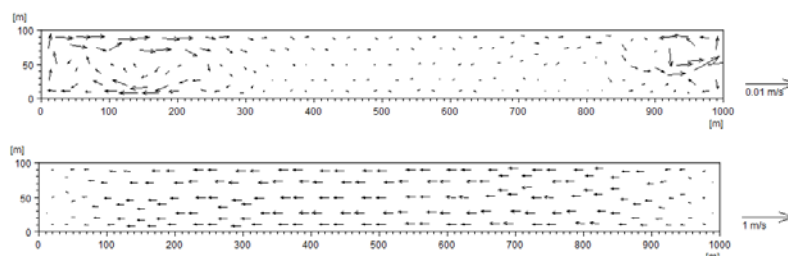
Figure 4.9   Forcing flow field for sand transport calculations
Upper figure: Forcing option as Depth-averaged 3D velocity field.
Lower figure: Forcing option from Bottom stress value.
NOTE: the vector scale is different for the two plots

In the case where the forcing velocity component is created by a depth-integration of the 3D flow field, the velocities are too small to create any sand transport in the basin.

In the case where the forcing velocity component is created by the bottom stress value, the velocities are sufficiently large to create a resulting sand transport that will cause the sand in the east side of the basin to be eroded and moved towards the west side of the basin as shown in Figure 4.10.
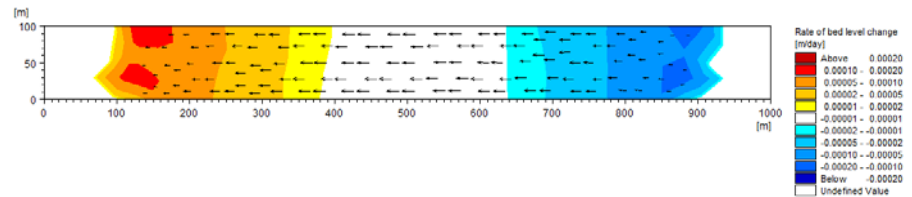
Figure 4.10   Initial bed level change rate using Bottom Shear Stress forcing

4.3.4   List of data and specification files

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

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

File name: Sim1.m3fm
Descrition: MIKE 3 Flow Model FM specification file, 3D flow option

File name: Sim2.m3fm
Descrition: MIKE 3 Flow Model FM specification file, bottom stress option

5 SAND TRANSPORT MODULE

The Sand Transport Module calculates the resulting transport of non-cohesive materials based on the mean horizontal flow conditions found in the hydrodynamic calculations and, if included, wave conditions from wave calculations.

5.1 Model Definition

You have the option to calculate sand transport using four different model types:

- Pure current
- Pure current - Multi fraction/Multi layer
- Waves and current
- 3D flow field

You may also set some additional model parameters for the simulation.

For more details on the four model types, see the scientific documentation. Note, that "3D flow field" can only be selected when the "Navier-Stokes equations" is selected for the governing equations in the hydrodynamic module.

5.1.1 General description

The Sand Transport Module calculates non-cohesive sediment transport. Two modes of sediment transport are described:

1. bed load transport
2. suspended load transport

A third category is normally referred to as wash load. Wash load is not included in this model.

For the "Pure current" model type and the "3D Flow field" model type, the bed load and suspended load is calculated separately. For the "Pure current - Multi fraction/Multi layer" model and "Waves and current" model it is the total load that is used.

5.1.2 Forcing parameters

In the three-dimensional flow you may calculate the mean horizontal velocity component to be used in the sand transport calculation from one of two hydrodynamic results

- Depth averaged velocity

• Bed shear stress

When "3D flow field" is selected for the model type, the horizontal velocity components are always calculated based on the bed shear stress from the hydrodynamic model.

5.1.3 Pure current

You must select one of two ways to describe the sand transport:

• Equilibrium
• Non-equilibrium

Selecting equilibrium the transport rates are calculated assuming equilibrium conditions.

Selecting the non-equilibrium the transport rates are calculated considering the conditions at the previous time steps.

For "Pure current" only one fraction of sediment can be simulated.

Layer thickness

You may choose to include a varying layer thickness as start condition for the simulation (see page 40). This option is useful when simulating sand transport in areas with rock bed present (i.e. cases with non-erodible bed and limited sand supply).

In case the local layer thickness Δh is less than the defined threshold thickness Δh_{crit}, the total transport rate Q_t is reduced according to a parabolic formulation, i.e.

$$Q_{t,reduced} = Q_t \cdot (\Delta h / \Delta h_{crit})^2 \qquad (5.1)$$

NOTE: In case of non-equilibrium pure current only the bed load is reduced using the same formula. The suspended load remains unchanged.

Fraction definitions

Five sediment theories are presently available in the model:

• Engelund and Hansen (total load)
• Van Rijn (bed load + suspended load)
• Engelund and Fredsoe (bed load + suspended load)
• Meyer-Peter and Muller (bed load)

- Kovacs and Parker (bed load)

Bed load

For bed load, a bed load factor, k_b, can be applied to the selected transport formula. The default value is 1, and values can vary between 0.5 and 2.0. Beyond this range the validity of the chosen formula could be questioned.

Suspended load

For suspended load a suspended load factor, k_s, can be applied to the selected transport formula. As for bed load the default value is 1, and values can vary between 0.5 and 2.0. Again, beyond this range the validity of the chosen formula could be questioned.

Total load

The Engelund and Hansen formula only predict total load, whereas information about both bed load and suspended load is required. The total load formula can still be applied by using the calibration factors k_b and k_s for bed load and suspended load, respectively, in order to differentiate between the two modes of transport. By specifying $k_b = 0.1$ and $k_s = 0.9$, it is understood that 10% of the transport takes place as bed load.

Kovacs and Parker

When "Kovacs and Parker" is selected for the bed load formula you can also specify the dynamic friction coefficient and the friction velocity factor. For more details see the scientific documentation. When bed morphology is included, the solution may become unstable if the dynamic friction coefficient μ_d is smaller than the tangent of the specified angle of repose in the bank erosion section.

Maximum concentration

To maintain stability, a maximum value of suspended sediment concentrations can be specified.

5.1.4 Pure current - Multi fraction/Multi layer

You must specify the number of fractions and number of layers. Both the number of fractions and number of layers must be 2 or greater. The layers are numbered in increasing order from the top and downward. Hence, the surface layer is layer 1.

You must also specify the maximum thickness of the surface layer and the maximum thickness of the sub-layers (between the surface layer and the bottom layer). There is no maximum thickness of the bottom layer.

You may include the influence of helical flow on the sand transport.

Fraction definitions

The total sediment load is calculated using the sediment theory by Wilcock and Crowe. For each fraction it is possible to specify a total load factor. The default values are 1, and values can vary between 0.5 and 2.0.

5.1.5 Waves and current

For combined wave and current the sand transport rates are found by interpolation in sediment transport tables.

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 a table with the same name and extension .crs exists in the same directory.

Layer thickness

You may choose to include a varying layer thickness as start condition for the simulation (see page 40). This option is useful when simulating sand transport in areas with rock bed present (i.e. cases with non-erodible bed and limited sand supply).

In case the local layer thickness Δh is less than the defined threshold thickness Δh_{crit}, the total transport rate Q_t is reduced according to a parabolic formulation, i.e.

$$Q_{t, reduced} = Q_t \cdot (\Delta h / \Delta h_{crit})^2 \qquad (5.2)$$

Sediment transport tables

The sediment transport tables has to be generated beforehand by the MIKE 21 Toolbox program "Generation of Q3D Sediment Tables". 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.

Remarks and hints

During the simulation the transport rates are found by linear interpolation in the transport 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
d	-	-	Program stops
gamma	-	-	Program stops
V^2/R	p.t. not used	p.t. not used	

5.1.6   3D flow field

The 3D flow field model uses a 3D transport equation for the suspended sediment concentration. The 3D flow field model can be used to for modelling various scenarios where vertical resolution is important, such as e.g. trench siltation, scour around monopiles and berms, breaker bar dynamics under waves.

For "3D flow field" only one fraction of sediment can be simulated.

Layer thickness

You may choose to include a varying layer thickness as start condition for the simulation (see Layer thickness). This option is useful when simulating sand transport in areas with rock bed present (i.e. cases with non-erodible bed and limited sand supply).

In case the local layer thickness Δh is less than the defined threshold thickness Δh_{crit}, the total transport rate Q_t is reduced according to a parabolic formulation, i.e.

$$Q_{t,reduced} = Q_t \cdot (\Delta h / \Delta h_{crit})^2 \qquad (5.3)$$

Fraction Definition

Bed load
Five sediment theories are presently available in the model for the bed load:

- Engelund and Hansen

- Van Rijn

- Engelund and Fredsoe

- Meyer-Peter and Muller

- Kovacs and Parker

For bed load, a bed load factor, k_b, can be applied to the selected transport formula. The default value is 1, and values can vary between 0.5 and 2.0. Beyond this range the validity of the chosen formula could be questioned.

The Engelund and Hansen formula only predict total load, whereas information about both bed load and suspended load is required. The total load formula can still be applied by using the calibration factors k_b and k_s for bed load and suspended load, respectively, in order to differentiate between the two modes of transport. By specifying $k_b = 0.1$ and $k_s = 0.9$, it is understood that 10% of the transport takes place as bed load.

Suspended load

For suspended load a suspended load factor, k_s, can be applied to the selected transport formula. As for bed load the default value is 1, and values can vary between 0.5 and 2.0. Again, beyond this range the validity of the chosen formula could be questioned.

Bed concentration

The suspended sediment concentration is calculated by solving a 3D transport equation for the concentration. In this case you must specify the formula used for calculating the reference bed concentration defined at a level of 2 sediment grain diameters above the bed. Here two options are available

- Engelund and Fredsøe

- Zyserman and Fredsøe

Kovacs and Parker

When "Kovacs and Parker" is selected for the bed load formula, you must specify the bed shear stress forcing type

- Total bed shear stress

- Skin friction bed shear stress

When Total bed shear stress is selected, the bed load and the reference bed concentration are derived from the bed shear stress from the hydrodynamic module using the Shields parameter θ formula

$$\theta = \frac{\tau_b}{(\rho_s - \rho)gd_{50}}$$ (5.4)

where

τ_b is the bed shear stress,
θ_s is the density of the sediment,
θ is the density of the water,
g is acceleration due to gravity and
d_{50} is the mean grain diameter.

When Skin friction bed shear stress is selected (assuming presence of unresolved bed forms), only the skin friction component of the bed shear stress is used in the calculation of the bed load and the reference bed concentration. The skin friction component of the Shields parameter θ', is estimated by the Engelund & Fredsøe (1982) relation

$$\theta' = \begin{cases} \min(0,06 + 0,3\theta^{1,5}, \theta) \ , \ \theta_{cr} \le \theta < 0,55 \\ \theta \qquad \qquad , \qquad \theta \ge 0,55 \end{cases} \tag{5.5}$$

When "Kovacs and Parker" is selected for the bed load formula you can also specify the dynamic friction coefficient and the friction velocity factor. For more details see the scientific documentation. When bed morphology is included, the solution may become unstable if the dynamic friction coefficient μ_d is smaller than the tangent of the specified angle of repose in the bank erosion section.

Maximum concentration

To maintain stability, a maximum value of suspended sediment concentrations can be specified.

5.2   Time Parameters

The sand transport calculation is activated at the start time step relative to the start of the simulation specified on the Time dialog.

Once the sand transport calculation is activated the time step for the sand transport calculations is the overall time step, specified on the Time dialog, multiplied by the time step factor.

In case the model type is pure current with non-equilibrium transport the time-integration is performed using a dynamic time step interval (see Solution Technique below).

5.3   Solution Technique

The simulation time and accuracy can be controlled by specifying the order of the numerical schemes that 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 transport (advection-dispersion) equation 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 equations 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.

5.3.1 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.

5.4 Sediment Properties

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.

5.4.1 Sediment Properties

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

When "Pure current" or "3D flow field" is selected for the model type description you must specify the mean grain diameter, the relative density and the critical Shields value. When "Pure current - Multi fraction/Multi layer" is selected for the model type, you must specify the grain diameter and the density for each fraction. The grain diameter and density are constant for the whole domain. When "Wave and current" is selected for the model type you must specify the mean grain diameter and the grading coefficient, σ_g.

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 piecewice constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data.

5.4.2 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 relative sediment density is 2.65.

5.5   Bed Resistance

The bed resistance can be specified as

- No bed resistance
- Chezy number
- Manning number
- Alluvial resistance
- Resistance from HD simulation

Selecting Alluvial resistance you must specify the Manning number. When "3D flow field" is selected for the model type the resistance information is always taken from the HD simulation.

5.5.1   General description

The bed resistance is the resistance against the flow.

Alluvial bed resistance

You can use an alluvial roughness coefficient, which is a function of the local depth at every time step. The Chezy number (C) or the Manning number can be updated as follows:

$$C = a \cdot H^b \tag{5.6}$$

where:
H is local water depth
a is the Resistance coefficient and
b is the Resistance power

Inclusion of the resistance updating effects simulated scour and deposition patterns. Flow is deflected more over shallow parts, and sediment transport increases due to increased bed shear stresses. In general, this reduces over-shoot effects when bed scour commences, and causes the topography of a point bar crest to become more rounded (as reported by Talmon, 1992).

The calculated Chezy coefficient is limited by a Minimum resistance and a Maximum resistance.

If hydrodynamic calibration of the bed resistance shows a certain relationship between local depth and (say) Chezy number, when running the morphological model an alluvial roughness coefficient should be specified using the same parameters.

5.5.2   Data

The Chezy number, C, Manning number, M, and Alluvial resistance 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 Chezy number or the Manning number 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, a piecewice constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data.

5.5.3   Recommended values

To avoid model instability when applying alluvial bed resistance, the maximum Chezy number should normally be less than 90 $m^{1/2}$/s.

5.6   Forcings

The model type determines the necessary forcings to calculate the sediment transport. In case of "Pure current" or "Pure current - Multi fraction/Multi layer" a flow field is needed only, but for "Waves and current" it is necessary to define the wave properties as well.

If the Hydrodynamic module is selected the flow field is always included directly from the ongoing simulation. The wave field can be included by reading values from a data file or directly from a coupled SW simulation.

5.6.1   Waves

You can define the waves in your calculations as

- No waves
- Wave field
- Wave field from SW simulation

No waves

If you choose no waves, the sediment transport rates will be found by interpolation in the transport tables using the input values $(H_{rms}/D)_{min}$, $(H_{rms})_{min}$ and $(T_p)_{min}$. The angle will be set to 0.0 deg.

Wave field

If you choose wave field you have four options to describe the waves

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

For all options you must specify the wave height, wave period and the angle to true north.

You must select to specify the wave height as the RMS wave height **or** the significant wave height.
You must select to specify the wave period as the peak wave period **or** the mean wave period.

For the case of varying waves, you have to prepare a data file containing the wave properties (mean heights, periods and angles to true north) before you set up the hydrodynamic simulation.

Data

Constant
For the case of constant waves the waves will be sinusoidal with no directional spreading. You must specify the wave height, wave period and the angle to true north. These values will be set for the entire domain in all time steps of the simulation.

Varying in time, constant in domain
For this case you have to prepare a time series file (dfs0) that contains the wave properties. The values in the data file will be set for the entire domain for each time step in the simulation. If the time step in the file is not equal to that of the simulation, the program will interpolate (see Figure 5.1).

Constant in time, varying in domain
For this case you have to prepare a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2) that contains the wave properties. The area in the data file must cover the model area. If a dfsu-file is used, a piecewice constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data. The first time step of the file will be used in all time steps of the simulation.

Varying in time and domain
For this case you have to prepare a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2) that contains the wave properties. The area in the data file must cover the model area. If a dfsu-file is used, a piecewice constant interpolation is used to map the data. If a dfs2-file is used, a 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 5.1).
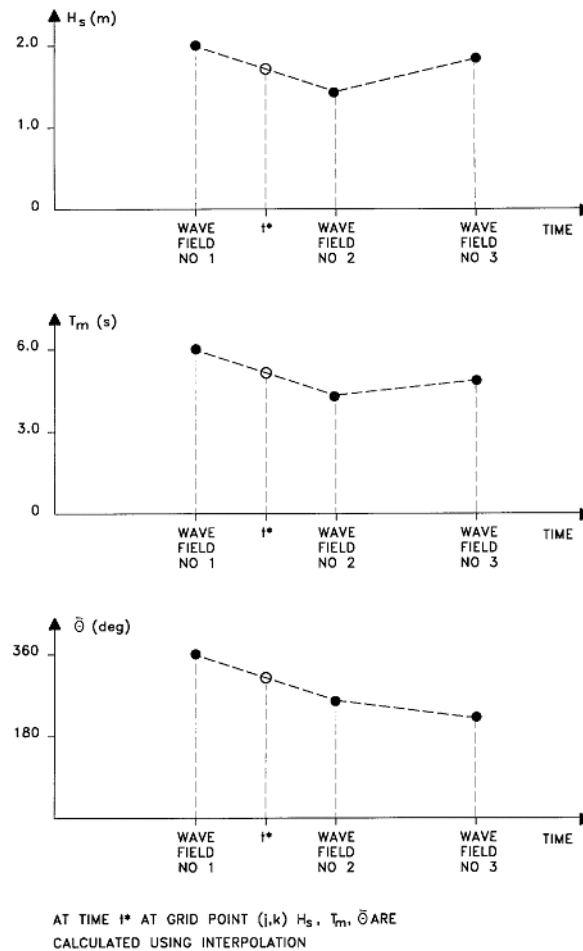

Figure 5.1   Interpolation of wave fields in time

5.6.2   Flow

If the Hydrodynamic module is not included in the simulation you can define the flow in your calculations as

- No flow
- Flow Field

For the flow field you must specify the surface elevation and vector velocity (the U and V velocity).

Data

You have to prepare a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2) that contains the flow properties. The area in the data file must cover the model area. If a dfsu-file is used, a piecewice constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data. The program will interpolate if the time step is not equal to that of the simulation.

5.7 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.

When "3D flow field" is selected for the model type, a 3D transport equation is solved for the suspended concentration. 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.

5.7.1 Dispersion

The dispersion can be formulated as

- 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.

Data

Selecting 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 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, a piecewice constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data.

When selecting the 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.

When "Pure current" is selected for the model type the file must be a 2D unstructured data file (dfsu) or a 2D data grid file (dfs2). When '3D flow field' i selected, 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, a piecewice constant interpolation is used to map the data. If a dfs3-file is used, a bilinear interpolation is used to map the data.

5.7.2 Recommended values

When more sophisticated eddy viscosity models are used, 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. 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 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 more information, see Rodi (1980).

5.8 Sources

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

In the Sand Transport Module the source concentrations of each component in every sources point can be specified. The number of sources, their generic names, location and discharges 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.

5.8.1 Source specification

The type of sources can be specified in two ways

- Specified concentration
- Excess concentration

The source flux is calculated as Q_{source}. 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 discharge out 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 discharge 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 discharge out 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.

5.8.2 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 usually is lower than the source concentration.

5.9 Initial Conditions

The model type (see Model Definition page 23) determines the necessary initial conditions. When "Pure current" is selected for the model type and Non-equilibrium is selected for the transport description or when you have selected "3D flow field" for the model type, the initial concentration of the suspended sediment has to be specified. When "Pure current" or "Wave and current" is selected for the model type and varying layer thickness is included you must specify the initial bed thickness of the erodible bed. When "Pure current - Multi fraction/Multi layer" is selected for the model type the layer thickness for each layer and the composition of the layer has to be specified. 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.

5.9.1 Fraction concentration

You can choose between the following two types of initial conditions boundary types

- Specified initial conditions
- Equilibrium conditions

When "3D flow field" is selected for the model type then "Equilibrium conditions" can not be used.

Data

The initial concentration (in component unit) for each fraction can be specified as

- Constant (in domain)

- Varying in domain

A typical background concentration is 10 g/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.

When "Pure current" is selected for the model type and Non-equilibrium is selected for the transport description, 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 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.

When "3D flow field" is selected for the model type, 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, the mesh in the data file must match exactly the mesh in the simulation. 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.

5.9.2 Layer thickness

When "Pure current" or "Wave and current" is selected for the model type and varying layer thickness is include you must specify the initial bed thickness of the erodible bed.

Data

The initial bed thickness 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 layer 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, 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 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 times of the file.

5.9.3 Layer thickness and composition

When "pure current - multi fraction/multi layer" is selected for the model type you must specify the initial bed thickness for each layer and the composition of each layer. The composition of a layer must be given as the content for each of the fractions in %. This means that for a layer the sum of the composition (content) in each point must be 100%. The layers are numbered in increasing order from the top and downward. Hence, the surface layer is layer 1.

Data

The initial thickness of 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 layer 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, 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 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 times of the file.

The composition (content in % of the fractions) of 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 content in % for each fraction 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, 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 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 times of the file.

5.10   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.

5.10.1   Boundary specification

You can choose between the following four boundary types

- Land
- Equilibrium conditions
- Specified values
- Zero gradient

When "3D flow field" is selected for the model type then "Equilibrium conditions" can not be used.

Data

If specified values is selected the format of the fraction concentration at the boundary can be 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 component concentration (in component unit) before you set up the hydrodynamic simulation.

When "Pure current" is selected for the model type and Non-equilibrium is selected for the transport description, the file must be a profile file (dfs1). 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).

When "3D flow field" is selected for the model type, 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.

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 inter-polation 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.

5.11 Morphology Module

The Morphology module is included provided the ST module is selected.

A more detailed description of the Morphology module can be found in sec-tion 6 MORPHOLOGY MODULE.

5.12 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 21 viewing/editing tool by clicking on the "View" button during and after the simulation.

5.12.1 Geographic view

This shows the geographical position of the output data.

5.12.2 Output type

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 only 2D field parameters can be selected unless "3D flow field" is selected for the model type. The discharge through a cross-section can also be selected.

In case the model type is Pure current, described by a non-equilibrium transport description, or 3D flow field, the mass budget for a domain can be selected as well.

NOTE:
- output of discharge is not available in the case of equilibrium pure current

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.

If mass budget is selected for the field type 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 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 and area series both options are available. For line series only interpolated values is available. See also Table 5.2.

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 values 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 5.2.

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

Table 5.2   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
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 5.3   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

The flood and dry can be treated in three different ways

- Whole area

- Only wet area

- Only real wet area

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. 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 5.2.
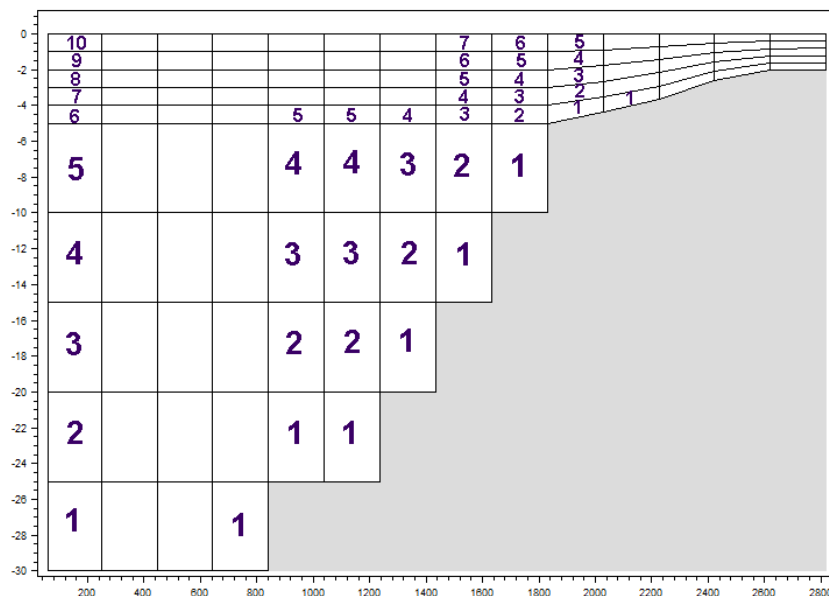
Figure 5.2   Example of layer numbers in point output specification in case of combined sigma-z level mesh.

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. 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. The third item is unused (but must be specified).

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.

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 5.3.
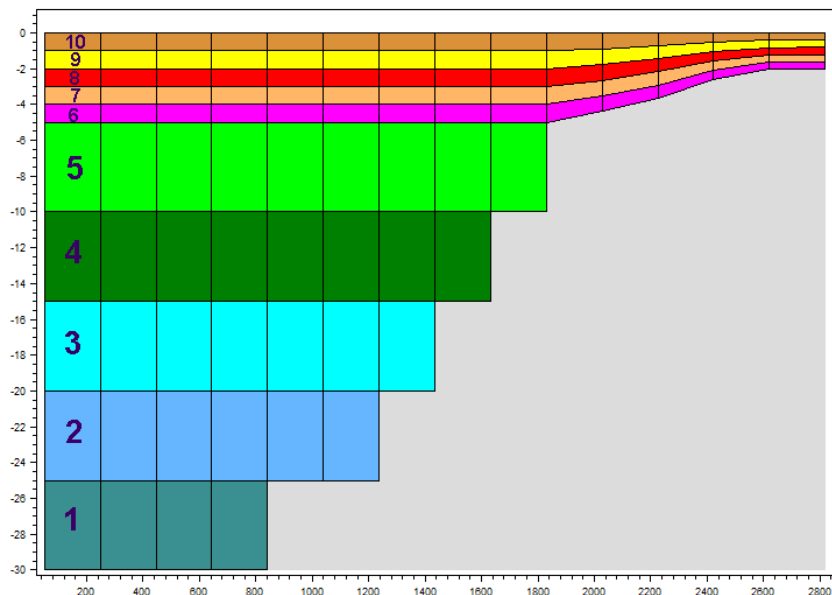

Figure 5.3   Example of layer numbers in volume output specification in case of combined sigma-z level mesh.

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.

NOTE: output of discharge is only available in the case of non-equilibrium pure current.

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.

NOTE: output of mass budget is only available in the case of non-equilibrium pure current.

5.12.3 Output items

2D (horizontal)

You can select basic output variables and additional output variables.

The basic variables are

- SSC - suspended sediment concentration (for each fraction)
- Bed thickness (for each layer)
- Bed distribution (for each layer and each fraction)
- Bed load, x-component
- Bed load, y-component
- Suspended load, x-component
- Suspended load, y-component
- Total load, x-direction
- Total load, y-direction
- Rate of bed level change
- Bed level change
- Bed level

The bed thickness is only available in case of varying layer thickness. For combined waves and current only the total load is available, not the concentration, not the bed load or the suspended load.

The additional variables are

- Bed load, magnitude
- Bed load, direction
- Suspended load, magnitude
- Suspended load, direction
- Total load, magnitude
- Total load, direction
- Acc. bed load, x-component
- Acc. bed load, y-component
- Acc. suspended load, x-component
- Acc. suspended load, y-component
- Acc. total load, x-component
- Acc. total load, y-component
- Acc. total load, x-component (for each fraction)
- Acc. total load, y-component (for each fraction)
- Sediment size surface mean
- Number of layers
- Deviation
- Centrifugal acceleration
- Curvature
- Surface elevation
- Still water depth
- Total water depth
- Depth averaged U-velocity
- Depth averaged V-velocity
- Wave height
- Wave period
- Wave direction

Only relevant variables are available depending on the model type (see Model Definition p.23).

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

Load directions are defined positive clockwise from true North (going towards). Wave directions are defined positive clockwise from true North (coming from).

When "Pure current - Multi fraction/Multi layer" is selected for the model type, the layers in the output are numbered in increasing order from the top and downward. Hence, the surface layer is layer 1.

3D

When you have selected "3D low field" for the model type you can get an 3D output file. You can select basic output variables and additional output variables.

The basic variables are

- SSC - suspended sediment concentration (for each fraction)

The additional variables are

- Velocity components
- CFL number

3D (bed parameters)

When you have selected "Pure current - Multi fraction/Multi layer" for the model type you can get an 3D output file showing the bed layers. The items in the file are

- Layer number
- Bed distribution (an item for each fraction)

The layers are numbered in increasing order from the top and downward. Hence, the surface layer is layer 1.

Mass Budget

When you have selected "Pure current" for the model type and "Non equilibrium" for the transport description or when you have selected "3D flow field" for the model type, you can select the mass budget calculation to be included for the flow and for the suspended sediment fraction.

For each 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 concentration the minimum value is 0, while the maximum value is specified on the Model Definition Dialog.

Discharge

When you have selected "Pure current" for the model type and "Non equilibrium" for the transport description or when you have selected "3D flow field" for the model type, you can select the discharge calculation to be included for the flow and for the suspended sediment load. The discharge of the suspended sediment load is calculated using the general transport equations and does NOT consider the Speedup factor.

When you have selected "Wave and current" for the model type, you can select the discharge calculation to be included for the flow and the total sediment load.
The discharge of the total sediment load is calculated based on the reconstructed fluxes and INCLUDES the effect of the Speedup factor. The values may be affected by the maximum limit of bed level change.

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. The discharge rates are given in solid volumes.

6 MORPHOLOGY MODULE

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.

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.

$$Z^{n+1} = Z^n + \Delta z^n \tag{6.1}$$

where:

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

Speedup factor

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

$$Z^{n+1} = Z^n + \Delta z^n \cdot Speedup \tag{6.2}$$

In which *Speedup* is a dimensionless speed-up factor. Note that the amount of suspended sediment is not affected by this. Only the bed is affected.

The layer thickness is updated in the same way.

6.2 Model Definition

To ensure a stable update of the bed you may introduce a limiting bed level change rate.

You may choose to exclude the morphological development of the bed by deactivating the feedback on the hydrodynamic, wave and sand transport calculation. Thus the calculated values will be based on the initial bed level thoughout 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, have to be the same as the time step of the simulation. A linear interpolation will be applied if the time steps differ.

You can specify a soft start interval during which the speedup factor is increased linearly from 1 to the specified value of the speedup factor. Soft start will only be applied to values larger than 1. By default the soft start interval is zero (no soft start).

6.2.1 Remarks and hints

Note that the layer thickness is always updated. This means the sand transport rates may be affected in locations where the layer thickness varies around the value for the threshold thickness during the simulation.

If the bed level change within one time step exceeds the threshold thickness mass budget errors may be introduced.

For combined wave and current the values in a discharge output will consider the speedup factor and may be affected by the maximum limit of bed level change.

It is possible to limit the overall area for morphology by using a 'speedup map'. This can improve results for combined one-line/2D morphological modelling but can also be used in normal 2D morphological modelling (i.e. without using the Shoreline Morphology module).

6.3   Time Parameter

The morphological update is activated at a Start time step relative to the start of the simulation specified on the Time dialog.

Once the morphological update is activated the bathymetry is updated for every overall time step specified on the Time dialog.

6.4   Slope Failure

It is possible to include the effect of slope failure on the morphological update of the bathymetry.

The slope failure mechanism can be defined in one of five ways:

- No slope failure
- Simple bank erosion
- Extended bank erosion
- General slope failure
- General slope failure incl. dry elements

The slope failure mechanism will come into effect if the local bed slope becomes steeper than a specified angle of repose.

6.4.1   Angle of repose

The format of the angle of repose can be specified as

- Constant.
  The angle of repose is constant over the entire model area.
- Varying in domain.
  The angle of repose varies over the model area.

For the case with values varying in domain you have to prepare a data file containing the angle of repose before you set up the morphological simula-tion. The file must be a 2D unstructured data file (dfsu) or a 2D data 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.

6.4.2   General description

The chosen type of slope failure determines which elements that are checked for slope failure.

If Simple bank erosion is selected, the slope failure is checked for real wet elements next to a dry element.

If Extended bank erosion is selected, the slope failure is checked for real wet elements or not real wet elements next to a dry element.

If General slope failure is selected, the slope failure is checked for all elements that are not totally dry, disregarding the neighbouring elements.

If General slope failure incl. dry elements is selected, the slope failure is checked for all elements in the domain.

Note that the inclusion of slope failure will increase the computation time.

6.5 Boundary Conditions

Boundary conditions must be defined for the morphological update in addition to the Boundary Conditions given for the sediment transport rates.

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 meningful names in the Domain dialog.

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.

6.5.1 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

6.6 Bed Level Sources

You can add/remove sediment in the domain by applying a bed level source. A bed level source can specify an extra bed level change besides the calculated morphological changes. By this it is possible to account for e.g. sand bypass by harbours (using pumps) or additional sand into the model from river outlets. A bed level source can also be used for limiting the bed level accretion in certain areas by automatic dredging and optional redistribution of

the dredged material. The sediment volume deficit will be included in the sediment budget.

You create a new source by clicking on the "New source" button. By selecting a source in the source list and clicking on the "Delete source" you can remove this source. For each source you can specify the name of the source and whether the source should be active or not. The specification of detailed information for each source is made subsequently. From the list view page you can go to the dialog for specification by clicking on the "Go to .." button or by selecting a source from the source list and clicking on the "Edit source" button.

6.6.1 Bed level source specification

You can define the source in one of two different ways:

- Standard
- Dredging

Depending on the type of source you need to specify an additional rate of bed level change or to specify a maximum allowed bed level together with a description of what to do with the excess material that may be removed from some elements.

Standard

A standard source is used to specify an additional rate of bed level change in the domain. The data can be specified as:

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

If the bed level source is specified by a constant additional rate of bed level change you have to specify the additional rate directly.

For the case with constant in time but varying in domain you have to prepare a data file containing the additional rate of bed level change. 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.

For the case with varying in time and domain you have to prepare a data file containing the additional rate of bed level change. 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 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.

Dredging

A dredging source can be used to control the maximum bed level in the domain area. By specifying the maximum allowed bed level in each element, the model will automatically remove the excess volume. The removed excess volume can be discarded or redistributed in the domain using a distribution map.

Maximum allowed bed level

The maximum allowed bed level can be specified as

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

If the maximum allowed bed level is specified by a constant the value is specified directly.

For the case with constant in time but varying in domain you have to prepare a data file containing the maximum allowed bed level. 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.

For the case with varying in time and domain you have to prepare a data file containing the maximum allowed bed level. 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 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.

Distribution factor

By specifying a distribution map you can redistribute the removed excess volume from the bed level source to other places in the domain. The resulting amount of material in each element is controlled by a distribution factor. The distribution factor can be specified in three ways:

- Constant (in time and domain)
- Constant in time, varying in domain

- Varying in time and domain

If the distribution factor is specified by a constant the value is specified directly.

For the case with constant in time but varying in domain you have to prepare a data file containing the distribution factor. 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.

For the case with varying in time and domain you have to prepare a data file containing the distribution factor. 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 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.

NOTE: The distribution factors within the distribution map do not need to sum up to 1 (or 100). The total excess volume of sediment is redistributed in the domain according to a weighted distribution factor in each element.

INDEX

V

W