MIKE 21 Flow Model FM

Sand Transport Module, incl. Shoreline Morphology

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 21 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 21: 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 21 Flow Model FM describes erosion, transport and deposition of sand under the action of currents and waves or pure current.

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 21 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 21 Flow Model FM. The user guide and the training guide (PDF-format) can be accessed from the MIKE 21 Documentation Index:

  MIKE 21 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

In addition a comprehensive training guide covering the Sand Transport Module and the Coastline Morphology Module within the MIKE 21/3 Coupled Model FM is available. The training guides (PDF-format) can be accessed from the MIKE 21 Documentation Index:

  MIKE 21/3 Coupled Model FM,
  Step-by-Step Training Guide: Coastal Application

  MIKE 21/3 Coupled Model FM, Shoreline Morphology
  Step-by-Step Training Guide: Headland Beach example

4 EXAMPLES

4.1 General

One of the best ways of learning how to use a modelling system such as MIKE 21 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_21\FlowModel_FM\ST\U_flume

- Equilibrium slope example:
  .\Examples\MIKE_21\FlowModel_FM\ST\EqSlope

- Shelter example:
  .\Examples\MIKE_21\FlowModel_FM\ST\Shelter

- Harbour example:
  .\Examples\MIKE_21\FlowModel_FM\ST\Harbour

- Tidal inlet example:
  .\Examples\MIKE_21\FlowModel_FM\ST\Inlet

- River example. From scratch to simulation:
  .\Examples\MIKE_21\FlowModel_FM\ST\Gorai

- Torsminde example. From scratch to simulation:
  .\Examples\MIKE_21\FlowModel_FM\ST\Torsminde

- Morphology example. From scratch to simulation:
  .\Examples\MIKE_21-3_Integrated_Models\CoupledModel_FM\Torsminde

- Shoreline morphology example:
  .\Examples\MIKE_21-3_Integrated_Models\CoupledModel_FM\Pier

- Shoreline morphology example:
  .\Examples\MIKE_21-3_Integrated_Models\CoupledModel_FM\Harbour

- Shoreline Morphology example. From scratch to simulation:
  .\Examples\MIKE_21-3_Integrated_Models\CoupledModel_FM\HeadlandBeach

4.2 U-Shaped Flume

4.2.1 Purpose of the example

This simplified example has been chosen to describe the influence of helical movements on 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.
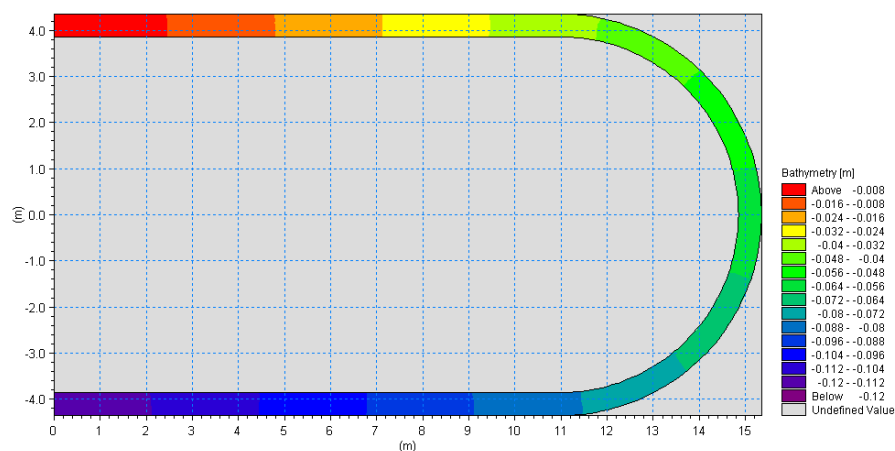
Figure 4.1   Computational domain and bathymetry

The problem is to investigate the morphological development of the bathymetry in the flume bending for uniform discharge conditions, when excluding and including the effect of helical movements, respectively.

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 m³/s inserted into the model.

4.2.2 Defining the problem

The main condition defining the hydrodynamic problem is:

- An unstructured quadrangular mesh with 600 elements and 707 nodes is used. The mesh is shown in Figure 4.2.

- 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 has been defined as a Chezy number with a constant value of 18.7 $m^{1/2}$/s.

- 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 helical flow is excluded from simulation 1, but included in simulation 2.

- 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 defined as Chezy number with a constant value of 18.7 $m^{1/2}$/s.

- 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.

- If helical flow is included equilibrium conditions are used for both initial state and at the boundaries.
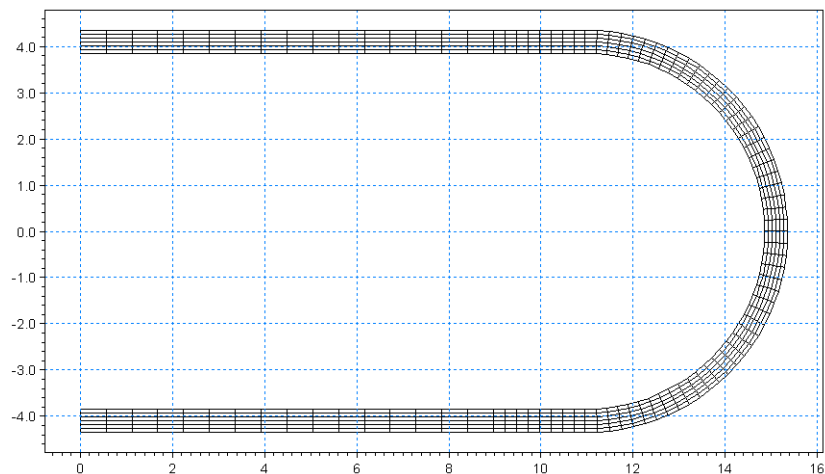
Figure 4.2   Computational mesh

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.
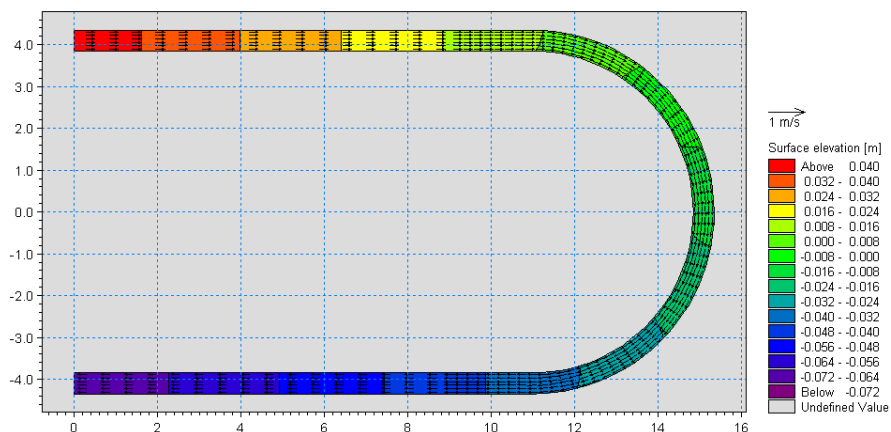
Figure 4.3   Contour plot of water surface, Vector plot of flow field.

A plot of the bed level change is shown in Figure 4.4.
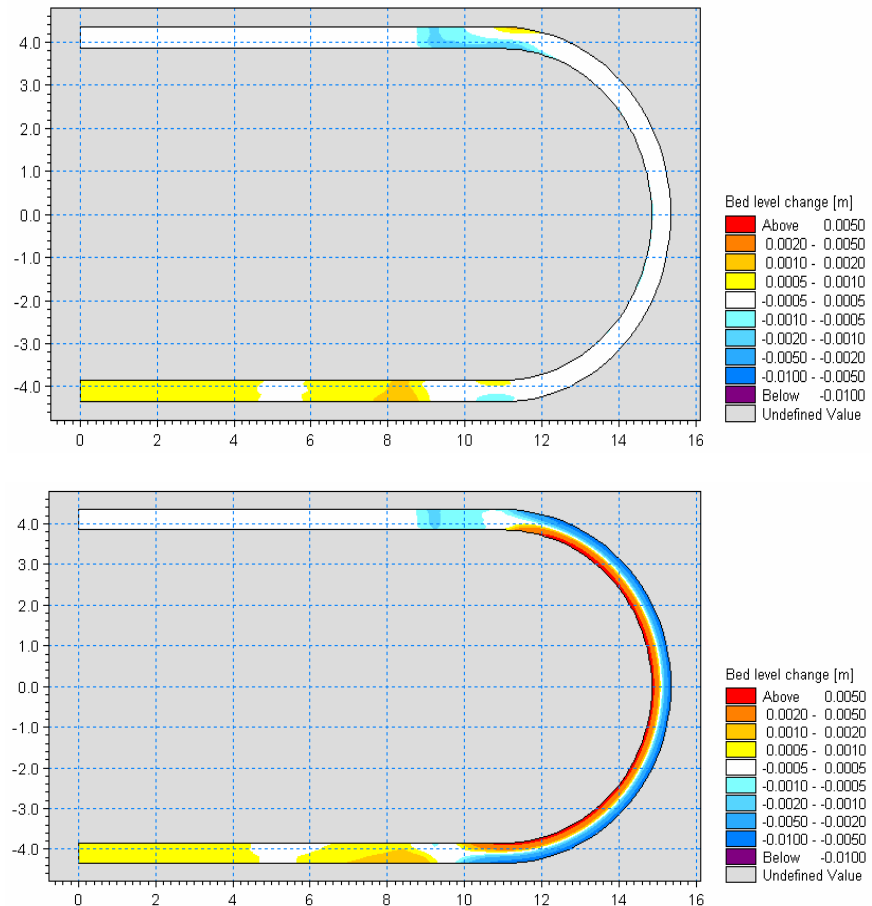
Figure 4.4   Contour plot of bed level change after 30 minutes.
Upper figure: excluding helical flow
Lower figure: including helical flow

Note that the evolution in the bending of the curve is very different for the two cases.

In the case where the influence of helical movements on the sediment transport has been omitted, the bed level change is influenced by pressure gradients only. Thus resulting in erosion of the bed at the inner part of the bending and accumulation of material at the outer part. Overall the bed level changes are small compared to the case where helical flow is included.

In the case where the influence of helical movements on the sediment transport is included, the bed level change is mostly influenced by the helical flow. This results in erosion at the outer part of the bending and accumulation of material at the inner part of the bending. This behaviour is the one that is observed in nature for meandering rivers.

4.2.4 List of data and specification files

The following data files (included in the ST\U_flume folder) are supplied with MIKE 21 Flow Model ST FM:

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

File name: Sim1.m21fm
Description: MIKE 21 Flow Model FM specification file, no helical flow

File name: Sim2.m21fm
Description: MIKE 21 Flow Model FM specification file, incl. helical flow

4.3 Equilibrium Slope

4.3.1 Purpose of the example

The aim of this example is to verify that MIKE 21 ST FM is able to predict the equilibrium slope of a straight flume.

The model is a straight flume, 30 m long and 2 m wide, that has a moveable bed. Initially the bed level is -0.4 m along the entire flume. Now a fixed discharge is entered in the upstream end, while the downstream water level is kept constant. As the bed of flume is initially horizontal an accelerating flow is created, causing erosion of the bed.

The problem is to simulate the equilibrium bed level of the flume under the given conditions.

4.3.2 Defining the problem

The main condition defining the hydrodynamic problem is:

- An unstructured quadrangular mesh with 180 elements and 244 nodes is used. The mesh is shown in Figure 4.5.
- An overall time step of 0.5 seconds is selected and the duration of the simulation is 30 minutes (3600 time steps).
- The eddy viscosity has been omitted.
- The bed resistance type has been defined as a Manning number M with a constant value of 32 $m^{1/3}$/s.
- Initially, the surface elevation is set to 0.0 m for the entire flume.
- The upstream boundary is defined by a constant discharge of 0.5 m^3/s (corresponding to q = 0.25 m^3/s/m). The downstream boundary is defined by a constant surface elevation of 0.0 m. A soft start of 120 seconds is used for the upstream boundary.

The main condition defining the sand transport problem is:

- The model type is selected as Pure current with an Equilibrium transport condition. The transport theory is selected to Engelund and Fredsøe.

- The sediment transport calculations start at timestep 600, 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 for every overall time step of 0.5 seconds.

- The porosity is set to 0.40 and the relative density to default 2.65. The bed consist of grains with d50=0.1 mm

- The bed resistance type for the sediment transport calculations is defined as a Manning number with a constant value of 32 $m^{1/3}$/s.

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

- The morphological calculations start at timestep 1200, which is 10 minutes after the start of the hydrodynamic calculations.
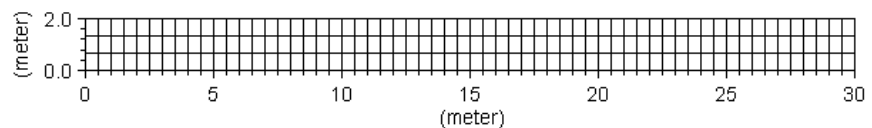
Figure 4.5   Computational mesh

4.3.3   Presenting and evaluating the results

The water level upstream adapts itself to the change in bed level during the morphological simulation. The change in water level in time is shown in Figure 4.6.
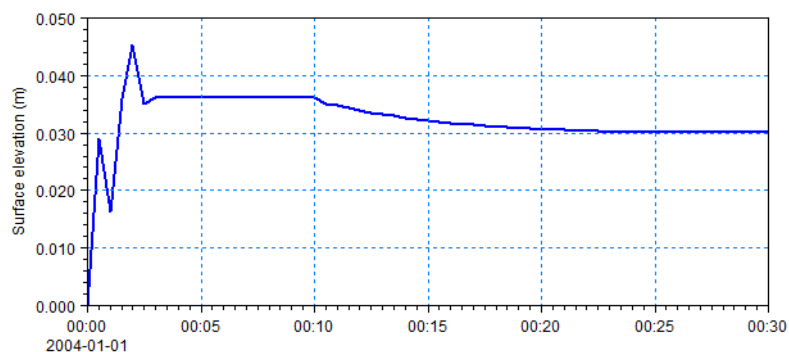
Figure 4.6   Water level at the upstream boundary during simulation.

As the water level downstream is fixed to 0.0 m, Figure 4.6 shows that the slope of the water level, I, is 0.03 m to 30 m, i.e. I_0 = 0.001.

The standard equation for natural depth, y_0, in a wide flume is

$$y_0 = \left(\frac{q}{M\sqrt{I_0}}\right)^{3/5} \tag{4.1}$$

Inserting the hydrodynamic conditions from the simulation you find the theoretical natural depth to y_0 = 0.43 m.

Figure 4.7 shows the development of the bed level during the simulation.
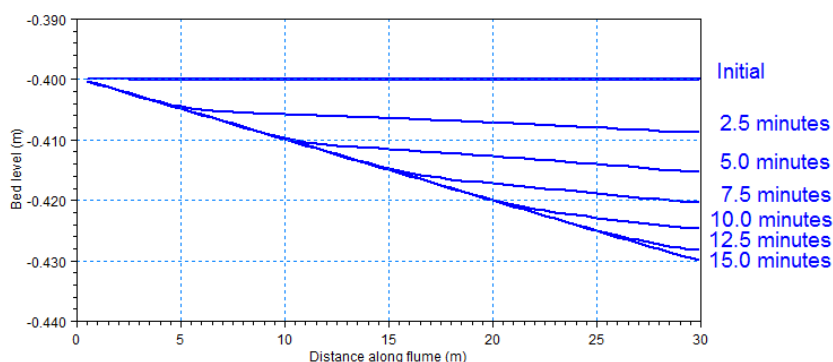
Figure 4.7   Longitudinal profile of the bed of a simulated flume at 6 times during a morphological simulation, showing the development of an equilibrium slope from an initially horizontal bed.

It is seen that the bed level at the downstream point erodes to a little less than 0.03 cm. This is in very good agreement with the theory.

4.3.4   List of data and specification files

The following data files (included in the ST\EqSlope folder) are supplied with MIKE 21 Flow Model ST FM:

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

File name: EqSlope.m21fm
Description: MIKE 21 Flow Model FM specification file

4.4 Sheltering of Jetty

4.4.1 Purpose of the example

This simplified example has been chosen to describe a case of sedimentation by a jetty protecting a natural bay. The sediment transport conditions in the area are influenced by both waves and currents.

The bay is being used as a natural harbour and a jetty has been constructed to shelter the bay from waves from NW. Due to the jetty blocking sediment transport a beach has been formed west of the jetty. The computational domain and bathymetry is shown in Figure 4.8.
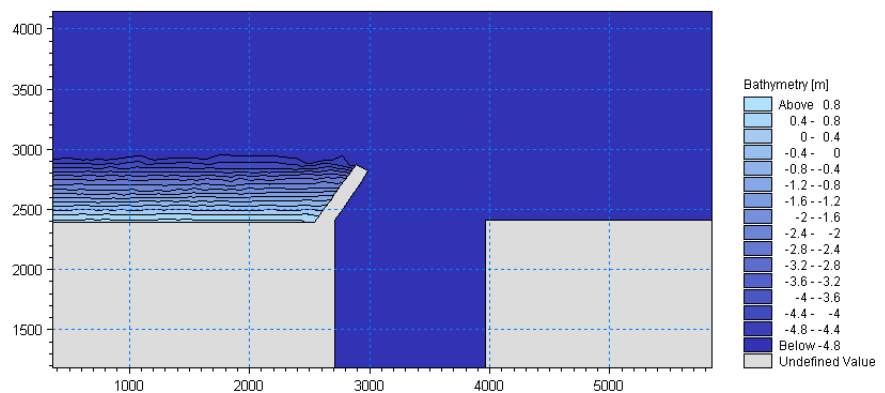
Figure 4.8   Computational domain and bathymetry

The problem is to determine the sedimentation pattern by the jetty and in front of the bay, as the sediment starts to bypass the jetty.

The test conditions are:

- The bay is placed at a constant water depth of 5 m and has vertical walls. The bay area is define by a square with an area of 1.44 km^2.

- A jetty is sticking out into the by-passing flow, sheltering the bay from waves from NW.

- To the left of the jetty a beach exists with gradually increasing water depth.

- The sediment in the area is graded with a mean grain diameter of 0.15 mm and a sediment spreading of 1.3.

- A north-westerly wind is blowing generating waves with a wave height of 1.0 m, wave period of 8 s and wave direction of 320 deg.N at the offshore boundary.

4.4.2   Defining the problem

The main condition defining the hydrodynamic problem is:

- An unstructured mesh with 1302 elements and 726 nodes is used. The mesh is shown in Figure 4.9.

- An overall time step of 600 seconds is selected and the duration of the simulation is 1 day and 7 hours (186 time steps).

- The maximum CFL number is set to 0.9.

- The horizontal eddy viscosity type has been chosen as the Smagorintsky formulation with a constant of 0.28.

- The bed resistance type has been chosen to Manning number and a constant value of 32 $m^{1/3}/s$ is applied.

- The driving force is a radiation stress field generating longshore currents. The radiation stress field driving the currents is generated by the Spectral Wave Module (SW) inducing a wave height of 1.0 m, wave period of 8 s and wave direction of 320 deg.N at the boundary.

- As boundary conditions the water levels on the left and right boundary is set to 0 m, which is the initial setting. The offshore boundary is set to land which means that the flux is 0 across the boundary.

The main condition defining the sand transport problem is:

- The model type is selected as Wave and Current.

- A sediment transport table has been generated that reflects the conditions for the example

- The sediment transport calculations start at timestep 36, which is 6 hours after the start of the simulation. At this time the flow field has almost reached equilibrium.

- The morphological calculations start at timestep 42, which is 7 hours after the start of the simulation.

- No feedback on the hydrodynamic conditions is chosen.

Figure 4.9   Computational mesh

4.4.3   Presenting and evaluating the results

A plot of the flow field is shown in Figure 4.10. The figure show that the maximum currents appear along the sloping beach and an eddy is formed inside the bay.

18:00:00 02-01-2004 Time Step 186 of 186.

Figure 4.10   Contour and vector plot of the flow field.

A plot of the total sand transport is shown in Figure 4.11. The transport is seen to be the largest by the beach.

Figure 4.11   Contour and vector plot of the sand transport field.

A plot of the initial bed level change is shown in Figure 4.12. The bed level changes the most by the beach. However some sand is seen to bypass the jetty and settle in front of and inside the bay.
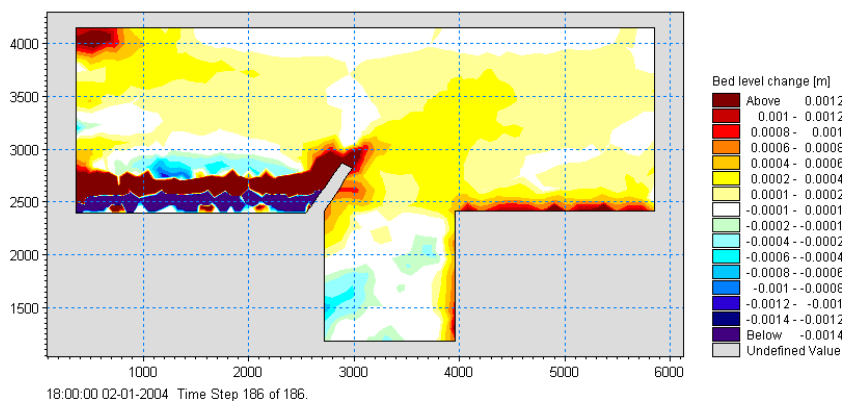
Figure 4.12   Contour plot of initial bed level change after 1 day.

4.4.4   List of data and specification files

The following data files (included in the ST\Shelter folder) are supplied with MIKE 21 Flow Model ST FM:

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

File name: Shelter.m21fm
Description: MIKE 21 Flow Model FM specification file

File Name: Waves.dfsu
Description: Wave properties and radiation stresses for the area

File Name: QsTable.lon & QsTable.crs
Description: Sediment transport tables

4.5   Harbour Example

4.5.1   Purpose of the example

This example illustrate the wave-driven current and littoral transport patterns in the vicinity of a harbour located on a sandy coast.

The computational domain and bathymetry is shown in Figure 4.13.
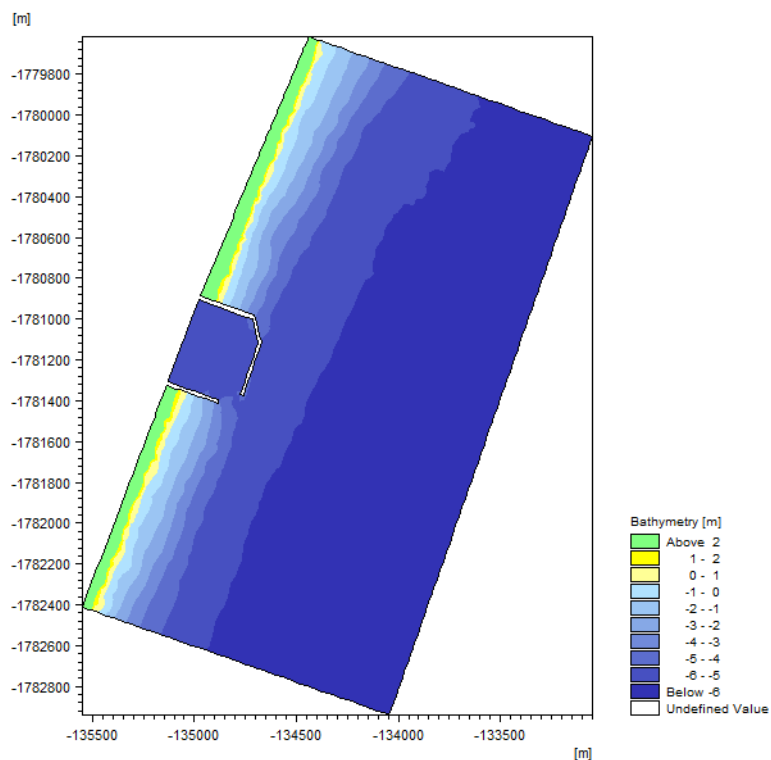
Figure 4.13   Computational domain and bathymetry

The problem is to determine the sedimentation pattern along the coast.

The test conditions are:

- The coastline normal is oriented approximately 115 deg.N.

- The sediment in the area is graded with a mean grain diameter of 0.22 mm and a sediment spreading of 1.25.

- Irregular, directional waves propagate towards the coast from 90 deg.N over a water level corresponding to MWL = 1.0 m. The wave characteristics at the offshore boundary of the model are H_{m0}= 2 m and T_p= 10 s.

4.5.2 Defining the problem

The main condition defining the hydrodynamic problem is:

- An unstructured mesh with 4378 elements and 2346 nodes is used. The mesh is shown in Figure 4.14.

- An overall time step of 60 seconds is selected and the duration of the simulation is 5 hours (300 time steps).

- The horizontal eddy viscosity type has been chosen as a constant value of 1.0 m^2/s.

- The bed resistance type has been chosen to Manning number and a constant value of 32 $m^{1/3}$/s is applied.

- The driving force is a radiation stress field generating longshore currents. The radiation stress field driving the currents is generated by the Spectral Wave Module (SW).Note that the wave conditions inside the harbour and just downstream in the vicinity of the harbour is not reliable, as SW does not consider diffraction in the calculation.

- The offshore boundary is set to land which means that the flux is 0 across the boundary whereas the updrift and downdrift boundaries is defined by the water level 1.0 m, which is the initial setting. At both updrift and downdrift boundary radiation stress correction is included.

The main condition defining the sand transport problem is:

- The model type is selected as Wave and Current.

- A sediment transport table has been generated that reflects the conditions for the example.

- The sediment transport calculations start after 3 hours (at timestep 180) when a steady flow field is obtained.
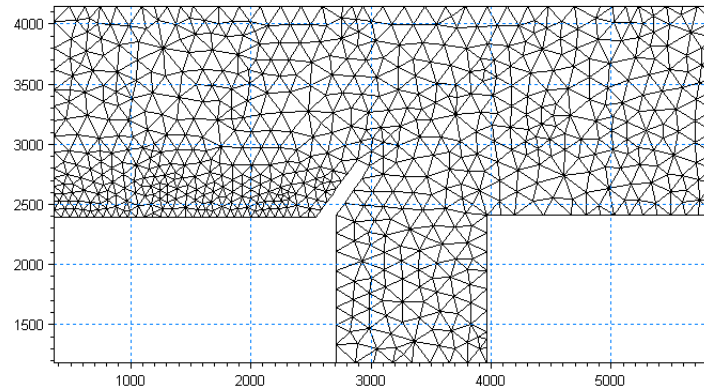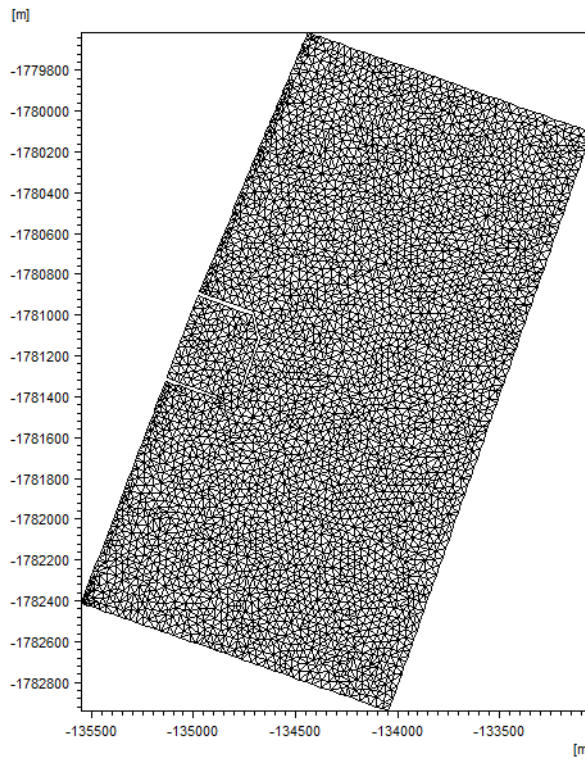
Figure 4.14   Computational mesh

4.5.3   Presenting and evaluating the results

A plot of the flow field is shown in Figure 4.15. The figure show that the maximum currents appear along the coast.
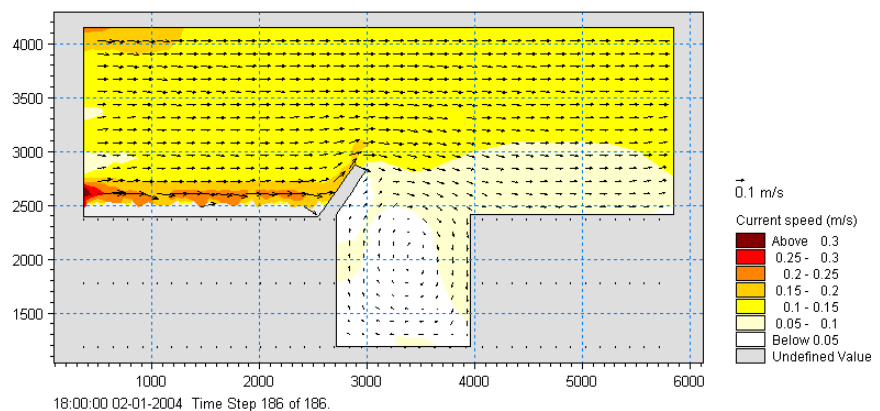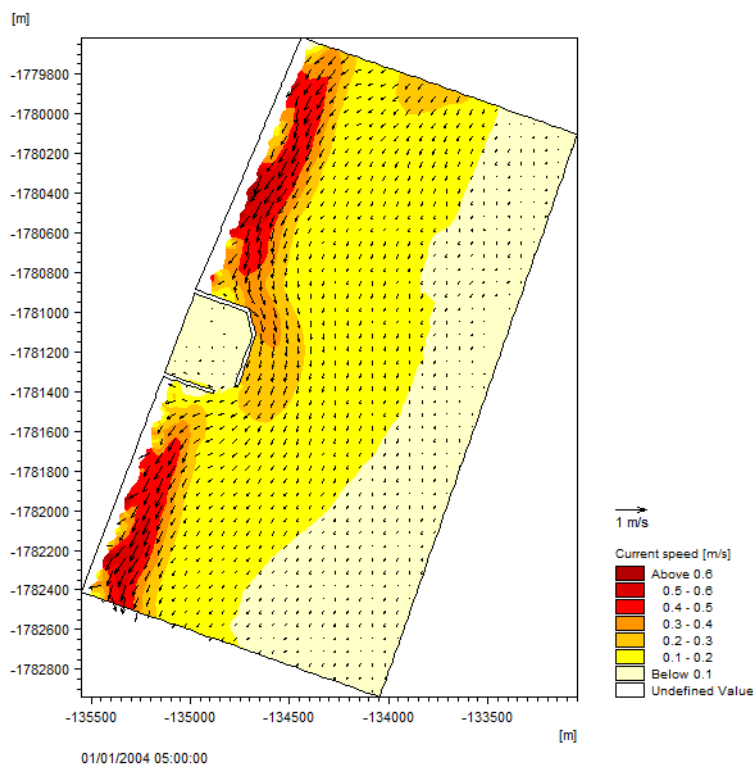
Figure 4.15   Contour and vector plot of the flow field.

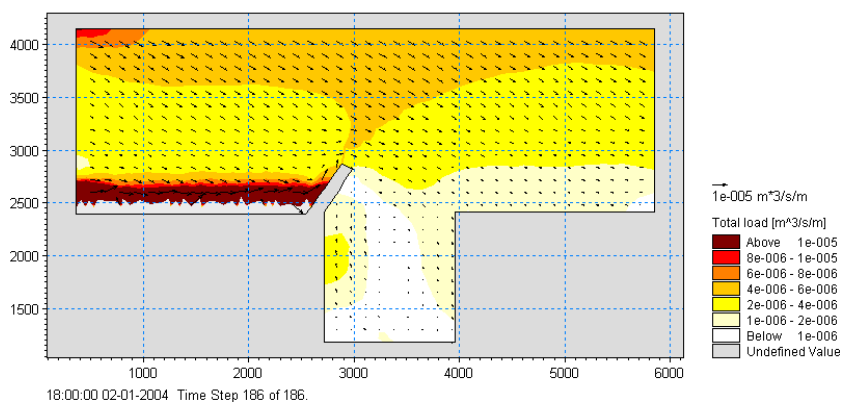A plot of the total sand transport is shown in Figure 4.16. The transport is seen to be the largest by the beach. At the harbour some sand transport is directed along the jetty. This sediment will initially settle outside the jetty but in time there is a risk that the sand will settle in front of the entrance of the harbour.
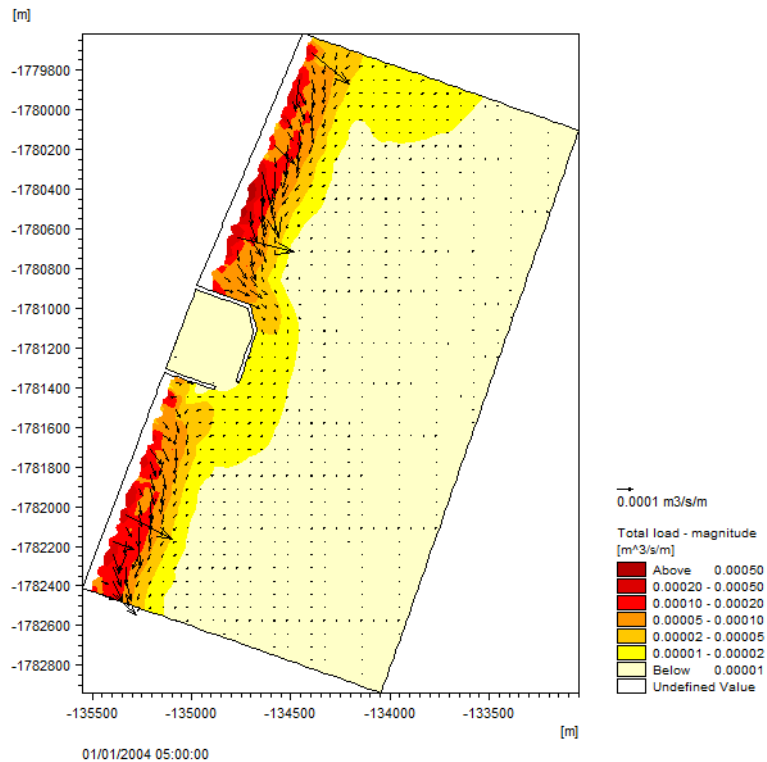
Figure 4.16   Contour and vector plot of the sand transport field.

A plot of the initial bed level change rate at the harbour is shown in Figure 4.17. The initial accumulation of sand will mostly appear upstream the harbour but some sedimentation is seen at the harbour entrance too. Just downdrift the harbour the bed will erode.
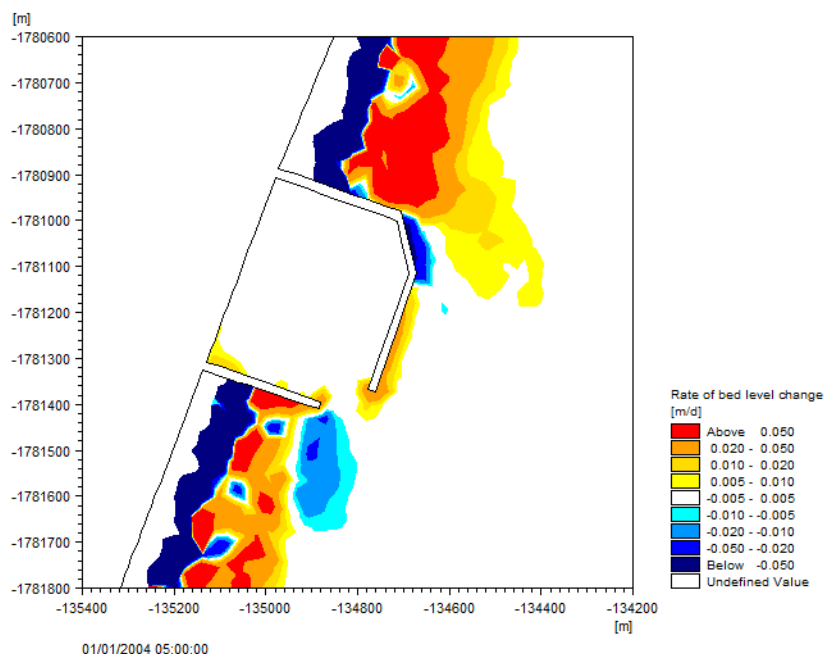
Figure 4.17   Contour plot of initial bed level change rate.

The same domain is also used in the Harbour example in the Coupled Model FM User Guide. The objectives of this example is to use an integrated MIKE 21/3 Flow Model FM setup to investigate the various options available in morphological simulations, e.g. the basic morphology calculations included in the Sand Transport FM module versus the Shoreline Morphology module.

The description of this example is included in the user guide for the Coupled Model. This is provided via the Documentation Index for MIKE 21:

MIKE 21/3 Coupled Model FM, User Guide

4.5.4   List of data and specification files

The following data files (included in the ST\Harbour folder) are supplied with MIKE 21 Flow Model ST FM:

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

File name: Harbour.m21fm
Description: MIKE 21 Flow Model FM specification file

File Name: Waves.dfsu
Description: Wave properties and radiation stresses for the area

File Name: TransportTable.lon & TransportTable.crs
Description: Sediment transport tables

4.6 Tidal Inlet Stability

4.6.1 Purpose of the example

This simplified example has been chosen to describe the influence of bank erosion due to slope failure on the morphological development of a tidal inlet.

The model covers a rectangular area of 46 m x 94 m. The ocean part has a maximum depth of 1 m that decrease linearly towards the inlet by a slope of 1:3 to the depth of 18.3 cm. From here the bathymetry is described by a stable Dean profile.The channel of the inlet is initially about 1 m long, 60 cm wide at mean tide level with side slopes of 1:1.4 and has a constant water depth of 13.2 cm. The bay area is 2700 m^2 with the same water depth as in the channel and vertical walls.

The computational domain and bathymetry is shown in Figure 4.18.
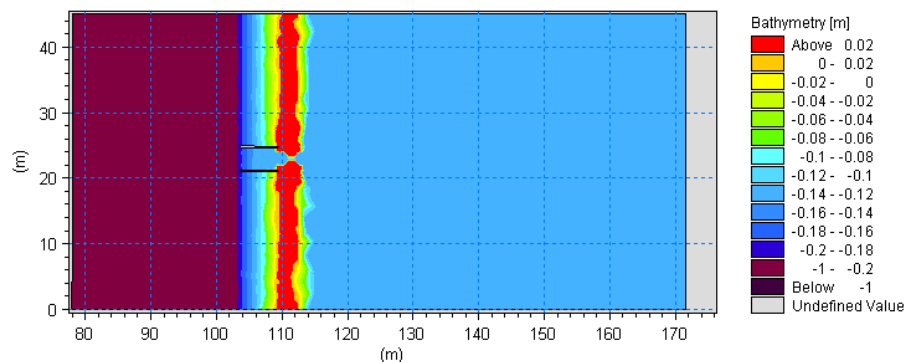
Figure 4.18   Computational domain and bathymetry

The problem is to simulate the morphological development of the inlet channel cross-section, when including the effect of bank erosion.

The test conditions are:

- The bathymetry is as described above. Two structures have been placed on the ocean side of the inlet to direct the flow.

- The sediment in the area is uniform with a constant grain diameter of 0.13 mm and a constant layer thickness.

- The water level is constant along the boundary and varies sinusoidal with a tidal range of 3.05 cm and a tidal period of 105.4 minutes.

4.6.2 Defining the problem

The main condition defining the hydrodynamic problem is:

- An unstructured mesh with 4263 elements and 2205 nodes is used. The size of the elements goes from 10 m^2 in the ocean to 0.002 m^2 in the inlet channel. The mesh is shown in Figure 4.19.

- An overall time step of 31 seconds is selected and the duration of the simulation is about 10.5 hours (1225 time steps) corresponding to 6 tidal periods.

- A low order solution technique is used, with a minimum time step of 0.001 seconds, a maximum time step of 31 seconds and a critical CFL number of 0.8.

- Flood and dry is enabled. The drying depth is set to 0.001 m, the flooding depth to 0.005 and the wetting depth to 0.01 m, respectively

- The horizontal eddy viscosity type has been chosen as the Smagorintsky formulation with a constant of 0.28.

- The bed resistance type has been defined as a Manning number M with a constant value of 60 $m^{1/3}$/s.

- Initially, the surface elevation is set to 0.03 m for the entire area as the simulation is started at high tide.

- The boundary is defined by a time varying water level with a tidal period of 105.4 minutes, a tidal range of 3.05 cm and a minumum water level of 0 m. A soft start of 60 seconds is used.
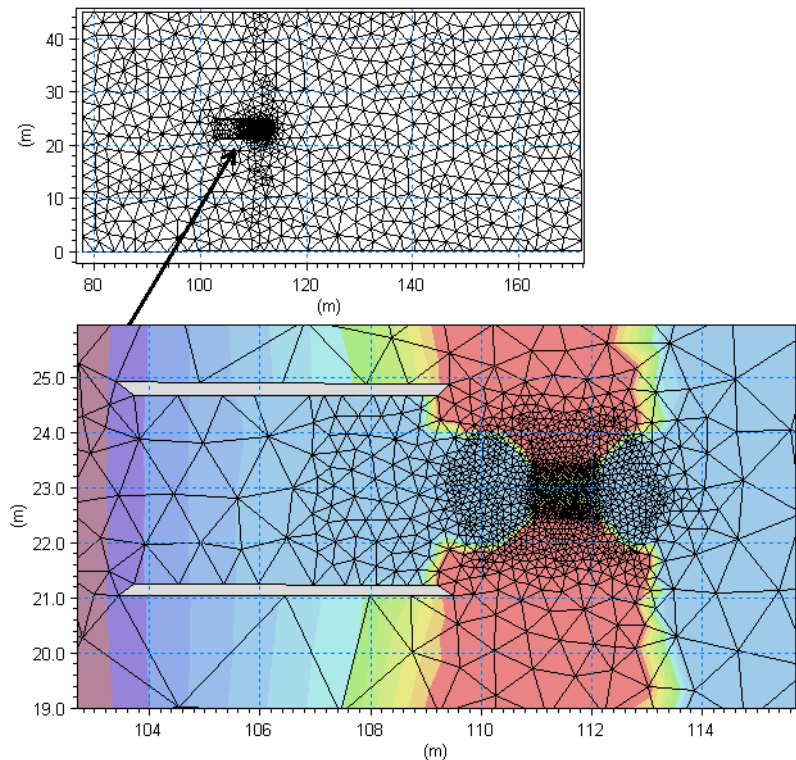
Figure 4.19   Computational mesh

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. The bed load transport and suspended load transport is weighted with 0.25 and 0.75, respectively.

- The sediment transport calculations start at overall timestep 102, which is one half tidal period after the start of the hydrodynamic calculations.

- The porosity is set to 0.4 and the relative density to 2.65. The bed consist of grains with d50=0.13 mm.

- The bed resistance type for the sediment transport calculations is taken from the hydrodynamic model.

- The morphological update is including feedback on the hydrodynamic conditions. The speedup factor is defined as 1. The maximum bed level change is set to 100 m/day.

- The morphological calculations start at overall timestep 204, which is one tidal period after the start of the hydrodynamic calculations.

- The slope failure type is defined as 'Simple bank erosion'. The Angle of repose is set to 25 deg.

4.6.3   Presenting and evaluating the results

The flow in the channel is ruled by the difference in surface elevation between ocean and bay. A plot of the flow field at flood tide is shown in Figure 4.20. It can be noticed that the maximum velocity is situated in the middle of the channel.
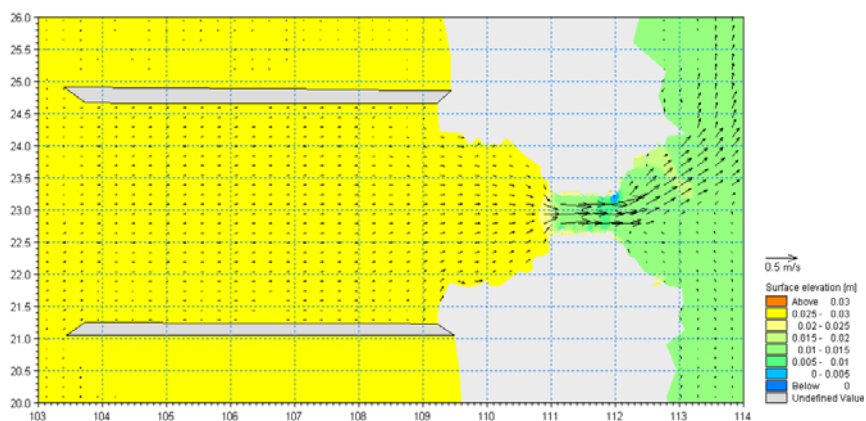
Figure 4.20   Surface elevation and current speed (magnitude and direction) at mid flood tide.

Figure 4.21 shows the sediment transport field at mid flood tide.
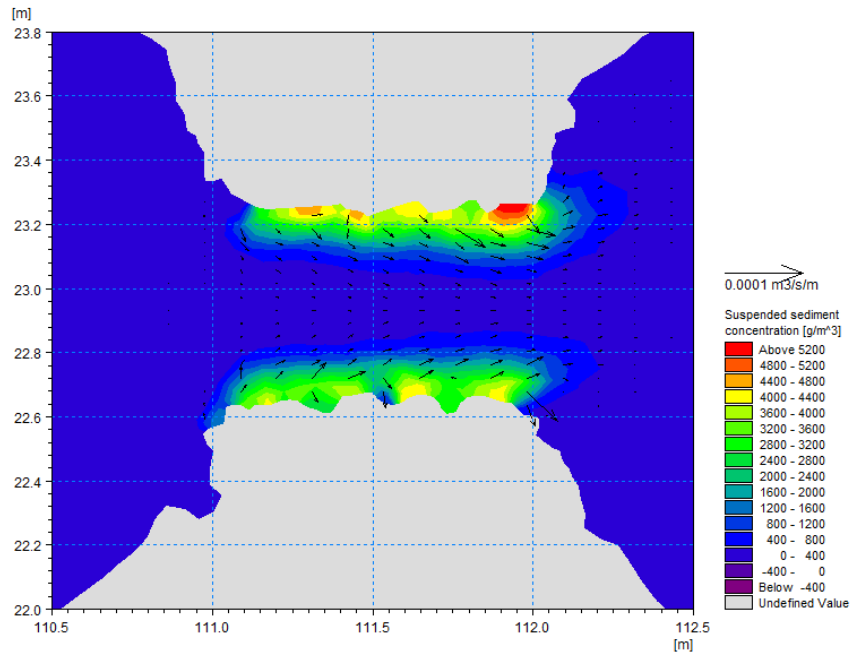
Figure 4.21   Total load transport at initial bathymetry and mid ebb tide

The increase of current speed in the narrow part of the channel will in turn result in an increase of sediment transport capacity, resulting in erosion in the area of acceleration. The cross-section area of the channel will start to increase and, if possible, the banks will erode. If the banks cannot erode, the dry land will remain fixed and the channel will deepen.

The bathymetry development for cross-section C, located at the middle of the channel, is shown in Figure 4.22. The result of a simulation where bank erosion has been omitted has been included for comparison.
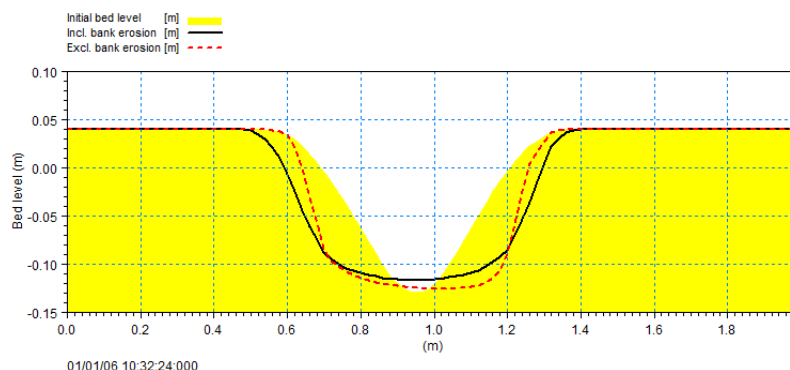
Figure 4.22   Inlet cross-section bathymetry after 5 tidal cykles.
Yellow filled area: initial bathymetry
Black solid line: simulated bathymetry with slope failure
Red dotted line: simulated bathymetry without slope failure

4.6.4   List of data and specification files

The following data files (included in the ST\Inlet folder) are supplied with MIKE 21 Flow Model ST FM:

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

File name: Waterlevel.dfs0
Description: Water level variation at the boundary

File name: Sim.m21fm
Description: MIKE 21 Flow Model FM specification file

4.7   Sand Transport Modelling for the Ganges and Gorai River

4.7.1   Purpose of the example

The objective of this example is to set up a morphological sand transport model based on MIKE 21 Flow model FM for the bifurcation of the Ganges and Gorai river, Bangladesh. The model is made to show how the morphological model can be applied to simulate the closure of the Gorai offtake.

An easy to follow Step-by-step guide is provided via the Documentation index for MIKE 21:

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

4.8 Sand Transport Modelling by Torsminde, Denmark

4.8.1 Purpose of the example

The objectives of this example is to setup a MIKE 21 Flow Model FM for Torsminde, Denmark, from scratch and to calibrate the model to a satisfactory level.

An easy to follow Step-by-step guide is provided via the Documentation index for MIKE 21:

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

4.9 Morphology Modelling by Torsminde, Denmark

4.9.1 Purpose of the example

The objectives of this example is to setup an Integrated MIKE 21/3 Flow Model FM for Torsminde, Denmark, from scratch and to calibrate the model to a satisfactory level.

An easy to follow Step-by-step guide is provided via the Documentation index for MIKE 21:

MIKE 21/3 Coupled Model FM,
Step-by-Step Training Guide: Coastal Application

4.10 Impact of Pier

4.10.1 Purpose of the example

The objectives of this example is to use an integrated MIKE 21/3 Flow Model FM setup to investigate the difference in results for two alternate methods: the basic morphology calculations included in the Sand Transport FM module versus the Shoreline Morphology module.

The description of this example is included in the user guide for the coupled model. This is provided via the Documentation Index for MIKE 21:

MIKE 21/3 Coupled Model FM, User Guide

4.11 Shoreline Morphology Modelling, Harbour

4.11.1 Purpose of the example

This example is similar to the Harbour Example, however this use an integrated MIKE 21/3 Flow Model FM setup to calculate the wave-driven current and littoral transport patterns in the vicinity of a harbour located on a sandy coast. The example illustrates the influence on the results of updating the bed during the simulation and shows how to combine a Shoreline model with basic 2D morphology. Furthermore the effect of including bed level sources or dune erosion is investigated.

The description of this example is included in the user guide for the coupled model. This is provided via the Documentation Index for MIKE 21:

MIKE 21/3 Coupled Model FM, User Guide

4.12 Shoreline Morphology Modelling, Headland beach

4.12.1 Purpose of the example

The objectives of this example is to setup a Shoreline Morphology model for a headland beach, from scratch and to calibrate the model to a satisfactory level.

An easy to follow Step-by-step guide is provided via the Documentation index for MIKE 21:

MIKE 21/3 Coupled Model FM, Shoreline Morphology
Step-by-Step Training Guide: Headland Beach example

5 SAND TRANSPORT MODULE

The Sand Transport Module calculates the resulting transport of non-cohesive materials based on the 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 three different model types:

- Pure current
- Pure current - Multi fraction/Multi layer
- Waves and current

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

For more details on the three model types, see the scientific documentation.

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 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 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.

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

Layer thickness

You may choose to include a varying layer thickness as start condition for the simulation (see page 56). 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.

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

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.

5.1.3 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.

5.1.4 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 56). This option is useful when simulating sand trans-

port 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.

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

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	

5.2   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) equations is performed using an explicit scheme. 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" 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.

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:

$$C = a \cdot H^b \tag{5.3}$$

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).
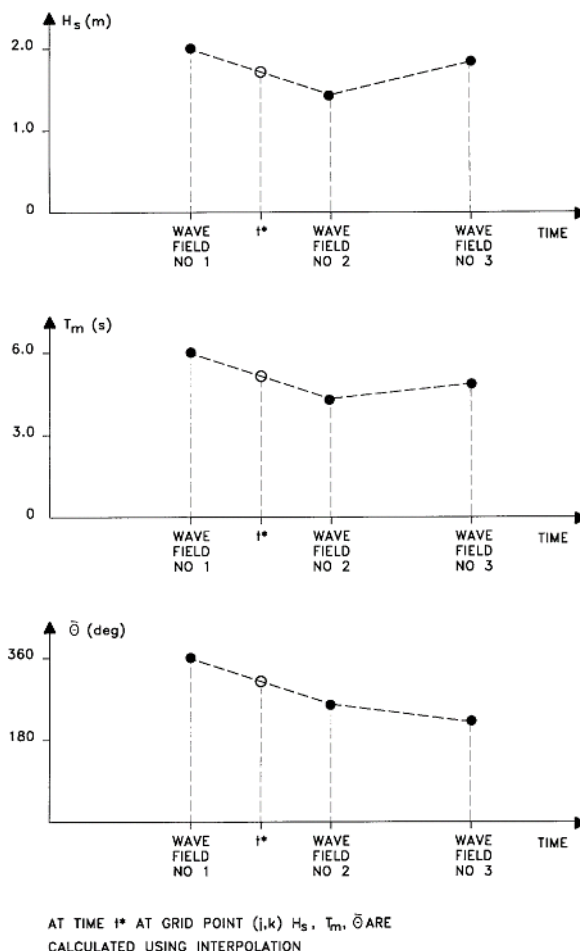
Figure 5.1   Interpolation of wave fields in time

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.

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

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, 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.

5.7.2 Recommended values

When more sophisticated eddy viscosity models are used, as the Smagorinsky or model, 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 $Q_{source} \cdot C_{source}$ 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 41) determines the necessary initial conditions. When "Pure current" is selected for the model type and Non-equilibrium is selected for the transport description 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

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.

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 piece-wise 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. 60)*).

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

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).

Interpolation type

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

- linear
- piece wise cubic

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

- normal
- reverse order.

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

Soft start interval

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

5.11 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 section 7 MORPHOLOGY MODULE.

5.12 Helical Flow Module

The helical flow (HE) module is included provided the module is selected in the Model Definition dialog.

A more detailed description of the Helical Flow module can be found in section 6 HELICAL FLOW MODULE.

5.13 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.

5.13.1 Geographic view

This shows the geographical position of the output data.

5.13.2 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 2D simulation 2D field parameters can be selected. 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, 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 val-

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

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

Output file

A name and location of the output file must be specified along with the file type. The file type depends on the Output format and the Data type as shown in Table 5.2.

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
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

Table 5.3   List of tools for viewing, editing and plotting results

File type	Viewing/editing tools	Plotting tools
dfs2	Grid Series Editor Data Viewer	Plot Composer Result Viewer Data 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 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

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). 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.

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.

5.13.3 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.41).

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 (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, 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, 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.

Coastline

When selecting output for the coastline you can select basic output variables and additional output variables.

The basic output variables are

- Coastline X

- Coastline Y

- Accumulated shoreline change

- Deposited volume

- Littoral drift

- Offshore transport

The extended output variables are

- Net offshore transport

- Net littoral drift

- Gross littoral drift

- Vertical area

- Shoreline change

- Actual deposited volume

- Dune foot x coordinate

- Dune foot y coordinate

- Dune foot z coordinate

- Dune erosion volume

- Dune volume

Littoral drift rates are defined positive along the coastline (i.e. in the direction of the x-axis of the dfs1 file).

NOTE: output of coastline variables is only available when using the Shoreline Morphology module in MIKE 21/3 Coupled Model FM. Output of extended output dune variables is only available when using the Parameterized Dune Erosion option.

6 HELICAL FLOW MODULE

The helical flow (HE) module is invoked provided helical flow is selected in the Sand Transport model (see Model Definition).

Helical flow is a secondary flow phenomenon in rivers. Whilst it generally does not have a strong influence on general flow patterns in rivers with large width/depth ratios, it can have a significant influence on sediment transport direction and hence morphological changes. Helical flow is the driving force for bend scour and plays an important role in confluence scour and char build up and migration.
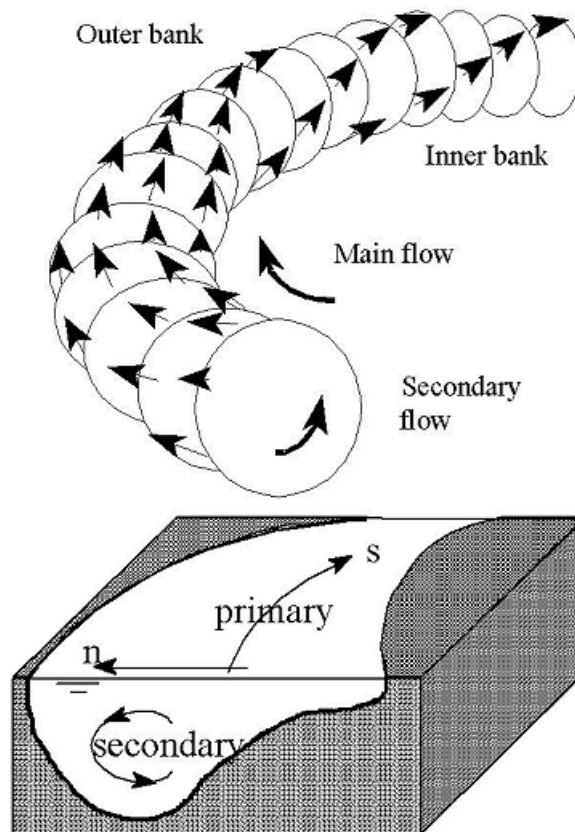

Figure 6.1   Illustration of helical flow

6.1 General Description

The helical flow pattern occurs in regions of curved flow, particularly river bends. It arises from an imbalance between pressure gradient and centripetal acceleration exerted on water moving along a curved path. Near the bed, helical flow is directed towards the centre of flow curvature. The magnitude of

helical flow (the transverse flow velocity component) rarely exceeds 5-10% of the main flow velocity in natural rivers.

6.1.1 Helical flow theory

The magnitude of the equilibrium secondary flow is proportional to flow depth and inversely proportional to the radius of curvature of the main stream line. This has been defined by de Vriend (1981):

$$i_s = u \cdot \frac{h}{R_s} \qquad (6.1)$$

where:
u is main flow velocity
R_s is radius of curvature of streamlines
i_s is helical flow intensity

This yields a bed shear stress direction given by

$$\tan \delta_s = -\beta \cdot \frac{h}{R_s} \qquad (6.2)$$

where δ_s is the angle between the bed shear stress and depth averaged shear stress (or flow), see Figure 6.2.
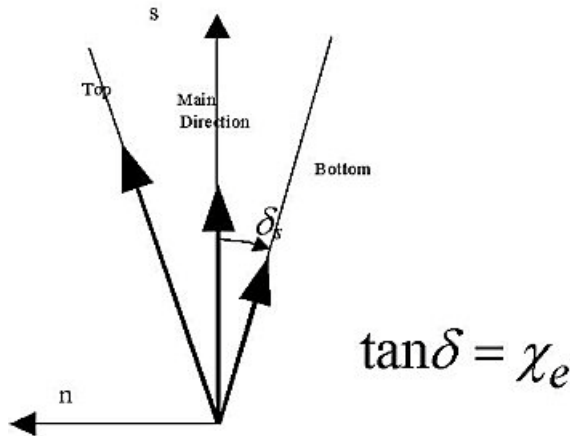

Figure 6.2   Definition of flow direction at the bed

β is defined as:

$$\beta = \alpha \cdot \frac{2}{\kappa^2}\left(1 - \frac{\sqrt{g}}{\kappa C}\right) \qquad (6.3)$$

where
κ is Karman constant, 0.4
g is gravity, 9.81 m/s^2
C is Chezy number and
α is a calibration constant (model input)

6.1.2 Recommended values

Theoretical values of β can be obtained from theoretical velocity profiles. They exhibit some scatter depending on what kind of profile is assumed. Measurement of the near bed flow direction is almost impossible due to migrating bed forms. The best value of β is obtained by calibration of the morphological model. According to Olesen (1987), β is in the range 4-12. This means that the model input parameter α is in the range 0.4 to 1.2 for a Chezy roughness of 40 m$^{1/2}$/s. α can be specified as a global constant or as a two-dimensional varying parameter.

6.1.3 Remarks and hints

As mentioned above, the effect of the helical flow is important in simulating sediment movement and morphology in river bends. This can be tested by excluding the helical sub-model from the simulation.

If a model region undergoes wetting and drying during a simulation, instability of the helical flow model can occur due to the rapid changes in stream line curvature. Helical flow in shallow areas is rarely important for morphological development. As such, in specific grid points where instabilities occur and where helical flow is unimportant it is recommended that a is set to zero.

6.2 Time Parameters

The helical flow calculation is activated at a Start time step relative to the start of the simulation specified on the Time dialog.

6.3 Solution Technique

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

The time integration of the transport (advection-dispersion) equations is performed using an explicit scheme. Due to the stability restriction using an explicit scheme the time step interval must be selected so that the CFL num-

ber 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.

6.3.1 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 Courant number. It must be in the range from 0 to 1. Alternatively, you can reduce the maximum time step interval. Note, that setting the minimum and maximum time step interval equal to the overall time step interval specified on the Time dialog, the time integration will be performed with constant time step. In this case the time step interval should be selected so the the CFL number is smaller than 1.

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

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

6.4 Initial Conditions

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

6.4.1 Deviation

Here you can either specify the initial concentration or assume the deviation to be in equilibrium condition.

Data

The format of the deviation 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 deviation (in component unit) before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). If a dfsu-file is used the mesh in the data file must match exactly the mesh in the simulation. If a dfs2-file is used the area in the data file must cover the model area. 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.

6.5 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 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 four boundary types

- Land

- Equilibrium conditions

- Specified values

- Zero gradient

Data

If specified values is selected the format of the deviation 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 fraction concentration before you set up the hydrodynamic simulation. 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).

7 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.

7.1 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.

$$Z^{n+1} = Z^n + \Delta z^n \tag{7.1}$$

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.

$$Z^{n+1} = Z^n + \Delta z^n \cdot Speedup \tag{7.2}$$

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.

7.2 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 de-activating 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).

7.2.1 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).

7.3   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.

7.4   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.

7.4.1   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.

7.4.2   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.

7.5 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.

7.5.1 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

7.6 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.

7.6.1 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.

8 SHORELINE MORPHOLOGY MODULE

The Shoreline Morphology Module combines a one-line model for the shoreline with a 2D description of the wave, current and sediment transport field. This model is an alternative morphological model within the Sand Transport module when applying this within the MIKE 21/3 Coupled Model FM in two dimensions.

Without the Shoreline Morphology model, the morphology in the Sand Transport Module simply updates the bed level in each mesh element according to the local sediment continuity equation. This could lead to instabilities for long-term simulations.

With the Shoreline Morphology model, the effect of the cross-shore transport within the shoreface is eliminated from the morphology calculations by integrating the sediment volume deposited during each time step across the shoreface and updating the morphology according to the continuity equation for the shoreline using a predefined coastal profile. Thereby the effect of the cross-shore transport is eliminated from the morphology allowing for long term simulations.

8.1 General Description

The model solves a modified version of the one-line equation for the shoreline (see Kaergaard and Fredsoe, 2013)

$$\frac{\Delta n}{\Delta t} = -\frac{vol}{dA_z} \tag{8.1}$$

where Δn is the distance the shoreline is moved perpendicular to the local shoreline orientation, Δt is the time step, vol is the volume of sediment deposited on the shoreface in front of the shoreline edge and dA_z is the area of the strip of shoreface in front of the shoreline edge projected onto the plane defined by the shoreline edge and the vertical axis, see Figure 8.1.
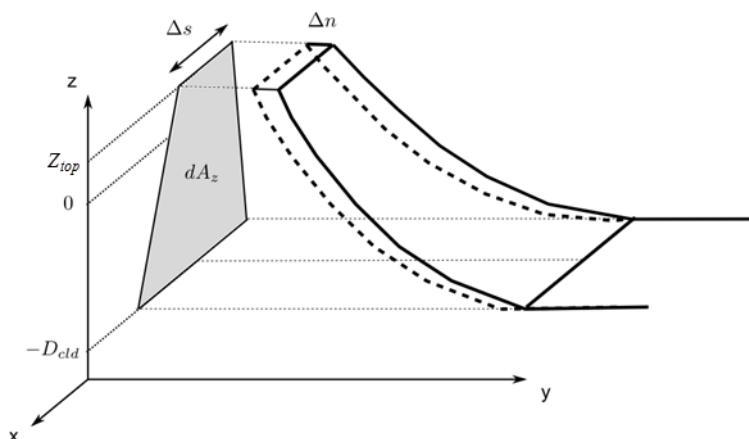
Figure 8.1   Sketch of a strip of shoreface with definition of model parameters

dA_z is the area over which the deposited sediment volume is distributed, thereby giving the movement of the shoreline. If the shoreline is curved, using the above equation will induce some sediment conservation problems; these are removed using an iterative procedure. The direction the shoreline is moved is perpendicular to the local orientation of a baseline. The baseline is defined by a set of points forming a polyline; see Figure 8.2 for an example.

The position of the baseline points is presently fixed and the local orientation of the baseline is defined by the user during model setup.

The shoreline model can be applied to problems over a longer time scale. This is due to the simplifications imposed on the morphological evolution of the coastal profile, when compared with the existing morphological model for sand transport in MIKE 21/3 Coupled FM for 2 dimensions. The shoreline evolution is based directly on the calculated sediment transport field from the area model MIKE 21 ST FM. The latter calculates the transport of non-cohesive sediments due to the action of waves and currents. Effects from coastal structures on shoreline evolution are inherently included in the new shoreline model because their effect on the waves and flow are included in the underlying models for wave transformation and hydrodynamics. Effects from dune erosion on shoreline evolution, even if the shoreline has not eroded all the way back, can be included via a special dune erosion option.

8.2   Shoreline Model

You need to define various input for the Shoreline model.

The graphical overview indicates the defined location and layout of the various spatial shoreline input data used in the simulation.

NOTE:
- The shoreline model can only be applied for domains defined in projected

coordinate systems

- (.xy) or (.xyz) input files must be provided in metric unit [m]

8.3　Solution Technique

The accuracy of the simulation of morphological changes can be controlled by specifying the order of the tolerance and the relaxation factor used in the numerical calculations as well as the maximum number of iterations.

You can also control if the volume of sediment transported across the off-shore boundary is to be included in the morphology calculations.

Finally, you can select between two types of morphological calculations:

- Only shoreline morphology
- Combined 2D and shoreline morphology (defined by Edge Map)
- Combined 2D and shoreline morphology (defined by Active Profile)

The latter two cases are a mixture of shoreline morphology for the coastline and 2D morphology in the surrounding area. I.e. morphological changes in elements that are NOT associated with a shoreline edge (via the Edge map)/ are outside the active profile are calculated using the standard 2D morphology method.

8.4　Baseline

The baseline determines the direction in which the shoreline can move during development by the local orientation of the line.

The baseline is presently fixed in time.

You define a baseline by an ascii file with extension xy (or .xyz) containing the coordinates of the baseline, and the map projection by which the coordinates are referring to.

8.4.1　General description

The baseline coordinates is used for deriving the local orientation by which the shoreline can move on/offshore during the simulation.

The baseline should be located behind the coastline and the baseline points must be defined by increasing order by the positive distance from the origin of the line, such that the seawards side is to the left.
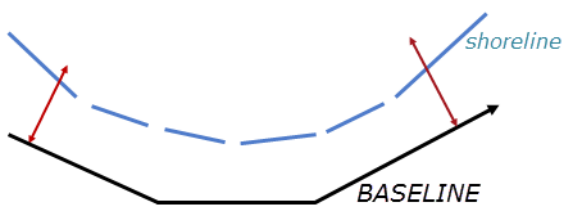
Figure 8.2   Sketch of baseline with indication of shoreline movements

8.4.2   Baseline specification

You have to prepare an ascii file with extension .xy (or .xyz) containing the geographical coordinates of the baseline. The file format is two (or three) space separated floats (real numbers) for the x- and y-coordinate on separate lines for each of the points (in case of three columns in the file x and y is derived from the first two columns).

You must also select the map projection in which you have specified the geographical coordinates.

NOTE: (.xy) or (.xyz) input files must be provided in metric unit [m]

8.4.3   Remarks and hints

The points on the baseline determines the resolution of coastline points as the points on the initial Coastline must be correlated with the points on the baseline such that the inner points on the coastline are defined in the middle of a baseline section.
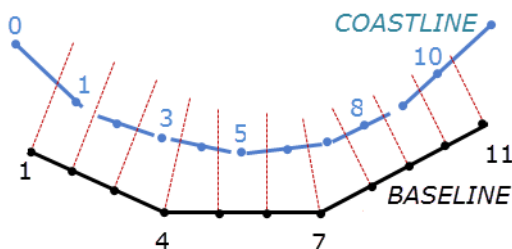
Figure 8.3   Sketch of relation between baseline points and coastline points

8.5 Coastline

The coastline defines the initial position of the local shoreline subject to erosion and deposition. The grid points along the coastline each represent a shoreline edge that can move on/off shore.

8.5.1 General description

The coastline defines the initial position of the shoreline.

The coastline should be located on the seawards side of the baseline and the coastline points must be defined by increasing order by the positive distance from the origin of the line, such that the seawards side is to the left.

Figure 8.4   Sketch of coastline positioned in front of baseline

8.5.2 Coastline specification

The coastline can be specified in one of two ways

- xy(z) file
- dfs1 file

For the last case the dfs1 file can be the result from a previous simulation in which case the coastline effectively acts as a hot start.

Data

In the case where the coastline is specified by a xy file you have to prepare an ascii file with extension .xy (or .xyz) containing the geographical coordinates of the coastline. The file format is two (or three) space separated floats (real numbers) for the x- and y-coordinate on separate lines for each of the points (in case of three columns in the file x and y are derived from the first two columns). You must also select the map projection in which you have specified the geographical coordinates.

NOTE: (.xy) or (.xyz) input files must be provided in metric unit [m]

In the case where the coastline is specified by a dfs1 file you have to prepare a profile file (dfs1) containing the geographical coordinates of the coastline. The x- and y-coordinate will per default be derived from item 1 and item 2, respectively, from the first time step in the file.

8.5.3 Remarks and hints

The points on the coastline must be correlated with the points on the baseline such that the inner points on the coastline are defined in the middle of a baseline section. The coastline must be defined by one, and only one, more point than the baseline.
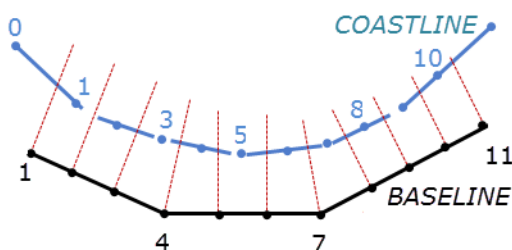
Figure 8.5   Sketch of relation between baseline points and coastline points

8.6 Edge map

The edge map defines which shoreline edge each element in the mesh belongs to. The sediment volume, deposited on the defined strip of shoreline and within the active profile, will contribute to the movement of the shoreline edge.

8.6.1 General description

The edge map defines which shoreline edge each mesh element belongs to. The values in the file will correlate to the point numbers/CoastID on the coastline.

The sediment volume, deposited on the defined strip of shoreline and within the active profile, will contribute to the movement of the shoreline edge. The edge map can contain zeros; in that case those elements do not belong to any shoreline edge and will not contribute to the movement of the shoreline.
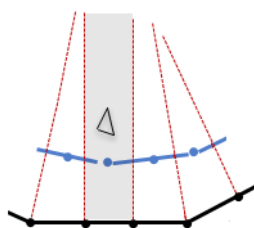
Figure 8.6   Sketch of mesh element and strips of shoreline edges. Elements in shaded area for CoastID 5 will hold the value 5

8.6.2   Edge map specification

You have to prepare a data file containing the grid code values by which the elements are assigned to the coastline points before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu). The area in the data file must cover the model area and preferably match the mesh resolution in that same area. Otherwise piecewise constant interpolation is used to map the data.

8.7   Profile

The profile defines a representative cross-shore profile which moves forth and back with the shoreline edge at the given location. The profile can vary along the coastline.

8.7.1   General description

The profile acts as a representative cross-shore profile which moves forth and back with the shoreline edge at the given location. A profile is defined by the distance and z-level along a line, going through the coastline point and perpendicular to the related baseline section. The first point in a profile is defined at dry land (negative distance and positive z-level) and the profile is directed seawards ending with a point described by positive distance and negative z-level.

Figure 8.7   Sketch of profile alignment with respect to coastline and baseline

When the shoreline erodes, the z-level in the 2D domain will decrease accordingly including any section of the profile that is above the defined dune height, but not below the defined closure depth.

When the shoreline experience accretion, the z-level in the 2D domain will increase accordingly including any section that is below the defined closure depth, but not above the defined top level.
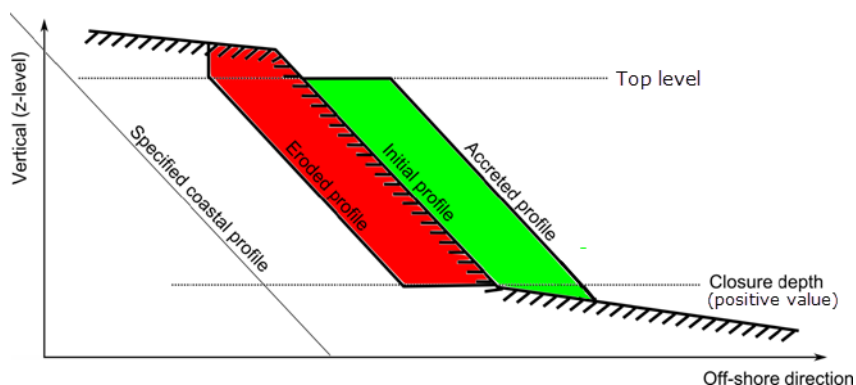
Figure 8.8   Sketch of profile with influence of profile parameters during morphological update

8.7.2   Profile specification

The cross-shore profile bathymetry can be specified in one of two file formats:

- Xy file
- Dfs2 file

When using a xy file, the same cross-shore profile is applied along the coastline during the simulation.

When using a dfs2 file the dfs2 file must contain profile information for all the cross-shore profiles. The cross-shore profile may vary along the coastline.

Per default the individual cross-shore profiles are kept constant during the simulation, however it is possible to apply pre-defined time varying profiles by specifying the profiles in a time-varying dfs2 file.

The Closure depth describes the maximum water depth below which the profile can increase, but not erode.

The Top level describes the bathymetry level over which the profile can erode, but not accumulate.

Data

In the case where the profile is specified by a xy file you have to prepare an ascii file with extension .xy containing the distance from the shoreline and z-level for all points in the cross-shore profile. The file format is two (or three) space separated floats (real numbers) for the distance and z-level on separate lines for each of the points (in case of three columns in the file distance and z-level are derived from the first two columns). The measuring unit of the distance and z-level is the same as the unit defined for the mesh projection.

In the case where the profile data is specified by a dfs2 file you have to prepare a grid series file (dfs2) containing items defining the distance from the shoreline and the z-level for all points in the cross-shore profile and along the coastline. The individual profile is defined along the x-axis and the given shoreline points along the y-axis. The number of points on the x-axis of the dfs2 file must correspond to the number of points in a profile, and the number of points on the y-axis must correspond to the number of points on the coastline-2.
In case the profiles vary 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.

8.8 Initial Conditions

It is possible to continue the shoreline evolution from a previous simulation by specifying the coastline by a dfs1 file containing the resulting coastline position.

Besides the coastline position, it is possible to apply initial conditions for the accumulated output. This way the accumulated output from the previous simulation can be accounted for in the output of the new simulation.

8.8.1 Specifications

There are three types of initial conditions

- Zero condition

- Accumulated coastline
- Accumulated coastline and transport

Data

In the case of Zero Condition the accumulated coastline change and transport rates are set to zero at simulation start.

In the case of Accumulated coastline you have to prepare a profile file (dfs1) with the accumulated coastline change. The file must have the same number of grid steps as the coastline file. In this case the accumulated transport rates are set to zero at simulation start and the resulting coastline position from the previous simulation should be entered into the Coastline dialog.

In the case of Accumulated coastline and transport you have to prepare a profile file (dfs1) with the accumulated coastline change, and start values of net and gross drift rates. The file must have the same number of grid steps as the coastline file. The resulting coastline position from the previous simulation should be entered into the Coastline dialog.

NOTE: A profile file with the necessary information for initial conditions can be obtained from the previous simulation by saving the results from the last time step in a separate output file of type coastline.

8.9 Dune Erosion

It is possible to consider the effect of dune erosion on the morphological shoreline changes.

8.9.1 General description

Dune erosion can be calculated were simplistic by only eroding the dune when the shoreline has eroded all the way back to the dune position.

The parameterized dune erosion module improves the description of dune erosion such that the dune erodes during high water level/high wave events even if the shoreline has not eroded all the way back.
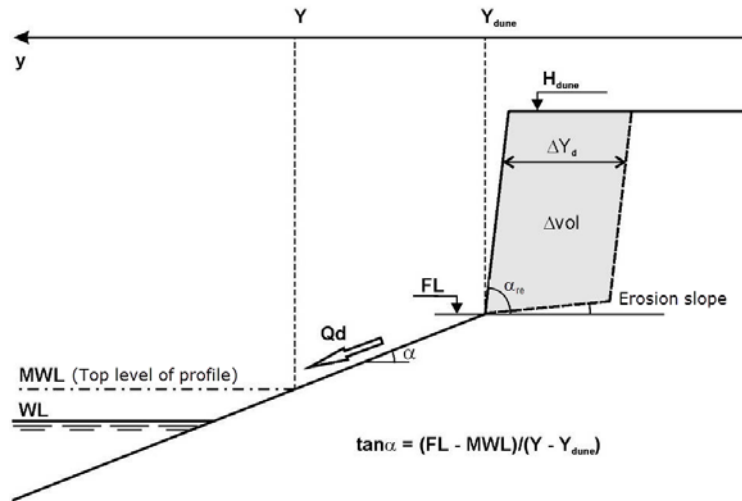
Figure 8.9   Definition sketch for the dune erosion model

The loss Q_d eroded from the dune is transferred to the coastline model as a source term so that the coastline advances correspondingly.

$$\Delta vol = Q_d \cdot \Delta t \qquad (8.2)$$

The dune front is defined by the angle of repose, α_{re}. As the dune front retreats the level of the dune foot increase along an angle defined by the erosion slope. The slope of the beach is restricted by a maximum value. In case the calculated beach slope exceed the maximum value, the dune foot level FL is reduced to a value corresponding to maximum slope.

8.9.2   Specifications

Select the dune erosion type as 'Parameterized dune erosion' to include the effect of dune erosion. Next you have to specify the initial location of the dune and dune erosion parameters.
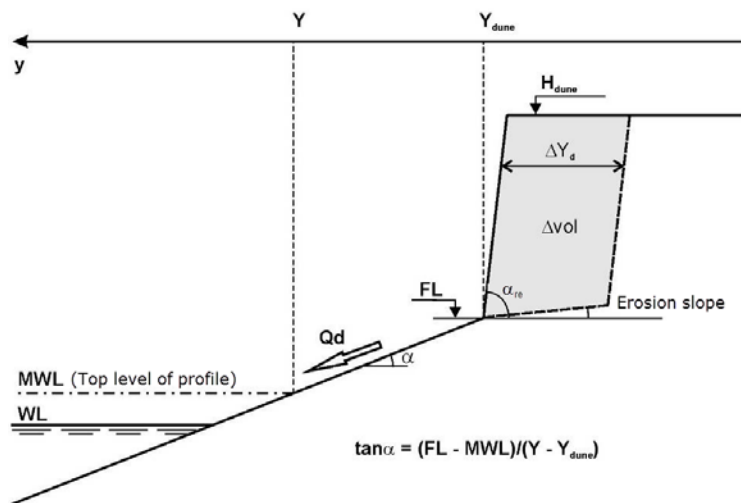
Figure 8.10   Definition sketch for the dune erosion model

The dune erosion parameters are described in Table 8.1 below.

Table 8.1   Dune erosion parameters

Parameter	Description
Erosion slope	z-level change due to dune foot level retreat
Dune height	dune crest level
Max. slope	maximum slope of shoreface before failure
Reference depth	mean water depth by which data for the dune erosion calculations are derived
Gamma	calibration factor for the estimate of wave run-up
Cs	calibration factor, usually in the order 0.001 to 0.003
Angle of repose	angle of the dune front
Min. active area	minimum area for active dune erosion (DVol/m)
Max. no. of iterations	calculation parameter
Tolerance	calculation parameter
Relaxation factor	calculation parameter

Location

You have to prepare an ascii file with extension .xyz containing the geographical coordinates and level of the dune foot. The file format is three space separated floats (real numbers) for the x- and y-coordinate and z-level on separate lines for each of the points. You must also select the map projection in which you have specified the geographical coordinates.

NOTE: (.xyz) input files must be provided in metric unit [m]

INDEX