MIKE 21/3 Coupled Model FM

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. For details please refer to your 'DHI Software Licence Agreement'.

LIMITED LIABILITY

The liability of DHI is limited as specified in Section III of 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 OUT OF THE USE OF OR THE INABILITY TO USE THIS DHI SOFTWARE PRODUCT, 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 COUNTRIES OR STATES 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 TO YOU. BY YOUR OPENING OF THIS SEALED PACKAGE OR INSTALLING OR USING THE SOFTWARE, YOU HAVE ACCEPTED THAT THE ABOVE LIMITATIONS OR THE MAXIMUM LEGALLY APPLICABLE SUBSET OF THESE LIMITATIONS APPLY TO YOUR PURCHASE OF THIS SOFTWARE.'

CONTENTS

1 About This Guide

1.1 Purpose

The main purpose of this User Guide is to enable you to use, MIKE 21/3 Coupled Model FM for applications to various hydraulic phenomena in lakes, rivers, estuaries, bays, coastal areas and seas.

The User Guide is complemented by the Online Help, and User Guides on the individual modules.

1.2 Assumed User Background

Although the MIKE 21/3 Coupled Model FM has been designed carefully with emphasis on a logical and user-friendly interface, and although the User Guide and Online Help contain modelling procedures and reference material, common sense is always needed in any practical application.

In this case, "common sense" means a background in coastal hydraulics and oceanography, waves, sediment transport and environmental processes, which is sufficient for you to be able to check whether the results are reasonable or not.

1.3 General Editor Layout

The MIKE Zero setup editor consists of three separate panes as shown in Figure 1.1 and described in the following sections.
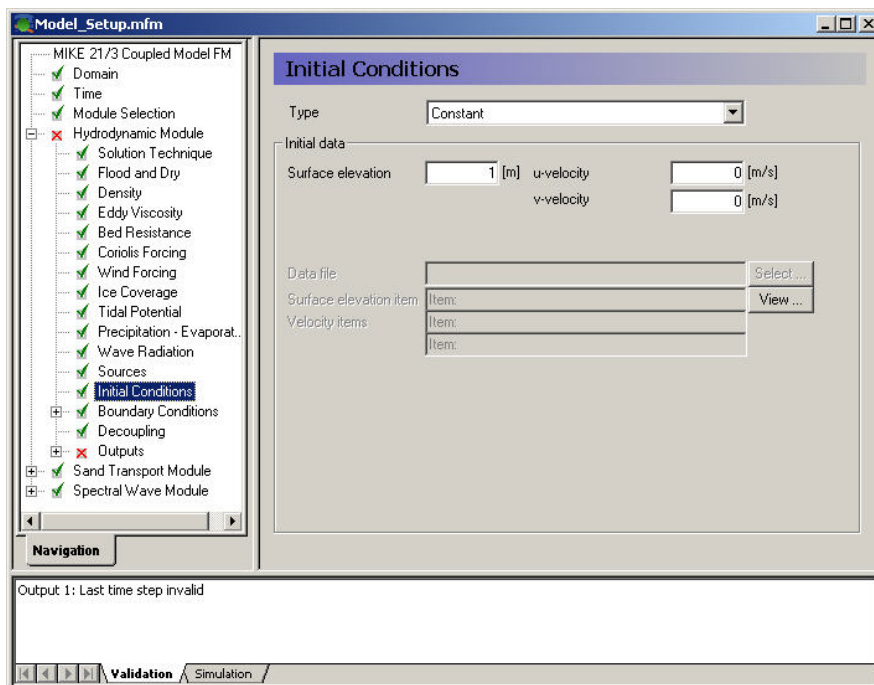
Figure 1.1   General Editor Layout

1.3.1   Navigation tree

To the left is a navigation tree, which 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, to show the optional GIS background layer 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/Simulation window

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

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

The Simulation window shows the progress of the simulation along with the contents of the -log file.

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

MIKE 21/3 Coupled Model FM is a truly dynamic modelling system for applications within coastal, estuarine and river environments.

MIKE 21/3 Coupled Model FM is composed of following modules:

- Hydrodynamic Module
- Transport Module
- ECO Lab / Oil Spill Module
- Mud Transport Module
- Particle Tracking Module
- Sand Transport Module
- Shoreline Morphology Module
- Spectral Wave Module

The Hydrodynamic Module and the Spectral Wave Module are the basic computational components of the MIKE 21/3 Coupled Model FM. Using MIKE 21/3 Coupled Model FM it is possible to simulate the mutual interaction between waves and currents using a dynamic coupling between the Hydrodynamic Module and the Spectral Wave Module. The MIKE 21/3 Coupled Model FM also includes a dynamic coupling between the Mud Transport Module and the Sand Transport Module and the Hydrodynamic Module and the Spectral Wave Module. Furthermore, combining the Hydrodynamic Module and the Spectral Wave Module with the Sand Transport Module opens up for coupling with the Shoreline Morphology Module. Hence, a full feedback of the bed level changes on the waves and flow calculations can be included.

Figure 2.1   Example of a flexible mesh in Tampa Bay, FL, where fully integrated models are used to address management needs

The Hydrodynamic Module simulates water level variations and flows in response to a variety of forcing functions in lakes, rivers, estuaries and coastal regions. The effects and facilities include

- Flooding and drying
- Momentum dispersion
- Bottom shear stress
- Coriolis force
- Wind shear stress
- Barometric pressure gradients
- Ice coverage
- Tidal potential
- Precipitation/evaporation
- Wave radiation stresses

- Sources and sinks

The Hydrodynamic Module can be used to solve both three-dimensional (3D) and two-dimensional (2D) problems. In 3D the model is based on the numerical solution of the three-dimensional incompressible Reynolds averaged Navier-Stokes equations invoking the assumptions of Boussinesq and of hydrostatic pressure. Thus, the model consists of continuity, momentum, temperature, salinity and density equations and is closed by a turbulent closure scheme. In 2D the model is based on the shallow water equations - the depth-integrated incompressible Reynolds averaged Navier-Stokes equations.

The Spectral Wave Module simulates the growth, decay and transformation of wind-generated waves and swell in offshore and coastal areas. The following physical phenomena can be taken into account:

- Wave growth by action of wind
- Non-linear wave-wave interaction
- Dissipation due to white-capping
- Dissipation due to bottom friction
- Dissipation due to depth-induced wave breaking
- Refraction and shoaling due to depth variations
- Wave-current interaction
- Effect of time-varying water depth and flooding and drying

The Spectral Wave Module includes two different formulations:

- Directional decoupled parametric formulation
- Fully spectral formulation

The directional decoupled parametric formulation is based on a parameterization of the wave action conservation equation. The parameterization is made in the frequency domain by introducing the zeroth and first moment of the wave action spectrum as dependent variables following Holthuijsen (1989). The fully spectral formulation is based on the wave action conservation equation, as described in e.g. Komen et al. (1994) and Young (1999), where the directional-frequency wave action spectrum is the dependent variable.

2.1   Application Areas

The application areas are generally problems where flow and transport phenomena are important with emphasis on river, coastal and marine applications, where the flexibility inherited in the unstructured meshes can be utilized.

MIKE 21/3 Coupled Model FM can be used for investigating the morphological evolution of the nearshore bathymetry due to the impact of engineering works (coastal structures, dredging works etc.). The engineering works may

include breakwaters (surface-piercing and submerged), groins, shoreface nourishment, harbours etc. MIKE 21/3 Coupled Model FM can also be used to study the morphological evolution of tidal inlets.

It is most suitable for medium-term morphological investigations (several weeks to months) over a limited coastal area. The typical dimensions are about 10 km in the alongshore direction and 2 km in the offshore direction. The computational effort can become quite large for long-term simulations, or for larger areas.

2.2 Computational Features

The main features of the MIKE 21/3 Coupled Model FM are as follows:

- Dynamic coupling of waves and currents
- Fully feedback of bed level changes on waves and currents
- Easy switch between 2D and 3D calculations (hydrodynamic module and process modules)
- Optimal degree of flexibility in describing bathymetry and ambient flow and wave conditions using depth-adaptive and boundary-fitted unstructured mesh

3   Getting Started

User Guides (PDF-format) for the different modules included in the MIKE 21/3 Coupled Model FM can be accessed from the MIKE 21 and MIKE 3 Documentation Index. These guides can be found as part for the documentation for MIKE 21 Flow Model FM, MIKE 3 Flow Model FM and the MIKE 21 Spectral Wave Model FM.

Step-by-step training guides for the most of the modules are also available. The training guides (PDF-format) can be accessed from the MIKE 21 and MIKE 3 Documentation Index:

MIKE 21 & MIKE 3 Flow Model FM, Hydrodynamic Module,
Step-by-Step Training Guide
(2d and 3D flow application)

MIKE 3 Flow Model FM, ECO Lab Module,
Step-by-Step Training Guide
(3D water quality application)

MIKE 21 Flow Model FM, ABM Lab - Drifter Example,
Step-by-step training guide
(Agent based modelling application)

MIKE 21 Flow Model FM, Mud Transport Module,
Step-by-Step Training Guide
(2D mud transport application)

MIKE 21 Flow Model FM, Sand Transport Module,
Step-by-Step Training Guide: River Application
(2D river sand transport application - pure current)

MIKE 21 Flow Model FM, Sand Transport Module,
Step-by-Step Training Guide: Coastal Application
(2D coastal sand transport application - waves and current)

MIKE 21 Flow Model FM, Particle Tracking Module,
Step-by-Step Training Guide
(2D Particle tracking application)

A comprehensive step-by-step training guide covering the MIKE 21/3 Coupled Model FM is also available. The training example relates to Torsminde Harbour on the west coast of Jutland, Denmark. Severe sedimentation and shoaling problems affects the harbour entrance and therefore investigations of the sediment transport pattern in the area have been conducted. 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

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

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 model such as MIKE 21/3 Coupled Model FM is through practice. Therefore, some 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 more parameters are changed.

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

- Harbour example:
  Examples\
  MIKE_21-3_Integrated_Models\CoupledModel_FM\Harbour
- Impact of pier example:
  Examples\ MIKE_21-3_Integrated_Models\CoupledModel_FM\Pier
- Torsminde example. From scratch to simulation:
  Examples\
  MIKE_21-3_Integrated_Models\CoupledModel_FM\Torsminde
- Shoreline Morphology example. From scratch to simulation:
  .\Examples\
  MIKE_21-3_Integrated_Models\CoupledModel_FM\HeadlandBeach

4.2 Harbour

4.2.1 Purpose of the example

This example illustrates the wave-driven current and littoral transport patterns in the vicinity of a harbour located on a sandy coast, as well as the influence on the results of updating the bed during the simulation.

The computational domain and bathymetry is shown in Figure 4.1.
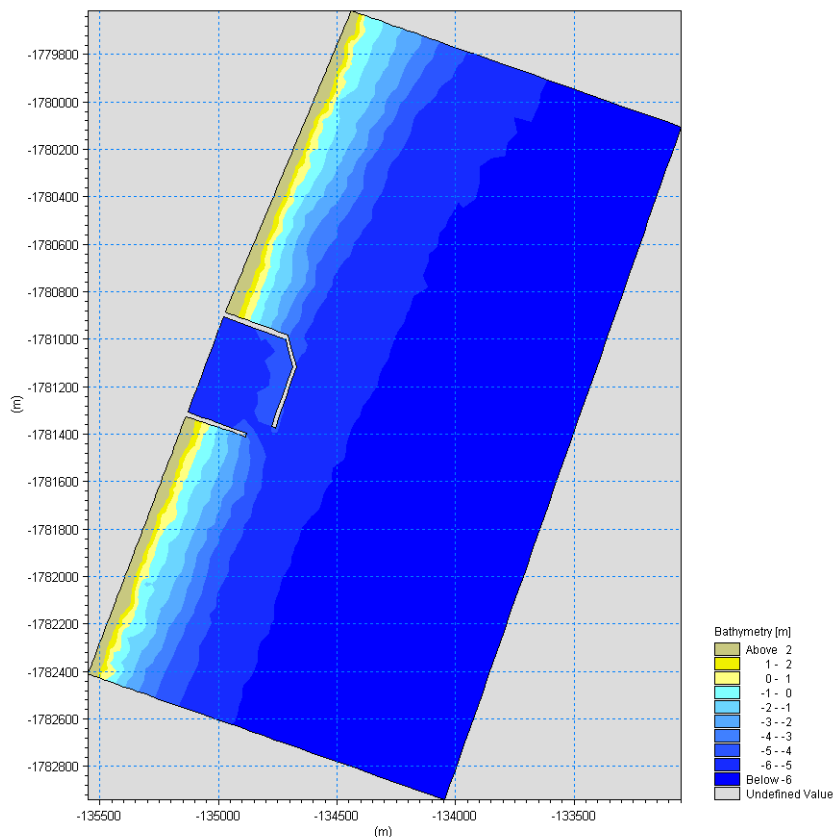
Figure 4.1   Computational domain and bathymetry

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

The test conditions are:

- The coastline normal is oriented approximately 115 degrees North
- 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 degrees North 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

4.2.2   Defining the problem

The main conditions defining the hydrodynamic model are:

- An unstructured mesh with 4378 elements and 2346 nodes is used. The mesh is shown in Figure 4.2
- An overall time step of 600 seconds is selected and the duration of the simulation is initially 5 hours (30 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 MIKE 21 Spectral Wave Module (SW). Note that the wave conditions inside the harbour and just downstream in the vicinity of the harbour are not reliable, as SW does not consider diffraction/reflection in the calculation. At both the updrift and downdrift boundary radiation stress correction is included
- The offshore boundary is set to land, which means that the flux is 0 across the boundary, whereas the updrift and downdrift boundaries are defined by the water level 1.0 m, which is the initial setting

The main conditions defining the wave model are:

- The quasi-stationary, directionally decoupled parametric formulation is used
- Per default, the wave calculation is carried for out every overall time step (i.e. 10 minutes in this example)
- The wave action spectrum is discretized by 11 directions in the interval from 60 degrees to 120 degrees
- The bottom friction type has been chosen to Nikuradse roughness with a constant value of 0.0006 m
- The waves at the offshore boundary is set to constant, whereas the upstream and downstream boundaries are set to lateral

The main conditions defining the sand transport model are:

- The model type is selected as 'Wave and Current'.
- A sediment transport table has been generated, which reflects the conditions for the example
- The sediment transport calculations start after 3 hours (at time step 18) when a steady flow and wave field is obtained
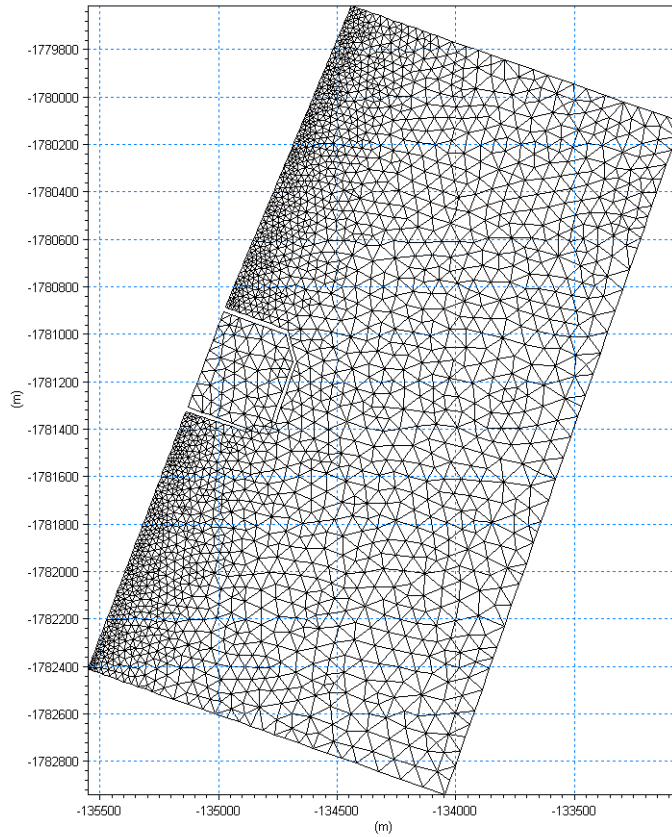
Figure 4.2   Computational mesh (plot generated using Data Viewer)

4.2.3   Initial transport field

A plot of the flow field at the start of the sand transport calculations is shown in Figure 4.3. The figure shows that the maximum currents appear along the coast.
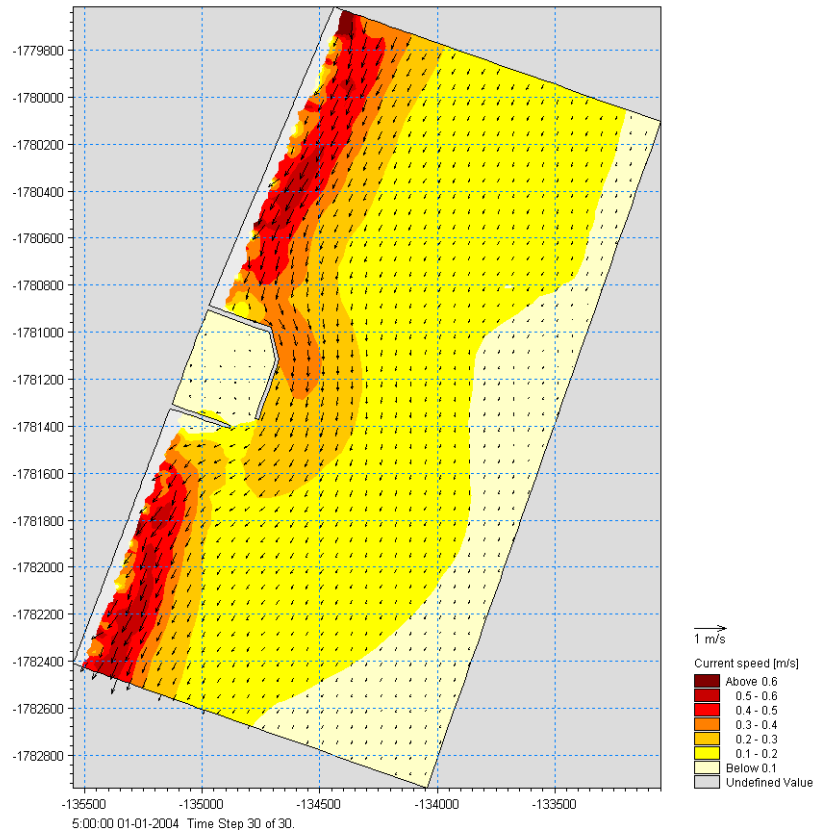
Figure 4.3   Contour and vector plot of the flow field
(plot generated using Data Viewer)

A plot of the initial total sand transport is shown in Figure 4.4. 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.
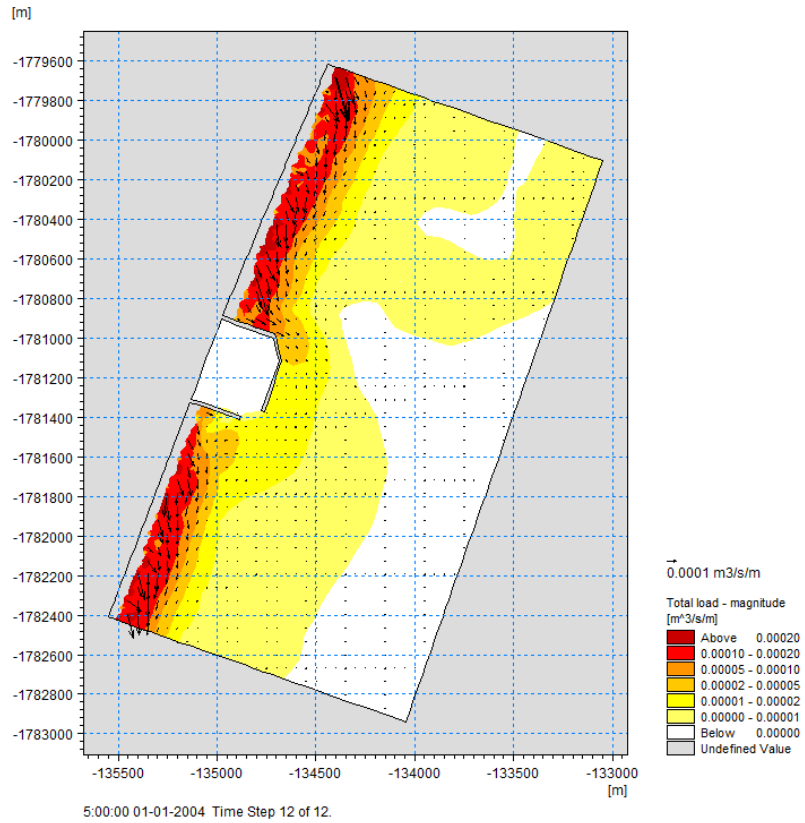
Figure 4.4   Contour and vector plot of the initial sand transport field

A plot of the initial bed level change rate at the harbour is shown in Figure 4.5. 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.
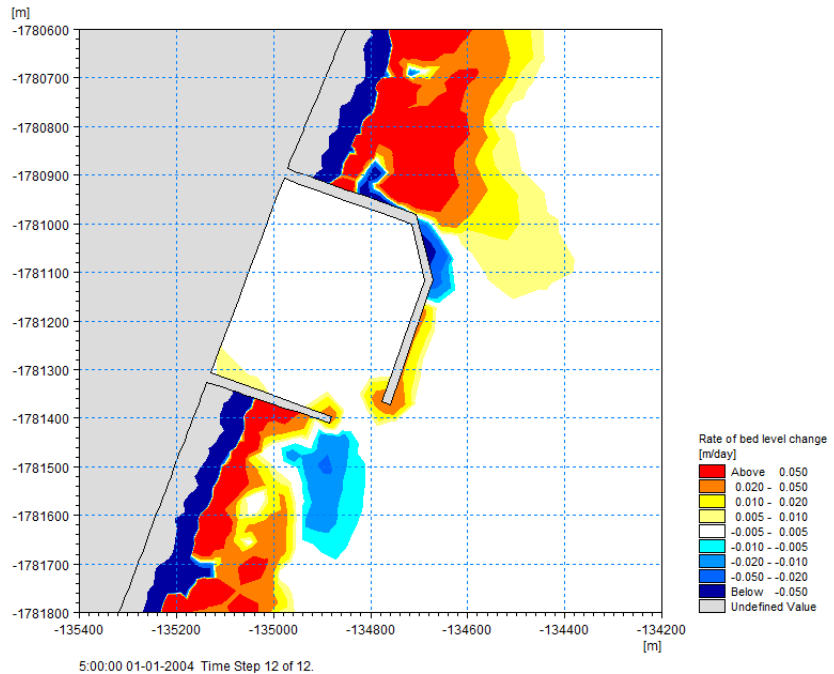
Figure 4.5   Contour plot of initial bed level change rate

4.2.4   Morphological evolution using speed-up factor

Now the model is set up to simulate the bed level change for 12 days. The same flow, wave and sediment conditions are applied.

Note that simulated bed level changes are most valid if the bed level is updated continuously and this is instantly reflected in the current, waves and sediment transport rates.

To save simulation time, a speed-up factor of 144 is applied on the morphological simulation. This means that for each 10 minutes in the simulation the bed level updates as if the duration was 1 day. The maximum bed level change rate is increased to 20 m/day in order to accommodate the artificially increased bed level change rates. The bed level is updated during the simulation and the bed level changes is reflected in the interaction between the currents, waves and sediment transport. This corresponds to calculating the waves once a day to account for the bathymetry changes.

Figure 4.6 shows the total bed level change after 12 days, using a speed-up factor. Figure 4.7 shows the total bed level change obtained by using the initial bed level change rate to extrapolate the long-term bed level change. It can be seen from the figures, that the maximum bed level change rate may limit the sedimentation. Applying feedback on the hydrodynamics from the sedimentation will enable the evolution of a bed topography that reflects the

actual conditions better, e.g. the formation of bars is better indicated in Figure 4.6.
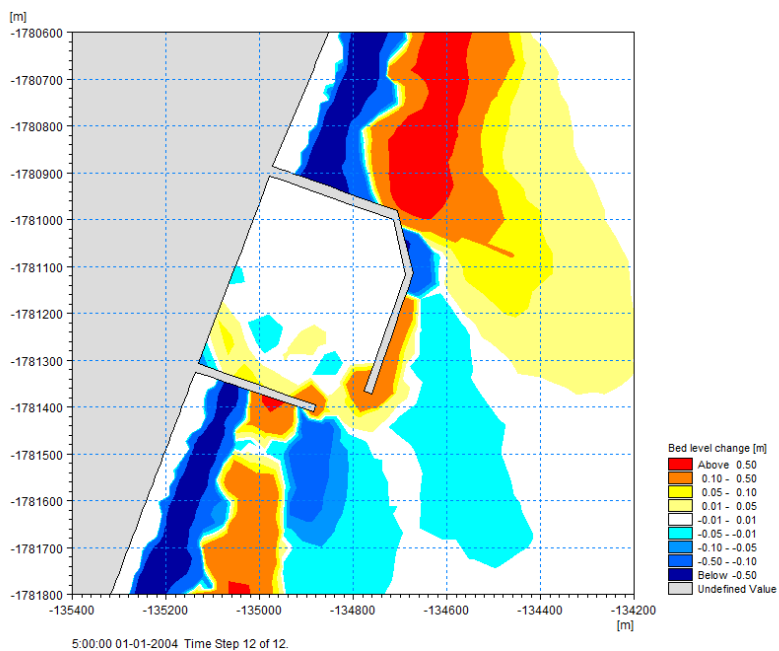
Figure 4.6   Contour plot of bed level change after 12 days using morphological modelling and applying speed-up factor 144.
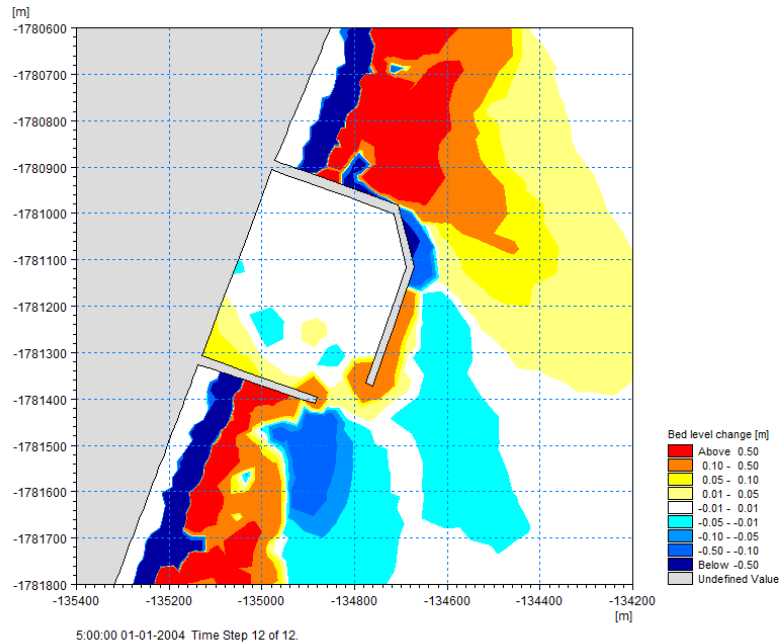
Figure 4.7   Contour plot of bed level change after 12 days using simple extrapolation of initial bed level change.

4.2.5   Morphological evolution using quasi-stationary formulation

Another way to save simulation time is to apply the quasi-stationary time formulation when solving the hydrodynamic equations. While calculating the morphological changes using no speed-up factor will require more time steps and hence longer simulation time, the decrease in simulation time for the hydrodynamics will reduce some of the additional time required. Note that this option should only be used in cases where the flow conditions are nearly constant.

In order to simulate the morphology changes during 12 days, the number of time steps is increased to 1746. The bed level is updated during the simulation for each time step and the bed level changes reflect the interaction between the currents, waves and sediment transport.

Figure 4.8 shows the total bed level change after 12 days, using quasi-stationary formulation with tolerance levels of 1.0E-05. For comparison Figure 4.9 shows the total bed level change using the in-stationary formulation which is default for the Coupled Model FM.

It can be seen from Figure 4.8 and Figure 4.9 that the calculated bed level changes are very similar, even if the time consumption of solving the flow conditions in the quasi-stationary mode is only 12% of the time consumption when solving the flow equations in the in-stationary mode.

The morphological development using a 'true' time scale (speed-up factor 1.0), as shown in Figure 4.8, shows a smoother transition between erosion and deposition areas than when using an artificial time scale (speed-up factor 144), as shown in Figure 4.6. Note that the sedimentation inside the harbour seems more realistic in the corner when the flow conditions have time to adapt to the new bed level conditions.
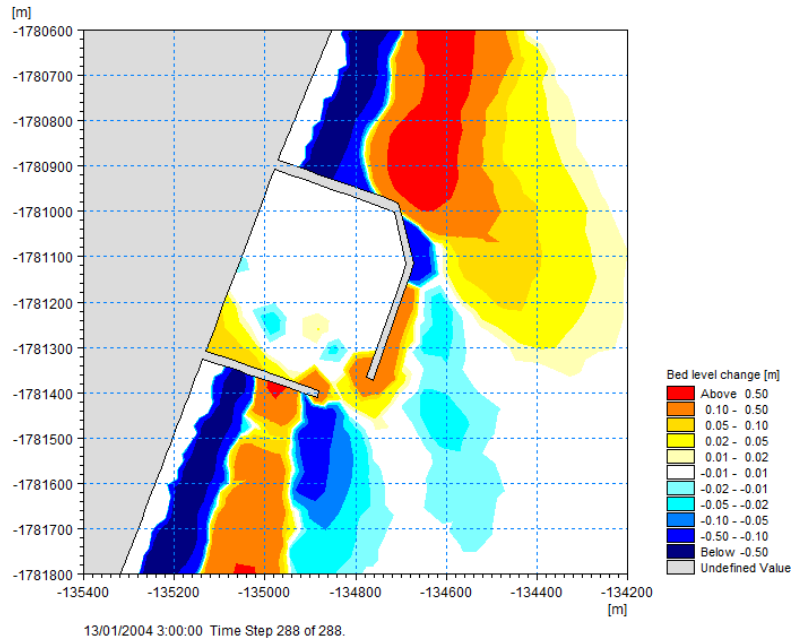
Figure 4.8   Contour plot of bed level change after 12 days using quasi-stationary time formulation in hydrodynamic modelling
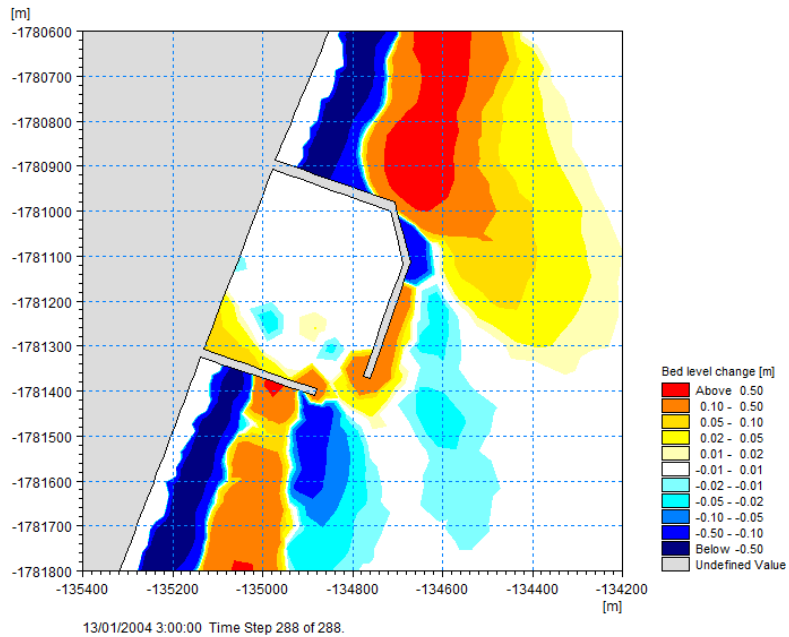
Figure 4.9　　Contour plot of bed level change after 12 days using instationary time formulation in hydrodynamic modelling

4.2.6　List of data and specification files

The following data files (included in the ST\Harbour folder) are supplied with MIKE 21/3 Coupled Model FM:

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

File name: Harbour.mfm
Description: MIKE 21/3 Coupled Model FM specification file (initial sand transport rates)

File name: HarbourB.mfm
Description: MIKE 21/3 Coupled Model FM specification file (morphological evolution using speed-up factor)

File name: HarbourC.mfm
Description: MIKE 21/3 Coupled Model FM specification file (morphological evolution using quasi-stationary hydrodynamics)

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

4.3   Impact of Pier

4.3.1   Purpose of the example

This example simulates the wave field, wave-driven current and related littoral transport patterns in the vicinity of a pier structure located on a sandy coast. The morphological impact of the pier structure is investigated by using two alternate methods: the basic Morphology module and the Shoreline Morphology module.
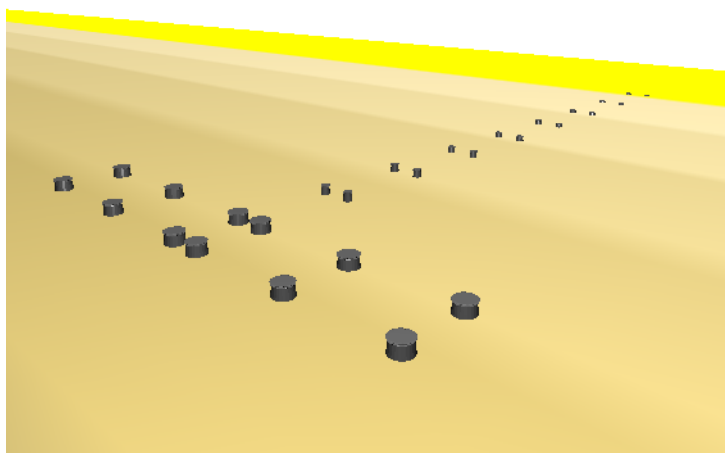
Figure 4.10   Illustration of pier structure

The problem is to determine the impact the presence of the pier structure has on the shoreline change and sedimentation pattern along the coast.

The test conditions are:

- The coastline normal is oriented 180 degrees North
- The cross-shore bathymetry has a uniform slope of 1:100, reaching from +1.5 m to -15.5 m.
- The sediment in the area is uniform with a mean grain diameter of 0.1 mm.
- Irregular, directional waves propagate towards the coast from 135 degrees North over a water level corresponding to MWL = 0.0 m. The wave characteristics at the offshore boundary of the model are Hm0= 1 m and Tp= 6 s

- 28 individual circular piles form the foundation of a pier structure extending to the -4 m contour in the near shore zone. The pier structure is illustrated in Figure 4.10 and the description of the individual piers are given in Table 4.1.

Table 4.1   Position and outline of pier structures
             Note: height of pier correspond to top being 1 m above water level)

X-position	Y-position	Diameter	Height
3000	0	5	1
3000	-50	5	1.5
3000	-100	5	2
3000	-150	5	2.5
3000	-200	5	3
3000	-250	5	3.5
3000	-300	5	4
3000	-350	5	4.5
3000	-400	10	5
3015	0	5	1
3015	-50	5	1.5
3015	-100	5	2
3015	-150	5	2.5
3015	-200	5	3
3015	-250	5	3.5
3015	-300	5	4
3015	-350	5	4.5
3015	-400	10	5
2900	-400	10	5
2900	-430	10	5.3
2950	-400	10	5
2950	-430	10	5.3
3065	-400	10	5
3065	-430	10	5.3
3115	-400	10	5
3115	-430	10	5.3

The computational domain and bathymetry is shown in Figure 4.11.
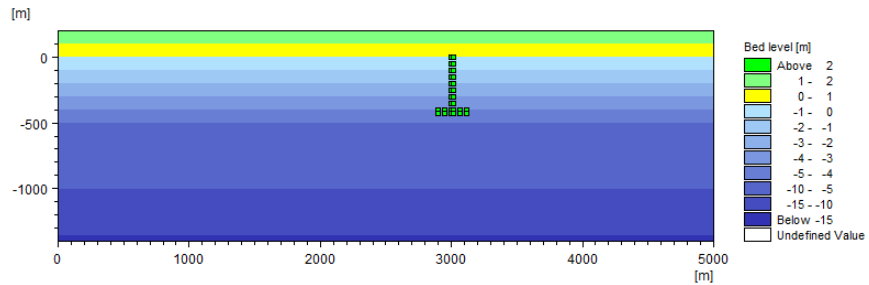
Figure 4.11   Computational domain and bathymetry.
Positions of piles are indicated.

4.3.2   Defining the problem

The main conditions defining the hydrodynamic model are:

- An unstructured mesh with 6400 quadrangular elements and 6565 nodes is used. The mesh is shown in Figure 4.12.
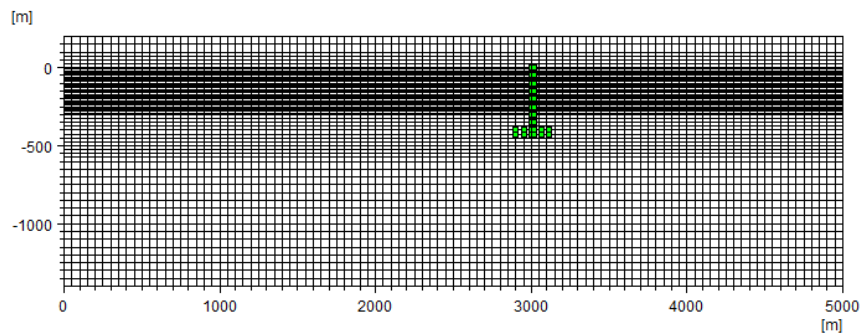
Figure 4.12   Computational mesh (plot generated using Data Viewer)

- An overall time step of 3600 seconds is selected with a maximum of 300 s for the internal time step.
- To see the long-term effect of the pier the duration of the simulation is to be 1 year (8784 time steps) using a quasi-stationary formulation.
- The initial conditions for the long-term morphology simulation are derived from a simulation using 48 time steps and in-stationary formulation.
- The horizontal eddy viscosity type has been chosen as a constant value of 0.28 m2/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 MIKE 21 Spectral Wave Module (SW).

- The offshore boundary has a specified discharge of 0 m³/s. The eastern and western boundaries are both defined by the water level 1.0 m. Radiation stress correction is included at both the eastern boundary and the western boundary.
- 28 piles are included as pier structures as defined in Table 4.1

The main conditions defining the wave model are:

- The quasi-stationary, directionally decoupled parametric formulation is used
- Per default, the wave calculation is carried for out every overall time step (i.e. 1 hour in this example)
- The wave action spectrum is discretized by a 360 degree rose with 36 directions
- The bottom friction type has been chosen to Nikuradse roughness with a constant value of 0.04 m
- The waves at the offshore boundary are set to constant (H_{m0}= 1 m, T_p= 6 s, from 135 deg.N), whereas the eastern and western boundaries are set to lateral.
- 28 piles are included as pier structures as defined in Table 4.1

The main conditions defining the sand transport model are:

- The model type is selected as 'Wave and Current'.
- A sediment transport table has been generated, which reflects the conditions for the example (d_{50} = 0.1 mm)
- The sediment transport calculations start at time step 0, as a steady flow field is entered as an initial condition.

The main conditions defining the basic morphology model are:

- The model use the default values of maximum bed level change and speedup factor (1.0 m/day and 1, respectively)
- Feedback on calculations are included
- The boundary condition 'Zero sediment flux gradient for outflow, zero bed change for inflow' are defined for all open boundaries

The main conditions defining the shoreline morphology model are:

- The baseline is defined as a line from (4000, 190) to (1000, 190) with a grid spacing of 100 m.
- The coastline is defined as a line with inner points from (3950,0) to (1050,0) with a grid spacing of 100 and additional points in the start and end to fulfil numerical requirements.
- The cross-shore profile is the same along the coastline and is defined from -200 m to 600 m from the shoreline. The closure depth is set to 5 m.
- The bathymetry values match the mesh values in the active region between 0 m and -5 m and is here defined by a slope of 1:100. The slope is steeper elsewhere. The profile is shown in Figure 4.13.
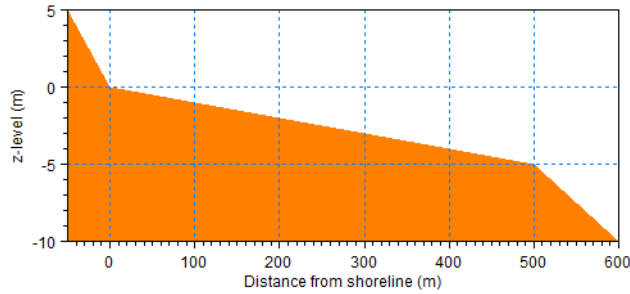
Figure 4.13   Cross-shore profile in Shoreline Morphology module (note distorted scale)

- An edge map is specified in order to associate element data in the mesh with a given cross-shore profile

An outline of the data dedicated for the Shoreline Morphology model is illustrated in Figure 4.14.
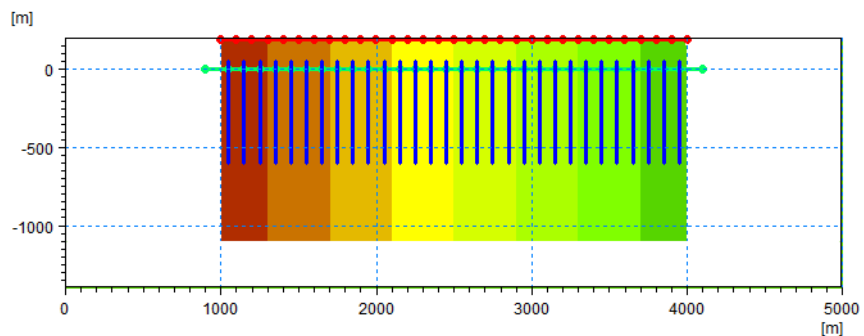
Figure 4.14   Overall layout of data within Shoreline Morphology model
Red line: baseline with points, Green line: coastline with points,
Blue lines: cross-shore profiles, Underlying contour: EdgeMap

4.3.3   Initial conditions

Calculating the morphological evolution for one year will require a long simulation time if the hydrodynamics are modelled by the in-stationary formulation. As the applied forcing (the incoming waves) is specified as constant, it is possible to use the quasi-stationary formulation for the simulation of the flow field, thus reducing the simulation time considerably.

A coupled setup with the Spectral Wave module and the Hydrodynamic module using the in-stationary formulation is applied to simulate the wave field and corresponding flow field for a period of two days. The computed velocities at the end of this simulation will describe the stationary flow field that can be used as initial conditions for the long-term morphology simulation.

The resulting initial flow conditions at the start of the morphology calculations are shown in Figure 4.3 and the corresponding wave conditions are shown in Figure 4.3a. The figures shows that the maximum wave generated currents appear within the extension of the structure and even if the pile structures affect the waves and currents, the impact is less than would be expected from a solid structure.
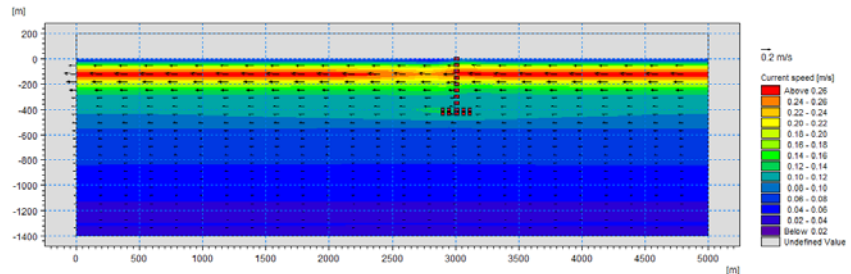
Figure 4.15   Contour and vector plot of the flow field
(plot generated using Data Viewer)
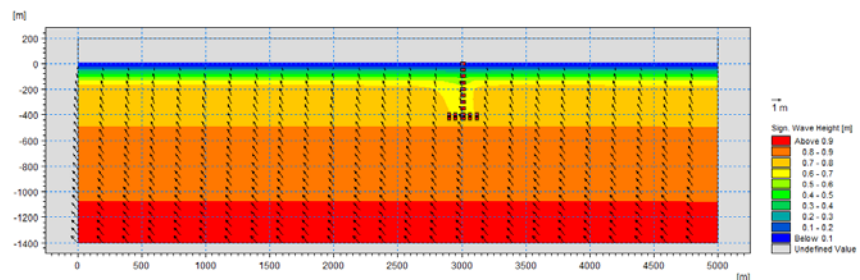
Figure 4.16   Figure 4.3 Contour and vector plot of the wave field
(plot generated using Data Viewer)

4.3.4   Quasi-stationary flow field

A similar model, now using the initial condition for the flow field found from the previous simulation and simulating for a period of 1 week, is applied to verify the use of the quasi-stationary formulation.

The resulting water level and current speed in point (2525m,-125m) from two simulations, using the in-stationary formulation and quasi-stationary formulation, respectively, is compared in Figure 4.17.

As the results in Figure 4.17 are very similar it is taken as a proofs that the quasi-stationary formulation is capable of simulating the flow conditions for the constant wave forcing with an adequate accuracy. This will save time in future simulations.

Note: a test shows the CPU time for the quasi-stationary setup to be more than 25 times faster than the CPU time for the in-stationary setup (290 CPU seconds versus 8451 CPU seconds, respectively).
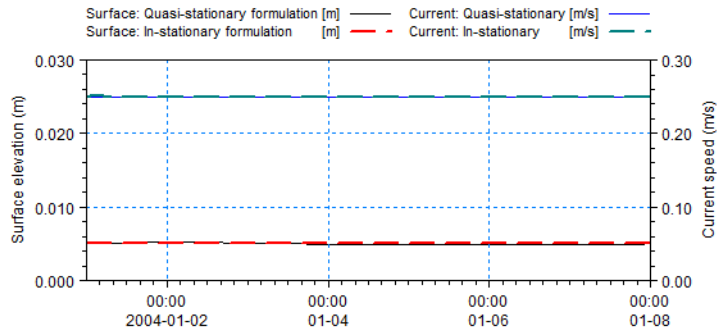
Figure 4.17   Simulated water level and current speed in position (2525m,-125m)
Solid lines: Quasi-stationary formulation
Stippled lines: Instationary formulation

4.3.5   Morphological evolution using basic morphology model

Now the Coupled model is set up to include the Sand Transport module and simulate the bed level change for a year.

First, the model setup does NOT include the Shoreline Morphology module. This implies that the bed level changes in each mesh element are updated according to the local sediment continuity equation.

The resulting bed level changes after a year of simulation is shown in Figure 4.18. A detail of the resulting bed level is shown in Figure 4.19.
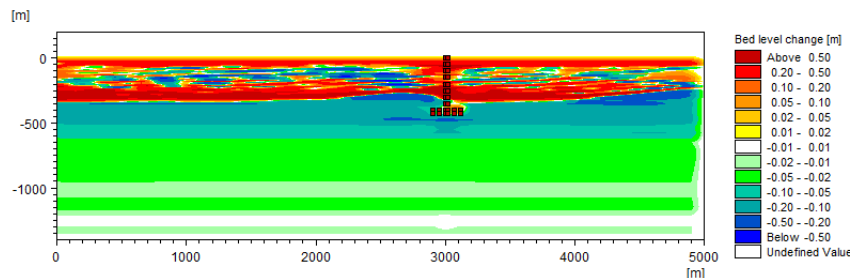
Figure 4.18   Simulated bed level change after one year using default morphology calculations
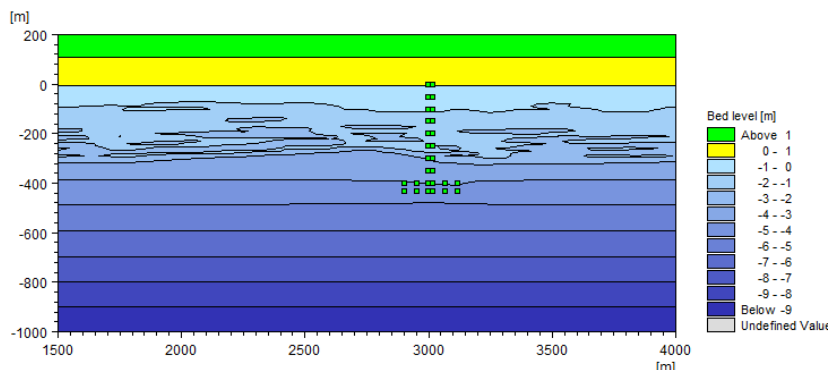
Figure 4.19 Simulated bed level after one year using default morphology calcula-
tions

The figure shows a mixed pattern of the bed level changes in the inner
surfzone and it is not possible to derive a direct effect of the pier on the shore-
line. The presence of the pier structure does affect the bed level changes
however most of the changes are related to the effect of cross-shore trans-
port rates.

4.3.6 Morphological evolution using shoreline morphology model

Next the Shoreline Morphology module is added to the setup.

The Shoreline Morphology module calculates the bed level change rates and
related volume change in the 2D domain, but instead of up-dating the bed
level in each mesh element the calculated change of sediment volume during
each time step is integrated across the shoreface for each coastline point.
The morphology is then updated according to the continuity equation for the
shoreline using a predefined coastal profile and redistributed to the related
mesh element.

Thereby the effect of the cross-shore transport is eliminated from the mor-
phology allowing for long term simulations. It has the effect that the simulated
bed level changes in the 2D domain are less realistic however it is possible to
review the resulting shoreline change.

Figure 4.20 shows the simulated bed level change in the 2D domain after one
year using Shoreline morphology calculations. Figure 4.21 shows a detail of
the resulting bed level in detail. These values are to be considered as internal
results as the main output of this model is the simulated shoreline change
shown in figure xy, however the results illustrate the accretion of the shoreline
behind the pier structure.
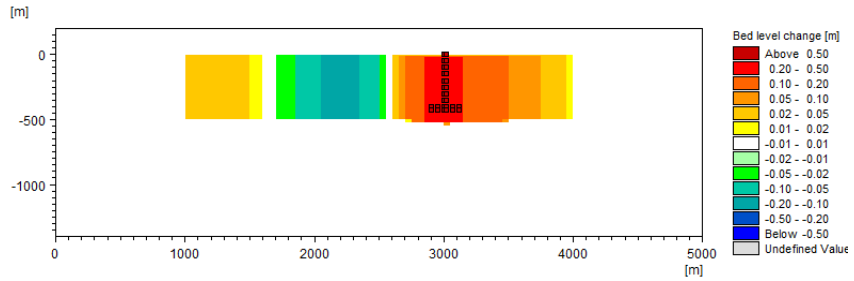
Figure 4.20 Simulated bed level change after one year using Shoreline Morphology module
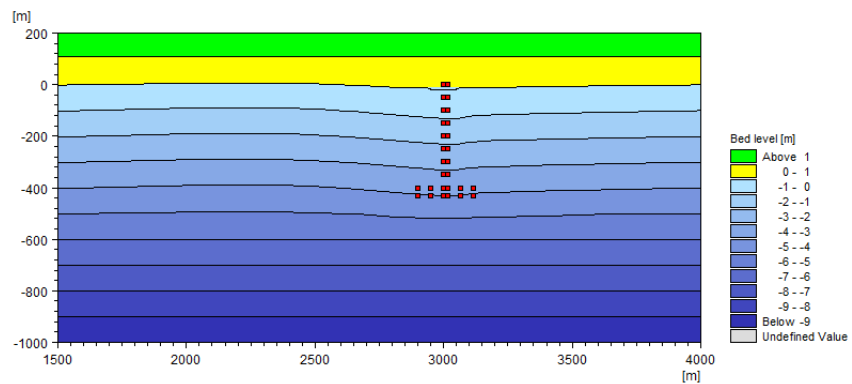
Figure 4.21 Simulated bed level after one year using Shoreline Morphology module
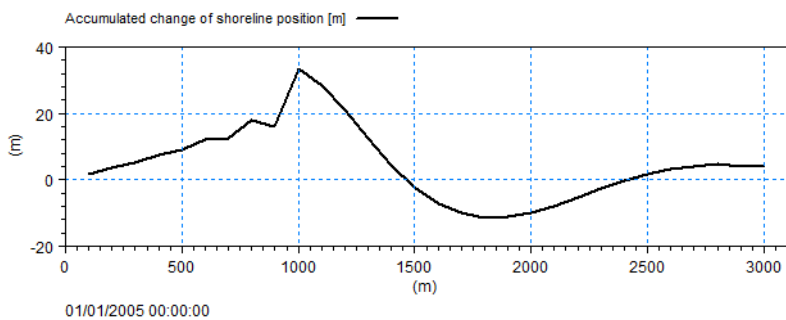
Figure 4.22 Simulated shoreline change after one year using Shoreline Morphology module.
Note: direction of x- and y-axis is shown 180 opposite the 2D domain due to the orientation of shoreline.

Both Figure 4.21 and Figure 4.22 show that the pier will affect the shoreline position when applying the shoreline morphology model.

4.3.7   List of data and specification files

The following data files (included in the ST\Pier folder) are supplied with MIKE 21/3 Coupled Model FM:

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

File Name: Table_01.lon & tTable_01.crs
Description: Sediment transport tables

File name: PierPositions.xyz
Description: Position of structures

File name: ImpactOfPiers_InitFlowField.mfm
Description: MIKE 21/3 Coupled Model FM specification file (creation of initial conditions)

File name: ImpactOfPiers_InStationary.mfm
Description: MIKE 21/3 Coupled Model FM specification file (validation of quasi-stationary formulation)

File name: ImpactOfPiers_QuasiStationary.mfm
Description: MIKE 21/3 Coupled Model FM specification file (validation of quasi-stationary formulation)

File name: ImpactOfPiers_DefaultMorphology.mfm
Description: MIKE 21/3 Coupled Model FM specification file (morphological evolution using default morphology module)

File name: ImpactOfPiers_ShorelineMorphology.mfm
Description: MIKE 21/3 Coupled Model FM specification file (morphological evolution using Shoreline morphology module)

File name: baseline.xyz
Description: ASCII file describing position of baseline in setup for Shoreline Morphology model

File name: coastline.xyz
Description: ASCII files describing position of coastline in setup for Shoreline Morphology model

File name: SimpleProfile.xy
Description: ASCII files describing position of cross-shore profile with respect to coastline in setup for Shoreline Morphology model

File name: EdgeMap.dfsu
Description: Data file used in setup for Shoreline Morphology model

4.4 Morphological Modelling for Torsminde Harbour, Denmark

4.4.1 Objective

The objective of this example is to set up a MIKE 21/3 Coupled Model FM for Torsminde harbour, 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:

.\Manuals\MIKE_21-3_Integrated_Models\CoupledModel_FM

4.5 Shoreline Morphology Modelling, Headland beach

4.5.1 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 BASIC PARAMETERS

5.1 Domain

Providing MIKE 21/3 Coupled Model FM with a suitable mesh is essential for obtaining reliable results from your model. Setting up the mesh includes selection of the appropriate area to be modelled, adequate resolution of the bathymetry, wave, wind and flow fields under consideration and definition of codes for open and closed boundaries. Furthermore, the resolution in the geographical space must also be selected with respect to stability considerations.

The MIKE 21/3 Coupled Model FM is based on flexible mesh approach. In case 3D simulation is selected a layered mesh is used: In the horizontal domain an unstructured mesh is used while in the vertical domain a structured mesh is used (see Figure 5.1). The vertical mesh is based on either sigma-coordinates or combined sigma/z-level coordinates. For the hybrid sigma/z-level mesh sigma coordinates are used from the free surface to a specified depth and z-level coordinates are used below. The different types of vertical mesh are illustrated in Figure 5.2. The elements in the sigma domain and the z-level domain can be prisms or bricks (hexahedrals) whose horizontal faces are triangles and quadrilateral elements, respectively. The elements are perfectly vertical and all layers have identical topology.
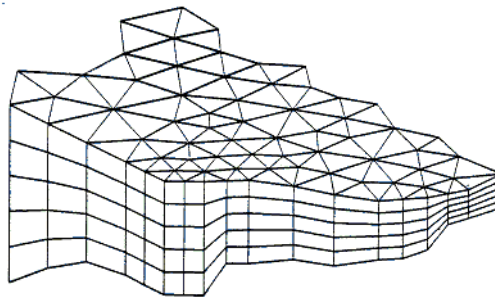
Figure 5.1   3D mesh using sigma coordinates
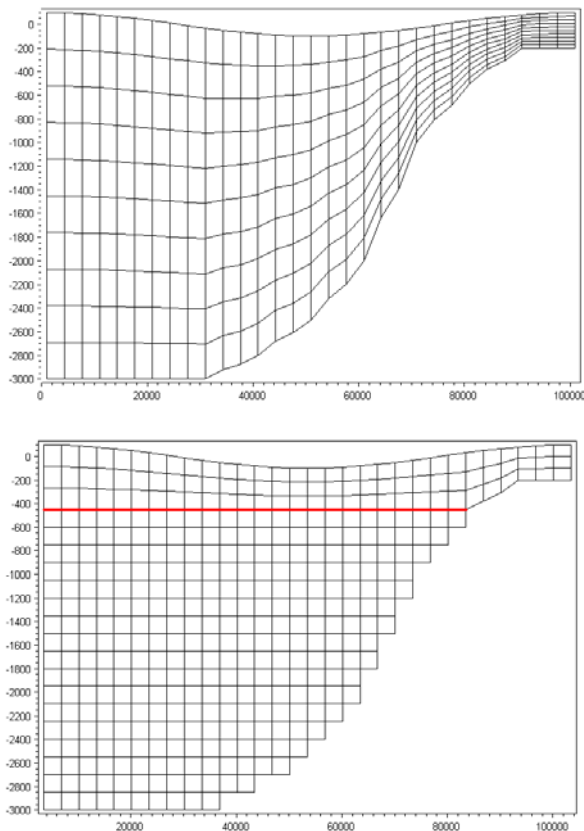
Figure 5.2   Illustrations of the different vertical grids.
Upper: sigma mesh, Lower: combined sigma/z-level mesh with simple
bathymetry adjustment. The red line shows the interface between the z-
level domain and the sigma-level domain.

5.1.1   Mesh and bathymetry

You generate your mesh file in the MIKE Zero Mesh Generator, which is a
tool for the generation and handling of unstructured meshes, including the
definition and editing of boundaries.

The mesh file is an ASCII file including information of the geographical posi-
tion and bathymetry for each node point in the mesh. The file also includes
information of the node-connectivity in the mesh.

5.1.2   Domain specification

Map projection

If the mesh is generated by the MIKE Zero Mesh Generator, the map projection is defined in the mesh file and is only shown for reference in the user interface. If the map projection information is not defined in the mesh file, you have to select the correct map projection corresponding to the data in the mesh file.

Minimum depth cutoff

If the bathymetry level in an element is above the minimum depth cutoff value then the minimum depth cutoff value is replacing the actual bathymetry value in the computations. Please note that the minimum depth cutoff value may be negative as the bathymetry levels is often so in the mesh file.

If you also apply a Datum shift - the depth cutoff is relative to the corrected depths.

For instance - you have a mesh file with values between +2 and -20 meters. You then shift these to a different datum with a shift of +1 meters. Your corrected bathymetry now ranges between +1 and -21 m. You can then cutoff all depths above -2m, leaving the bathymetry used in the model to range between -2 and -21 m.

Datum shift

You can use any convenient datum for setting up the mesh of your model. This can be Chart Datum (CD), Lowest Astronomical Tide (LAT) or Mean Sea Level (MSL). The actual datum is unimportant.

What is important, however, is that for each simulation you must provide the model with the correct height of the model reference level relative to the datum used in the setup of your bathymetry. Specifying the datum shift does this. In this way it is possible to carry out simulations using a range of different water levels without having to alter the mesh file.

If you do not plan to apply different water levels in different simulations it is recommended that you set up your bathymetry with the datum that you plan use in the simulations, thus having a datum shift of 0 m.

Note: A datum shift of e.g. 2 m (-2 m) means the water depth is increased (decreased) by 2 m in all node points.

Mesh decomposition

To improve the performance of the numerical scheme it is possible to include reordering of the mesh (renumbering of the element and node numbers). This

can significantly speed up the computational time by optimizing the memory access.

To improve the performance of the numerical scheme a domain decomposition technique is applied. If reordering is included the reordering is applied at subdomain level.

Note: When reordering is applied the numbering of the nodes and elements in the output files has been changed compared to the information in the mesh file. The information in the log file corresponds to the new ordering.

PLEASE NOTE:
When executing a flexible mesh model in decoupled mode the interpretation of the decoupled files depends on the choice of re-ordering and number of sub-domains used for the simulation. In case you change the number of sub-domains when creating and using the decoupled files, reordering should be omitted.

Dimension

In MIKE 21/3 Coupled Model FM you can choose whether to simulate in 2D or 3D. If you choose to simulate in 3D you have to specify the Vertical mesh of the grid.

5.1.3 Vertical mesh

In the vertical domain a layered mesh is applied. Two different types of the mesh can be used:

- Sigma
- Combined sigma/z-level

Sigma

In the sigma domain the vertical distribution of the layers can be specified in three different ways:

- Equidistant
- Layer thickness
- Variable
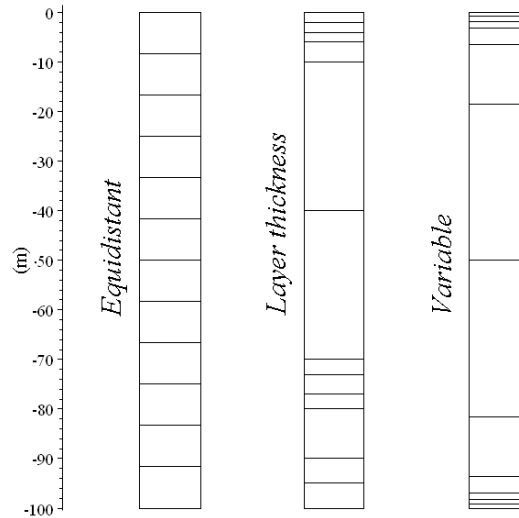
Figure 5.3   Examples of vertical distribution of layers for a water column 100 m deep. A number of 12 layers has been applied for all three options.
Left column: Equidistant distribution
Middle column: Layer thickness distribution
Right column: Variable distribution

For all three options you must specify the number of vertical layers (elements).

Selecting equidistant distribution, the layers are distributed equidistant across the water depth.

Selecting layer thickness distribution, the fraction of each layer's thickness across the water depth must be specified. Note that the sum of the values for the layer thickness must be equal to 1.

Selecting variable distribution, you must specify three vertical distribution parameters:

1. sigma_c (σ_c)
   σ_c is a weighting factor between the equidistant distribution and the stretch distribution. The range is $0 < \sigma_c \leq 1$. The value 1 corresponds to equidistant distribution and 0 corresponds to stretched distribution. A small value of σ_c can result in linear instability.

2. theta (θ)
   θ is the surface control parameter. The range is $0 < \theta \leq 20$.

3. b is the bottom control parameter. The range is $0 \leq b \leq 1$.

The variable s-coordinates are obtained using a discrete formulation of the general vertical coordinate (s-coordinate) system proposed by Song and Haidvogel (1994).

If θ *<<1* and *b=0* an equidistance vertical resolution is obtained. By increasing the value of θ *,* the highest resolution is achieved near the surface. If θ *>0* and *b=1* a high resolution is obtained both near the surface and near the bottom.

A detailed description of the equations behind the variable distribution can be found in the scientific documentation for MIKE 21 & MIKE 3 Flow Model FM, Hydrodynamic and Transport Module.
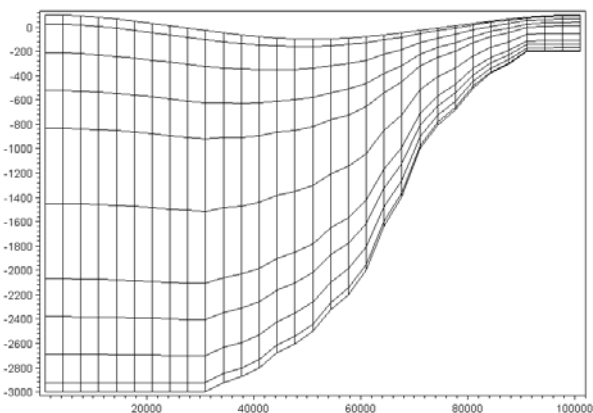
Figure 5.4   Example of vertical distribution using layer thickness distribution. Number of layers: 10, thickness of layers 1 to 10: 0.025,0.075,0.1,0.1,0.2,0.2,0.1,0.1,0.075,0.025
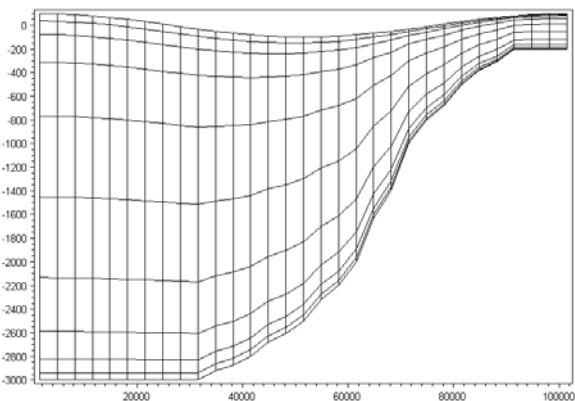
Figure 5.5   Example of vertical distribution using variable distribution. Number of layers: 10, sigma_c = 0.1, theta = 5, b = 1

Combined sigma/z-level

For the combined sigma/z-level mesh sigma coordinates are used from the free surface to a user specified level (sigma depth) and below that z-level coordinates are used.
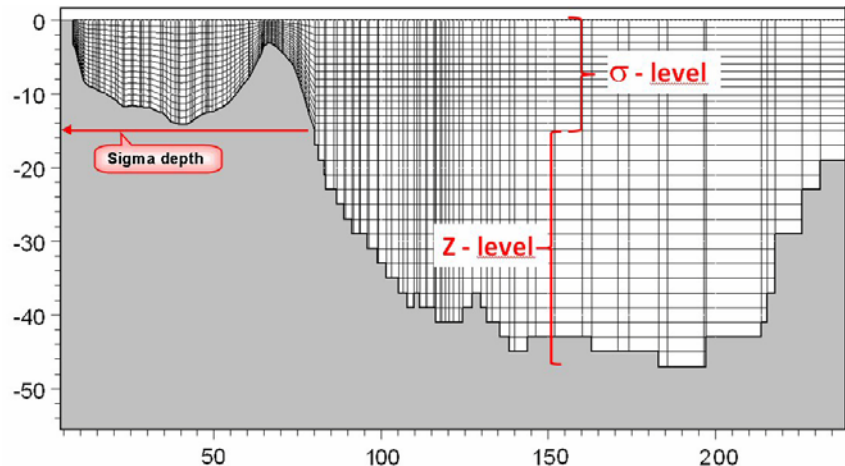
Figure 5.6   Example of vertical distribution using combined sigma/z-level

When flood and dry is included in the simulation, the flooding and drying (see Flood and Dry *(p. 58)*) is restricted to areas within the sigma domain. Therefore the sigma depth must be selected so that the minimum water level during the simulation does not become lower that sigma depth.

The specification of the mesh in the sigma domain is done as described in the previous section (see). In the z-level domain the vertical distribution of the layers can be specified in two different ways:

- Equidistant
- Layer thickness

For both options you must specify the number of vertical layers (elements) and the sigma depth.

Selecting Equidistant distribution the constant layer thickness must be specified.

Selecting Layer thickness distribution the thickness of each layer must be specified. Layer 1 correspond to the bottom layer, layer 2 correspond to the second layer from the bottom and so on.

The type of bathymetry adjustment can be specified in two ways:

- Simple adjustment

- Advanced adjustment

When selecting simple adjustment, the bottom depth is rounded to the nearest depth except when the bottom depth is below the minimum z-level. Here a bottom fitted approach is applied to take into account the correct depth.

When selecting advanced adjustment, a bottom fitted approach is applied in the whole z-level domain which allows the correct depth to be taken into account. Using the advanced adjustment you must also specify a minimum layer thickness. Normally, it can be specified as 1/100 of the constant layer thickness when the option Equidistant distribution is selected and correspondingly 1/100 of the minimum layer thickness when the option Layer thickness distribution is selected.

Depth correction
When combined sigma/z-level mesh is applied for the vertical discretization the correction bathymetry is limited. If "Simple adjustment" is selected for the bathymetry adjustment, depth correction and morphological changes due to sediment transport are not allowed. If "Advanced adjustment" is selected for the bathymetry adjustment only small corrections of the bathymetry are possible. The correction factor for the layer thickness is not allowed to be less than zero or larger than one except for the bottom cell.

A detailed description of the bottom fitted approach can be found in the scientific documentation for MIKE 21 & MIKE 3 Flow Model FM, Hydrodynamic and Transport Module.

Remarks and hints

Although, a bottom fitted approach can be applied in which the correct depth is taken into account, the flow at the bottom still needs to pass obstacles either sidewards or upwards whenever encountered. Therefore smoothing of bathymetry may improve the results.

5.1.4   Boundary names

When you generate your mesh using the MIKE Zero Mesh Generator you already have defined a code value for open water boundaries. Figure 5.7 shows the definition of codes in a simple application.
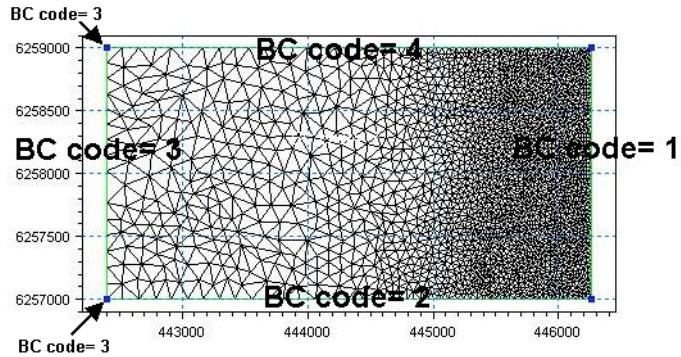
Figure 5.7   The definition of boundary codes in a mesh is made in the Mesh Generator

In this case three open boundaries have been detected from the mesh file specified in the domain parameters; code 2, code 3 and code 4. In the main Boundary names dialog you can re-name the code values to more appropriate names, see Figure 5.8.

Figure 5.8   Change of default code names (from the mesh file) to more appropriate names

5.2   Time

The period to be covered by the simulation is specified in this dialog. You have to specify the simulation start date, the overall number of time steps and the overall time step interval. The overall discrete time steps specified on this page are used to determine the frequency for which output can be obtained from the different modules and to synchronize the coupling between the different modules.

The simulation always starts with time step number 0 and the simulation start date is the historical data and time corresponding to time step 0. The simulation end date is presented for reference.

5.2.1   Remarks and hints

Each of the various modules in MIKE 21/3 Coupled Model FM use an internal time step, see Figure 5.9.
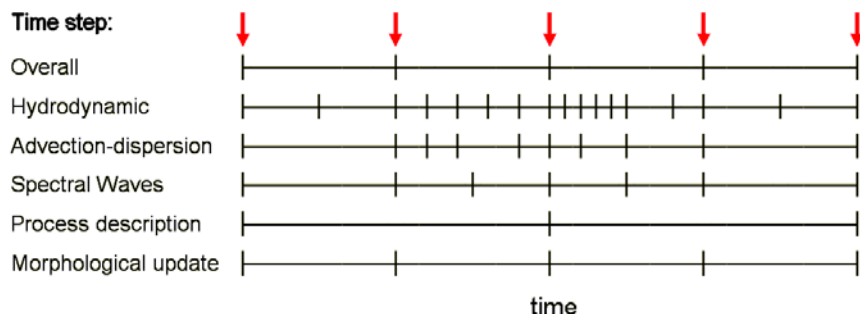
Figure 5.9   Overall time step related to internal time step

The time steps for the hydrodynamic calculations, the advection-dispersion calculations and the spectral waves calculations are dynamic and each determined to satisfy stability criterias.

All time steps within the simulation for the various modules are syncronized at the overall discrete time step. The time step for the hydrodynamic calculations is syncronized at the time step for the advection/dispersion calculations. For the Sand Transport module and the MIKE ECO Lab module the time step for updating the process description can be a multipla of the overall time step.

5.3   Module Selection

The MIKE 21/3 Coupled Model FM comprises various modules, which you can select to be included in the simulation.

You can make your selection to explicitly include the following modules

- Hydrodynamic
- Transport
- MIKE ECO Lab / Oil Spill
- Mud Transport
- Particle Tracking
- Sand Transport
- Spectral Waves

by ticking one, or more, of the check boxes.

In case the Hydrodynamic model and the Sand Transport model both have been selected, you may choose to dedicate the morphology solution in the Sand Transport model to Shoreline morphology.

The two basic computational components, the Hydrodynamic Module and the Spectral Wave Module, can be used as stand-alone modules. The Sand Transport Module can also be used as a stand-alone module when the wave-

current mode is applied. In this case the flow and current conditions have to be specified as input to the model.

5.3.1 Shoreline morphology

It is not possible to calculate shoreline morphology in the Sand Transport model and at the same time apply the other process modules, such as the Transport Module, the ECO Lab/Oil Spill Module, the Mud Transport Module and the Particle Tracking Module.

6 HYDRODYNAMIC MODULE

The hydrodynamic module calculates the resulting flow and distributions of salt, temperature, subject to a variety of forcing and boundary conditions. Baroclinic effect due to salt and temperature variations, and turbulence, are considered as subordinated to the HD module and are set up here.

6.1 Basic Equation

The Hydrodynamic Module for 2D simulations includes two time formulations

- Quasi-stationary formulation
- Instationary formulation

Per default the hydrodynamic solver is assumed to be dynamic, thus using the instationary formulation. For idealized cases with nearly constant conditions, the simulations can be speed up by applying the Quasi-stationary formulation.

6.2 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.

For 2D simulation the time integration of the shallow water equations and the transport (advection-dispersion) equations is performed using an explicit scheme. For 3D simulation the time integration of the shallow water equations and the transport equations is performed using a semi-implicit scheme, where the horizontal terms are treated explicitly and the vertical terms are treated implicitly. Due to the stability restriction using an explicit scheme the time step interval must be selected so that the CFL number is less than 1. A variable time step interval is used in the calculation of both the shallow water equations and the transport equations determined so that the CFL number is less than a critical CFL number in all computational nodes. The CFL number for the transport equations is usually less restrictive than the CFL number for the shallow water equations. Hence, a larger time step interval is normally used for the transport equations than the shallow water equations. 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 and the time step interval for the shallow water equations is synchronized to match the time step interval for the transport equations.

In the quasi-stationary flow solution time is removed as an independent variable and a steady state solution is calculated at each time step. The flow solution is iterated in a pseudo time-domain until stationary conditions has been met.

6.2.1 CFL number

For the shallow water equations in Cartesian coordinates the Courant-Friedrich-Lévy (CFL) number is defined as

$$CFL_{HD} = (\sqrt{gh} + |u|)\frac{\Delta t}{\Delta x} + (\sqrt{gh} + |v|)\frac{\Delta t}{\Delta y} \qquad (6.1)$$

where h is the total water depth, u and v are the velocity components in the x- and y-direction, respectively, g is the gravitational acceleration, Δx and Δy are a characteristic length scale in the x- and y-direction, respectively, for an element and Δt is the time step interval. The characteristic length scale, Δx and Δy, is approximated by the minimum edge length for each element and the water depth and the velocity component is evaluated at element center.

For the transport equations in Cartesian coordinates the CFL number is defined as

$$CFL_{AD} = |u|\frac{\Delta t}{\Delta x} + |v|\frac{\Delta t}{\Delta y} \qquad (6.2)$$

6.2.2 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 stability of the numerical scheme should be secure if the CFL number less than 1. 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. Therefore the default value of the critical CFL number is set to 0.8. 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 (p. 49), 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.

For both the time integration of the shallow water equations and the transport equations, 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.

6.2.3 Quasi-stationary formulation

The quasi-stationary HD solver is suited for use in simulations where the wave and flow climate changes slowly (compared to the time it takes to iterate over a solution). This is often the case for simulation of idealized cases where a few sea states are used to describe the forcing conditions for equilibrium sand transport.

NOTE: This option should only be used in cases where the flow conditions are nearly constant. E.g. it is usually not a viable solution for a setup with tidal variations.

The forcings from e.g. the SW module depends on the HD solution. The number of time steps determines how many times this interdependence is activated, i.e. how many times the forcings must be updated during each overall time step. Usually this will be 1 for idealized stationary conditions.

The steady state solution is obtained as the steady state solution solving the instationary basic equations with steady boundary conditions and forcing. The same time integration procedure as used in the instationary mode is applied. However, the time step used in this iteration procedure is determined automatically to satisfy the stability restriction.

The input time step interval is the maximum duration to iterate within each overall time step.

The iteration procedure is stopped when the root-mean-square norm of a vector, containing the increment rate of a certain parameter between two iteration steps, is smaller than a user-specified tolerance. This requirement should be fulfilled for both the surface elevation and current velocity.

$$\frac{\left| S^{k+1} - S^k \right|}{dt} < TOL_1 \tag{6.3}$$

$$\frac{\left| \overline{V}^{k+1} - \overline{V}^k \right|}{dt} < TOL_2 \tag{6.4}$$

where *TOL1* is the tolerance of the change rate in surface elevation and *TOL2* is the tolerance of the change rate in current speed.

Recommended values

For idealized cases where the forcings don't change, the number of time steps where forcings must be updated can be set to 1. For cases where the forcings are specified by a few scenarios, the number of time steps should be increased.

If the tolerance is too high the algorithm may stop before the steady state solution is obtained.

6.2.4   Output of convergence information

For monitoring the convergence of the iteration procedure used in the Quasi-stationary formulation it is possible to write output of the overall convergence information and the domain convergence information.

Overall

For the output of overall convergence information you can select the following three options

- No output
- Standard output
- Standard output and point output

For the standard output the following items for each iteration step is written to a dfs0-file

- The time step
- The iteration number
- The RMS-norm of the change in water level between two iterations steps (dh/dt)
- The max-norm of the change in water level between two iterations steps (dh/dt)
- The RMS-norm of the change in current speed between two iterations steps (duv/dt)
- The max-norm of the change in current speed between two iterations steps (duv/dt)
- The time step interval

If "Standard output and point output" is selected the water level in a number of user-specified points is also written to the dfs0.

You must specify the name of the output file and if you choose "Standard output and point output" you must also specify the location of the points. 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 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. The third item is unused (but must be specified). The last item (the remaining of the line) is the name specification for each point. The point values in the output file are determined by piecewise linear interpolation. Hence, the point values are the discrete values for the element in which the specified point is located.

Domain

For the output of domain convergence information you can select the following three options

- No output
- Output after end calculation
- Intermediate output

If domain output is selected the following items is written to a dfsu-file

- The surface elevation
- The total water depth
- The U velocity
- The V velocity
- dh/dt
- duv/dt

You must specify the name of the output file and if you choose "Intermediate output" and you must also specify the frequency by which information should be stored.

6.3   Depth Correction

It is possible to define a correction, Δz, to the defined mesh bathymetry to be included in the simulations.

This utility can be applied for e.g. a morphology hotstart where bed changes need to be accounted for in a continuation of a previous simulation or in tsunami modelling to simulate the bed level fluctuations during an earthquake.

If included in the simulation, a data file with the depth correction values must be specified.

Note: It is not possible to apply a time-varying depth correction map in a simulation where morphological update of the bed level is applied as well.

Data

The format of the bathymetry correction can be specified as

- Varying in space, constant in time
- Varying in space, varying in time

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.

In case the bathymetry correction is defined as constant in time, the first time step in the bathymetry correction data file is applied as an initial condition in the simulation. In case the bathymetry correction is defined as varying in time, the bathymetry correction data must cover the complete simulation period.

6.3.1 General description

The corrected still water depth, *dcorrected*, is defined as

$$d_{corrected} = d_{initial} - \Delta z \tag{6.5}$$

where $d_{initial}$ is the initial water depth. A positive value for Δz will decrease the still water depth and a negative value will increase the still water depth.

At the initial time step the still water depth is corrected and the initial conditions for the surface elevation or water depth is satisfied. For the following time steps using time varying depth correction the still water depth is corrected and the surface elevation is corrected correspondingly to secure mass conservation.

6.4 Flood and Dry

The flood and dry can be specified in three different ways

- No flood and dry
- Standard flood and dry
- Advanced flood and dry (floodplain)

If your model is located in an area where flooding and drying occur, you must enable the flood and dry facility by selecting "Standard flood and dry" or

"Advanced flood and dry (floodplain)" (2D only). In this case you have to specify a drying water depth, a flooding water depth and a wetting depth.

When the water depth is less than the wetting depth the problem is reformulated and only if the water depth is less than the drying depth the element/cell is removed from the simulation. The flooding depth is used to determine when an element is flooded (i.e. re-entered into the calculation). The reformulation is made by setting the momentum fluxes to zero and only taking the mass fluxes into consideration.

If flooding and drying is not enabled, you should specify a minimum depth cut-off (see Section 5.1.2, Domain specification), which is less than zero. If the total water depth becomes less than zero a blow-up is detected and the simulation halted.

6.4.1   General description

The approach for treatment of the moving boundaries (flooding and drying fronts) problem is based on the work by Zhao et al. (1994) and Sleigh et al. (1998). When the depths are small the problem is reformulated and only when the depths are very small the elements/cells are removed from the calculation. The reformulation is made by setting the momentum fluxes to zero and only taking the mass fluxes into consideration.

The depth in each element/cell is monitored and the elements are classed as dry, partially dry or wet. Also the element faces are monitored to identify flooded boundaries.

- An element face is defined as flooded if the following two criteria are satisfied. Firstly, the water depth at one side of face must be less than a tolerance depth, h_{dry} , and the water depth at the other side of the face larger than a tolerance depth, h_{flood} . Secondly, the sum of the still water depth at the side for which the water depth is less than h_{dry} and the surface elevation at the other side must be larger than zero.

- An element is dry if the water depth is less than a tolerance depth, h_{dry} , and no of the element faces are flooded boundaries. The element is removed from the calculation.

- An element is partially dry if the water depth is larger than h_{dry} and less than a tolerance depth, h_{wet} , or when the depth is less than the h_{dry} and one of the element faces is a flooded boundary. The momentum fluxes are set to zero and only the mass fluxes are calculated.

- An element is wet if the water depth is greater than h_{wet} . Both the mass fluxes and the momentum fluxes are calculated.

In case the water depth become negative, the water depth is set to zero and the water is subtracted from the adjacent elements to maintain mass balance. In addition the conserved variables hu and hv in the adjacent element, where mass is subtracted, is corrected so that the velocities u and v remain the

same. When mass is subtracted from the adjacent elements the water depth at these elements may become negative. Therefore an iterative correction of the water depth is applied (max. 10 iterations).

When "Advanced flood and dry (floodplain)" is selected the momentum equation is suppressed as the water depth tends to the wetting depth. The suppression starts at two times the wetting depth. Additionally, the bed resistance is treated implicitly by calculating the bed resistance source term based on the solution estimated at the new time step. Finally, a correction of the velocities/fluxes is applied when the CFL number, estimated based on the calculated solution at the new time step, becomes larger than 1. In this case the velocities/fluxes are reduced so that the CFL number becomes less than 0.5.

Note: When an element is removed from the calculation, water is removed from the computational domain. However, the water depths at the elements, which are dried out, are saved and then reused when the element becomes floodded again.

6.4.2 Recommended values

The default values are: drying depth h_{dry} = 0.005m, flooding depth h_{flood} = 0.05m and wetting depth h_{wet} = 0.1m.

The wetting depth, h_{wet} , must be larger than the drying depth, h_{dry} , and flooding depth, h_{flood} , must satisfy

$$h_{dry} < h_{flood} < h_{wet} \tag{6.6}$$

In case of floodplain simulations the values for flooding depth, wetting depth and drying depth should be decreased. A reduction factor in the order of 50 is recommended.

6.4.3 Remarks and hints

For very small values of the tolerance depth, h_{wet} , unrealistic high flow velocities can occur in the simulation and give cause to stability problems. In this case the "Advanced flood and dry (floodplain)" option should be selected.

6.5 Density

The density is assumed to be a function of salinity and temperature. If you select barotropic mode, both temperature and salinity (TS) will be constant and the density will not be updated during the simulation. When the density is considered to be a function of temperature and/or salinity (baroclinic mode), the transport equation for the temperature and/or salinity must be solved. The solution of these equations is automatically invoked and the density is updated from the TS fields.

The specification of the setup for the solution of additional transport equations is described in the Section 7 TEMPERATURE/SALINITY MODULE.

The density is calculated using UNESCO's standard equation of state for sea water. These relations are applicable for temperatures ranging from –2.1 ºC to 40.0 ºC and salinities in the range from 0 to 45 PSU (Practical Salinity Unit).

The reference temperature and salinity can be used to improve the accuracy of the density calculation, as these are subtracted from the TS fields before the density calculation. If density is a function of temperature alone, the density is calculated using the actual temperature and the reference salinity, and vice versa for salinity.

6.5.1 Recommendation

Density gradients can be included in the hydrodynamic simulation if the water mass is well mixed, such that the temperature and salinity and thus the density is almost constant over the depth. The density gradient is counteracted by the water-level gradients and in estuaries density gradients may result in set-up of the water level of several cm. In most cases the effect of the density gradients in 2D shallow water equations is small and hence the density terms can be neglected. If the density variation over the depth is significant you should apply the 3-dimensional flow model.

6.5.2 Remarks and hints

If the density is included, the CPU time for the simulation is increased significantly as one or two additional transport equations are solved.

6.6 Eddy Viscosity

The decomposition of the prognostic variables into a mean quantity and a turbulent fluctuation leads to additional stress terms in the governing equations to account for the non-resolved processes both in time and space. By the adoption of the eddy viscosity concept these effects are expressed through the eddy viscosity and the gradient of the mean quantity. Thus the effective shear stresses in the momentum equations contain the laminar stresses and the Reynolds stresses (turbulence).

6.6.1 Horizontal eddy viscosity

The horizontal eddy viscosity can be specified in three different ways

- No eddy
- Constant eddy formulation

● Smagorinsky formulation

Selecting the constant eddy formulation you must specify the eddy coefficient, and selecting Smagorinsky formulation you must specify the Smagorinsky coefficient.

Data

The format of the eddy viscosity coefficient, or the Smagorinsky coefficient, can be specified as

● Constant (in domain)
● Varying in domain

For the case with values varying in domain the values vary in the horizontal domain only. In case of a 3D simulation the values are constant in the vertical domain.

For the case with values varying in domain you have to prepare a data file containing the eddy viscosity coefficient or the Smagorinsky coefficient 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 piecewice constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data.

Eddy parameters

Using the Smagorinsky formulation you must specify a minimum and maximum value for the eddy viscosity.

6.6.2   Vertical Eddy Viscosity

The vertical eddy viscosity can be specified in four different ways

● No eddy
● Constant eddy formulation
● Log law formulation
● k-ε formulation

Selecting the constant eddy formulation you must specify the eddy coefficient, ν_t .

The log law formulation uses a parabolic eddy coefficient, scaled with local depth and bed and surface stresses.

Selecting the k-ε formulation the eddy viscosity is determined as function of the turbulent kinetic energy (TKE), k, and the dissipation of TKE, ε . The two

additional transport equations must be solved for the TKE and the dissipation of TKE. The solution of these equations is automatically invoked. The specification of the setup for the solution of the additional transport is described in TURBULENCE MODULE.

Data

Using the constant eddy formulation the eddy viscosity coefficient ν_t can be specified in two different ways

- Constant (in domain)
- Varying in domain

For the case with values varying in domain the values are constant in the vertical domain and only varying in the horizontal domain. You have to prepare a data file containing the eddy viscosity coefficient 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 piecewice constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data.

Damping

When using the constant eddy formulation or the log-law formulation you can include a dependency on the vertical stratification by a Richardson number type damping. In that case you must specify the two coefficents a and b in the generalization of the Munk-Anderson formulation (see Section 6.6.3 General description).

Eddy parameters

Using the constant eddy or the log-law formulation you may specify a minimum and maximum value for the eddy viscosity.

6.6.3 General description

In many applications a constant eddy viscosity can be used for the horizontal stress terms. Alternatively, Smagorinsky (1963) proposed to express sub-grid scale transports by an effective eddy viscosity related to a characteristic length scale. The sub-grid scale eddy viscosity is given by

$$A = c_s^2 \, l^2 \sqrt{2 S_{ij} S_{ij}} \tag{6.7}$$

where c_s is a constant, l is a characteristic length and the deformation rate is given by

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \quad (i, j = 1, 2) \tag{6.8}$$

For more details on this formulation, the reader is referred to Lilly (1967), Leonard (1974), Aupoix (1984), and Horiuti (1987).

For the vertical stress terms several formulations for the eddy viscosity can be applied: a constant viscosity, vertically parabolic viscosity (log-law) and k-ε model (Rodi, 1980).

Log law
The eddy viscosity derived from the log-law is calculated by

$$\nu_t = U_\tau h + \left(c_1 \frac{z+d}{h} + c_2\left(\frac{z+d}{h}\right)^2\right) \tag{6.9}$$

where $U_\tau = max(U_{\tau s}, U_{\tau b})$, c_1 and c_2 are two constants, d is the still water depth and h is the total water depth. $U_{\tau s}$ and $U_{\tau b}$ are the friction velocities associated with the surface and bottom stresses, $c_1 = 0.41$ and $c_2 = -0.41$ gives the standard parabolic profile.

In applications with stratification the effects of buoyancy can be included explicitly. This is done through an introduction of Richardson number dependent damping of the eddy viscosity coefficient, when stable stratification occurs. The damping is a generalisation of the Munk-Anderson formulation (see Munk and Anderson, 1948)

$$\nu_t = \nu_t^*(1 + aRi)^{-b} \tag{6.10}$$

where ν_t^* is the undamped eddy viscosity and Ri is the local gradient Richardson number

$$Ri = -\frac{g}{\rho_0}\frac{\partial\rho}{\partial z}\left(\left(\frac{\partial u}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial z}\right)^2\right)^{-1} \tag{6.11}$$

$a = 10$ and $b = 0.5$ are empirical constants.

k-ε model
In the k-ε model the eddy-viscosity is derived from turbulence parameters k and ε using

$$\nu_t = c_\mu \frac{k^2}{\varepsilon} \tag{6.12}$$

where k is the turbulent kinetic energy per unit mass (TKE), ε is the dissipation of TKE and c_m is an empirical constant.

6.6.4   Recommended values

The Smagorinsky coefficient, C_S, should be chosen in the interval of 0.25 to 1.0.

When using the model for inland flooding the typical eddy viscosity value should be

$$E < 0{,}02 \cdot \frac{l^2}{\Delta t} \tag{6.13}$$

where l is a characteristic element length and Δt the time step.

A minimum value for the eddy viscosity can be chosen to zero, but more useful is a value in the order of the molecular viscosity. When applying k-ε model, it is recommended you use minimum values, which are consistent with the expression for determination of the eddy viscosity, i.e. $k_{min}=1.0 \cdot 10^{-7}$ m^2/s^2 and $\varepsilon_{min}=5 \cdot 10^{-10}$ m^2/s^3, yielding $=1.8 \cdot 10^{-6}$ m^2/s.

6.6.5   Remarks and hints

In the same way as for the bed resistance you can use the eddy coefficients to damp out numerical instability (see section 6.7 Bed Resistance). You should only use this as a last resort to your stability problem: The schematisation of the bathymetry and the boundary conditions are usually the primary causes for a blow-up.

When you use the Smagorinsky formulation of the turbulence the CPU time for a simulation is increased.

When you use the k-ε model formulation of the turbulence the CPU time for the simulation is increased significantly as two transport equations are solved.

6.7   Bed Resistance

In the case of 2D simulation the bed resistance can be specified in four different ways

- No bed resistance
- Chezy number
- Manning number
- Wave induced bed resistance

Selecting one of the two middle options you must specify the Checy number and the Manning number, respectively. Selecting the last option you must specify the effective grain diameter, the relative density of the bed material and some wave induced bed resistance parameters.

In the case of 3D simulation the bed resistance can be specified in four different ways

- No bed resistance
- Quadratic drag coefficient
- Roughness height
- Wave induced bed resistance

Selecting one of the middle options you must specify the quadratic drag coefficient and the roughness height, respectively. Selecting the last option you must specify the grain diameter, the relative density of the bed material and some wave induced bed resistance parameters.

Data

The format of the Chezy number, C, the Manning number, M, and the effective grain diameter, d, can be specified in one of three ways

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

For the case with values varying in domain you have to prepare a data file containing the Chezy number, the Manning number or the grain diameter 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 piecewice constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data. If the data is 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 hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

The drag coefficient, the roughness height, k_s, and the grain diameter, d_{50}, can be specified in one of three ways

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

For the case with values varying in domain you have to prepare a data file containing the drag coefficient, the roughness height or the grain diameter

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 piecewice constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data. If the data is 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 hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

Wave induced bed resistance

Using the wave induced bed resistance formulation you must specify some related parameters.

The type of calculation can be one of two:

- Parameterized
- Non-Parameterized

The applied wave height value can be restricted by the water depth. In case this option is enabled you need to specify the maximum value of the wave height/depth ratio.

You must also specify the Speed-up factor by which the bed resistance from the waves is recalculated. The default value is 1. Note that a value above 1 may lead to instabilities in the results.

6.7.1 General description

The bottom stress, $\overline{\tau_b}$, is determined by a quadratic friction law

$$\frac{\overline{\tau_b}}{\rho_0} = c_f\, \overline{u_b} \left| \overline{u_b} \right| \tag{6.14}$$

where c_f is the drag coefficient, $\overline{u_b}$ is the flow velocity above the bottom and ρ_0 is the density of the water.

For two-dimensional calculations $\overline{u_b}$ is the depth-average velocity and the drag coefficient can be determined from the Chezy number, C, or Manning number, M

$$c_f = \frac{g}{C^2} \tag{6.15}$$

$$c_f = \frac{g}{(Mh^{1/6})^2} \tag{6.16}$$

where h is the total water depth and g is the gravitational acceleration.

Please note that the relation between the Manning number and the bed roughness length, k_s, can be estimated using the following

$$M = \frac{25.4}{k_s^{1/6}}$$ (6.17)

Also note that the Manning number used here is the reciprocal value of the Manning´s n described in some textbooks.

For three-dimensional calculations $\overline{u}_b$ is the velocity at a distance Δz_b above the sea bed and the drag coefficient is determined by assuming a logarithmic profile between the seabed and the point at the distance Δz_b above the sea-bed

$$c_f = \frac{1}{\left(\frac{1}{\kappa}\ln\left(\frac{\Delta z_b}{z_0}\right)\right)^2}$$ (6.18)

where κ=0.4 is the von Kármán constant and z_0 is the bed roughness length scale. When the boundary surface is rough, z_0 depends on the roughness height,

$$z_0 = mk_s$$ (6.19)

where m is approximately 1/30.

The wave induced bed resistance can be determined from

$$c_f = \left(\frac{U_{fc}}{\overline{u}_b}\right)^2$$ (6.20)

where U_{fc} is the friction velocity calculated by considering the conditions in the wave boundary layer. For a detailed description of the wave induced bed resistance see Fredsøe (1984) and Jones et. al (2014).

6.7.2 Recommended values

If the relative variation of the water depth in a 2D simulation is considerable you should specify Manning numbers. Values in the range 20-40 $m^{1/3}$/s are normally used with a suggested value of 32 $m^{1/3}$/s if no other information is available.

If you use Chezy numbers values in the range 30 - 50 $m^{1/3}$/s are common for coastal and marine applications.

If you use Wave induced bed resistance the default value of the sediment density is 2.65, the default value of wave height/depth ratio is 0.85 and the default update value is 1.

Note that due to of the definition of the resistance numbers a smal resistance number corresponds to a high bed resistance and vice versa.

For a 3D simulation the default value of roughness height is 0.05 m. Normally the roughness height lies in the range between 0.01 – 0.3 m.

By definition a small value of the roughness height correspond to low friction and vice versa.

6.7.3 Remarks and hints

If the boundary conditions at one of your boundaries are inaccurate and you therefore have stability problems (blow-ups) at this boundary, you can specify a small area along the boundary with a very high resistance in order to remove noise generated by the boundary. Manning numbers in the range 5 - 10 $m^{1/3}$/s or roughness heights in the order of 1 m have been applied successfully. However, this method should only be used if it is impossible to improve the boundary conditions. Furthermore, the simulation results in the area which high bed resistance should be used with caution.

6.8 Coriolis Forcing

The effect of the coriolis force can be included in three different ways

- No Coriolis force
- Constant in domain
- Varying in domain

If the constant in domain option is selected, the Coriolis force will be calculated using a constant specified reference latitude (in degrees).

If the varying in domain option is selected, the Coriolis force will be calculated based on the geografical information given in the mesh file.

6.9 Wind Forcing

It is possible to take into account the effect of the wind on the flow field. The format of the wind data can be specified as

- Constant. The wind is blowing from the same direction and with the same magnitude for the whole simulation period and over the entire model area.

- Varying in time and constant in domain. The magnitude and direction varies during the simulation period but is the same over the entire model area.

- Varrying in time and domain. The magnitude and direction varies during the simulation period and over the model area.

NOTE: the directions are given in degrees and measured clockwise from true north to where the wind is blowing from.

Data

For the case with wind varying in time but constant in domain you have to prepare a data file containing the wind speed and direction (in degrees from true North) before you set up the hydrodynamic simulation. The data file must be a time series file (dfs0). This can be done by entering the data in an ASCII file using your normal editor and then reading this file into the standard data file format using the MIKE Zero Time Series Editor. The wind data must cover the complete simulation period. The time step of the wind 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.

For the case with wind varying both in time and domain you have to prepare a data file containing the wind speed components and air pressure before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). This can be done by either using one of the two MIKE 21 wind generating programs (cyclone generated wind and pressure, or wind generated on the basis of digitised pressure fields). Or you can enter the data in an ASCII file using your normal editor and then reading this file into the standard data file format using the MIKE Zero Grid Series Editor. 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. The wind data must cover the complete simulation period. The time step of the wind 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.

Neutral pressure

For the case with wind varying in both time and domain, you have to specify a reference or neutral pressure level.

Soft start interval

You can specify a soft start interval during which the wind speed is increased from 0 to the specified wind speed in order to avoid shock waves being gen-

erated in the model. By default the soft start interval is zero (no soft start). The wind direction is not altered by the soft start.

6.9.1 Wind friction

The wind friction can be specified either as a constant or varying with the wind speed. In the latter case, the friction is linearly interpolated between two values based on the wind speed and if the wind speed is below the lower limit or above the upper limit the friction is given the value corresponding to that limit.

6.9.2 General description

You can include the effects of a wind blowing over the model area in the following way. The surface stress, $\overline{\tau}_s$, is determined by the wind speed above the water. The stress is given by the following empirical relation

$$\overline{\tau}_s = \rho_a c_d \left| u_w \right| \overline{u}_w \tag{6.21}$$

where ρ_a is the density of air, c_d is the empirical drag coefficient of air, and $\overline{u}_w = (u_w, v_w)$ is the wind speed 10 m above the sea surface.

Please notice that the direction of the wind is given in degrees blowing **from** (relative to true north (see Figure 6.1).
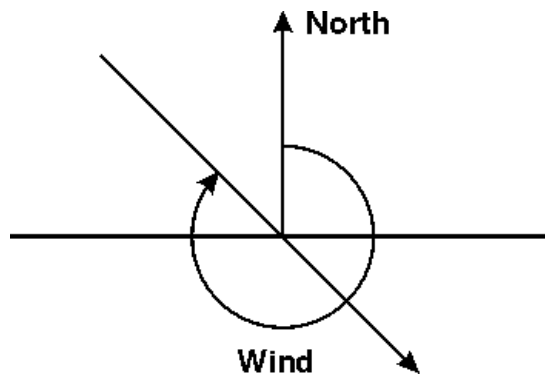
Figure 6.1   Definition of wind direction

The drag coefficient can either be a constant value or depend on the wind speed. The empirical formulae proposed by Wu (1980, 1994) is used for the parametrization of the drag coefficient.

$$c_d = \begin{cases} c_a & w_{10} < w_a \\ c_a + \dfrac{c_b - c_a}{w_b - w_a} \cdot (w_{10} - w_a) & w_a \leq w_{10} \leq w_b \\ c_b & w_{10} > w_b \end{cases} \tag{6.22}$$

where c_a, c_b, w_a and w_b are empirical factors and w_{10} is the wind speed 10 m above the sea surface.

The default values for the empirical factors are $c_a = 1.255 \cdot 10^{-3}$, $c_b = 2.425 \cdot 10^{-3}$, $w_a = 7$ m/s and $w_b = 25$ m/s. The value gives generally good results for open sea applications. Field measurements of the drag coefficient collected over lakes indicate that the drag coefficient is larger than open ocean data. For a detailed description of the drag coefficient see Geenaert and Plant (1990).

6.9.3 Remarks and hints

You can use the drag coefficient as a parameter in your model calibration.

6.10 Ice Coverage

It is possible to take into account the effects of ice coverage on the flow field. The ice coverage can be specified in four different ways

- No coverage
- Specified ice concentration
- Specified ice thickness
- Specified ice concentration and thickness

For the two cases where the ice concentration (the area coverage ratio of the ice) is specified the effect of the ice is taken into account in areas where the ice concentration becomes larger than a user-specified critical concentration (default value 0.9). For the case where only the ice thickness is specified the effect of the ice is taken into account in areas where the ice thickness is larger than zero.

When specifying ice thickness approximately 90% of the ice is submerged into the water. The precise value is found from ρ_{ice}/ρ_{water} , where $\rho_{ice} = 917$ kg/m^3 and ρ_{water} is the local water density. Note that the water level will be suppressed accordingly.

If included in the simulation, maps of the ice concentration and/or thickness must be specified.

In areas where the sea is covered by ice the wind stress is excluded that is the wind speed is set to zero. Additionally, the effect of the ice roughness on the flow field can be included. For this case you must specify the roughness height.

Ice coverage data

You have to prepare a data file containing the ice concentration and/ot thickness 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 data file must cover the model area. If a dfsu-file is used, piecewice constant interpolation is used to map the data and if a dfs2-file is used, bilinear interpolation is used. 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.

Critical concentration

In areas where the ice concentration becomes larger than the specified value of the critical concentration, the effect on the flow field is taken into account.

Roughness data

The format of the ice roughness height, k_s, 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 drag coefficient or the roughness height 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 data file must cover the model area. If a dfsu-file is used piecewice constant interpolation is used to map the data and if a dfs2-file is used bilinear interpolation is used.

6.11 Tidal Potential

The tidal potential is a force, generated by the variations in gravity due to the relative motion of the earth, the moon and the sun. The forcing acts throughout the computational domain. The forcing is considered as the sum of several harmonic terms, each representing a specific constituent, that are specified separately.

The format of the tidal potential data can be specified as

- Specified from dialog
- Specified from file

Data

The tidal potential is defined by the number of constituents that should be included and each constituent is described by a number of parameters, as shown in Table 6.1, where the parameters for the M_2 tidal constituent are shown as an example. The default is 11 constituents comprising M_2, O_1, S_2, K_2, N_2, K_1, P_1, Q_1, M_m, M_f and S_{sa}. There is no limit on the number of constituents that can be defined. The values for other constituents can be found in standard tidal books as for example (Pugh, 1987).

Table 6.1   Specification of the M_2 tidal constituent

Name	M_2	-	
Species	2	-	Semi-diurnal, i_0=2, Diurnal, i_0=1, Seasonal, i_0=0
Constituent	1	-	
Amplitude	0.242334	m	
Earthtide	0.693	-	Factor based on Love numbers
Period scaling	3600		Multiplies Period to obtain seconds
Period	12.42		Period of the constituent
Nodal no. 1	1.0	-	Nodal modulation factor
Nodal no. 2	-0.037	-	Nodal modulation factor
Nodal no. 3	-2.1	o	Nodal factor, angle
Argument 1	0	-	Doodson number, i_1
Argument 2	0	-	Doodson number, i_2
Argument 3	0	-	Doodson number, i_3
Argument 4	0	-	Doodson number, i_4
Argument 5	0	-	Doodson number, i_5
Phase	0	o	Phase angle correction

6.11.1 Remarks and hints

The tidal potential is a weak force and is usually significant in deep enclosed water bodies only as e.g. Mediterranean Sea or Gulf of Mexico or in large scale models of e.g. the Pacific Ocean. In model setups with open boundaries, these usually dominates the flow.

6.12   Precipitation-Evaporation

In applications where the rain fall is important for the flow, you can include precipitation in your simulation. The precipitation can be included in three different ways

- No precipitation
- Specified precipitation
- Net precipitation

The net precipitation is the precipitation minus the evaporation. Thus evaporation corresponds to negative net precipitation. Selecting the No precipitation or the Specified precipitation option, evaporation can also be included seperately in three different ways

- No evaporation
- Specified evaporation
- Computed evaporation

For the case with computed evaporation the evaporation rate is calculated as part of the latent heat flux. This option can only be selected if both density variations and heat exchange are included.

Selecting the specified precipitation (or net precipitation) option you must specify the precipitation rate. Selecting the specified evaporation option you must specify the evaporation rate.

Data

The format of the precipitation rate (or net precipitation rate) and evaporation rate can be specified as

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

If specified precipitation/evaporation is selected the precipitation/evaporation ´rates must be positive. If net precipitation is selected and the precipitation rate is negative it will act as evaporation.

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

For the case with precipitation rate varying both in time and domain you have to prepare a data file containing the precipitation rate 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 piecewice 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.

For the case with evaporation rate varying in time but constant in domain and for the case with evaporation rate varying both in time and domain the data file needed must be prepared in the same way as described above for the precipitation input.

Soft start interval

You can specify a soft start interval during which the precipitation/evaporation rate is increased linearly from 0 to the specified values of the precipitation/evaporation rate. By default the soft start interval is zero (no soft start).

6.13 Infiltration

The effect of infiltration and leakage at the surface zone may be important in cases of flooding scenarios on otherwise dry land.

6.13.1 Infiltration specification

The effect of infiltration can be defined in one of three ways:

- No infiltration
- Net infiltration
- Constant infiltration with capacity

Net infiltration

The net infiltration rate can be specified in two ways:

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

Constant infiltration with capacity

In this case you can only specify the infiltration in one way:

- Constant in time, varying in domain

Before selecting the data file you have to define the type of two parameters specified in the data file (item 3 and item 5).

The specified data file must contain the following 5 items:

1. Infiltration rate
2. Porosity of infiltration zone
3. Depth OR Level value describing the extent of the infiltration zone
4. Leakage rate (specified as type infiltration)
5. Initial water volume in Percentage of capacity (interval: 0-100 [%]) OR as a Water content (interval: 0-porosity[()])

Data

You have to prepare a data file containing the necessary item(s) 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 piecewice constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data. In case the values are 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 hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

6.13.2 General description

The most direct way of including infiltration is to specify the net infiltration rate.

It is however also possible to calculate the net infiltration rate by a simplified model that describes the infiltration from the free surface zone to the unsaturated zone and the leakage from the unsaturated zone to the saturated zone. This way the model can e.g. account for a decreased storage capacity due to previous rainfall events.

The model assumes the following:

- The unsaturated zone is modelled as an infiltration zone with constant porosity over the full depth of the zone.
- The infiltrated volume from the free surface zone and to the unsaturated zone is based on a constant flow rate, $V_{infiltration} = Q_i \times \Delta t$, where Q_i is the prescribed infiltration rate.
- The leaked volume from the unsaturated zone and to the saturated zone is based on a constant flow rate, $V_{leakage} = Q_l \times \Delta t$, where Q_l is the prescribed leakage rate.
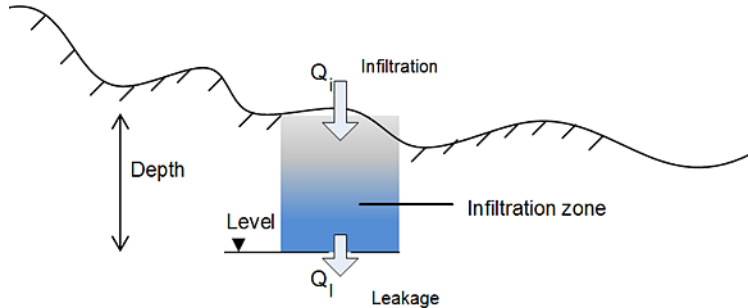
Figure 6.2   Definition of infiltration with storage and leakage:
In the case of net infiltration, $Q_{infiltration} = Q_{leakage}$, the model assumes an infinite storage volume in the infiltration zone.

NOTE: The infiltrated flow volume cannot exceed the amount of water available in the free surface water zone nor the difference between the water capacity of the infiltration zone and the actual amount of water stored there. It is possible that the infiltration flow completely drains the free surface zone from water and thus creates a dried-out point in the 2-dimensional horizontal flow calculations. In case the infiltration is described by Constant infiltration with capacity it is also possible that the infiltration zone becomes fully saturated so infiltration cannot take place (in which case Q_i is set to 0).

6.14   Wave Radiation

The second order stresses due to breaking of short period waves can be included in the simulation. The radiation stresses act as driving forces for the mean flow and can be used to calculate wave induced flow.

The radiation stresses can be included by reading values from a data file or directly from a coupled SW simulation.

If included in the simulation, a data file with the three components of the radiation stress divided by the density of the water, Sxx, Syy and Sxy, must be specified.

The data file containing the wave radiation stresses can be generated by the wave models MIKE 21 SW or MIKE 21 PMS.

Data

In case you choose to import a data file you have to prepare a data file containing the three components of the wave radiation stress divided by the den-

sity of water 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 piecewice 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.

Soft start interval

You can specify a soft start interval during which the wave radiation stresses are increased linearly from 0 to the specified values of the radiation stresses in order to avoid shock waves being generated in the model. By default the soft start interval is zero (no soft start).

6.14.1 Recommendations

In order not to generate a "shock" in the computations it is recommended to apply these stresses in combination with a soft-start (warm-up) period.

6.14.2 Remarks and hints

Since the wave radiation stresses describe the average flow over one wave period, the stresses are connected to a certain water depth. Application of the wave radiation stresses in simulations with time varying water depths (e.g. tide and/or storm surges) is possible, but the error introduced by the changes in water depth should be considered.

If the "Flood and Dry" facility is applied, the user should be sure that the wave radiation stresses are well defined in all grid points which will be flooded during the simulation. In general, it is not recommended to use wave radiation stresses together with the "Flood and Dry" facility.

6.15 Sources

The effects of rivers, intakes and outlets from power plants, etc. can be included in the simulation using point sources.

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

There are two different methods for specification of the sources. In the list view you can 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.

In the geographic view it is also possible to create a new source by double-clicking a position on the map or by selecting "New source" from the context menu. The name of the source must then be specified and the geographical position of the source can be edited. The additional information for the source is made subsequently.

6.15.1 Source specification

Up to four types of sources can be selected

- Simple source
- Standard source
- Connected source
- Jet

The type 'Jet' can only be selected for 3D calculations.

Simple source
For the simple source option only the source contribution to the continuity equation is taken into account. For this option you have to specify only the magnitude of the source. If the magnitude of the source is positive, water is discharged into the ambient water and if the magnitude is negative water is discharged out of the ambient water.

Standard source
For the standard source option both the source contribution to the continuity equation and the momentum equations is taken into account. For this option you have to specify both the magnitude of the source and the velocity by which the water is discharged into the ambient water. Note that the contribution to the momentum equation is only taken into account when the magnitude of the source is positive (water is discharge into the ambient water).

Connected source
For the connected source option both the source contribution to the continuity equation and the momentum equations is taken into account. For this option you have to specify the number of the source which the source is connected to .The magnitude of the source is then obtained as the magnitude of the source specified for the connected source, but with opposite sign. You also have to specify the velocity by which the water is discharged into the ambient water. A connected source should be defined after the source it connects to. Note that the contribution to the momentum equation is only taken into account when the magnitude of the source is positive (water is discharge into the ambient water).

Jet

For the jet option you have to specify the magnitude of the source. The magnitude of the source has to be positive. If the magnitude is negative the source is treated as a simple source, where water is discharged out of the ambient water. In addition you have to specify the diameter of the jet, the horizontal angle (clockwise from true North) and vertical angle (relative to horizontal, with 90 degree indicating flow toward the surface) and the maximum travel distance of the jet. You can also specify if an upstream approach should be applied for determination of the ambient flow conditions, and in that case you need to specify the minimum upstream distance. For more details on jet source see General description *(p. 83)*.

Note: For a MPI application the communication overhead in connection with jet sources is significant. The inclusion of a large number of jet sources in a simulation can therefore significantly increase the computational time.

Note: When running a simulation in decoupled mode the details of the jet source are not resolved and small deviation in the concentration field can be expected.

Location

You must select the map projection (Longitude/Latitude, UTM, etc.) in which you want to specify the horizontal location of the source point.

In case of a 3D simulation you must specify the horizontal coordinates and the vertical position. The vertical position can be specified in four different ways:

- Layer number
- Distance below surface
- Distance above bed
- z-coordinate

When layer number is selected you must specify the number of the layer in which the source should be placed. The layers are numbered 1 at the lowest layer and increasing upwards. In case of a combined sigma-z level mesh some of the bottom layers may be inactive, i.e. the bed layer number may be above 1, see Figure 6.3. In case a source is defined by a layer number which is below the active bed layer, the source will be applied into the active bed layer.
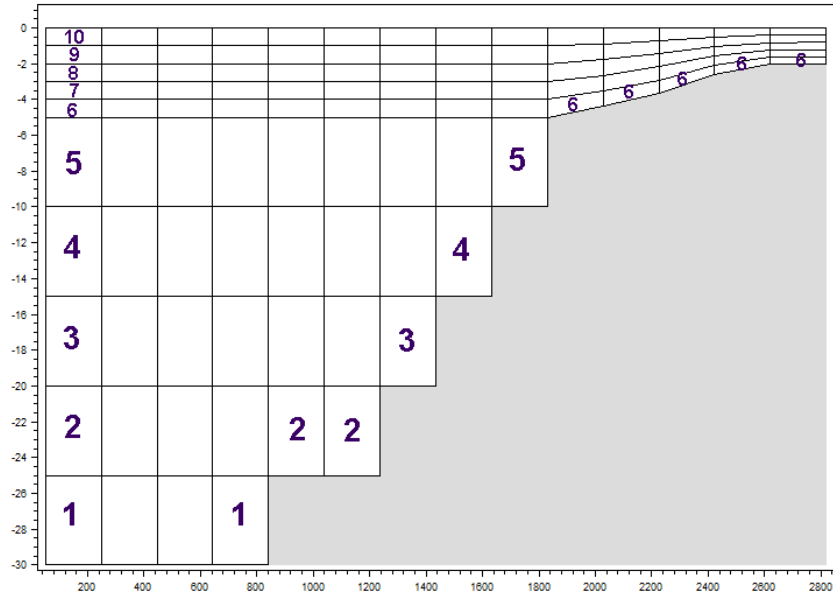
Figure 6.3   Example of layer numbers in source specification for combined sigma-z level mesh

When Distance below surface or Distance above bed is selected you must specify the distance. When z-coordinate is selected the z-coordinate of the source must be specified given in the coordinate system in which the bathymetry is defined and corrected corresponding to the datum shift specified on the domain page.

Data

The format of the source information can be specified as

- Constant in time
- Varying in time

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

6.15.2 Output

If output is included you must specify the name of the output file. The file will, for each jet source, contain information of the calculated jet.

The output file contains the following seven items for each jet source:

1. x end position - x-position when the jet/plume becomes passive

2. y end position - y-position when the jet/plume becomes passive

3. z end position - z-position when the jet/plume becomes passive

4. Characteristic width - diameter of the of the jet/plume when it becomes passive

5. Dilution factor - centerline dilution of a tracer or pollutant mass when the jet/plume becomes passive

6. Distance - the distance in horizontal plane from the initial jet/plume position to the end position.

7. Stop code - the stop code is an integer between 1 and 6 correspond to the six stop criteria listed in General description.

6.15.3 General description

The treatment of a jet/plume is based on dynamic coupling of near field integral jet model and the far field hydrodynamic model (MIKE 3 Flow Model FM).

The near field solution is determined using the integral jet model described by Jirka (2004). The jet model determines the steady state solution of the jet/plume by solving conservation equations for flux and momentum, salinity and temperature (if included) as well as the equations for the trajectory of the jet and the equation of state for the centerline density. The steady state solution at a certain point in time depends on the geometry of the jet efflux, the efflux characteristics (discharge, salinity, temperature) and the calculated ambient flow conditions (surface elevation, current conditions and stratification) from the far field model. The jet model employs an entrainment closure approach that distinguishes between the separate contributions of transverse shear and of azimuthal shear mechanisms. It further contains a quadratic law turbulent drag force mechanism as suggested by a number of recent detailed experimental investigations on the dynamics of transverse jets into crossflow. When the jet becomes passive it has reached its final position and becomes part of the ambient flow.

The effect of the diluted jet/plume on the far field model is represented by introducing a number of entrainment sinks along the centerline of the predicted jet trajectory and by introducing a source at the final position of the jet trajectory. This approach was proposed by Choi and Lee (2007). For each section in the jet calculation the entrainment of water (the volume flux) is calculated and is then introduced as a sink in the corresponding element in the far field model. If advection-dispersion calculation of temperature and salinity, passive substances, for cohesive or non-cohesive sediment transport or for ecological modelling (MIKE ECO Lab) is included in the simulation the tracer mass flux for the sinks is computed as the volume flux times the local solute

concentration. The discharge of water at the final position of the jet is determined as the sum of the total entrained amount of ambient water and the specified effluent discharge. The tracer mass flux at the release point is determined as the sum of the entrainment tracer fluxes and the effluent discharge times the effluent tracer concentration. This approach for representing the jet/plume in the far field model secures conservation of water and tracer mass. A distribution of the released material in the far field model at the final position of the jet is applied to get the right dilution in the far field model. Here a Gaussian distribution is used covering a circular plane corresponding to the final jet radius and perpendicular to the jet trajectory at the final position of the jet. If the jet reaches the surface/bottom level, a uniform distribution is applied covering a circular area over the bed/surface corresponding to the final jet radius.

The determination of the steady state jet solution and the corresponding sink and source information is performed at every hydrodynamic time step.

The ambient flow conditions can be determined either as the local flow conditions at the jet location or as the upstream ambient flow conditions. The upstream option can be used to avoid unrealistic feed-back between the jet solution and the ambient flow in cases with dominant advection effects on the released material from the ambient flow. For the upstream ambient flow condition, the conditions are obtained at a point defined by distance from the jet location in the upstream flow direction. The distance is the maximum of the characteristic length determined from the mesh and a user-specified minimum upstream distance. The characteristic length is here determined as 2.3 times the square root of the local element area at the initial release point. The upstream point for determined the ambient flow conditions is not allowed to be located on land. In that case the ambient flow conditions are determined at the last upstream point.

In order to determine, when the jet/plume becomes passive and thus a part of the ambient flow, a number of stop criteria are invoked:

1. If the jet reaches the water surface

2. If the jet reaches the bottom

3. If the jet momentum is smaller than or close to the ambient flow induced momentum - the jet excess momentum becomes less than 1% of the ambient flow induced momentum.
   (This happens in strong cross flows)

4. If the jet dies out in the stagnant conditions - In the absence of strong ambient flows, if the jet velocity drops below 1 cm/s.

5. If the jet experiences buoyancy reversal twice.
   (This happens in stratified environments with large density gradients)

6. If the trajectory of the jet exceeds a maximum travel distance as specified by the user. The travel distance is here defined as the distance in horizontal plane from the initial jet/plume position to the end position.

7. If the jet exhibits strong curvature, such as going into strong opposing ambient flow.

8. If the jet reaches a domain boundary (open or close). It releases in the first set of cells on its trajectory before the boundary.

9. If the jet reaches dried cells. It releases in the last set of wet cells on its trajectory before the dried point.

6.16   Structures

The horizontal dimension of structures is usually much smaller than the cell (element) sizes used in the computational grid. Therefore, the effect of structures is modeled by a subgrid technique (the flow past a structure is modelled by considering the upstream and downstream water level). The layers are not considered separately. Six different types of structures can be included in the simulations:

- Weirs
- Culverts
- Dikes
- Gates
- Piers
- Turbines

Furthermore you have the option to include composite structures by combining a weir and one or more culverts.

Note: A structure will only come into effect if placed in an area where water flow would exist without the structure.

6.16.1   Weirs

Depending on the choice of property page you can see a Geographic View or a List View of the weirs.

There are two different methods for specification of the weir. In the List View you can create a new weir clicking on the "New weir" button. By selecting a weir in the Weir list and clicking on the "Delete weir" button you can remove this weir. For each weir you can specify the name of the weir. The specification of detailed information for each weir 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 weir in the Weir list and clicking on "Edit weir" button.

In the Geographic View it is also possible to create a new weir by double-clicking a position on the map. This position will be the first point defining the cross-section for the weir or by selecting "New weir". The name of the weir must then be specified and the geographical position of the weir can then be edited. The additional information for the weir is made subsequently.

Weir specification

The location and geometrical layout of the weirs must be specified.
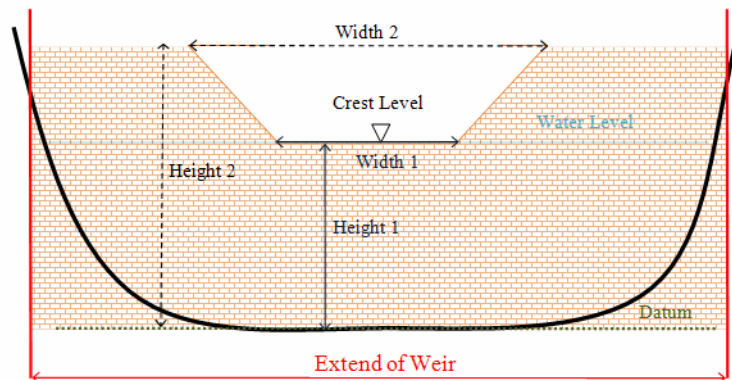
Figure 6.4   Setup definition of contracted weir

Location

Weirs are defined in the domain as a cross section specified as a list of points (a minimum of two points required). The section is composed of a sequence of line segments. The line segments are straight lines between two successive points. The 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 centers of the faces crosses one of the line segments. The geographical coordinates are taken from the dialog or from an ASCII file. The file format is two space separated floats (real numbers) for the x- and y-coordinate on separate lines for each of the points. The faces defining the line section are listed in the log-file.

You must also select the map projection (Longitude/Latitude, UTM, etc.) in which you want to specify the location of the line section for the weir.

For more information please see Location of a structure .

Attributes

The required formula to be applied for the specific weir-structure is defined in the attributes. A range of formulas are available:

- Broad Crested Weir formula
- Weir formula 1

- Weir formula 2 (Honma formula)

Valve

Valve regulation of the structure flow can be defined as part of the structure definition.

Four different valve regulation types are available:

- None: No valve regulation applies (flow is not regulated).
- Only Negative Flow
  Only flow in negative flow direction is allowed. Valve regulation does not allow flow in positive flow direction and the flow through the structure will be zero in this case.
- Only Positive Flow
  Only flow in positive flow direction is allowed. Valve regulation does not allow flow in negative flow direction and the flow through the structure will be zero in this case.
- No Flow
  No flow is allowed in the structure. Valve regulation closes completely the structure.

For more information please see Flow direction *(p. 115)*.

Alpha zero

Alpha zero is the water level difference at which the discharge calculation is described by a linear variation. If the water level difference is below this value the discharge gradients are suppressed.
The default setting is 0.01 meter. If a structure shows oscillatory behavior it is recommended to increase this value slightly.

For more information please see Alpha zero value *(p. 116)*.

Distribution

For weirs, culverts and Composite structures the total flux trough the structure is calculated based on the upstream and downstream surface elevation. The upstream and downstream surface elevation is determined as the mean value of the surface elevation at the real wet elements to the left and the right of the section of structure faces. Real wet elements are elements where the water depth is larger than wetting depth. The distribution of the calculated discharge along the section of structure faces can be specified in two ways

- Uniform
- Non-uniform

When non-uniform distribution is selected the discharge will be distributed as it would have been in a uniform flow field with the Manning resistance law applied, i.e. is relative to $h^{5/3}$, where h is the depth. This distribution is, in most

cases, a good approximation. This does not apply if there are very large variations over the bathymetry or the geometry. The distribution of the discharge only includes the faces for with the element to the left and the right of the face is a real wet element. In no elements on the downstream side of the structure are real wet elements the distribution is determined based on the upstream information.

For Composite structures the distribution for the first structure is applied.

Head Loss Factors
The factors determining the energy loss occurring for flow through hydraulic structures.

Following head loss factors shall be defined (for positive and negative flow direction):

- Inflow (contraction loss)
- Outflow (expansion loss)
- Free Overflow

Head Loss Factors are applied in structure flow calculation only for broad crested weir and special weir types.

For more information please see Head Loss Factors *(p. 90)* and Flow direction *(p. 115)*.

Geometry
The weir geometry defines the geometrical shape of the active flow area of the weir. Geometry is defined as a Level-width relationship, where the Level/Width table defines the Weir shape as a set of corresponding set of levels and flow widths. Values in the levels column must be continuous, increasing values.

For more information please see Level-width relationship *(p. 114)*.

Levels are defined relative to the datum (starting from the crest or sill level and up). E.g. for a horisontal weir positioned at invert level -10 m (bed level) and extending 6 m above the bed, the weir could should be defined by L_0= -4 m and L_1= 0 m.

Datum defines an offset which is added to the level column in the level/width table during computation. This is typically used if the weir geometry has been surveyed with respect to a local benchmark.

NOTE: Weir geometry shall only be defined for Broad Crested Weir.

General description

Broad Crested Weir

For a broad crested weir the user describes the shape of the "hole" through a level/width table (see Figure 6.5). The datum value for the structure may be used to shift the levels by a constant amount. The latter is typically used if the weir geometry has been surveyed with respect to a local benchmark.

Level	Width
L_0	0
L_1	W_1
L_2	W_2
L_3	W_3
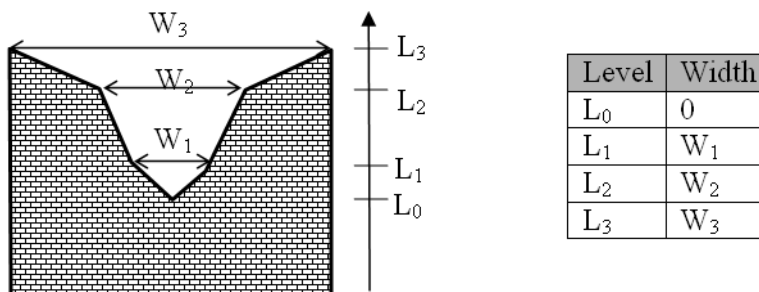
Figure 6.5   Definition sketch of broad crested weir geometry

The standard formulations for flow over a broad crested weir are established automatically by the program on the basis of the weir geometry and the user specified head loss and calibration coefficients. These formulations assume a hydrostatic pressure distribution on the weir crests. Different algorithms are used for drowned flow and free overflow, with an automatic switching between the two.

Weir formula 1

For the Weir formula 1 description the parameters are given by Figure 6.6. The width is perpendicular to the flow direction. Typically the invert level coincides with the overall datum.

Weir formula 1 is based on a standard weir expression, reduced according to the Villemonte formula:

$$Q = WC(H_{us} - H_w)^k \left[1 - \left(\frac{H_{ds} - H_w}{H_{us} - H_w} \right) \right]^{0,385} \tag{6.23}$$

where Q is discharge through the structure, W is width, C is weir coefficient, k is the weir exponential coefficient, H_{us} is upstream water level, H_{ds} is downstream water level and H_w is weir level (see Figure 6.6). The invert level is the lowest point in the inlet or outlet section respectively.
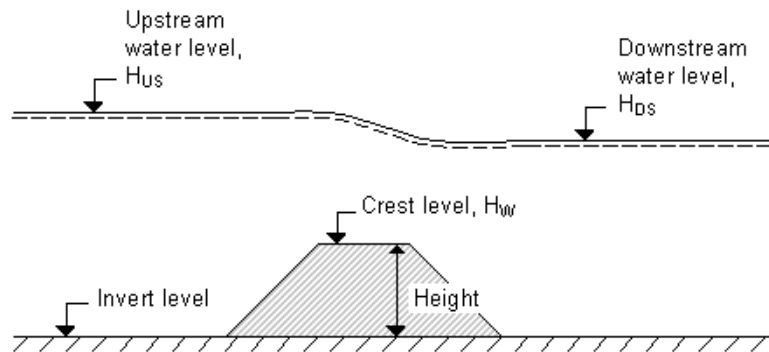
Figure 6.6   Definition sketch for Weir Flow

Weir formula 2

For the Weir formula 2 the geometry is given by a crest level and a width. The crest level is taken with respect to the global datum. The width is perpendicular to the flow direction.

Weir formula 2 is the Honma formula:

$$
Q = \begin{cases} C_1 W (H_{us} - H_w) \sqrt{(H_{us} - H_w)} & \text{for} \quad (H_{ds} - H_w)/H_{us} < 2/3 \\ C_2 W (H_{ds} - H_w) \sqrt{(H_{us} - H_{ds})} & \text{for} \quad (H_{ds} - H_w)/H_{us} \geq 2/3 \end{cases}
\tag{6.24}
$$

where Q is the discharge through structure, W is the width, C_1 is the weir coefficient 1, and C_2, the weir coefficient 2, is calculated according to $C_2 = 1,5\sqrt{3}\,C_1$. H_{us} is the upstream water level, H_{ds} is the downstream water level and H_w is weir level (see Figure 6.6).

Head Loss Factors

The flow description generally used for a structure is given by:

$$
\Delta H = \zeta_t \frac{V_s^2}{2g}
\tag{6.25}
$$

where ΔH is the energy loss over the structure, ζ_t is the total head loss coefficient and V_s is the mean cross sectional velocity at the structure.

The total head loss coefficient (ζ_t) is composed of entrance (ζ_1) and exit (ζ_2) coefficients. The coefficients are generally related to the input parameters for Inflow (ζ_{in}) and Outflow (ζ_{out}) and the changes in velocity (v) and area (A):

$$\zeta_t \;=\; \zeta_1 + \zeta_2 \;=\; \zeta_{in}\!\left(\frac{v_1}{v_s}\right) + \zeta_{out}\!\left(\frac{A_s}{A_2}\right)^2 \tag{6.26}$$

where suffix '1' and '2' represents velocity and area on inflow and outflow side of structure respectively, and 's' represents the velocity and area in the structure itself.

However, in the present implementation, upstream and downstream cross sections are not extracted and accordingly, tabulated relations on cross section areas as function of water levels are not known. Instead, upstream and downstream areas are set to a large number resulting in a full loss contribution from the head loss factors defined. Viz,

$$\zeta_t = \zeta_1 + \zeta_2 = \zeta_{in} + \zeta_{out} \tag{6.27}$$

Care must be taken when selecting loss coefficients, particularly in situations where both subcritical and supercritical flow conditions occur. When flow conditions change from subcritical to supercritical (or the Froude number FR becomes greater than 1), the loss coefficients ζ_{in} and ζ_{out} (specified in the Head Loss Factors box) are modified:

- If FR > 1 in upstream h-point then ζ_{in} = ζ_{in} / 2
- If FR > 1 in downstream h-point then ζ_{out} = ζ_{out} / 2

The critical flows (and orifice flows for culverts as well) are multiplied by the critical flow correction factor, α_c, specified as the Free Overflow Head Loss Factor. Typically a value of 1.0 is used.

Note: it is not allowed to have a weir crest level below the upstream or downstream ground level.

6.16.2 Culverts

Depending on the choice of property page you can see a Geographic View or a List View of the culverts.

There are two different methods for specification of the culverts. In the List View you can create a new culvert clicking on the "New culvert" button. By selecting a culvert in the Culvert list and clicking on the "Delete culvert" button you can remove this culvert. For each culvert you can specify the name of the culvert. The specification of detailed information for each culvert 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 culvert in the Culvert list and clicking on "Edit culvert" button.

In the Geographic View it is also possible to create a new culvert by double-clicking a position on the map or by selecting "New culvert" from the context menu. This position will be the first point defining the cross-section for the culvert. The name of the culvert must then be specified and the geographical position of the culvert can then be edited. The additional information for the culvert is made subsequently.

Culvert specification

The location and geometrical layout of the culverts must be specified.
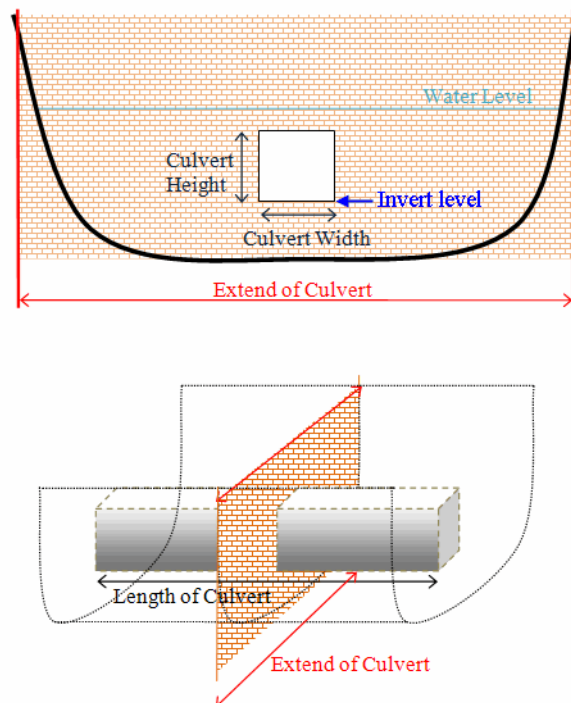
Figure 6.7   Setup definition of culvert

Location

Culverts are defined in the domain as a cross section specified as a list of points (a minimum of two points required). The section is composed of a sequence of line segments. The line segments are straight lines between two successive points. The 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 centers of the faces crosses one of the line segments. The geographical coordinates are taken from the dialog or from an ASCII file. The file format is two space separated floats (real numbers) for the

x- and y-coordinate on separate lines for each of the points. The faces defining the line section are listed in the log-file.

You must also select the map projection (Longitude/Latitude, UTM, etc.) in which you want to specify the location of the line section for the culvert.

For more information please see Location of a structure *(p. 113)*.

Attributes

A number of parameters define the culvert characteristics:

- Upstream Invert
  Invert level at upstream location of the culvert.

- Downstr. Invert
  Invert level at downstream location of the culvert.

  NOTE: Upstream and downstream location is not necessarily the inlet and outlet location, respectively.

- Length
  Length of the culvert.

- Manning's n
  Manning's bed resistance number along the culvert (for friction loss contribution). n = 1/M (Manning number)

- No. of Culverts
  'No. of Culverts' is a number identifying how many culverts exist at the specific culvert location with identical geometrical definition.

  An example; Five identical shaped draining pipes are placed just next to each other in an earth dam, and in order not to make 5 individual culvert definitions - one for each pipe - the 'No. of Culverts' in this case can be defined as 5 and the simulation engine will recognize that 5 culverts of identical shape and size are located here and flow calculations will take this into account accordingly.

- Alpha zero
  Alpha zero is the water level difference at which the discharge calculation is described by a linear variation. If the water level difference is below this value the discharge gradients are suppressed.
  The default setting is 0.01 meter. If a structure shows oscillatory behavior it is recommended to increase this value slightly.

  For more information please see Alpha zero value *(p. 116)*.

Valve

Valve regulation of the structure flow can be defined as part of the structure definition.

Four different valve regulation types are available:

- None: No valve regulation applies (flow is not regulated).

- Only Positive Flow
  Only flow in positive flow direction is allowed. Valve regulation does not allow flow in negative flow direction and the flow through the structure will be zero in this case.

- Only Negative Flow
  Only flow in negative flow direction is allowed. Valve regulation does not allow flow in positive flow direction and the flow through the structure will be zero in this case.

- No Flow
  No flow is allowed in the structure. Valve regulation closes completely the structure.

For more information please see Flow direction *(p. 115)*.

Section Type
A culvert structure can be defined as either a Closed or an Open section type.

For more information please see Closed / Open Section switch *(p. 95)*.

Distribution

For weirs, culverts and Composite structures the total flux trough the structure is calculated based on the upstream and downstream surface elevation. The upstream and downstream surface elevation is determined as the mean value of the surface elevation at the element to the left and the right of the section of structure faces. Here only real wet elements. Real wet elements are elements where the water depth is larger than wetting depth. The distribution of the calculated discharge along the section of structure faces can be specified in two ways

- Uniform
- Non-uniform

When non-uniform distribution is selected the discharge will be distributed as it would have been in a uniform flow field with the Manning resistance law applied, i.e. is relative to $h^{5/3}$, where h is the depth. This distribution is, in most cases, a good approximation. This does not apply if there are very large variations over the bathymetry or the geometry. The distribution of the discharge only includes the faces for with the element to the left and the right of the face is a real wet element. In no elements on the downstream side of the structure are real wet elements the distribution is determined based on the upstream information.

For Composite structures the distribution for the first structure is applied.

Head Loss Factors

The factors determining the energy loss occurring for flow through hydraulic structures.

Following head loss factors shall be defined (for positive and negative flow directions):

- Inflow (contraction loss)
- Outflow (expansion loss)
- Free Overflow
- Bends

For more information please see Head Loss Factors *(p. 96)*.

Geometry

The culvert geometry defines the geometrical shape of the active flow area of the culvert. The cross sectional geometry of a culvert can be specified as:

- Rectangular
  The width and height specify the geometry.

- Circular
  The geometry is specified by the diameter.

- Irregular Level-Width Table
  The geometry is specified using a level/width table. Values in the level column must be increasing.

The Level/Width table defines the Weir shape as a set of corresponding levels and flow widths. Values in the levels column must be continuous, increasing values.

General description

Closed / Open Section switch

A culvert structure can be modelled as an open section if required by setting the Closed / Open switch. An example where this may be used is a "long" weir where the friction along the length of the weir is of importance and/or the flow areas at the entrance and exit are significantly different.

If set to open the culvert will never run full or partially full, therefore only those flow conditions which represent a free water surface are modelled. When the water level is higher than the soffit the hydraulic parameters are calculated based on a section extended vertically upwards with a width equal to that at the soffit. For example, in the case of a rectangular section the height value is essentially redundant as the cross-section will be modeled as an open section of constant width.

In the case of a circular section, this switch is invalid and will be set to closed.

Head Loss Factors

The total head loss, *DH* through a culvert is given by:

$$\Delta H_{loss} = \frac{Q^2}{2g}\left(\frac{\zeta_1}{A_{s_1}^2} + \frac{\zeta_f + \zeta_b}{A_{s_a}^2} + \frac{\zeta_2}{A_{s_2}^2}\right) \tag{6.28}$$

where A_s is the mean cross section area along the length of the culvert and Q is the discharge, $\zeta 1$ is the entrance or contraction loss, $\zeta 2$ is the outlet or expansion loss, ζf is the Friction loss calculated using the Manning formula and ζb is the bend loss coefficient.

$$\zeta_1 = \zeta_{in}\left(1 - \frac{A_s}{A_1}\right) \tag{6.29}$$

$$\zeta_2 = \zeta_{out}\left(1 - \frac{A_s}{A_2}\right)^2 \tag{6.30}$$

Cross section areas, A_1 and A_2 are not processed and extracted in the present implementation and hence, defined as an infinite value. Contraction and expansion losses are therefore assumed to be applied in full using the defined inflow and outflow loss coefficients, ζ_{in} and ζ_{out}.

$$\zeta_f = \frac{2gLn^2}{R^{4/3}} \tag{6.31}$$

where L is the culvert length, n is Manning's coefficient and R is the mean hydraulic radius along the culvert. The Manning's n-value depends on the interior surface of the culvert. Table values can be found in litterature. For e.g. a concrete culvert n would typically range from 0.011 to 0.017 .

The bend loss coefficient, ζb, is provided for situations where head losses other than from the above occur, for example bends, damaged culverts, trapped debris. For straight culverts in good condition a value of zero would apply.

The critical flows (and orifice flows for culverts as well) are multiplied by the critical flow correction factor, α_c, specified as the Free Overflow Head Loss Factor. Typically a value of 1.0 is used.

6.16.3 Dikes

Depending on the choice of property page you can see a Geographic View or a List View of the dikes.

There are two different methods for specification of the dike. In the List View you can create a new dike clicking on the "New line" button. By selecting a dike in the Dike list and clicking on the "Delete line" button you can remove this dike. For each dike you can specify the name of the dike and whether the dike should be active or not. The specification of detailed information for each dike is made subsequently. From the List View page you can navigate to the dialog for specification by clicking on the "Go to .." button.

In the Geographic View it is also possible to create a new dike by selecting "Add New line" from the context menu. Click once to add a point and twice to add the end point of the dike. You may edit the individual dike by selecting and moving the specified points (point, click and drag) or using the functionalities from the context menu (point, right-click). The additional information for the dike is entered subsequently.

Dike specification

The data for the dike and the location and geometry of the dike must be specified.

The discharge over the dike can be specified in three different ways

- Empirical formula
- Specify the overtopping discharge
- Calculate the overtopping discharge from a user-specified table

Empirical formula
When "Empirical formula" is selected you have to specify the critical level difference and the weir coefficient.

When the water level gradient across a structure is small, the corresponding gradient of the discharge with respect to the water levels is large. This in turn may result in a very rapid flow response to minor changes in the water level upstream and downstream. As a way of controlling this effect, a critical level difference has been introduced. The critical water level difference defines the water level difference below which the discharge gradients are suppressed. The default setting is 0.01 meter. If a structure shows oscillatory behavior it is recommended to increase this value slightly.

Specify the overtopping of discharge
When "Specify the overtopping discharge" is selected the discharge can be specified as

- Constant
- Varying in time and constant along curve
- Varying in time and varying along curve

When "Constant" is selected you have to specify the constant discharge and the start time and end time for which the discharge should be applied.

When "Varying in time and constant along curve" is specified you have to prepare a data file containing the discharge before you set up the simulation. The file must be a dfs0 file. The data must cover the complete simulation period when "Varying in time and varying along curve" is specified for the format. 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.

When "Varying in time and varying along curve" is specified you have to prepare a data file containing the discharge before you set up the simulation. The file must be a dfs1 file, where the number of grid points corresponds to the number of points, which is used to define the location of the dike. The data must cover the complete simulation period when "Varying in time and varying along curve" is specified for the format. 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.

The discharge is positive when the flow occurs from the left of the dike to the right of the dike when positioned at the first point and looking forward along the cross-section line, see Figure 6.8.

Calculate the overtopping discharge from a user-specified table

When "Calculate the overtopping discharge from a user-specified table" is selected, you must enter the file name for the file containing the discharge table. This table should contain the discharge (m^3/s/m) as function of the discrete values of the freeboard (m), the significant wave period (m) and the peak wave period (s). For the description of the format of the file see ASCII format for the discharge table *(p. 98)*. The freeboard is defined as the crest level minus the surface elevation. The surface elevation, the significant wave height and the peak wave period are determined as the values to the left of the dike when positioned at the first point and looking forward along the cross-section line.

The discharge is positive when the flow occurs from the left of the dike to the right of the dike when positioned at the first point and looking forward along the cross-section line, see Figure 6.8.

ASCII format for the discharge table

This is a general format which can be generated by most spread sheets or text editors.

The first part of the ASCII file must contains the header Information. The header information consists of three lines each with three space separated items:

- The first line contains the number of discrete freeboards, *nf*, the minimum freeboard, *fmin*, and the maximum freeboard, f_{max}.
- The second line contains the number of discrete significant wave heights, *nh*, the minimum significant wave height, h_{min}, and the maximum significant wave height, h_{max}.
- The third line contains the number of discrete peak wave periods, *nt*, the minimum peak wave period, t_{min}, and the maximum peak wave period, t_{max}.

Data follows after the header information. The data consist of the discharge (m³/s/m), qi,j,k , as function of discrete values of the freeboard, f_i, significant wave height, h_j, and peak wave period, t_k.

$$f_i = f_{min} + (i-1)\frac{f_{max} - f_{min}}{nf - 1} \qquad i = 1, nf$$

$$h_j = h_{min} + (j-1)\frac{h_{max} - h_{min}}{nh - 1} \qquad j = 1, nh \qquad (6.32)$$

$$t_k = t_{min} + (k-1)\frac{t_{max} - t_{min}}{nt - 1} \qquad k = 1, nt$$

The structure of the table can be expressed as follows:

nf f_{min} f_{max}
nh h_{min} h_{max}
nt t_{min} t_{max}
q1,1,1 q2,1,1 q3,1,1 ... q*nf*,1,1
q1,2,1 q2,2,1 q3,2,1 ... q*nf*,2,1
...
q1,nh,1 q2,nh,1 q3,nh,1 ... qnf,nh,1
q1,1,2 q2,1,2 q3,1,2 ... qnf,1,2
q1,2,2 q2,2,2 q3,2,2 ... q*nf*,2,2
...
q1,nh,2 q2,nh,2 q3,nh,2 ... qnf,nh,2
...
q1,1,nt q2,1,nt q3,1,nt ... qnf,1,nt
q1,2,nt q2,2,nt q3,2,nt ... qnf,2,nt
...
q1,nh,nt q2,nh,1 q3,nh,nt ... qnf,nh,nt

Location and geometry
The location in the horizontal domain of a structure may be given by a number of geo-referenced points which together make up a polyline. The poly-line defines the width of the structure perpendicular to the flow direction. 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 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 centers of the faces crosses one of the line segments. If two faces in a triangular element are part of the same face section, the face section is corrected so that these two faces are excluded from the face section and instead the third face in the triangle is applied.
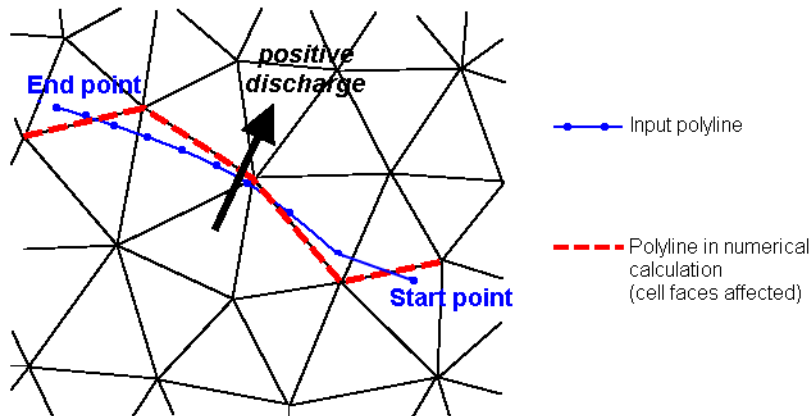

Figure 6.8   The location of a dike. Note the affected cell faces.

The crest level of the structure can be specified as:

- Constant
- Varying in domain

When Constant is specified you have to specify the constant crest level. The geographical coordinates and the crest level, when "Varying in domain" is selected for the crest level, are taken from the dialog or from an ASCII file. The file format is three space separated floats (real numbers) for the x- and y-coordinate and the crest level on separate lines for each of the points. The faces defining the line section are listed in the log-file.

You must also select the map projection (Longitude/Latitude, UTM, etc.) in which you want to specify the location of the line section for the structure.

Crest level correction

It is possible to specify a correction of the crest level. The format for the crest level correction can be specified as:

- Constant
- Constant in time and varying along curve
- Varying in time and varying along curve

When "Constant in time and varying along curve" and "Varying in time and varying along curve" is specified you have to prepare a data file containing crest level correction before you set up the simulation. The file must be a dfs1 file, where the number of grid points corresponds to the number of points,

which is used to define the location of the dike. The data must cover the complete simulation period when "Varying in time and varying along curve" is specified for the format. 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.

General description

When the empirical formulation is used the discharge, Q, over a section of the dike corresponding to an element face with the length (width), W, is based on a standard weir expression, reduced according to the Villemonte formula:

$$Q = WC(H_{us} - H_w)^k \left[1 - \left(\frac{H_{ds} - H_w}{H_{us} - H_w} \right) \right]^{0,385} \tag{6.33}$$

where Q is discharge through the structure, W is width, C is weir coefficient, k is the weir exponential coefficient, H_{us} is upstream water level, H_{ds} is downstream water level and H_w is weir level taken with respect to the global datum (see Figure 6.9). The value of the weir exponent is 1.5 and the default value of the weir coefficient is 1.838.
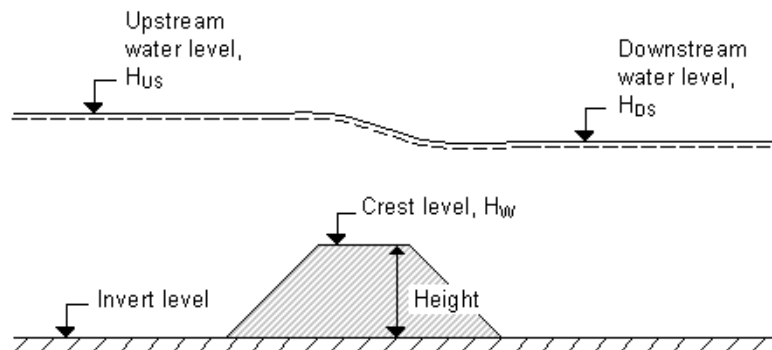

Figure 6.9   Definition sketch for Dike Flow

Note: The crest level applied in the calculations is defined as the highest of three values: the user-specified crest level, the ground level in upstream element or the ground level in downstream element.

6.16.4 Gates

Depending on the choice of property page you can see a Geographic View or a List View of the gates.

There are two different methods for specification of the gates. In the List View you can create a new gate clicking on the "New gate" button. By selecting a gate in the Gate list and clicking on the "Delete gate" button you can remove this gate. For each gate you can specify the name of the gate and whether the gate should be active or not. The specification of detailed information for each gate is made subsequently. From the List View page you can navigate to the dialog for specification by clicking on the "Go to .." button or by selecting a gate in the Gate list and clicking on "Edit gate" button.

In the Geographic View it is also possible to create a new gate by double-clicking on a position on the map or by selecting "New gate" from the context menu. This position will be the first point defining the line-section for the gate. The name of the gate must then be specified and the geographical position of the gate can then be edited. The additional information for the gate is entered subsequently.

Gate specification

The location of the gate and the control information for operation of the gate is specified.

Location

Gates are defined in the domain as a line section specified as list of points (a minimum of two points is required). The section is composed of the sequence of line segments. The line segments are straight lines between two successive points. The cross section in the numerical calculations is defined as a section of the nearest element faces. The face is included in the section when the line between the two element centers of the faces crosses one of the line segments. The geographical coordinates are taken from the dialog or from an ASCII file. The file format is two space separated floats (real numbers) for the x- and y-coordinate on separate lines for each of the points. The faces defining the line section are listed in the log-file.

You must also select the map projection (Longitude/Latitude, UTM, etc.) in which you want to specify the location of the line section for the gate.

For more information please see Location of a structure *(p. 113)*.

Gate Data

You must define the geometry of the gate along with the operation type.

The geometry can be described in two ways:

- Full water column
- Subset of column

If a subset of the column is requested you have to define the top level and bottom level in the water column set that is to be affected by the gate.

NOTE: The water level at the gate is derived by the average water level of the adjacent elements as shown in Figure 6.10.

The operation of the gate can be defined in three ways:

- User defined
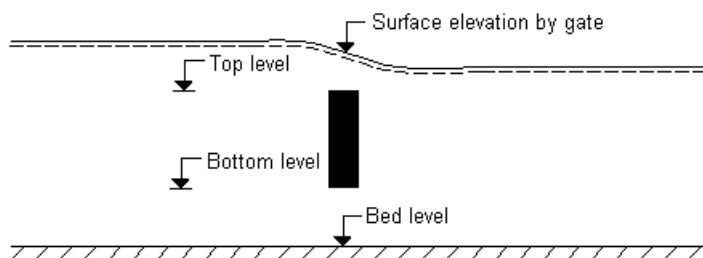- Water level control
- Water level difference control

Figure 6.10   Definition sketch for Gate Flow

User defined

In case the operation is to be user defined the gate is controlled by a factor. If the factor is 1 the gate is open and if the factor is 0 the gate is closed. When the factor is in the interval between 0 and 1 the gate is partly closed. The format of the control factor can be specified as:

- Constant in time
- Varying in time

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

Water level control

In case the operation is to be controlled by surface elevation, you have to define the location of the control point from where the controlling water level is extracted.

The gate is controlled based on the given control values. Two modes of operation are possible.

If you have specified a Close level greater than or equal to the Open level then:

When the simulated water level at the control point becomes lower than the value of the defined open level, the gate will open within the duration of the defined open interval. When the simulated water level at the control point becomes higher than the value of the defined close level, the gate will close within the duration of the defined close interval. Initially the gate is open.

If you have specified a Close level lower than the Open level then:

When the simulated water level at the control point becomes higher than the value of the defined open level, the gate will open within the duration of the defined open interval. When the simulated water level at the control point becomes lower than the value of the defined close level, the gate will close within the duration of the defined close interval. Initially the gate is closed.
If after the initial opening, you do not want the gate to ever close, specify a close level which can never be reached (say a level lower than the bathymetry).

An instant internal control factor will be applied using linear interpolation during the open/close interval.

Water level difference control

In case the operation is to be controlled by surface level difference you have to define the locations of the two control points from where the controlling water levels are extracted. The water level difference is regarded as positive if the water level in control point 1 is higher than the water level in control point 2.

The gate is controlled based on the given control values. When the simulated water level difference becomes lower than the value of the defined open level, the gate will open within the duration of the defined open interval. When the simulated water level difference becomes larger than the value of the defined close level, the gate will close within the duration of the defined close interval. Initially, the gate is closed.

An instant internal control factor will be applied using linear interpolation during the open/close interval.

General description

Gate in 2D simulation

The location of the gate is defined as a section of element faces. At a gate face with the length, Δl, the normal flux is calculated as the sum of the normal flux for a solid wall, where the length is determined as $(1-c)*\Delta l$, and the normal flux for a standard wet face, where the length is $c*\Delta l$. Here, c is a weighting factor with a value between 0 and 1, where 0 correspond to a closed gate and 1 correspond to an open gate. If the gate is defined as the full water column, c is equal to the user-defined gate control factor. If the gate geometry is defined as a subset of the water column the vertical location of the gate is

defined by the top level, z_{top}, and bottom level of the gate, z_{bottom}, (see Figure 6.10). In this case c is equal to the user-defined gate control factor multiplied by the fraction, f:

$$f = \frac{z_{top} - z_{bottom}}{s_{gate} - z_{gate}} \tag{6.34}$$

Here surface elevation at the gate, s_{gate}, is obtained as the mean value of the surface elevation at elements to the left and to the right of the gate face and the bed level at the gate, z_{gate}, is obtained as the mean value of the bed level at elements to the left and to the right of the gate face.

Gate in 3D simulation

The location of the gate is defined as a section of element faces. At a gate face with the area, ΔA, the normal flux is calculated as the sum of the normal flux for a solid wall, where the area is determined as $(1-c)*\Delta A$, and the normal flux for a standard wet face, where the area is $c*\Delta A$. Here, c is a weighting factor with a value between 0 and 1, where 0 correspond to a closed gate and 1 correspond to an open gate. If the gate is defined as the full water column, c is equal to the user-defined gate control factor. If the gate geometry is defined as a subset of the water column the vertical location of the gate is defined by the top level, z_{top}, and bottom level of the gate, z_{bottom}, (see Figure 6.10). In this case c is equal to the user-defined gate control factor multiplied by the fraction, f, corresponding to the fraction of the element face, which is blocked by the gate.

6.16.5 Piers

The effect of bridge piers is modeled as sub-grid structures using a simple drag-law to capture the increasing resistance imposed by the piers as the flow speed increases.

Depending on the choice of property page you can see a Geographic View or a List View of the piers.

There are two different methods for specification of the piers. In the List View you can create a new pier by clicking on the "New pier" button. By selecting a pier in the Pier list and clicking on the "Delete pier" button you can remove this pier. For each pier you can specify the name of the pier and whether the pier should be active or not. The specification of detailed information for each pier is made subsequently. From the List View page you can navigate to the dialog for specification by clicking on the "Go to .." button or by selecting a pier in the Pier list and clicking on the "Edit pier" button.

In the Geographic View it is also possible to create a new pier by double-clicking on a position on the map or by selecting "New pier" from the context menu. The name of the pier must then be specified and the geographical

position of the pier can then be edited. The additional information for the pier is made subsequently.

Pier specification

The location and geometrical layout of the piers must be specified.

Location
You specify the horizontal coordinates in which the pier should be placed.

You must also select the map projection (Longitude/Latitude, UTM, etc.) in which you want to specify the horizontal location of the pier point.

Pier Data
First you specify the angle of the pier, the streamline factor and the number of vertical pier sections. Then you specify the geometry.

The angle is measured from projection north to the alignment, positive clockwise and in degrees, see Figure 6.11. Note that projection north is not necessary the same as geographical north.
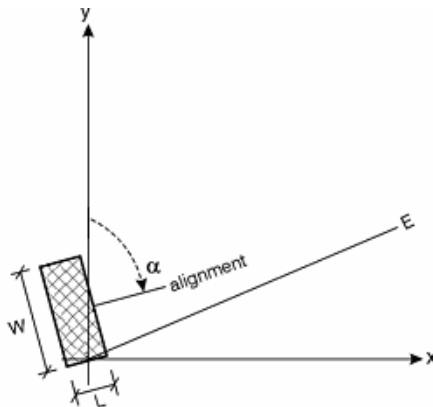
W : width of pier section
L : length of pier section
α : angle between projection north and alignment
x,y : the MIKE 3 grid coordinate system

Figure 6.11   Definition of pier angle

The streamline factor is a factor that is multiplied on the total drag force to take into account the increased flow velocity due to the blocking of piers. A typical value is 1.02.

The number of sections means the number of pier segments, i.e. the number of vertical sections with different geometrical layout.

Geometry

Additionally, you specify the geometry for each section. You have to specify the type of pier. The type can be one of three:

- Circular
- Rectangular
- Elliptical

For each section you specify a number of parameters depending on the type of section. For a circular section the height and the width (diameter) must also be specified. For a rectangular section the height, the width, the length and the radius of the corners must be specified. Finally, for an elliptical section the height, the width and the length must be specified. The sections are ordered so that the first section is at the top of the pier structure and the last section is at the bottom of the pier structure (at the sea bed), see Figure 6.12.
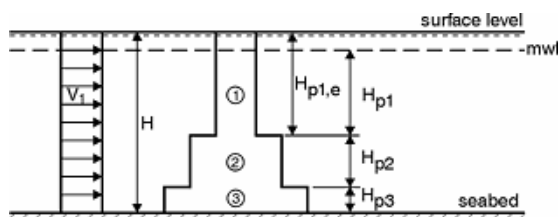
Example : Effective height for pier section:

$$H_{p1} = \max \left\{ (H - H_{p2} + H_{p3}), 0 \right\}$$
$$H_{p2} = \max \left\{ (H - H_{p3} - H_{p1e}), 0 \right\}$$
$$H_{p3} = \min \ (H_{p3}, H)$$

Figure 6.12   Definition of pier sections

General description

The effect on the flow due to the piers is modeled by calculating the current induced drag force on each individual pier.

The effective drag force, F, is determined from

$$F = \frac{1}{2} \rho_w \gamma C_D A_e V^2 \tag{6.35}$$

where ρ_w is the density of water, γ is the streamline factor, C_D is the drag coefficient, A_e is the effective area of pier exposed to current and V is the current speed. The sign of F is such that a positive force acts against the current direction.

6.16.6 Turbines

The effect of tidal turbines is modelled as sub-grid structures using a simple drag-law to capture the increasing resistance imposed by the turbine blades as the flow speed increases. Turbines are assumed always to have their axis aligned with the flow direction.

Depending on the choice of property page you can see a Geographic View or a list view of the turbines. On the property page output it is possible to specify if turbine information should be written to an output file.

There are two different methods for specification of the turbines. In the List View you can create a new turbine by clicking on the "New turbine" button. By selecting a turbine in the Turbine list and clicking on the "Delete turbine" you can remove this turbine. For each turbine you can specify the name of the turbine and whether the turbine should be active or not. The specification of detailed information for each turbine is made subsequently. From the List View page you can navigate to the dialog for specification by clicking on the "Go to .." button or by selecting a turbine in the Turbine list and clicking on the "Edit turbine" button.

In the Geographic View it is also possible to create a new turbine by double-clicking a position on the map or by selecting "New turbine" from the context menu. The name of the turbine must then be specified and the geographical position of the turbine can then be edited. The additional information for the turbine is made subsequently.

Turbine specification

The location of the turbine and the turbine data is specified.

Location
You specify the horizontal coordinates in which the turbine should be placed.

You must also select the map projection (Longitude/Latitude, UTM, etc.) in which you want to specify the horizontal location of the turbine point.

Turbine Data
You must specify the diameter of the turbine and the z-coordinate of the centroid of the turbine.

The resistance imposed by the turbine blades can be specified in one of two ways:

- Fixed drag coefficient
- Tabulated drag and lift coefficient

When "Fixed drag coefficient" is selected, the turbine is assumed to have the axis aligned with the flow direction. Hence no lift force is calculated.

When "Tabulated drag and lift coefficient" is selected you must also specify the orientation of the turbine, $\theta_{alignment}$. The orientation is measured from projection north to the alignment, positive clockwise in degrees, see Figure 6.13. Note that the projection north is not necessarily the same as geographic north.

You must also specified if current correction should be applied (see General description *(p. 110)*).

It is also necessary to specify a correction factor. The correction factor is multiplied on the drag and lift.

The format of the correction factor can be specified as:

- Constant in time
- Varying in time

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

Fixed drag coefficient
In case of a fixed drag coefficient you specify a constant value.

Tabulated drag and lift coefficient
In case of tabulated values you have to specify the drag and lift coefficients as a function of current speed, V, and direction, θ. The definition of θ is shown in Figure 6.13.

Direction and speed
You have to specify the extent and range of the grid defining the tabulated speed and current values.

Drag coefficient
You have to specify the drag coefficient for each table value in the grid.

Lift coefficient
You have to specify the lift coefficient for each table value in the grid.

Output

If output is included you must specify the name of the output file. The file will, for each turbine, contain the drag and lift force and the average current speed and current direction used in the calculation of the drag force.

General description

The effect to the flow due to the turbines is modeled by calculating the current induced drag and lift force on each individual layer.

The effective drag force, F_D and lift force, F_L, are determined from

$$F_D = \frac{1}{2}\rho_w \alpha C_D A_e V_0{}^2$$

$$F_L = \frac{1}{2}\rho_w \alpha C_L A_e V_0{}^2$$

(6.36)

where ρ_w is the density of water, α is a correction factor, C_D is the drag coefficient, C_L is the lift coefficient, A_e is the effective area of turbine exposed to current and V_0 is the upstream current speed.

For three-dimensional calculations the current speed used to determine the force is the average current speed over the water column covered by the turbine and the shear stress is equally distributed over water column covered by the turbine.

When no current correction is applied the upstream current velocity is approximated by the local velocity, V_{local},. The local velocity is the velocity in the cell, where the turbine is located. When current correction is included the upstream current speed is approximated by

$$\frac{2}{1 + \sqrt{1 - \gamma}} V_{local}$$

(6.37)

where $\gamma = \alpha C_D A_E / h \Delta s$. Here h is the water depth and Δs is grid distance. When the drag coefficient is specified as a function of the upstream current speed the corrected current speed is determined by iteration. The grid distance is determined as the width perpendicular to the flow direction of the element, where the turbine is located. For more detail see (Kramer et al. (2014)).

Definition of angles in calculation
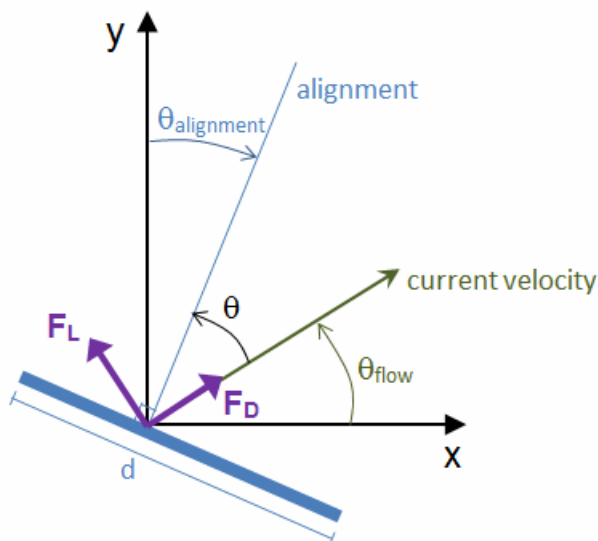
d   : diameter of turbine
θ_{flow} : current direction
$\theta_{alignment}$: angle between projection north and alignment
θ : angle between alignment and flow

Figure 6.13   Definition of turbine angles

6.16.7 Composite structures

Composite structures can be defined by combining a weir and/or one or more culverts. A composite structure can only be composed of a combination of Weirs and/or Culverts.

An example of a composite structure could be a bridge with multiple waterways. Such a structure can be described by a number of culverts, each defining an individual waterway. Additionally, for a potential bridge deck overtopping a weir can be included to describe such overflow.

A set of structures forming a composite structure are recognized by the program from the location definitions. Locations must be completely identical for all the structures forming the composite structure. That is, the table of coordinates defining the structure locations must be exactly identical (number of coordinates and coordinate values) for all structures defined.

Examples of composite structures are given below.

Example 1 - A wide weir with a small opening

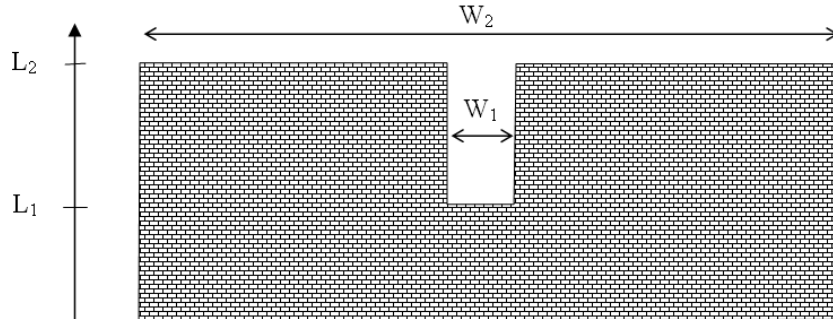Consider a weir as illustrated in Figure 6.14.

Figure 6.14   Wide weir with a small opening

There are a number of possibilities when modeling this in MIKE 21:

1. One weir

2. Two weirs with width W_1 and width (W_2-W_1), respectively

3. Three weirs with widths W_1 and two with width (W_2-W_1)/2, respectively

Using the first approach is only appropriate if the weir can be contained within a single grid cell. The second approach may be used if the weir spans multiple cells, keeping in mind that the flow over the highest crest (L_2) is uniformly distributed over all the affected cells. The third approach will give the best representation of the flow. Note that the location needs to be defined for each of the segments for case 2 and 3.

Example 2 - A wide weir with multiple culverts
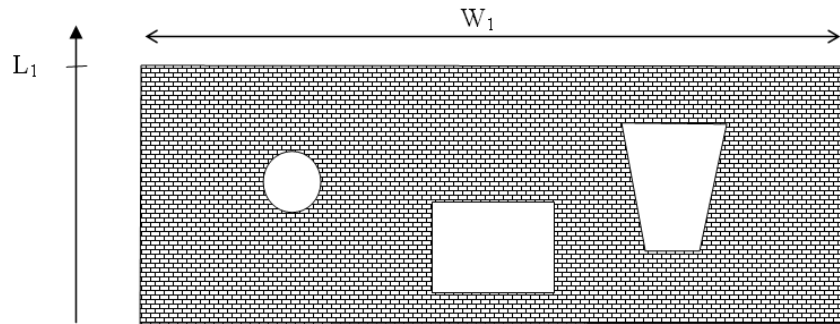
Consider a structure as illustrated in Figure 6.15.

Figure 6.15   Wide weir with multive culverts

The composite structure should be implemented as four separate structures:

1. A weir with a constant crest level L1 and a location defined by the full extent of the weir

2. A circular culvert and a location defined by the full extent of the weir

3. A rectangular culvert and a location defined by the full extent of the weir

4. A irregular culvert described by a level/width table
    and a location defined by the full extent of the weir

Note that the location needs to be defined for each of the four structure components separately. The location line should correspond to the maximum width of the structure component while still obeying the minimum requirement with respect to intersecting a line segments connecting cell centres.

6.16.8 Remarks and hints

Location of a structure

The location of a structure may be given by a number of geo-referenced points which together make up a poly-line. The poly-line defines the width of the structure perpendicular to the flow direction.

-   For a weir and a gate the poly-line will describe the location of the crest in the horizontal plan view.

-   For a culvert the poly-line should be given as a line perpendicular to the flow direction and further the line should intersect the mid point of the culvert.
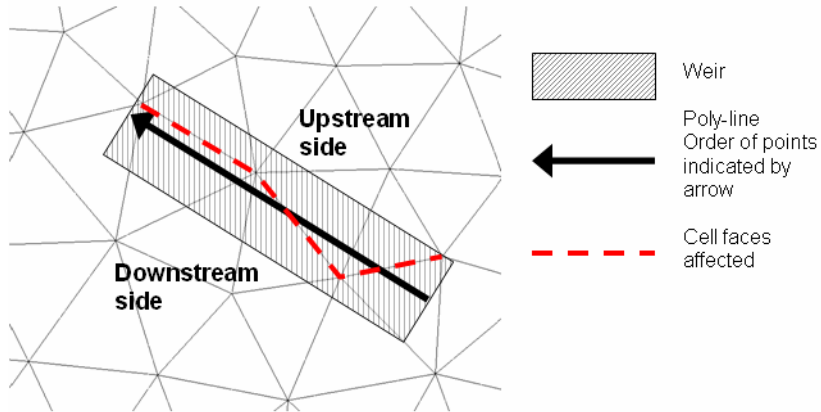
Figure 6.16   The location of a weir or gate. Note the affected cell faces.
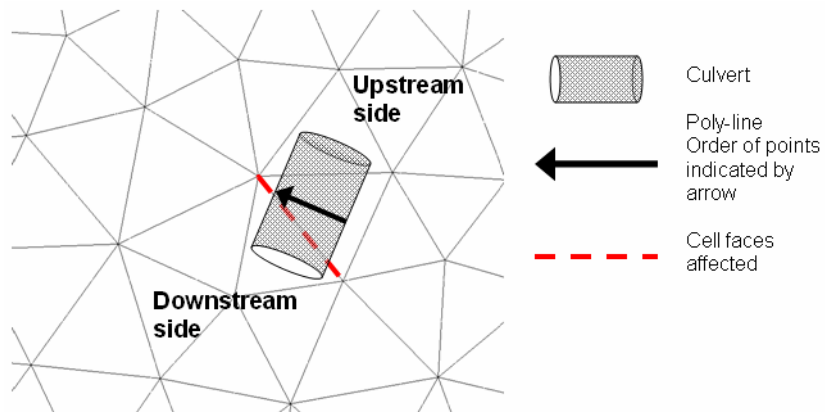

Figure 6.17   The location of a culvert. Note the affected cell face.

A cell face is affected if the defining poly-line intersects the line segment connecting the mid points of the two adjacent grid cells. The flow through the structure is evenly distributed along the affected cell faces.

Level-width relationship

The geometry of a structure may be defined by a level-width relation, where the Level/Width table defines the shape of the active flow area as a set of corresponding levels and flow widths. Values in the levels column must be continuous, increasing values.

Level	Width
L_0	0
L_1	W_1
L_2	W_2
L_3	W_3

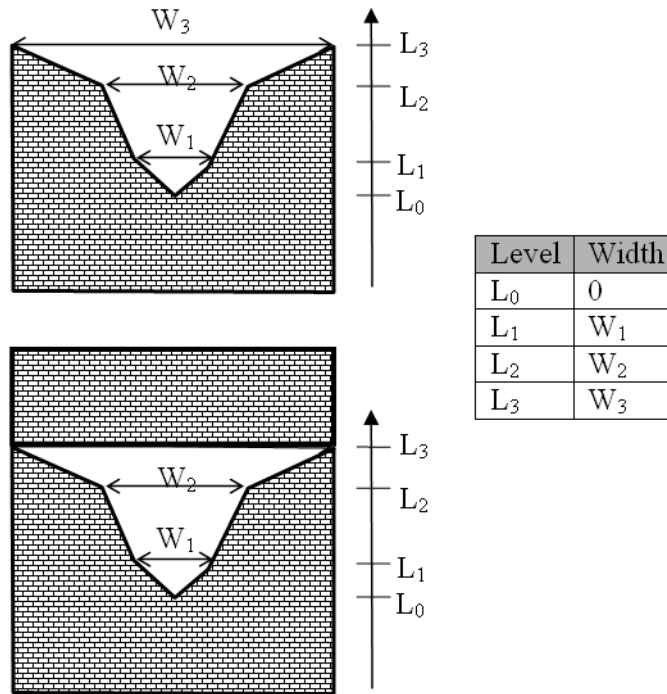Figure 6.18   Definition sketch of level-width relationship
Upper: broad crested weir
Lower: Irregular culvert

Flow direction

Positive and Negative flow directions as referred to in Valve and Head loss coefficient definitions follows the definition as presented in Figure 6.19. That is, positive flow direction is defined as the left perpendicular to the direction of a structure line schematization.
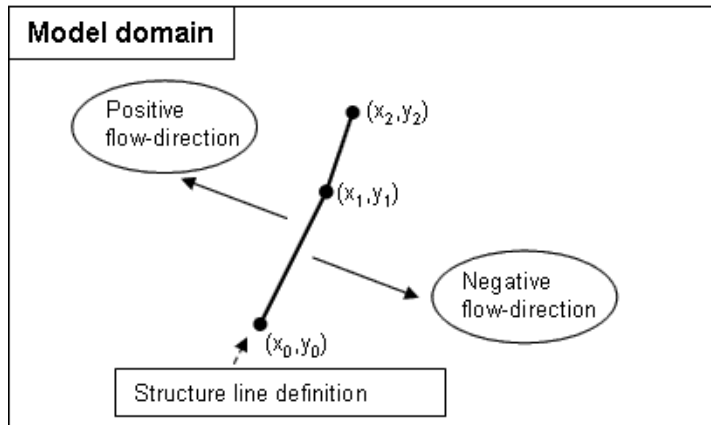
Figure 6.19   Positive and Negative flow direction definition at structures

Alpha zero value

When the water level gradient across a structure is small the corresponding gradient of the discharge with respect to the water levels is large. This in turn may result in a very rapid flow response to minor changes in the water level upstream and downstream.

As a way of controlling this effect an Alpha zero value has been introduced. The Alpha zero value defines the water level difference below which the discharge gradients are suppressed. The default setting is 0.01 meter. If a structure shows oscillatory behavior it is recommended to increase this value slightly.

6.17   Waves

If you have defined a dike and calculate the overtopping discharge from a user-specified table (see page 97) or if you have defined the bed resistance by Wave induced bed resistance (see page 67) you can define the related waves to be used in your calculations as

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

No waves

If you choose no waves, the overtopping discharge is derived assuming the minimum significant wave height, *hmin*, and minimum peak wave period, *tmin*, defined in the user specified table.

The wave induced bed resistance cannot be applied without specifying any wave data.

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 6.20).

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 6.20).

AT TIME t* AT GRID POINT (j,k) H_s, T_m, $\bar{\Theta}$ ARE
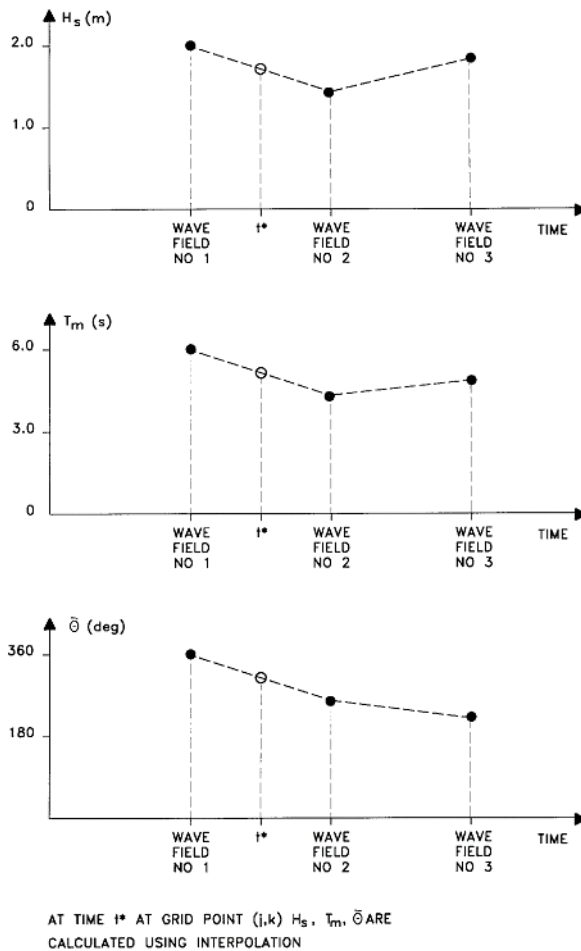CALCULATED USING INTERPOLATION

Figure 6.20   Interpolation of wave fields in time

6.18   Initial Conditions

The initial values for the hydrodynamic variables can be specified in three different ways

- Constant.
- Spatial varying surface elevation.
- Spatially varying water depth and velocities

For the last case 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 flow field.

To avoid generation of shock waves, it is recommended that your initial surface elevation roughly matches the boundary conditions at the start of the simulation.

Data

For the case with spatially varying surface elevation you have to prepare a data file containing the surface elevation 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, piecewice constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data. In case the input data file contains a single time step, this field is used. In case the file contains several time steps, e.g. from the results of a previous simulation, the actual starting time of the simulation is used to interpolate the field in time. Therefore the starting time must be between the start and end time of the file.

For the case with spatially varying surface elevation and velocities you have to prepare one or two data files before you set up the hydrodynamic simulation.

In case of a 2D simulation one file is necessary. The file must contain the total water depth and the velocity components in the x- and y-directions. 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.

In case of a 3D simulation two files are necessary. One containing the total water depth and one containing the velocity components in the x- and y-directions and the vertical velocity in the sigma coordinate system before you set up the hydrodynamic simulation. The first file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2) and the second file must be a 3D unstructured data file (dfsu) or a 3D grid data file (dfs3). The area in the data file must cover the model area. If a 2D dfsu-file is used piecewice constant interpolation is used to map the data. If a 3D dfsu file is used the mesh in the data file must match exactly the mesh in the simulation. If a dfs2-file or a dfs3-file is used bilinear interpolation is used to map the data.

In case the input data file contains a single time step, this field is used. In case the file contains several time steps, e.g. from the results of a previous simulation, the actual starting time of the simulation is used to interpolate the field in time. Therefore the starting time must be between the start and end time of the file.

6.19 Boundary Conditions

Initially, the set-up editor scans the mesh file for boundary codes (sections), and 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. 48)*).

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

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

6.19.1 Boundary specification

For hydrodynamic boundaries there are seven types:

- Land (zero normal velocity)
- Land (zero velocity)
- Specified velocities
- Specified fluxes
- Specified level
- Specified discharge
- Flather condition

It should be noted that each boundary section must contain at least 2 nodes and that flux and velocity boundary conditions cannot work if the initial bathymetry at the boundary is dry.

For stationary solid boundaries two types of boundary conditions can be applied. For the "Land (normal velocity)" the full slip boundary conditions is assumed to hold, that is, the normal velocity component is zero, while for the "Land (zero velocity)" the no slip condition is assumed to hold, that is, both the normal and tangential velocity components are zero. By default the code value for land boundaries automatically identified by the Mesh Generator is set to one. For these boundaries the full slip boundary condition are applied. If the no slip boundary conditions should be applied for these boundaries the code values should change to a value larger than one.

If you select the specified discharge option, the discharge will be distributed as it would have been in a uniform flow field with the Manning resistance law applied, i.e. is relative to $h^{5/3}$, where h is the depth. This distribution is, in most cases, a good approximation. This does not apply if there are very large gradients in temperature or salinity at the boundary or very large variations over the bathymetry or the geometry. When discharge boundary is selected, it is

possible to specify the approach by which the boundary condition should be imposed.

The Flather (1976) condition is one of the most efficient open boundary conditions. It is very efficient in connection with downscaling coarse model simulations to local areas (see Oddo and Pinardi (2007)). The instabilities, which are often observed when imposing stratified density at a water level boundary, can be avoided using Flather conditions.

Approach

When discharge boundary is selected, it is possible to specify whether the boundary condition should be imposed using a weak or a strong formulation. Fore more detail see General description *(p. 126)*.

Data

Velocity boundary

If a velocity boundary is selected the velocities in the x- and y-direction can be specified in three different ways

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

For the case with velocities varying in time but constant along the boundary you have to prepare a data file containing the velocity components in the x- and y-direction 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.

For the case with velocities varying both in time and along the boundary you have to prepare a data file containing the boundary values before you set up the hydrodynamic simulation. In case of a 2D simulation the file must be a profile file (dfs1). In case of a 3D simulation the file must be a dfs2 file or a 2D dfsu file containing information from a vertical plane. The dfsu option is only possible when the simulation is using a Sigma formulation and the dfsu file is extracted from a simulation (3D dfsu file) using Sigma formulation. The mapping from the input data file to the boundary section is described in Interpolation type. 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.

Flux boundary

If a flux boundary is selected the fluxes (the depth-integrated velocities) in the x- and y-direction can be specified in three different ways

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

For the case with fluxes varying in time but constant along the boundary you have to prepare a data file containing the flux components in the x- and y-direction 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 type of interpolation.

For the case with fluxes varying both in time and along the boundary you have to prepare a data file containing the boundary values before you set up the hydrodynamic simulation. The file must be a profile file (dfs1). The mapping from the input data file to the boundary section is described in Interpolation type. 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 type of interpolation.

Level boundary

If a level boundary is selected, the format of the water level (surface elevation, in m) can be specified as:

- Constant (in time and along boundary)
- Variable in time and constant along boundary
- Variable in time and along boundary
- Rating curve

For the case with water level varying in time but constant along the boundary you have to prepare a data file containing the water level 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 type of interpolation.

For the case with water level varying both in time and along the boundary you have to prepare a data file containing the boundary values before you set up the hydrodynamic simulation. The file must be a profile file (dfs1). The mapping from the input data file to the boundary section is described in Interpolation type. 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 type of interpolation.

For the case with water levels defined by a Rating curve you have to define a data file containing the relation between discharge and water level. The file must be a time series file (dfs0) describing the water level by the relative item axis and the discharge by an item.

Discharge boundary

If a discharge boundary is selected, the format of the total discharge can be specified as:

- Constant (in time)
- Variable in time
- Rating curve

For the case with discharge varying in time you have to prepare a data file containing the total discharge 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 time interpolation.

For the case with discharge defined by a Rating curve you have to define a data file containing the relation between discharge and water level. The file must be a time series file (dfs0) describing the water level by the relative item axis and the discharge by an item.

Flather boundary

If "Flather condition" is selected, the velocities in the x-and y-direction and the water level (surface elevation) must be specified. For specification of these parameters see "Velocity boundary" and "Level boundary". If discharge constraint is included the total discharge must also be specified. For specification of the total discharge see "Discharge boundary".

Rating curve

A rating curve may be selected at either a discharge or a water level boundary. The boundary value is determined from the rating curve table using the appropriate variable (water level or discharge) in the adjacent cell. Depending on the base boundary type one of the two above approaches are applied.

The use of the Rating curve format type is only recommended for downstream boundaries where water flows out of the model.

A Rating curve is a dfs0 file with axis type 'Relative item axis'. The data represent the relation between discharge (as an item) and waterlevel (specified in the relative axis).
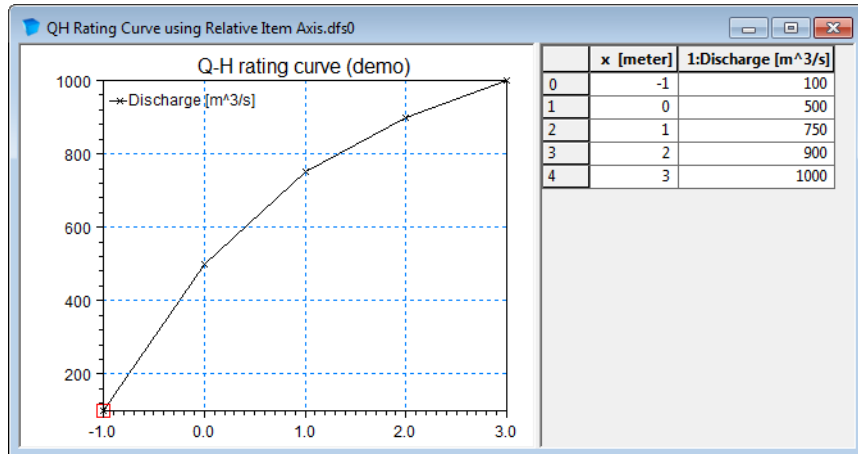
Figure 6.21   Example of Rating curve

Vertical profile

When you have selected a velocity boundary, flux boundary or a discharge boundary you must also the specify the velocity profile over the water depth in case of a 3D simulation:

- Uniform
- Logarithmic

When a dfs2 file or a dfsu file is specified for a velocity boundary, the velocity profile is obtained from the boundary data.

Interpolation type

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

- linear
- piecewise 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

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.

Discharge constraint

To ensure that the discharge through the boundary is correct when using a Flather boundary, the specified velocities can be corrected. The correction is performed by adding a uniform correction to the normal velocity components.

Discharge constraint is ment to correct inaccurate spatial interpolation when taking water level and current from a coarser outer model and applying them as Flather boundary conditions.

Coriolis/wind correction

Using level boundary conditions in the presence of Coriolis and wind force, may in some cases gives unrealistic results. If you specify a constant water level variation (horizontal water level) you will get a large inflow together with a large outflow at the same boundary, especially in a steady state situation, as the water level should actually be tilted. Also if you specify a variation of the water level along the boundary, the data very often do not include the effect of the local wind, e.g. if the level variation is obtained from tidal predictions. For these cases you can include tilting to improve the boundary conditions.

If tilting is included, a correction of the water level in each point along the boundary is calculated based on the steady state Navier Stokes equations. The correction takes into account the effect of Coriolis and wind force. This non-linear approach provides the best estimate in cases where you have a gentle slope on the seabed. If the seabed is very jaggered at the boundary, you may have to smooth the bathymetry.

The difficulty in using the tilt facility lies in the specification of the tilting point. Presently, the tilting point is selected as the deepest point along the boundary. If the still water depth along the line is constant, the centre point is selected.

If you have two adjacent boundaries, you should also be careful not to create a conflicting situation in the corner.

By default Coriolis/wind correction is not included. Note that coriolis/wind correction is not possible for boundaries with less than 3 node points.

Pressure correction

When level boundaries is selected in combination with spatially varying wind, the level boundaries can optionally be adjusted to take into account the variation of the pressure.

By default pressure correction is included.

Radiation stress correction

When level or discharge boundaries is selected in combination with wave radiation, the boundaries values can be optionally adjusted to take into account the effect of wave radiation.

For level boundaries the correction is assumed to be zero at the deepest point along the boundary.

By default radiation stress correction is not included.

6.19.2 General description

The flux, velocity and Flather boundary conditions are all imposed using a weak approach. A ghost cell technique is applied where the primitive variables in the ghost cell are specified. The water level is evaluated based on the value of the adjacent interior cell and the velocities are evaluated based on the boundary information. For a discharge boundary the transverse velocity is set to zero for inflow and passively advected for out flow. The boundary flux is then calculated using an approximate Riemann solver.

The level boundary is imposed using a strong approach based on the characteristic theory (see e.g. Sleigh et al. (1998)).

The discharge boundary condition is imposed using both a weak formulation based on ghost cell technique described above and a strong approach based on the characteristic theory (see e.g. Sleigh et al. (1998)). Note that using the weak formulation for a discharge boundary the effective discharge over the boundary may deviate from the specified discharge.

Convention of direction of boundary

When specifying a line series at the boundary it is important to know how the model defines the first and last node of the boundary.

The rule is: Follow the shoreline with the discretized domain on the left hand side, see example in Figure 6.22. When a boundary is reached, this is the first node of the boundary.
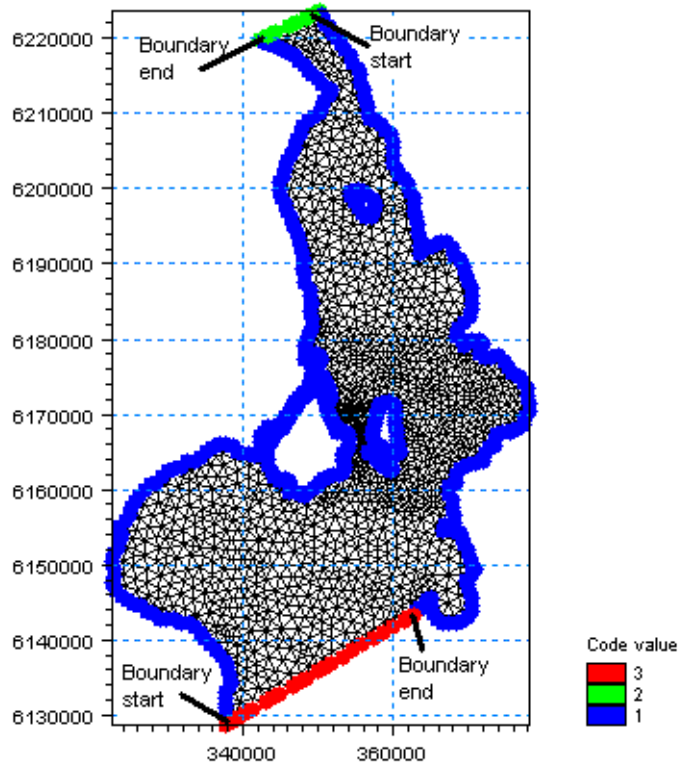
Figure 6.22   Convention of boundary start and end points.
Boundaries are defined by codes in the mesh file.

Convention of direction of discharge

By definition, discharge is positive for flow towards left when positioned at the first point and looking forward along the boundary line, i.e. referring to the Convention of direction of boundary positive discharge values will describe flow into the domain whereas negative discharge values will desribe flow out of the domain.

Correction of water level

The correction of the water level due to spatially varying air pressure is adjusted according to

$$\eta = \eta^* - \frac{\rho_{air} - \rho_{neutral}}{\rho g} \qquad (6.38)$$

where ρ_{air} is the air pressure, $\rho_{neutral}$ is the neutral pressure, g is the gravitational acceleration, ρ is the density of the water, η^* is the specified surface elevation and η is the corrected surface elevation applied as boundary condition.

Interpretation of dfs2 files

When using dfs2 files for boundary conditions, the y-axis in the dfs2 file is interpreted as the vertical axis given relative to a level defined by the y-axis origin. The z-level for the rows in a dfs2 file is found by

$$z(j) = \text{Origin(y)} - dy \cdot (NY - j) \qquad (6.39)$$

where dy is the grid-spacing in y-axis direction, NY is the grid size -1 and Origin(y) is the level for the upper row.

The origin for the spatial axis is usually 0, i.e. per default the lowest level corresponds to $-dy*NY$ and the upper level correspond to MWL=0m. In case the boundary contains levels below or above this interval the values will correspond to the nearest level.
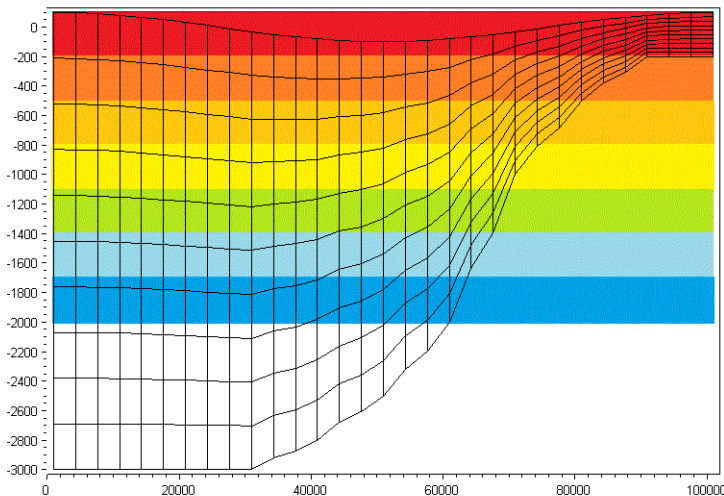

Figure 6.23   Example of dfs2 boundary with equidistant sigma mesh
Y-axis from 0 to NY=6, Origin,y = 100m, dy=300m
Values in levels below -2000m will correspond to value in level -2000m

6.20   Temperature/Salinity Module

The Temperature/Salinity (TS) module is invoked from the specification of the density, provided baroclinic density (density depends on temperature and/or salinity) is selected (see Section 6.5 Density).

A more detailed description of the Temperature/salinity module can be found in section 7 TEMPERATURE/SALINITY MODULE.

6.21   Turbulence Module

The turbulence module is invoked from the specification of vertical eddy viscosity, provided k-ε model is selected (see Section 6.6 Eddy Viscosity).

A more detailed description of the Turbulence module can be found in TURBULENCE MODULE.

6.22   Decoupling

In many applications a number of simulations are performed, where the model parameters for the add-on modules are changed, but where the model parameters for the basic hydrodynamic simulations are unchanged.

If the basic flow information from a hydrodynamic simulation is saved in a number of special decoupling output files, it is possible to re-run the simulation in a decoupled mode, where instead of calculating the flow information, it is read from these files. Running a simulation in decoupled mode can significantly reduce the computational time.

If you choose to save decoupling data you have to specify two output file names for a 2D simulation and three output file names for a 3D simulation. You must also specify the frequency (time step factor) by which data is going to be saved. Finally, you must give the name of the specification file for the decoupled simulation. Setting up and run a simulation using this specification file you can not change the basic parameters in the time and domain dialog and the parameters in the hydrodynamic model, but you are only allowed to change the parameters in the add-on modules.

6.22.1   Decoupling output files

The layout of the decoupled dfsu files depends on the way the model setup is executed:

* Using Shared memory approach (OpenMP) the output from the entire domain will be saved in the specified dfsu files.

* Using Distributed memory approach (MPI) the output from the individual sub-domains are initially saved in separate dfsu files. These files are merged at the end of the simulation into the specified dfsu files.

PLEASE NOTE:
When executing a flexible mesh model in decoupled mode you often use the same parallellization approach as used in the generation of the decoupled files. In case you mix the MPI and OpenMP approach when creating and using the decoupled files, reordering should be omitted.

6.22.2 Remarks and hints

It is not possible to generate decoupled files if the hydrodynamic solution is using Quasi-stationary formulation.

Note, that in decoupled mode, the time step is not restricted by the CFL stability condition for solution of flow equations, but only by the stability restriction for the solution of the transport equations.

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

6.23.1 Geographical view

This dialog shows the geographical position of the output data.

6.23.2 Output specification

For each selected output file the field type, the output format, the treatment of flood and dry, the output file (name and location) and time step must be specified. Depending on the output format the geographical extend of the output data must also be specified.

Field type

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

Output format

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

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

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

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

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

If Inundation is selected for the field type, you have to specify the domain for which the inundation should be calculated. The file type will be an area series dfsu file.

Table 6.2   List of tools for viewing, editing and plotting results

Output format	File type	Viewing/editing tools	Plotting tools
Point series	dfs0	Time Series Editor	Plot Composer
Line series	dfs1	Profile Series Editor	Plot Composer
Area series	dfsu	Data Viewer	Data Viewer
Volume series	dfsu	Data Viewer	Data Viewer

Output file

A name and location of the output file must be specified along with the type of data (file type).

Treatment of flood and dry

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

- Whole area

- Only wet area

- Only real wet area

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.

Disregarding the choice of treatment of output data, all adjacent elements, including dry elements, are considered in the interpolation calculation.

Time step

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

Point series

You must specify the type of interpolation. You can select discrete values or interpolated values.

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

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

If "discrete values" is selected for the type of interpolation, the point values are the discrete values for the elements in which the points are located. The element number and the coordinates of the center of the element are listed in the log-file.

If "interpolated values" is selected for the type of interpolation, the point values are determined by 2nd order interpolation. The element in which the point is located is determined and the point value is obtained by linear interpolation using the vertex (node) values for the actual element. The vertex values are calculated using the pseudo-Laplacian procedure proposed by Holmes and Connell (1989). The element number and the coordinates are listed in the log-file.

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

Layer number

The layer number selected for discrete values in the point output is defined from the lowest active layer (=1) increasing upwards. In case the mesh is a type sigma mesh the number of active layers in the water column will always be the same in any point in the domain. In case the mesh is a combined sigma-z level mesh the number of active layers may vary in the domain. An example is shown in Figure 6.24.
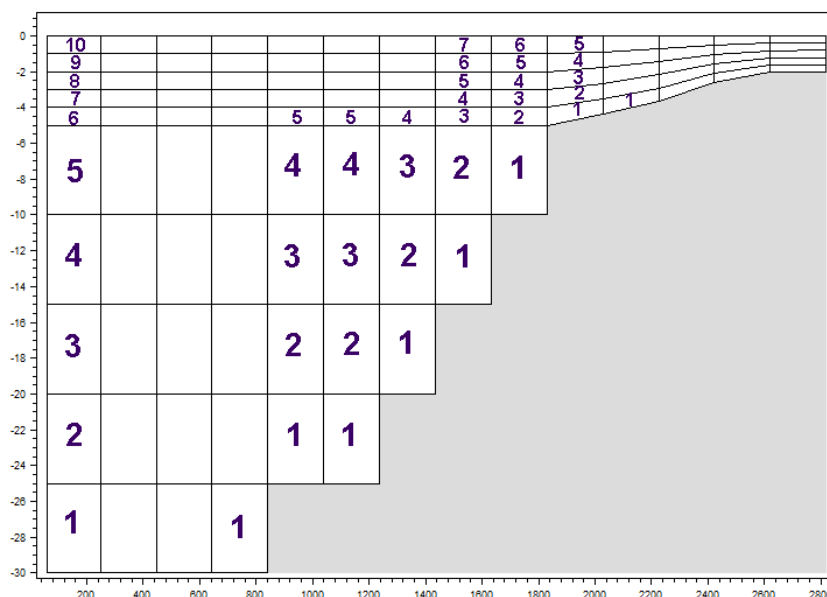
Figure 6.24   Example of layer numbers in point output specification in case of combined sigma-z level mesh.

Line series

You must specify the first and the last point on the line and the number of discrete points on the line. 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. For 3D field data the third item must be a float (real number) for the z-coordinate. For 2D field data the third item is unused (but must be specified). If the file contains information for more than two points (more than two lines) the information for the first two points will be used.

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

The values for the points on the line are determined by 2nd order interpolation. The element in which the point is located is determined and the point value is obtained by linear interpolation using the vertex (node) values for the actual element. The vertex values are calculated using the pseudo-Laplacian procedure proposed by Holmes and Connell (1989). The element number and the coordinates are listed in the log-file.

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

Area series

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. For 3D field data the third item must be a float (real number) for the z-coordinate. For 2D field data the third item is unused (but must be specified).

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

Volume series

The discrete field data within a polygon in the horizontal domain can be selected and within a specified range of layers in the vertical domain. The closed region in the horizontal domain is bounded by a number of line segments. You must specify the coordinates of the polygon 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 vertex 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. For 3D field data the third item must be a float (real number) for the z-coordinate. For 2D field data the third item is unused (but must be specified). You must also specify the range of layers (first and last Layer number) which should be stored in the output 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.

Layer number

The layer number(s) selected for the volume output are numbered 1 at the lowest layer and increase upwards. In case of a combined sigma-z level

mesh only the elements containing water are saved in the output. An example is shown in Figure 6.25.
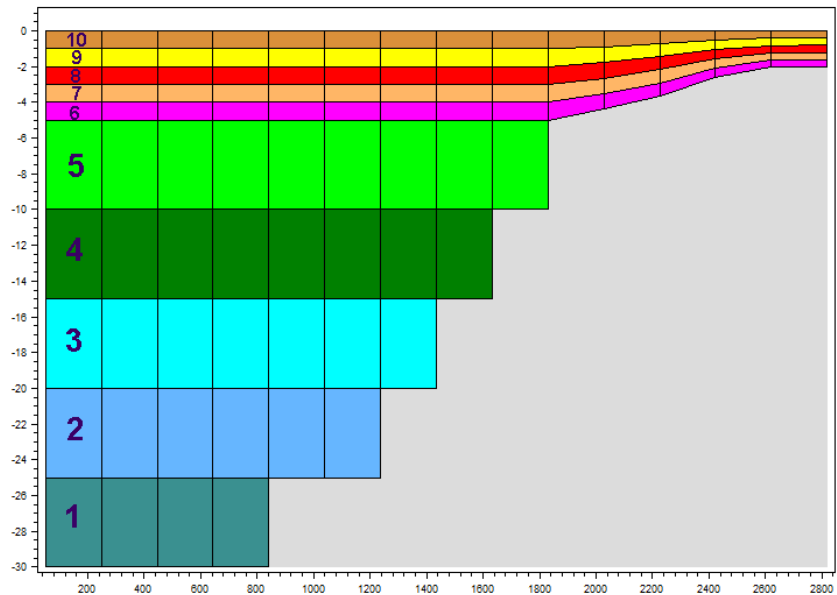

Figure 6.25   Example of layer numbers in volume output specification in case of combined sigma-z level mesh.

Cross section series

You must specify the first and the last point between which the cross section is defined. The cross section is defined as a section of element faces. The face is included in the section when the line between the two element centers of the faces crosses the line between the specified first and last point. 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). If the file contains information for more than two points (more than two lines), the information for the first two points will be used. 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.

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.

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.

6.23.3 Output items

Field variables

All output is optional, i.e. the user is free to select among the variables.

Flow directions in horizontal domain are given in degrees positive clockwise from true North (going against). Flow directions in vertical domain are given in degrees positive clockwise from the upward pointing z-axis.

The convergence angle is the angle from true North to projection North (positive clockwise).

Mass Budget

You can select the mass budget calculation to be included for the flow and for the temperature and the salinity if calculation of the temperature/salinity is included. For each selected component the following items are included in the output file:

- Total area - total volume/energy/mass within polygon
- Wet area - volume/energy/mass in the area within polygon for which the water depth is larger than the drying depth.
- Real wet area - volume/energy/mass in the area within polygon for which the water depth is larger than the wetting depth
- Dry area - volume/energy/mass in the area within polygon for which the water depth is less than the drying depth
- Transport - accumulated volume/energy/mass transported over lateral limits of polygon
- Source - accumulated volume/energy/mass added/removed by sources within polygon

- Process - accumulated volume/energy/mass added/removed by processes within polygon
- Error - accumulated volume/energy/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/energy/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 temperature and salinity the maximum and minimum values are specified on the Equation dialog.

For temperature the relative energy (relative to the energy at 0 ºC), E (J), in a volume of water V (m³) is calculated by

$$E = c_{water}\rho_{water}T \cdot V \tag{6.40}$$

where T (ºC) is the temperature, c_{water} = 4217.0 J/(K·kg) is the specific heat for water and ρ_{water} = 1010.0 kg/m³ is the density of water.

For salinity the mass of salt, M (kg), in a volume of water V (m³) is calculated by

$$M = \rho_{water}S \cdot V / 1000 \tag{6.41}$$

where S (in PSU) is the salinity and ρ_{water} = 1010.0 kg/m³ is the density of water.

Discharge

You can select the discharge calculation to be included for the flow and for the temperature and the salinity if calculation of the temperature/salinity is included. Each selected component will result in a number of output items.

You can select between two types of output items:

- Basic
- Extended

The basic output items is as follows:

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

For temperature the relative energy (relative to the energy at 0 ºC), E (J), in a volume of water V (m^3) is calculated by

$$E = c_{water}\rho_{water}T \cdot V \qquad (6.42)$$

where T (ºC) is the temperature, c_{water} = 4217.0 J/(K·kg) is the specific heat for water and ρ_{water} = 1010.0 kg/m^3 is the density of water.

For salinity the mass of salt, M (kg), in a volume of water V (m^3) is calculated by

$$M = \rho_{water}S \cdot V/1000 \qquad (6.43)$$

where S (PSU) is the salinity and ρ_{water} = 1010.0 kg/m^3 is the density of water.

Inundation

You can select to output an inundation map for the flow simulation. The following items may be selected for the output file

- Maximum water depth
- Time of maximum water depth
- Maximum current speed
- Time of maximum current speed
- Duration of depth above threshold

In case the latter item is selected, you have to specify a Depth threshold value as well.

7 Coupling of the Flow, Wave and Process Modules

The detailed description of the specification for the different modules can be found in the User Guides for MIKE 21 Flow Model FM, MIKE 3 Flow Model FM and the MIKE 21 Spectral Wave Module FM.

The overall discrete time step is specified on the Time dialog. This value is used to determine the frequency for which output can be obtained from the different modules and to synchronize the coupling between the different modules.

The time step for the Spectral Wave Module (SW), the Sand Transport Module (ST) and the Shoreline Morphology Module (SM) can be specified as a multiple of the overall time and a time step factor.

For the Transport Module (TR), the ECO Lab / Oil Spill Module (ELOS), the Mud Transport Module (MT) and the Sand Transport Module (ST) using the non-equilibrium pure current formulation the time-integration of the suspended components are performed with the same variable time step as specified for the transport calculations in the Hydrodynamic Module (HD).

The following fully dynamic interactions between the Hydrodynamic Module and the Spectral Wave Module are possible

- In the hydrodynamic module it is possible to include the radiation stress field from the spectral wave simulation
- In the Spectral Wave Module it is possible to include the water level variation from the hydrodynamic simulation
- In the Spectral Wave Module it is possible to include the current field variation from the hydrodynamic simulation

The application of the Shoreline Morphology Module requires coupling with the Sand Transport Module and Hydrodynamic Module while no other process modules are selected.

8   LIST OF REFERENCES

Aupoix, B., (1984), Eddy Viscosity Subgrid Scale Models for Homogeneous Turbulence, in *Macroscopic Modelling of Turbulent Flow*, Lecture Notes in Physics, Proc. Sophie-Antipolis, France.

Battjes, J.A. and J.P.F.M. (1978): Energy loss and set-up due to breaking of random waves, in Proc. 16th Int. Conf. On Coastal Eng., ASCE, NY, 569-587.

Battjes, J.A. and M.J.F. Stive (1985): Calibration and verification of a dissipation model for random breaking waves, Geophys. Res., 90, 9159-9167.

Bidlot, J., P. Janssen, and S. Abdalla (2007): A revised formulation of ocean wave dissipation and its model impact. Technical Report Memorandum 509, ECMWF, Reading, U. K.

Bouws, E. and G.J. Komen (1983): On the balance between growth and dissipation in an extreme, depth-limited wind-sea in the southern North Sea. J. Phys. Oceanogr., 13, 1653-1658.

Booij, N., R.C. Ris and L.H. Holthuijsen (1999): A third-generation wave model for coastal regions. 1. Model description and validation. J. Geophys. Res., 104, 7649-7666.

Charnock, H. (1955): Wind Stress on a water surface. Quart. J. Roy. Meteorol. Soc., 81, 639-640.

Choi, K.W. and J.H.W. Lee, (2007), Distributed Entrainment Sink Approach for Modeling Mixing and Transport in the Intermediate Field, *J. Hydraulic Eng.* 133 (7), 804-815.

Collins, J.I. (1972): Prediction of shallow water spectra. J. Geophys. Res., 77, 2693-2707.

Delhez, E. J. M., Deelersijder, E., Maouchet, A., Beckers, J. M.(2003), On the age of radioactive tracers, *J. Marine Systems*, 38, pp. 277-286.

Dingler, J. R (1974): Wave formed ripples in nearshore sands. PhD. Thesis, Univ. of California, San Diego, CA, 136pp.

Dingler, J. R., and D. L. Inman (1976): Wave formed ripples in nearshore sands. Proc. 15th International Conference on Coastal Engineering, Honolulu, Hawaii, ASCE, 2109-2126.

Donelan, M. A., J. Hamilton and W. H. Hui (1985): Directional Spectra of Wind-Generated Waves. Phil. Trans. R. Soc. Lond., A 315, 509-562.

Donelan, M.A. (1990): Air-Sea Interaction. The Sea, B. LeMehaute and D. Hanes (Eds.), John Wiley and Sons, 239-292.

Donelan, M.A., F.W. Dobson, S.D. Smith and R.J. Anderson (1993): On the dependence of sea surface roughness on wave development. J. Phys. Oceanogr., 23, 2143-2149.

Donelan, M.A., W.M. Drennan, and K. B. Katsaros (1997): The air-sea momentum flux in conditions of wind sea and swell. J. Phys. Oceanogr., 15, 2087-2099.

Eldeberky, Y. and Battjes, J.A. (1995): Parameterization of triad interaction in wave energy models, Proc. Coastal Dynamics Conf. '95, Gdansk, Polen, 140-148.

Eldeberky, Y. and Battjes, J.A. (1996): Spectral modelling of wave breaking: Application to Boussinesq equations, J. Geophys. Res., 101, No. C1, 1253-1264.

Flather, R. A. (1976), A tidal model of the northwest European continental shelf, Memories de la Societe Royale des Sciences de Liege 6 (10), 141-164.

Fredsøe, J. (1984), Turbulent boundary layers in Combined Wave Current Motion. J. Hydraulic Engineering, ASCE, Vol 110, No. HY8, pp. 1103-1120.

Garratt, J.R. (1977): Review of drag coefficients over oceans and continents, Monthly Weather Review, 105, 915.

Geernaert, G.L., S.E. Larsen, and F. Hansen (1987): Measurements of the wind stress, heat flux and turbulence intensity during storm conditions over the North Sea. J. Geophys. Res., 92C, 13,127 - 13,139.

Geernaert, G.L., K.B. Katsaros and K. Richter (1986): Variation of the drag coefficient and its dependency on sea state. J. Geophys. Res., 91C, 6, 7667-7669.

Geernaert G.L. and Plant W.L (1990), Surface Waves and fluxes, *Volume 1 - Current therory*, Kluwer Academic Publishers, The Netherlands.

Goda Y. (1969), Re-analysis of laboratory of data on wave transmission over breakwaters, Rep. Port and Harbour Res. Inst., Vol. 8, No. 3, pp 3-18.

Goda, Y, H. Takeda and Y. Moriya (1967), Laboratory investigation of wave transmission over breakwaters, Rep. port and Harbour Res. Inst., No. 13, 38 pp.

Günther, H., S. Hasselmann and P.A.E.M. Janssen (1992): The WAM model cycle 4. DKRZ Technical Report No 4, Hamburg.

Gulev, S.K. and L. Hasse (1998): North Atlantic wind waves and wind stress fields from voluntary observing ship data. J. Phys. Oceanogr., 28, 1107-1130.

Hasselmann, K., T.P. Barnett, E. Bouws, H. Carlson, D.E. Cartwright, K. Enke, J.A. Wing, H. Gienapp, D.E. Hasselmann, P. Kruseman, A. Meerburg,

P. Müller, D.J. Olbers, K. Ricter, W. Sell and H. Walden (1973): Measurements of wind-wave growth and swell decay during the Joint North Sea Wave Project (JONSWAP). Dtsch. Hydrogr. Z. Suppl. A (12), 95 pp.

Hasselmann, K. (1962): On the nonlinear energy transfer in a gravity-wave spectrum, Part 1, General theory. J. Fluid Mech., 12, 481-500.

Hasselmann, S., and K. Hasselmann (1985): Computation and parameterizations of Non-linear Energy Transfer in a Gravity-wave Spectrum, Part 1: A New Method for Efficient Computations of the exact Non-linear Transfer Integral, J. Phys. Oceanog., 15, 1369-1377.

Hasselmann, S., and K. Hasselmann, (1985), Computation and parameterizations of Non-linear Energy Transfer in a Gravity-wave Spectrum, Part 2: Parameterisations of non-linear energy transfer for applications in wave models, J. Phys. Oceanog., 15, 1378-1391.

Hersbach, H. and P.A.E.M. Janssen (1999), Improvement of the short-fetch behaviour in the Wave Ocean Model (WAM). J. Atmos. & Ocean Tech., 16, 884-892.

Holmes, D. G. and Connell, S. D. (1989), Solution of the 2D Navier-Stokes on unstructured adaptive grids, AIAA Pap. 89-1932 in Proc. AIAA 9th CFD Conference.

Holthuijsen, L.H., N. Booij, and T.H.T Herbers (1989), A prediction model for stationary, short crested waves in shallow water with ambient currents. Coastal Eng., 13, 23-54.

Holthuijsen L.H., Herman A., Booij N. and Cieslikiewicz W. (2002), Diffraction in SWAN, Proceedings 28th International Conference Coastal Engineering, Cardiff, 405-412.

Horiuti, K. (1987), Comparison of Conservative and Rotational Forms in Large Eddy Simulation of Turbulent Channel Flow, *Journal of Computational Physics*, 71, pp 343-370.

Janssen, J.A.M. (1997): Does wind stress depend on sea-state or not? – A statistical error analysis of HEXMAX data. Boundary-Layer Meteorology, 83, 479-503.

Janssen, P.A.E.M. (1989): Wave induced stress and the drag of airflow over sea waves. J. Phys. Oceanogr., 19, 745-754.

Janssen, P.A.E.M. (1991): Quasi-linear theory of wind wave generation applied to wave forecasting. J. Phys. Oceanogr., 21, 1631-1642.

Janssen, P.A.E.M. (1992): Experimental evidence of the effect of surface waves on the airflow. J. Phys. Oceanogr., 22, 1600-1604.

Janssen, P.A.E.M. (1998): On the effect of ocean waves on the kinetic energy balance and consequences for the initial dissipation technique. J. Phys. Oceanogr., 30, 1743-1756.

Jirka, Gerhard H. (2004), Integral Model for Turbulent Buoyant Jets in Unbounded Stratified Flows. Part I: Single Round Jet. *Environmental Fluid Mechanics,* 4: 1-56.

Johnson, H.K. (1998): On modeling wind-waves in shallow and fetch limited areas using the method of Holthuijsen, Booij and Herbers. J. Coastal Research, 14, 3, 917-932.

Johnson, H.K. and H. Kofoed-Hansen (2000): Influence of Bottom Friction on Sea Surface Roughness and Its Impact on Shallow Water Wind Wave Modelling. J. Phys. Oceanogr., 30, 1743-1756.

Johnson, H.K., J. Højstrup, H.J. Vested and S.E. Larsen (1998): On the Dependence of Sea Surface Roughness on Wind Waves. J. Phys. Oceanogr., 28, 1702-1716.

Johnson, H.K., H.J. Vested, H. Hersbach, J. Højstrup, and S.E. Larsen, (1999): On the coupling between wind and waves in the WAM model, To appear in Journal of Atmospheric and Oceanic Technology.

Jones, O., Zyserman, J.A. and Wu, Yushi (2014), Influence of Apparent Roughness on Pipeline Design Conditions under Combined Waves and Current, *Proceedings of the ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering.*

Kærgaard, K. H., & Fredsøe, J. (2013). A numerical shoreline model for shorelines with large curvature. Coastal Eng., Vol. 74, 2013, 19– 32

Kahma, K.K. and C.J. Calkoen (1994): Growth curve observations, In Dynamics and Modelling of Ocean Waves by Komen et al., Cambridge University Press, 174-182,

Katsaros, K. B, M.A. Donelan and W.M. Drennan (1993): Flux measurements from a SWATH ship in SWADE. J. Mar. Sys., 4, 117-132.

Kitaigorodskii, S.A. (1970): The physics of air-sea interaction. Translated for Russian by A. Baruch, Israel Program for Scientific Translations, Jerusalem, 237pp.

Kitaigorodskii, S.A. (1998): The dissipation subrange in wind wave spectra. Geophysical, 34 (3), 179-207.

Kofoed-Hansen, H; and J.H. Rasmussen (1998): Modelling of non-linear shoaling based on stochastic evolution equations. Coastal Engineering, 33, 203-232.

Kofoed-Hansen, H, H K Johnson, P Astrup and J Larsen (2000): Prediction of waves and sea surface roughness in restricted coatsal waters. Paper presented at the 27th International Conference of Coastal Engineering, 16-21 July 2000, Sydney, Australia.

Komen, G.J., P.A.E.M. Janssen, V. Makin, K. Mastenbroek and W. Oost (1998): Review: On the sea state dependence of the Charnock parameter, J. Glob Atmos. Ocean System, 5, 367-388.

Komen, G.J., Cavaleri, L., Doneland, M., Hasselmann, K., Hasselmann S. and Janssen, P.A.E.M. (1994) Dynamics and modelling of ocean waves. Cambrigde University Press, UK, 560pp.

Kramer S.C., M. D. Piggott, J. Hill, L. Kregtin, D. Pritchard, B. Elsaesser (2014), The modelling of tidal turbine farms using multi-scale, unstructured mesh models, Proceedings of the 2nd International Conference on Environmental Interactions of Marine Renewable Energy Technologies (EIMR2014), 28 April - 02 May 2014, Stornoway, Isle of Lewis, Outer Hebrides, Scotland.

Kudryatsev, V.N. and V.K. Makin (1996): Transformation of wind in the coastal zone. KNMI, Scientific Report WR 96-04.

Large, W.G. and S. Pond (1981): Open ocean momentum flux measurements in moderate to strong winds. J. Phys. Oceanogr., 14, 464-482.

Leonard, A. (1974), Energy Cascades in Large-Eddy Simulations of Turbulent Fluid Flows, *Advances in Geophysics*, 18, pp 237-247.

Lilly, D.K. (1966), On the Application of the Eddy Viscosity Concept in the Inertial Subrange of Turbulence, *NCAR Manuscript No. 123*, National Center for Atmospheric Research, Boulder, Colorado.

Makin V.K. and V.N. Kudryatsev (1999): Coupled sea surface-atmosphere model. 1. Wind over waves coupling. J. Geophys. Res., 104C, 7613-7623.

Makin V.K., V.N. Kudryatsev and C. Mastenbroek (1995): Drag of the sea surface. Boundary-Layer Meteorol., 73, 159-182.

Mastenbroek, C., G.J.H. Burgers, P.A.E.M. Janssen (1993): The dynamical coupling of a wave model and a storm surge model through the atmospheric boundary layer. J. Phys. Oceanogr., 23, 1856-1866.

Merzi, N. and W.H. Graf (1985): Evaluation of the drag coefficient considering the effects of mobility of the roughness elements. Ann. Geophys., 3, 473-478.

Monbaliu, J. (1994): On the use of the Donelan wave spectral parameter as a measure for the measure for the roughness of wind waves. Boundary-Layer Meteorol, 67, 277-291.

Munk, W., Anderson, E. (1948), Notes on the theory of the thermocline, *Journal of Marine Research*, 7, 276-295.

Nelson, R.C. (1987): Design wave heights on very mild slopes: An experimental study, Civil. Eng. Trans., Inst. Eng., Aust., 29, 157-161.

Nelson, R.C. (1994): Depth limited wave heights in very flat regions, Coastal Eng., 23, 43-59.

Nielsen, P. (1979): Some basic concepts of wave sediment transport. Series paper 20 Institute of Hydrodynamic and Hydraulic Engineering, Technical University of Denmark,160pp.

Nordeng, T.E. (1991): On the wave age dependent drag coefficient and roughness length at sea. J. Geophys. Res., 96, 7167-7174.

Oddo P. and N. Pinardi (2007), Lateral open boundary conditions for nested limited area models: A scale selective approach, Ocean Modelling 20 (2008) 134-156.

Oost, W.A. (1998): The Knmi HEXMAX stress data – a reanalysis. Boundary-Layer Meteorology, 86, 447-468.

Perrie, W. and B. Toulany (1990): Fetch relations for wind-generated waves as a function of wind-stress scaling, J. Phys. Oceanogr., 20, 1666-1681.

Phillips, O.M. (1960): On the dynamics of unsteady gravity waves of finite amplitude, Part 1. J. Fluid Mech., 9, 193-217.

Phillips, O.M. (1981): The structure of short gravity waves on the ocean surface. In: Spaceborn Synthetic Aperture Radars for Oceanography, the Johns Hopkins Press.

Plant, W. J. (1982): A relationship between wind stress and wave slope, J. Geophys. Res., 87(C3), 1961–1967.

Rasmussen, J.H. (1999): Deterministic and stochastic modelling of surface gravity waves in finite water depth. Ph.D. thesis, Department of Hydrodynamic and Water Resources (ISVA), Technical University of Denmark.

Resio, D.T., B. Tracy, C.L. Vincent, and J.H. Rasmussen (1999): Non-linear energy fluxes and the finite-depth equilibrium range in wave spectra. Submitted to J. Phys. Oceanogr.

Ris, R.C. (1997): Spectral modelling of wind waves in coastal areas, PhD. thesis, Delft 1997.

Ris, R.C., L.H. Holthuijsen and N. Booij (1999): A third-generation wave model for coastal regions. 2. Verification. J. Geophys. Res., 104C, 7667-7681.

Rodi, W. (1980), Turbulence Models and Their Application in Hydraulics - A State of the Art Review, *Special IAHR Publication*.

Ruessink, B.G., Walstra, D.J.R. and Southgate, H.N. (2003): Calibration and verification of a parametric wave model on barred beaches, Coastal Eng., 48, 139-149.

Schneggenburger, C., 1998: Spectral wave modelling with nonlinear dissipation. Phd. dissertation, GKSS-Forschungszentrum Geesthacht GmbH. 117pp.

Shemdin, O., K. Hasselmann, S. V. Hsiao and K. Heterich (1978): Nonlinear and linear bottom interaction effects in shallow water. Turbulent Fluxes through the sea surface, Wave Dynamics and Prediction. NATO Conf. Ser. V, 1, 347-365.

Sleigh, P.A., Gaskell, P.H., Bersins, M. and Wright, N.G. (1998), An unstructured finite-volume algorithm for predicting flow in rivers and estuaries, Computers & Fluids, Vol. 27, No. 4, 479-508.

Smagorinsky (1963), J. General Circulation Experiment with the Primitive Equations, *Monthly Weather Review*, 91, No. 3, pp 99-164.

Smith, S.D. (1988): Coefficients for sea surface wind stress, heat flux and wind profiles as function of wind speed and temperature. J. Geophys. Res., 93C, 15,467 - 15,472.

Smith, S.D. and E.G. Banke (1975): Variation of the sea surface drag coefficient with wind speed. Quart. J. Roy. Meteor. Soc, 101, 665-673.

Smith, S.D. (1980): Wind stress and heat flux over the open ocean in gale force winds. J. Phys. Oceanogr.,10, 709-726.

Smith, S.D., R.J.Anderson, W.A. Oost, C. Kraan, N. Maat, J. DeCosmo, K.B. Katsaros, K.L. Davidson, K. Bumke, L.Hasse, and H.M. Chadwick (1992): 'Sea surface wind stress and drag coefficients: The HEXOS results. Boundary-Layer Meteorology, 60, 109-142.

Smith, S.D., Katsaros, K.B., Oost, W.A. and Mestayer, P. (1996): The impact of the HEXOS Programme. Boundary-Layer Meteorol.,78, 109-142.

Schneggenburger, C., 1998: Spectral wave modelling with nonlinear dissipation. Phd. dissertation, GKSS-Forschungszentrum Geesthacht GmbH. 117pp.

Song, Y and Haidvogel D., (1994) A semi-implicit ocean circulation model using a generalized topography-following coordinate system, Journal of Comp. Physics, 115, pp. 228-244.

Sørensen, O.R., M. Rugbjerg, L.S. Sørensen and H. Kofoed-Hansen (2004): A third-generation spectral wind-wave model using an unstructured finite volume technique. Manuscript in preparation.

Taylor, P.K. and M.J. Yelland (1999): The dependence of sea surface roughness on the height and steepness of the waves. Manuscript submitted to. J. Phys. Oceanography, July 1999.

Taylor, P.A. and R.J. Lee (1984): Simple guidelines for estimating wind speed variations due to small-scale topographic features, Climatol Bull, 18, 3-32.

Toba, Y., N. Iida, H. Kawamura, N. Ebuchi and I.S.F. Jones (1990): Wave dependence on sea-surface wind stress. J. Phys. Oceanogr., 20, 705-721.

Tolman, H. L.(2003): Treatment of unresolved islands and ice in wind wave models. Ocean Modelling, 4, 219-231.

Tolman, H.L. (2002): Limiters in third-generation wind wave models. Global Atmos. and Ocean. System, 8, 67-83.

Tolman, H. L. (2002): Alleviating the Garden Sprinkler Effect in wind wave models. Ocean Modelling, 4, 269-289.

Tolman, H.L. and D. Chalikov (1996): Source terms in a third-generation wind wave model. J. Phys. Oceanogr., 26, 2497-2518.

Troen, I. and E.L. Petersen (1989): European Wind Atlas, Risø National Laboratory. 656pp.

U S Army Corps of Engineers, Coastal Engineering Research Center (1973, 1984), Shore Protection Manual.

Vledder, G. van (1999): Source term investigation in SWAN. Report prepared for Rijkswaterstaat, the Netherlands, by Alkyon, the Netherlands. Report A162.

WAMDI-group, The: S. Hasselmann, K. Hasselmann, E. Bauer, P.A.E.M. Janssen, G.J. Komen, L. Bertotti, P. Lionello, A. Guillaume, V.C. Cardone, J.A. Greenwood, M. Reistad, L. Zambresky and J.A. Ewing (1988): 'The WAM model - a third generation ocean wave prediction model', J. Phys. Oceanogr., 18, 1775-1810.

Weber, S.L. (1991): Bottom friction for wind sea and swell in extreme depth-limited situations, J. Phys. Oceanogr., 21, 149-172.

Weber, S.L. (1988): The energy balance of finite depth gravity waves. J. Geophys. Res. 93, C4, 3601-3607.

Wu, J. (1980): Wind stress coefficients over sea surface near neutral conditions. A revisit, J. Phys. Oceanogr., 10, 727-740.

Wu, Jin (1994), The sea surface is aerodynamically rough even under light winds, *Boundary layer Metheorology*, 69, 149-158.

Wu, Jin (1980), Wind-stress Coefficients over sea surface and near neutral conditions - A revisit, *J. Phys. Oceanogr.*, 10, 727-740.

Yelland, M. and P.T. Taylor (1996): Wind stress measurements from the open ocean. J. Phys. Oceanogr., 26, 1712-1733.

Yelland, M., B.I. Moat, P.K. Taylor, R.W. Pascal, J. Hutchings and V.C. Cornell (1998): Wind stress measurements from the open ocean corrected for airflow distortion by the ship. J. Phys. Oceanogr., 28, 1511-1526.

Young, I. and L.A. Verhagen (1996): 'The growth of fetch-limited waves in water of finite depth. Part I and II.', Coastal Engineering, 29 pp.47-99.

Young, I. and R.M. Gorman (1995): 'Measurements of the evolution of ocean wave spectra due to bottom friction', J. Geophys. Res, 100, C6, pp.10987-11004.

Young, I.R. (1999): Wind generated ocean waves, in Elsevier Ocean Engineering Book Series, Volume 2, Eds. R. Bhattacharyya and M.E. McCormick, Elsevier.

Zhao, D.H., Shen, H.W., Tabios, G.Q., Tan, W.Y. and Lai, J.S. (1994), Finite-volume 2-dimensional unsteady-flow model for river basins, *Journal of Hydraulic Engineering*, ASCE, 1994, 120, No. 7, 863-833.

INDEX