MIKE
Powered by DHI

MIKE 3 Wave Model FM

Hydrodynamic Module

User Guide

PLEASE NOTE

LIMITED LIABILITY

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

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

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

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

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

CONTENTS

1 About This Guide

1.1 Purpose

The main purpose of this User Guide is to enable you to use MIKE 3 Wave Model FM, Hydrodynamic Module, for determination and assessment of wave dynamics in ports, harbours and coastal areas. The User Guide is complemented by the Online Help.

1.2 Assumed User Background

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

In this case, “common sense” means a background in coastal hydraulics and oceanography, which is sufficient for you to be able to check whether the results are reasonable or not. This User Guide is not intended as a substitute for a basic knowledge of the area in which you are working: Mathematical modelling of hydraulic phenomena.

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

1.3 General Editor Layout

The MIKE 3 Wave Model FM setup editor consists of three separate panes.

1.3.1 Navigation tree

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

1.3.2 Editor window

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

For sections containing spatial data - e.g. internal wave generation, boundaries and output - a geographic view showing the location of the relevant items

will be available. The current navigation mode is selected in the bottom of this view, it can be zoomed in, zoomed out or recentered. A context menu is available from which the user can select to show the bathymetry or the mesh and to show the legend. From this context menu it is also possible to navigate to the previous and next zoom extent and to zoom to full extent. If the context menu is opened on an item - e.g. an internal wave generation zone - it is also possible to jump to this item's editor.

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

1.3.3 Validation window

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

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

1.4 Online Help

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

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

- Open the On-line Help system for browsing manually after a specific help page:
  Open the On-line Help system by selecting "Help Topics" in the main menu bar.

2 Introduction

MIKE 3 Wave Model FM is a non-hydrostatic wave model that is capable of simulating wave processes from deep water to nearshore. This model enhances the versatility and applicability of the MIKE software in offshore, coastal and port engineering.

2.1 Short Description

MIKE 3 Wave Model FM is a phase-resolving wave model based on the 3D Navier-Stokes equations and with the free surface described by a height function. The numerical techniques applied are based on an unstructured (flexible) mesh approach.

2.2 Application Areas

The model is to be applied in the following areas:

- Ports and terminals
  - Wave agitation caused by short and long waves
  - Input to dynamic ship mooring analysis (MIKE 21 MA)
- Coastal areas
  - Non-linear wave transformation
  - Surf and swash zone hydrodynamics
  - Wave breaking and run-up
  - Coastal flooding
  - Tsunami (transient) modelling
- Coastal structures
  - Wave overtopping
  - Wave transmission (and reflection) through porous structures
- Offshore environments
  - Transformation of steep non-linear waves
  - 3D wave kinematics for structural load calculations

3 Getting Started

The purpose of this chapter is to give you a general check list, which you can use for determination and assessment of wave dynamics in ports, harbours and coastal areas using MIKE 3 Wave Model FM.

The work will normally consist of the six tasks listed below:

1. Defining and limiting the wave problem
2. Collecting data
3. Setting up the model
4. Calibrating and verifying the model
5. Running the production simulations
6. Presenting the results

Each of these six tasks are described for a "general wave study" in the following sections. For your particular study only some of the tasks might be relevant. Please note that whenever a word is written in *italics* it is included as an entry in the *Online Help* and in the Reference Manual.

3.1 Defining and Limiting the Wave Problem

3.1.1 Identify the wave problem

When preparing to do a wave study you have to assess the following before you start to set up the model:

- What are the "wave conditions" under consideration in the "area of interest"?
- What are the "important wave phenomena"?

The following phenomena should be taken into consideration:

  - Shoaling
  - Refraction
  - Diffraction
  - Partial reflection/transmission
  - Bottom dissipation
  - Wave breaking
  - Run-up
  - Wind-wave generation
  - Frequency spreading
  - Directional spreading
  - Wave-wave interaction
  - Wave-current interaction

MIKE 3 Wave Model FM can handle these phenomena with the exception of wind-wave generation.

3.1.2 Check MIKE 3 Wave FM capabilities

Next, check if MIKE 3 Wave Model FM is able to solve your problem. This you can do by turning to Section 2, which gives a short description of MIKE 3 Wave Model FM and an overview of the type of applications for which MIKE 3 Wave Model FM can be used, and by consulting the Scientific Documentation.

3.1.3 Define computational domain

Draw up your model domain on a sea chart showing the area of interest and the area of influence. This is normally an iterative process as on one hand you should keep the model domain as small as possible, while on the other hand you have to include the total area of influence. You also have to consider the location of porous structures.

The choice of the discrete resolution in the geographic and spectral space depends on the wave conditions for which simulations are to be performed and on the bathymetry and forcing fields (current, water level):

- Discrete resolution in the geographical space must be selected to provide adequate resolution of the bathymetry.
- Discrete resolution in the geographical space and the vertical space must be selected to provide adequate resolution at the wave field under consideration.

3.1.4 Check computer resources

Finally, before you start to set up the model, you should check that you are not requesting unrealistic computer resources:

- The CPU time required should be estimated.
- The Disk Space required should be estimated.

It is recommended to run the MIKE 3 Wave Model FM examples included in the installation for assessment of the computational speed on your PC and to assess the disk space consumption.

3.2 Collecting Data

This task may take a long time if, for example, you have to initiate a monitoring program. Alternatively it may be carried out very quickly if you are able to

use existing data which are immediately available. In all cases the following data should be collected:

- Bathymetric data such as charts from local surveys or, for example, from the Hydrographic Office, UK, or MIKE C-MAP
- Wave data, which might be measurements (existing or planned specifically for your model), observations, wave statistics, etc.
- Information on type of structures for assessment of the reflection properties
- Calibration and validation data; these might be measured wave parameters at selected locations, observations, etc.

3.3 Setting up the Model

3.3.1 What does it mean

"Setting up the model" is actually another way of saying transforming real world events and data into a format which can be understood by the numerical model MIKE 3 Wave Model FM. Thus generally speaking, all the data collected have to be resolved on the spatial grid selected.

3.3.2 Mesh and bathymetry

Providing MIKE 3 Wave Model FM with a suitable mesh and bathymetry is essential for obtaining reliable results from your model. Setting up the mesh includes the appropriate selection of the area to be modelled, adequate resolution of the bathymetry and wave field under consideration and definition of codes for specification of location of porosity zones and sponge layers. Furthermore, the resolution in the geographical space must also be selected with respect to stability considerations.

Describing the water depth in your defined model domain is one of the most important tasks in the modelling process. A few hours less spent in generating the mesh covering the bathymetry may later on mean extra days spent in the calibration process.

The mesh file including your bathymetry is generated by MIKE Zero Mesh Generator, which is a tool for the generation and handling of unstructured meshes, including the definition and editing of boundaries. If a structured mesh consisting of quadrilateral elements is used, the mesh can also be generated using the MIKE Zero Bathymetry Editor.

3.3.3 Porosity

The location of the porous structures zones can be determined from the boundary information.

3.3.4 Sponge layer

The location of the sponge layers is can be determined from the boundary information.

3.3.5 Wave data

In most cases you will force the model by waves generated inside the model domain. The *internal wave generation* of waves allows you to absorb all waves leaving the model domain (radiation type boundaries).

3.4 Calibrating and Verifying the Model

3.4.1 Purpose

Having completed all the tasks listed above you are ready to do the first time-domain wave simulation and to start the calibration of the model. The purpose of the calibration is to tune the model in order to reproduce known/measured wave conditions. The calibrated/tuned model is then verified by running one or more simulations for which measurements are available without changing any tuning parameters. This should ensure that simulations can be made for any wave conditions similar to the calibration and verification wave conditions with satisfactory results. However, you should never use simulation results, whether verified or not, without checking if they are reasonable or not.

3.4.2 Verification

The situations which you select for calibration and verification of the model should cover the range of situations you wish to investigate in the production runs. However, as you must have some measurements/observations against which to calibrate and, as the measurements are often only available for short periods, you may only have a few situations from which to choose. When you have finished the calibration you can run one more simulation for which you have measurements (or other data) without changing the calibration parameters. If you then get a satisfactory agreement between the simulation results and the measurements you can consider your calibration to be successful.

3.4.3 Calibration parameters

When you run your calibration run for the first time and compare the simulation results to your measurements (or other information) you will, in many cases, see differences between the two. The purpose of the calibration is then to tune the model so that these differences become negligible. You can change the following model specifications in order to reduce the differences:

- Wave conditions
- Porosity
- Bed resistance
- Bathymetry

Recommendations on how the specification can be changed are given in the Reference Manual.

3.5 Running the Production Simulations

As you have calibrated and verified the model you can get on to the "real" work, that is doing your actual investigation. This will, in some cases, only include a few runs.

3.6 Presenting the Results

Throughout a modelling study you are working with large amounts of data and the best way of checking them is therefore to look at them graphically. Only in a few cases, such as when you check your bathymetry along a boundary or you want to compare simulation results to measurements in selected locations, should you look at the individual numbers. Much emphasis has therefore been placed on the capabilities for graphical presentation in MIKE Zero and it is an area which will be expanded and focused on even further in future versions. Essentially, one plot gives more information than scores of tables. A good way of presenting the model results is using contour plots of e.g. the calculated wave disturbance coefficient by using the Plot Composer or Result Viewer tool in MIKE Zero. Instantaneous pictures/videos of the simulated surface elevations can also be generated. For 3D visualisation MIKE Animator Plus is recommended.

4 Examples

4.1 General

One of the best ways of learning how to use a modelling system like MIKE 3 Wave Model FM is through practice. Therefore, examples have been included which you can go through yourself and which you can modify, if you like, to see what happens if some of the parameters are changed.

The specification data files for the example are included with the installation of MIKE Zero

- Ronne Harbour:
  .\Examples\MIKE_3\WaveModel_FM\HD\Ronne
- Breaking Waves on a Plane Beach:
  .\Examples\MIKE_3\WaveModel_FM\HD\Plane_Beach
- Breakwater Overtopping
  .\Examples\MIKE_3\WaveModel_FM\HD\Breakwater_Overtopping
- Coastal Flooding in Capbreton:
  .\Examples\MIKE_3\WaveModel_FM\HD\Capbreton

4.2 Ronne Harbour

4.2.1 Purpose of the example

The purpose of this example is to simulate the wave disturbance in Rønne harbour, Denmark, situated in the Baltic Sea. Of special interests is wave disturbance at the cruise terminal, see Figure 4.1. This example corresponds to the "Rønne Harbour" example in the User Guide for MIKE 21 Boussinesq Waves.

The event to be simulated corresponds to a situation occurring 10 hours per year (on average) and is characterised by having a significant wave height of H_{m0}= 2.65 m with a spectral peak period of T_p = 8.6 s. The waves are synthesized based on a mean JONSWAP spectrum, as the minimum wave period is set to T_{min} = 5.0s. Please note that the truncated wave spectrum is not rescaled, i.e. the incoming wave height is less than 2.65 m.

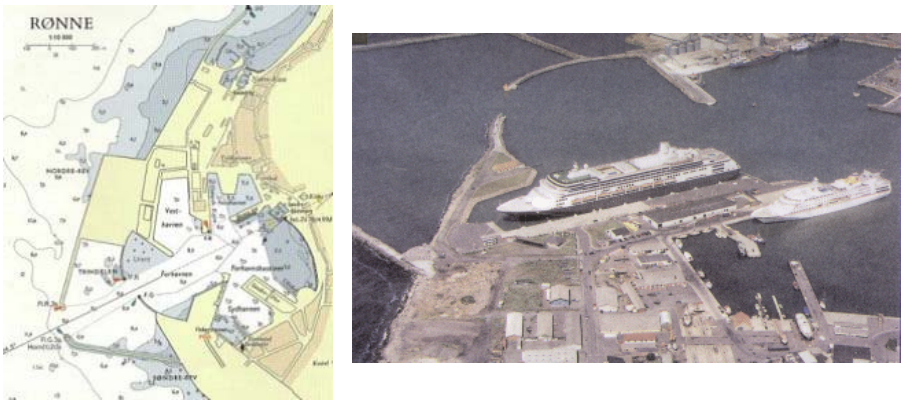

Figure 4.1 Right panel shows the new cruise ship terminal in Rønne harbour, Denmark. Left panel shows the harbour layout before the cruise terminal was constructed

4.2.2 Model setup

Simulations are performed using both a structured and an unstructured Mesh. The structured horizontal mesh consists of 21110 uniform quadrilateral elements (see Figure 4.2) and corresponds exactly to the mesh used in the MIKE 21 Boussinesq Wave example. Hence, there is a staircase approximation of the boundaries. The unstructured horizontal mesh consists of 55926 triangular elements (see Figure 4.3). Here a boundary fitted mesh is used. A non-equidistant (sigma_c=0.1, b=0 and theta=2) vertical discretization with 3 layers is applied. The total number of elements in the 3D structured and unstructured mesh is 63330 and 167778, respectively. The simulation period is 12 minutes.

For the simulation with structured mesh the incoming waves are specified using a relaxation zone: Line from (x,y)=(55.0m, 547.5m) to (x,y)=(55.0m, 277.5m) and the width of the ramp-up zone is 20m.

For the simulation with unstructured mesh the incoming waves are specified using a relaxation zone: Line from (x,y)=(479475.697m, 6105479.58m) to (x,y)=(479653.884m, 6105045.67m) and the width of the ramp-up zone is 20m.

At the harbour breakwater porosity layers are applied for simulation of partial wave reflection. Porosity values in the range 0.40-0.81 are used. The location of the porosity layers is shown in Figure 4.4. In the innermost part of the harbour a 50 wide sponge layer is used to absorb the waves here. In reality the waves will break at a small beach here, but this will not affect the waves in the central part of the harbour. For the simulation with structured mesh the porosity and the sponge layers is specified using dfs2-file. For the simulation with unstructured mesh the porosity and the sponge layers is specified based on

the boundary information. A plot showing the boundary codes is shown in Figure 4.5.

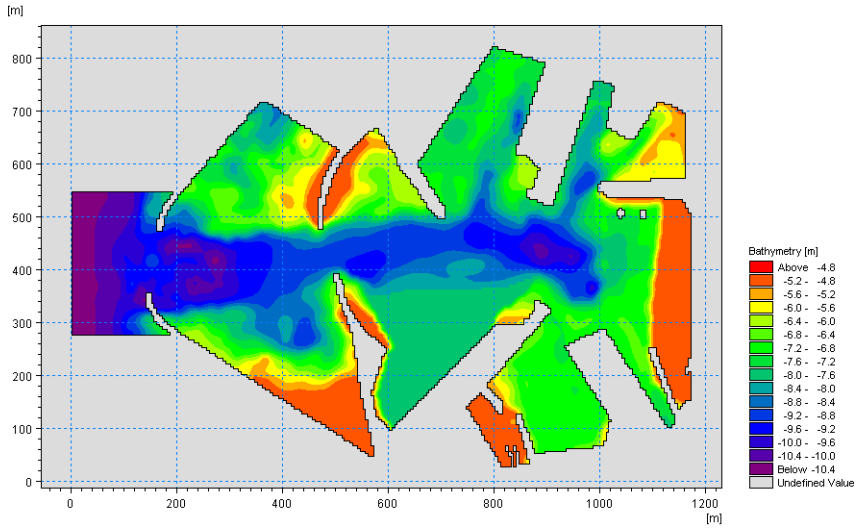

Figure 4.2 Computational domain (structured mesh)

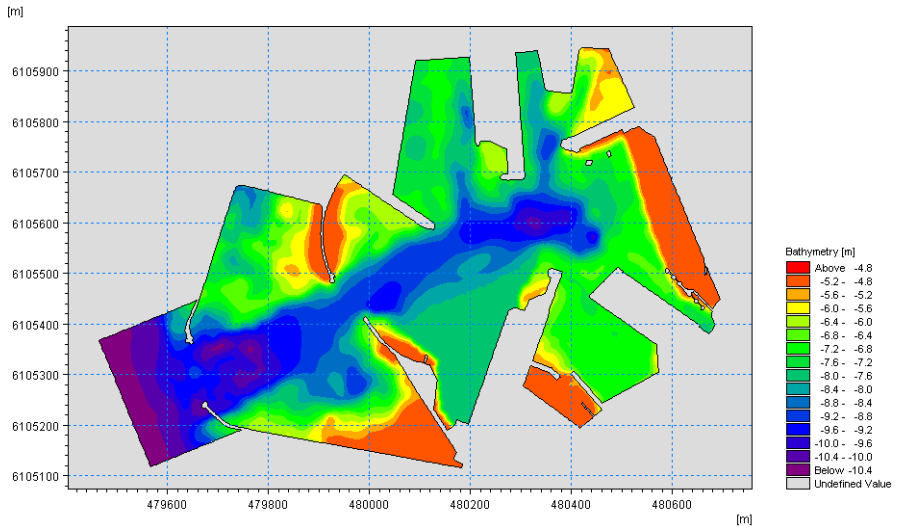

Figure 4.3 Computational domain (unstructured mesh)

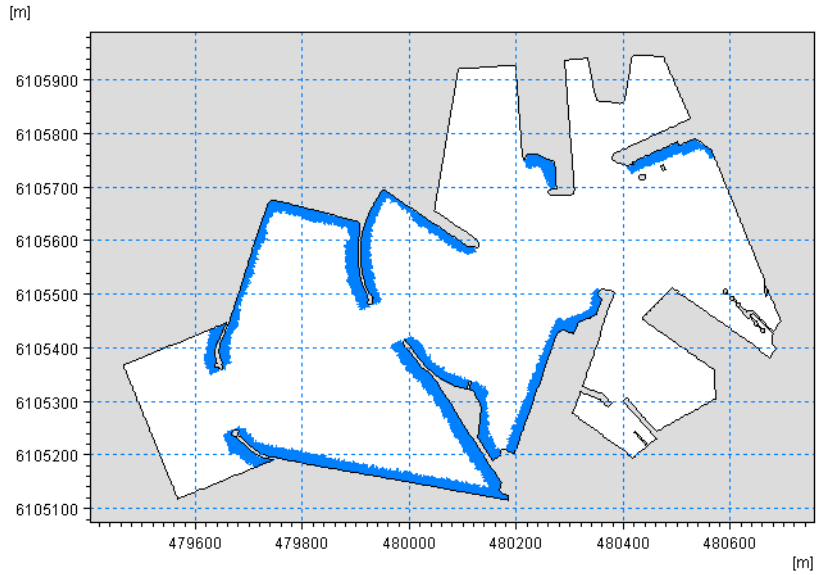

Figure 4.4 Porosity map (unstructured mesh)

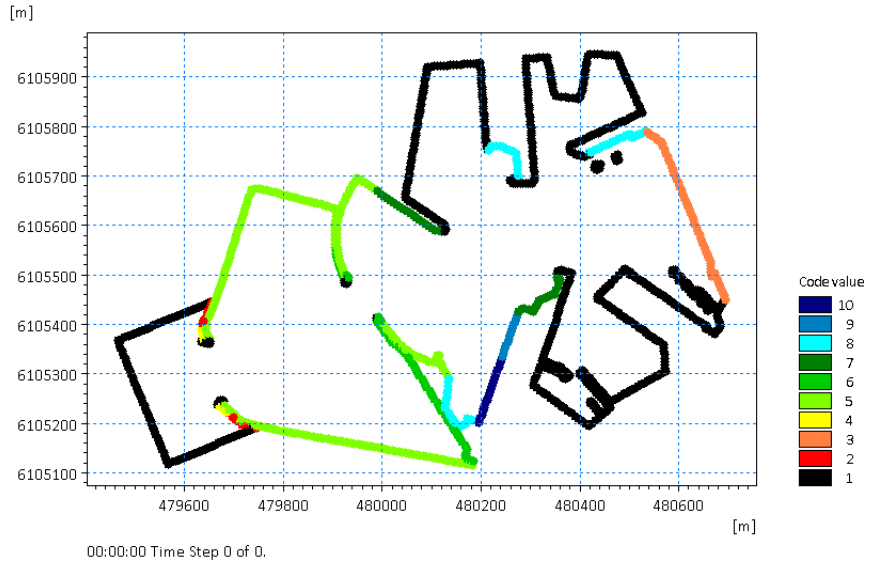

Figure 4.5 Boundary codes for specification of the sponge layer (code 2 and 3) and of the porosity (code 4-10) for the simulation using unstructured mesh.

4.2.3 Model results

The model results are presented in Figure 4.6 and Figure 4.7. Here contour plots of the simulated wave disturbance coefficients (after 12 minutes) are shown.

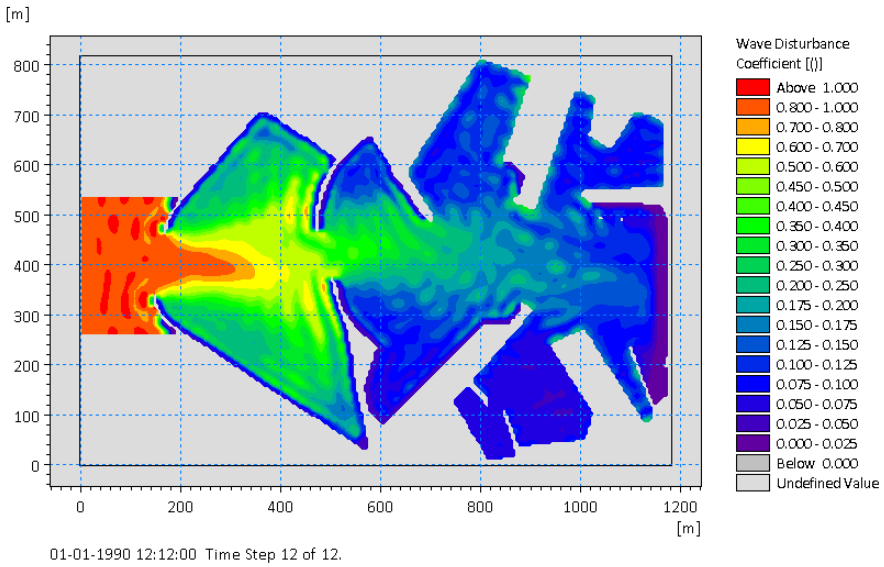

Figure 4.6 Contour plot showing the simulated wave disturbance coefficients. MIKE 3 Wave Model FM with structured mesh.

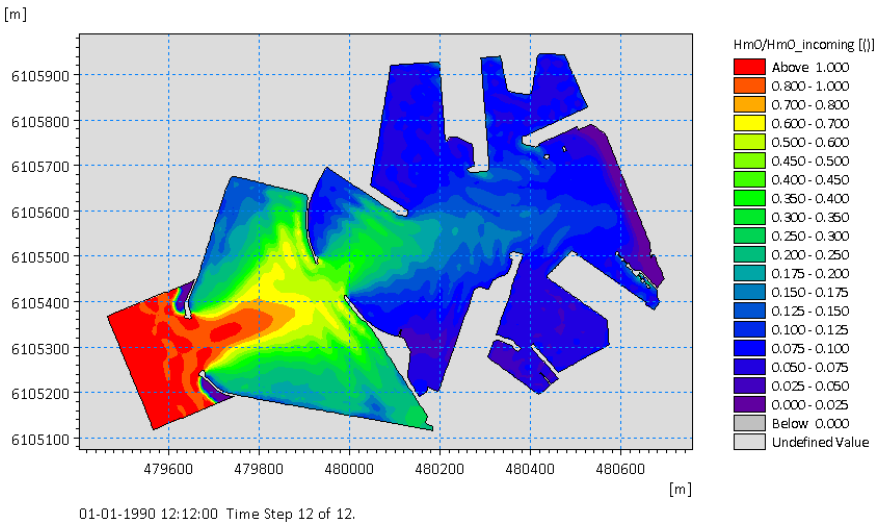

Figure 4.7 Contour plot showing the simulated wave disturbance coefficients. MIKE 3 Wave Model FM with unstructured mesh.

4.2.4 List of data and specification files

The following data files (included in the \Ronne folder) are supplied with MIKE 3 Wave Model FM:

File name: bathy_layout1998.dfs2
Description: Mesh file including mesh and bathymetry (structured)

File name: tri_layout1998_36m2.mesh
Description: Mesh file including mesh and bathymetry (unstructured)

Name: sponge.dfs2
Description: Sponge layer coefficients

Name: porosity.dfs2
Description: Porosity coefficients

File name: Quad_irregular.m3wfm
Description: MIKE 3 Wave Model FM specification file (structured mesh)

File name: Tri_irregular.m3wfm
Description: MIKE 3 Wave Model FM specification file (unstructured mesh)

4.3 Breaking Waves on a Plane Beach

4.3.1 Purpose of the example

Wave breaking and wave run-up on a gently sloping plane beach is considered in this example. The example concentrates on shoaling of regular waves and spilling type of wave breaking.

The experimental data by Ting and Kirby (1994) is used to validate MIKE 3 Wave Model FM. Ting and Kirby (1994) presented measurements for both spilling breakers and plunging breakers on a plane sloping beach. They looked at the wave breaking, undertow and turbulence.

4.3.2 Model setup

The model setup follows the experimental setup by Ting and Kirby (1994). The slope of the plane beach was 1/35 starting at a depth of 0.40m (see Figure 4.8). This test case is a one-dimensional flow problem. Hence, a one-element wide channel is used in the simulation. The horizontal mesh consists of quadrilateral elements with an edge length of 0.02m. An equidistant vertical discretization with 12 layers is applied. The incoming waves are specified using a relaxation zone: Line from (x,y)=(5.0m, 0.02m) to (x,y)=(5.0m, 0.0) and the width of the ramp-up zone is 3.0m. The waves are generated using the stream function wave theory with a wave period of 2.0s and a wave height

of 0.125m. Horizontal and vertical eddy viscosity has been applied using the k-ε formulation. Bed friction is applied with a roughness height of 0.0001m.

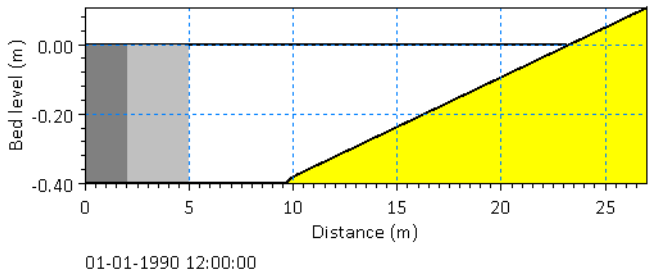

Figure 4.8 Sketch illustrating numerical setup

4.3.3 Model results

Figure 4.9 shows a line series of the simulated surface elevation on top of the bathymetry. The wave breaking and wave run-up processes are clearly seen on this figure. The cross-shore variation of the wave crest elevation, wave trough elevation and mean water level are shown in Figure 4.10.

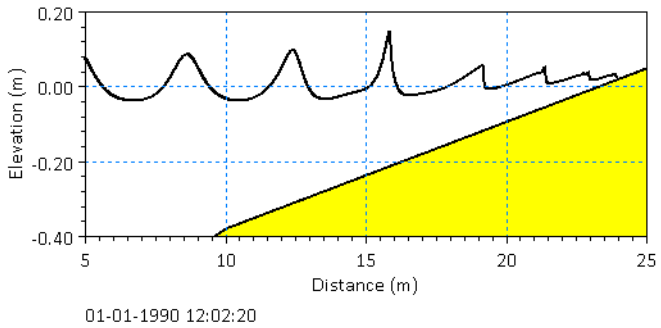

Figure 4.9 The cross-shore variation of surface elevation for the test of Ting and Kirby (1994) with spilling breakers (T=2s).

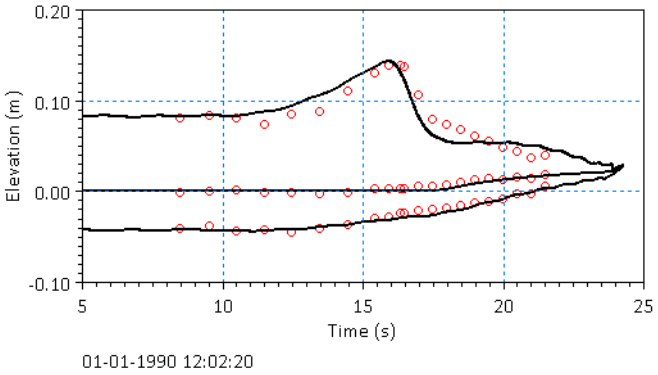

Figure 4.10 The cross-shore variation of the wave crest elevation, wave trough elevation and mean water level for the test of Ting and Kirby (1994) with spilling breakers (T=2s). Black line: MIKE 3 Wave Model FM; Red circles; experimental data.

The modelled and measured undertow is compared in Figure 4.11. The locations of the measurements at position A-H are x=[8.735m, 15.945m, 16.665m, 17.275m, 17.885m, 18.495m, 19.110m, 19.725m]. It is seen that MIKE 3 Wave Model FM does a fair job a predicting the undertow at profiles A, F, G and H. At locations B, C, D and E the model over-predicts the undertow velocities in the lower part of the water. This is likely related to the wave being too large before breaking and breaking slightly further off-shore in MIKE 3 Wave Model FM compared to the measurements.

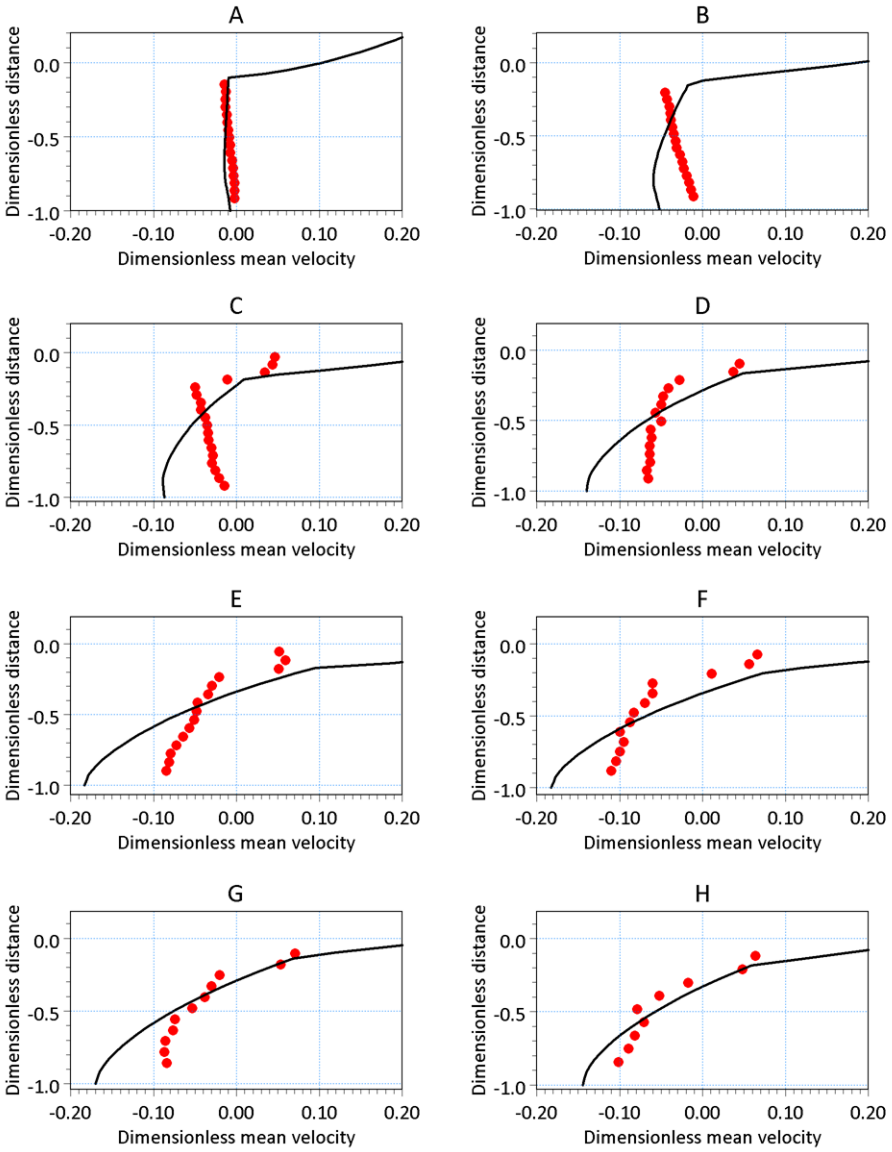

Figure 4.11 Comparison between measured and modelled undertow at the 8 locations A-H.
Black line: MIKE 3 Wave Model FM; Red circles; experimental data

The definition of the parameters shown in Figure 4.11 is described below.

The dimensionless mean velocity is defined by $u_{mean}/(\sqrt{gh_{mean}})$ and the dimensionless distance is defined by $(z - s_{mean})/h_{mean}$. The mean water depth, h_{mean}, is determined as the still water depth plus the calculated mean surface elevation, s_{mean}, and u_{mean} is the calculated mean velocity.

4.3.4 List of data and specification files

The following data files (included in the \Plane_Beach folder) are supplied with MIKE 3 Wave Model FM:

File name: quad_0.2m.mesh
Description: Mesh file including mesh and bathymetry

File name: regular_spilling.m3wfm
Description: MIKE 3 Wave Model FM specification file

4.4 Breakwater Overtopping

4.4.1 Purpose of the example

This example shows how MIKE 3 Wave Model FM can be used to simulate wave overtopping over both an impermeable breakwater and a porous breakwater. The example also shows the setup of a 3D porosity zone map and calculation of reflection coefficients from the porous breakwater.

Bruce et al. (2009) presented results from physical model tests of wave overtopping over breakwater structures. The main focus was on porous breakwaters with different types of armour layers. As a starting point a set of reference tests were made with an impermeable breakwater. Bruce et al. (2009) performed a number of experiments varying the water depth and the wave conditions. In this installation example, the case with still water depth h=0.222m and irregular waves with significant wave height Hm0=0.074m, and peak wave period Tp=1.56s is simulated.

4.4.2 Model setup

The geometry of the model setup follows the flume experiments in Bruce et al. (2009). It consists of a flume with a length of 16m and a width of 1m. As the experiments can be considered to be two dimensional the width of the flume is only resolved with one element. The length of the flume is resolved with quadrilateral elements with an edge length of 0.025m. The water depth is resolved with 10 non-equidistant sigma layers for the impermeable case and 30 non-equidistant sigma layers for the porous case.

For the impermeable breakwater case the toe of the impermeable breakwater is placed at a distance of 10m from the wave maker. The breakwater has a slope at 1:1.5 and the breakwater crest level is at 0.2812m which gives a free board of Rc=0.0592m (see Figure 4.12). For the porous breakwater case the water depth is constant in the whole domain, and the breakwater is modelled using a porosity map. In the experimental setup the breakwater was composed of three materials; core, filter layer and armour layer (see Figure 4.13). The thickness of the layers was related to the diameter of the applied armour

units in the experiments. For the selected case with an armour layer composed of natural rocks, the stones had a diameter, d50 = 0.03m. The corresponding grain diameters for the filter and core material were 0.014m and 0.007m. The values of the linear and nonlinear friction parameters are set to α = 500 and β = 2 and the porosity is set to 0.4 in all three zones.

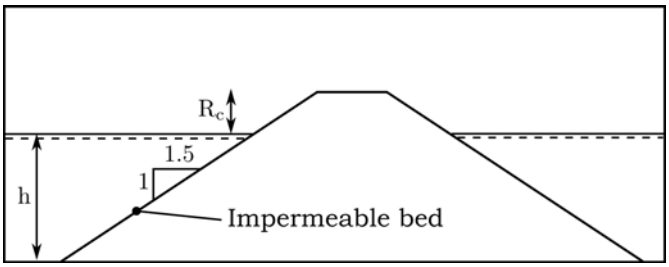

Figure 4.12 Layout of the impermeable breakwater structure following the experimental setup given in Bruce et al. (2009).

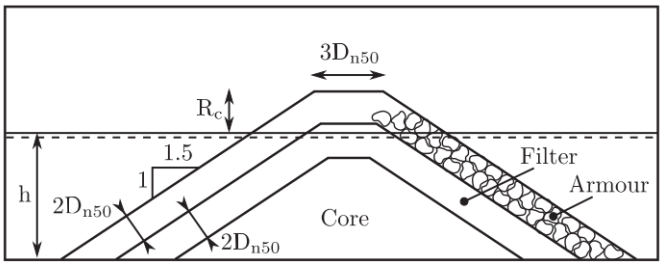

Figure 4.13 Layout of the porous breakwater structure following the experimental setup given in Bruce et al. (2009).

The porosity map that defines the breakwater is generated by running a simulation with only one time step. The initial water depth in this pre-processing step should be large enough to cover the entire volume of the porous breakwater. In the output dialog, only porous zones are selected. This will generate one dfsu file that only includes an item for the porous zones. The user will now have to modify this file using the Data Manager in order to define the porous zone value in the relevant elements. If the breakwater contains three different porous zones the elements that are inside these zones should be given the values 1, 2, and 3. This dfsu file is saved and named e.g. porosity_zones.dfsu and is subsequently used in the porosity dialog in the final model setup by selecting "Porosity zones (3D map)". Here, values for stone diameter, porosity, and resistance parameters can be assigned to each zone. A close-up of the 3D porosity zone map is shown in Figure 4.14.

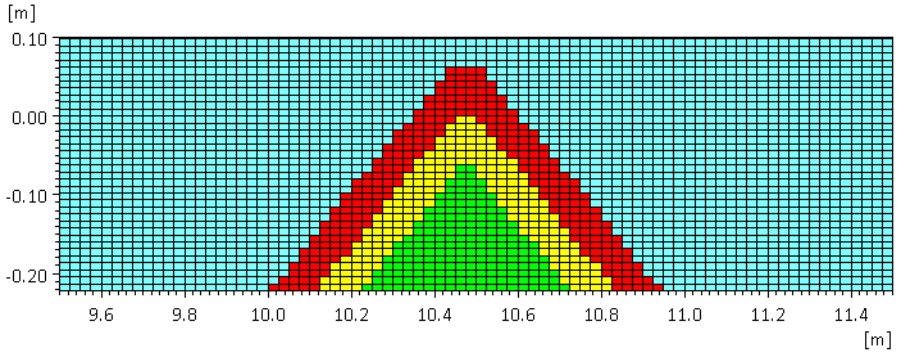

Figure 4.14 3D porosity map (dfsu-file) with three different porous zones. Yellow: Core; Green: filter layer; Red: armour. layer

The incoming waves are specified using a relaxation zone: Line from (x,y)=(3.0m, 0.0m) to (x,y)=(3.0m, 1.0) and the width of the ramp-up zone is 2.8m.

4.4.3 Model results

The overtopping is measured by adding a discharge output line at the crest of the breakwater. For the porous breakwater case "Cross section excluding porosity" should be selected. In Figure 4.15 and Figure 4.16 is shown the time series of the accumulated discharge for the impermeable breakwater and the porous breakwater. In Figure 4.17 is shown a snapshot of the surface elevation variation for the impermeable breakwater case at a time where overtopping occur.

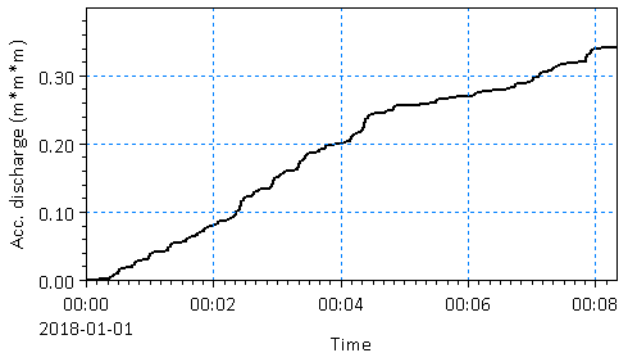

Figure 4.15 Time series of the accumulated discharge for the impermeable breakwater case.

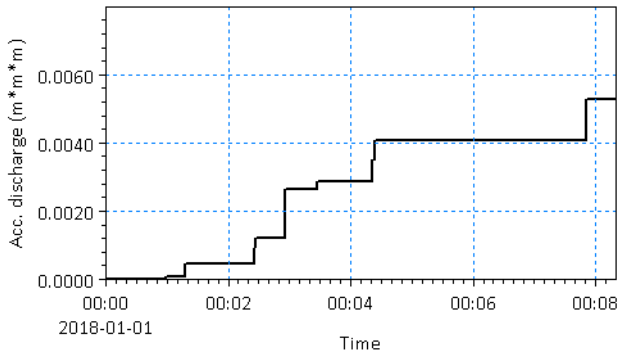

Figure 4.16 Time series of the accumulated discharge for the porous breakwater case.

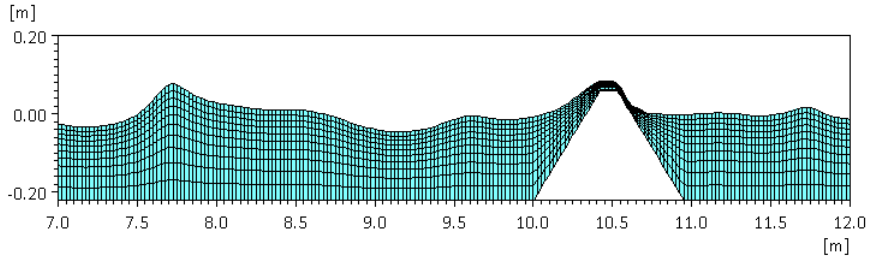

Figure 4.17 A snapshot of the surface elevation variation for the impermeable breakwater case

Calculating reflection from breakwater

The example setup includes output of the surface elevation at five wave gauges in front of the breakwater. These are placed with a non-equidistant spacing in order to be used for a reflection analysis. With this, the incident and reflected wave spectra can be separated and the reflection coefficient can be computed as the ratio between the reflected and incident wave height.

An example of performing the reflection analysis is included based on the WS reflection analysis included in the Mike Zero Wave Analysis Tools. In this case the reflection coefficient is found to be 0.462.

4.4.4 List of data and specification files

The following data files (included in the \Breakwater_Overtopping folder) are supplied with MIKE 3 Wave Model FM:

File name: quad_0.025m_breakwater.mesh
Description: Mesh file including mesh and bathymetry for the case with an impermeable breakwater

File name: quad_0.025m.mesh
Description: Mesh file including mesh and bathymetry for the case with a porous breakwater

File name: porosity_zones.dfsu
Description: 3D porosity map defining the three zones

File name: Irregular_impermeable.m3wfm
Description: MIKE 3 Wave Model FM specification file

File name: Irregular_porous.m3wfm
Description: MIKE 3 Wave Model FM specification file

File name: WsReflectionAnalysisPar1.wsra
Description: Wave reflection analysis

4.5 Coastal Flooding in Capbreton

4.5.1 Purpose of the example

The purpose of this example is to simulate the coastal flooding during a storm event in the town of Capbreton, France, situated on the Atlantic coast (Figure 4.18).

MIKE 3 Wave Model FM is used to calculate the overtopping rates in the entrance channel during this event.

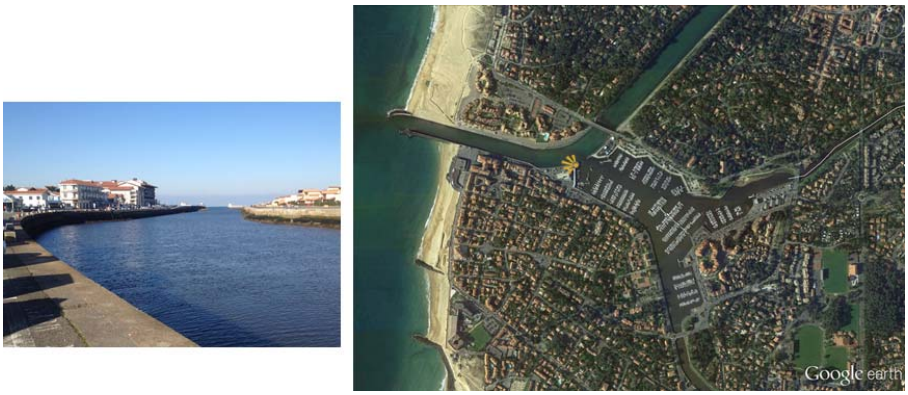

Figure 4.18 Left panel shows a picture of the entrance channel, taken from the yellow marker on the right panel. Right panel shows a satellite picture of the city of Capbreton (Source: Google Earth).

4.5.2 Model setup

The domain covers the entrance channel as well as the coastal areas south of the port (see Figure 4.2). The model size is 1600m by 1700m. Land is included as part of the bathymetry to allow coastal flooding in the model. Simulations are performed using a mesh with more than 100.000 uniform quadrilateral elements with a grid spacing of 5m.

The orientation of the domain area is defined to align the model grids with the entrance channel walls. This orientation is also adapted to the main direction of the incoming waves.

A wave generation line is located along the 10m depth contour (relative to NGF: general levelling of France). The simulated event corresponds to the peak of the main recent and widely documented storm on this coast: storm Christine, which happened from 2nd to 4th March 2014.

In front of the entrance channel, the peak of the storm is characterised by a significant wave height of H_{m0} = 2.55m, a spectral peak period of T_p = 18.3s, and a main wave direction of MWD = 290°N. The associated water level is 2.68m NGF. The water level is changed by specifying the initial surface elevation using a 2D input file, where the surface elevation is 2.68 m. The waves are irregular, and generated based on a mean JONSWAP spectrum. A wave absorbing sponge layer is applied along the southern boundary to absorb the waves going out of the model area. Sponge layers are also placed at the north and east boundary to absorb the waves propagating up the river. Note, that the reference level for the internal generation and the sponge layers should be set to 2.68m.

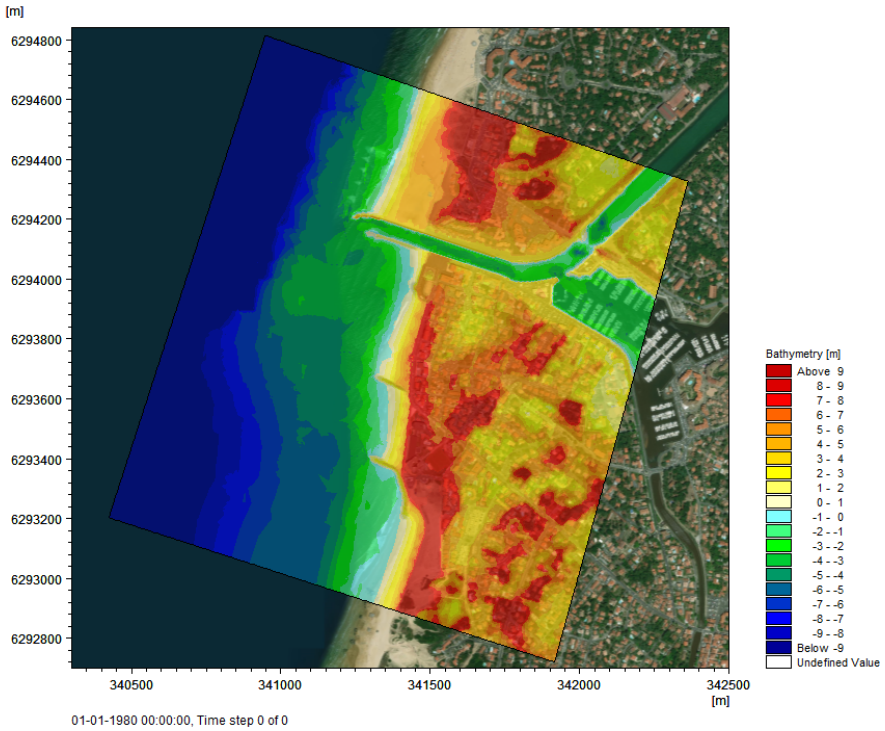

Figure 4.19 Domain and bathymetry for Capbreton harbour

The Capbreton example runs for about 30 minutes using a 3.2 GHz PC with 16 cores, to simulate 10 minutes (MPI parallelisation with 16 subdomains).

4.5.3 Model results

Figure 4.20 shows a 2D visualisation of the simulated instantaneous surface elevation at the moment of the arrival of the first generated waves in the entrance channel. You can make a similar plot using the file Area.plc after model execution.

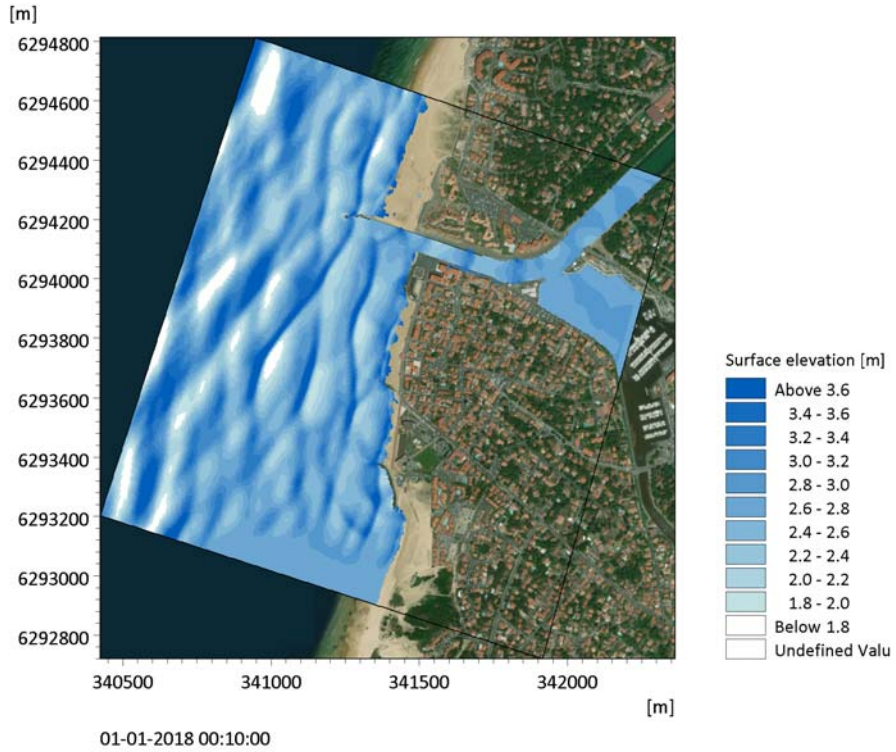

Figure 4.20 Model result: 2D visualisation of the instantaneous surface elevation after 10 minutes.

Figure 4.21 shows a 3D visualisation of the simulated instantaneous surface elevation. Flooding due to wave overtopping is seen both on the northern and southern quays of the channel. A screen shot of a video taken from the southern quay during the storm is also presented.

Figure 4.21 Wave overtopping in Capbreton harbour. Upper panel shows a 3D picture of the simulated instantaneous surface elevation in Capbreton entrance channel. Lower panel shows a screenshot of a video taken during the storm on the southern quay of the entrance channel (Source: YouTube, Mars 2014, retrieved from http://www.youtube.com/watch?v=6YuNtXfUHMM)

The wave overtopping in the entrance channel is calculated using discharge output lines along the vertical walls of the channel. Figure 4.22 shows the time series of the accumulated discharge across the two output lines, located on the map. You can make a similar plot using the file Results.plc after model execution. The discharge is positive when the flow occurs towards land. The estimated overtopping discharges across the quays can then be used as inputs for sources in a MIKE FLOOD model of Capbreton combining the effects of wave overtopping, tide, and of the two rivers flowing into the port.

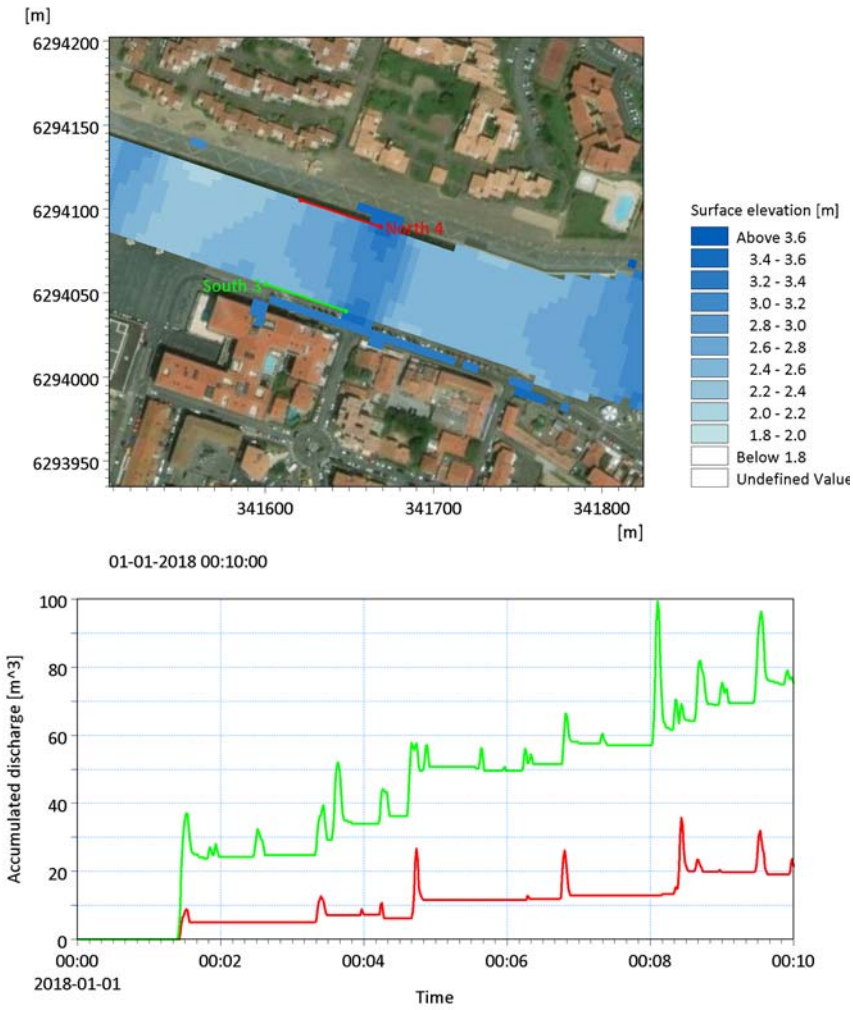

Figure 4.22 Upper panel shows a "zoomed-in" view of Figure 4.20 in the entrance channel with the location of two discharge output lines along the quays. Lower panel shows time series of the accumulated discharges across these lines over the duration of the simulation.

4.5.4 List of data and specification files

The following data files (included in the \Capbreton folder) are supplied with MIKE 3 Wave Model FM:

File name: Capbreton.mesh
Description: Mesh file including mesh and bathymetry

File name: Capbreton.m3wfm
Description: MIKE 3 Wave Model FM specification file

File name: Area.plc
Description: Plot Composer file for visualisation of the simulated instantaneous surface elevation in the entire domain

File name: Results.plc
Description: Plot Composer file for visualisation of the simulated instantaneous surface elevation in the entrance channel and two time series of accumulated discharges

File name: Capbreton.jpg+Capbreton.jpgw
Description: Georeferenced background satellite image file

5 BASIC PARAMETERS

5.1 Domain

Providing MIKE 3 Wave 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 and flow fields under consideration and definition of codes for porosity zones, sponge layers and closed boundaries. Furthermore, the resolution in the geographical space must also be selected with respect to stability considerations.

MIKE 3 Wave Model FM is based on the flexible mesh approach. 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.

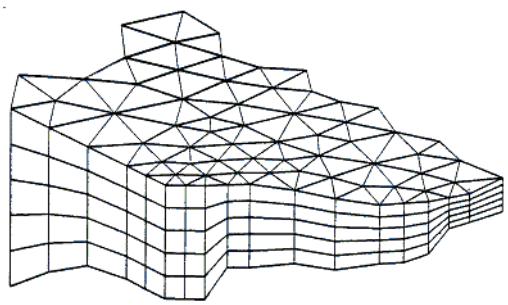

Figure 5.1 3D mesh using sigma coordinates

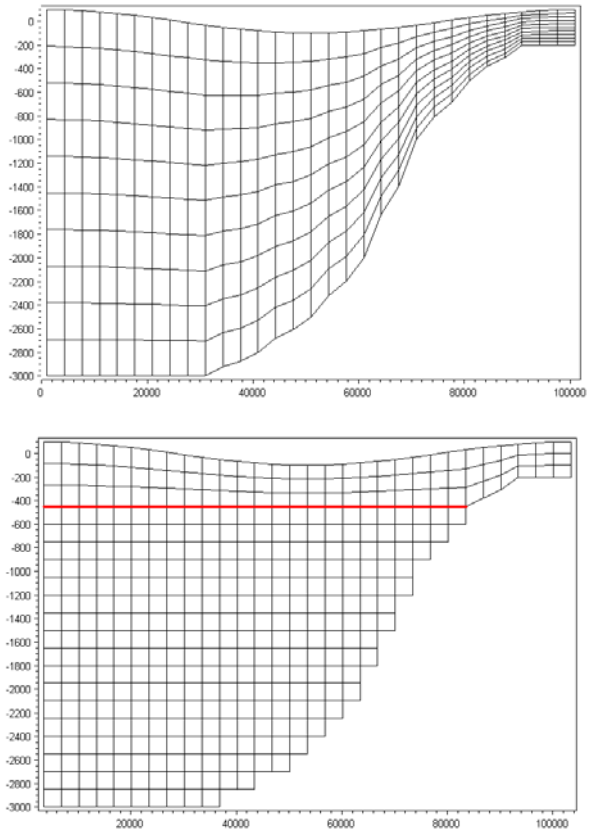

Figure 5.2 Illustrations of the different vertical grids.
Upper: sigma mesh, Lower: combined sigma/z-level mesh with simple bathymetry adjustment. The red line shows the interface between the z-level domain and the sigma-level domain.

5.1.1 Mesh and bathymetry

The mesh and bathymetry can be specified either using a mesh file or a bathymetry data file.

Mesh file

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 map projection and of the geographical position and bathymetry (Bed elevation) for each node point in the mesh. The file also includes information of the node-connectivity in the mesh.

Bathymetry data file

You generate your bathymetry data file in the MIKE Zero Bathymetry Editor, which is a tool for the generation of structured meshes. An example with a step-by-step description of how to use the Bathymetry Editor for creating a bathymetry data file is included with the installation. Please find this example in your installation folder under Examples\MIKEZero\BatEdit.

The bathymetry data file is a dfs2 file which contains the bathymetry (Bed elevation) and the following geographical information of the computational domain

- The map projection
- The geographical position of the grid origin
- The grid orientation.

The grid orientation is defined as the angle between true north and the y-axis of the model measured clockwise. A mnemonic way of remembering this definition is by thinking of NYC, which normally means **N**ew **Y**ork **C**ity, but which for our purpose means "from **N**orth to the **Y**-axis **C**lockwise", see Figure 5.3.

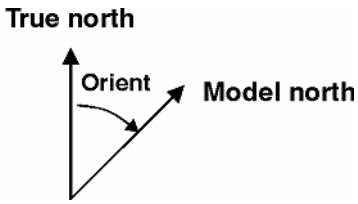

Figure 5.3 Definition of model orientation

The bathymetry data file also contains information of the true land value. True land value is the minimum value you have specified for land points when you prepared the bathymetry. All grid points with a depth value equal to or greater than the value you specify will be excluded from the computational domain. The value representing land is the forth element in the custom block called M21_Misc which consists of 7 elements of type float. The computational mesh is shown in the graphical view.

The bathymetry data file does not contain any information of the boundaries. Therefore starting at the grid origin and going counter-clockwise a boundary code (2, 3, 4 …) is set for each open boundary section of the grid. The boundary codes can be seen by clicking on the graphical view and select "Show mesh".

The bathymetry data file can also be converted to a mesh file using the MIKE Zero Mesh Converter Tool. This will allow you to edit the boundary codes using the MIKE Zero Data Viewer. Note, that a simulation using a mesh file will give different result than using the bathymetry data file directly due to the

difference in the determination of the bed elevation in the cell center (see Bed elevation).

Bed elevation

Using a mesh file for specification of the mesh and bathymetry, the bed elevation is given at the nodes (vertices) of the elements. The governing equations are solved using a cell-centred finite volume approach. Here the bed level is required at the cell center. This bed level is determined as the mean value of the node values.

Using a bathymetry data file for specification of the mesh and bathymetry, the bed elevation at the cell center of the elements is determined as the bed elevation specified in the input data file. The bed elevation at the nodes is determined as the area-weighted mean values of the bed elevation in the elements connected to the node and the weight factors are the areas of the connected elements.

5.1.2 Domain specification

Map projection

When the mesh is specified using a mesh file generated by the MIKE Zero Mesh Generator or the mesh is specified using a bathymetry data file generated by the MIKE Zero Bathymetry Editor the map projection is defined in the input data file and is only shown for reference in the user interface. If the map projection 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.

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

In most wave applications it is recommended to use sigma coordinates for the vertical discretization. The advantage using sigma coordinates is their ability to accurately represent the bathymetry and provide consistent resolution near the bed.

Sigma

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

- Equidistant
- Layer thickness

- Variable

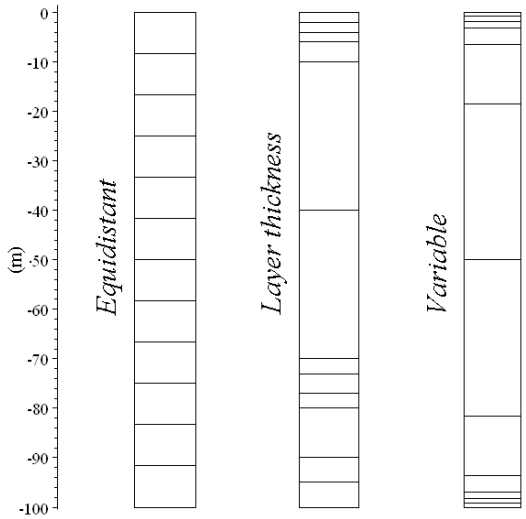

Figure 5.4 Examples of vertical distribution of layers for a water column 100 m deep. A number of 12 layers has been applied for all three options.
Left column: Equidistant distribution
Middle column: Layer thickness distribution
Right column: Variable distribution

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 3 Wave Model FM, Hydrodynamic Module.

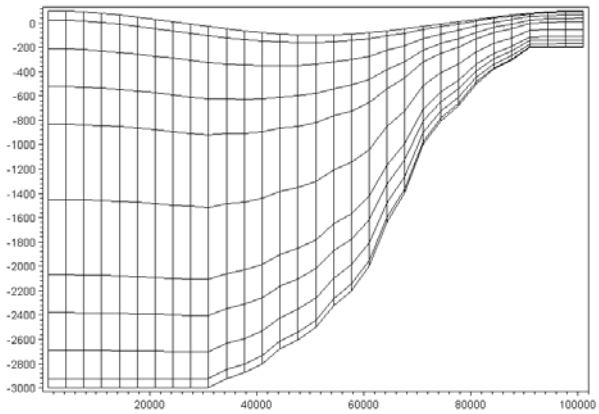

Figure 5.5 Example of vertical distribution using layer thickness distribution. Number of layers: 10, thickness of layers 1 to 10: 0.025,0.075,0.1,0.1,0.2,0.2,0.1,0.1,0.075,0.025

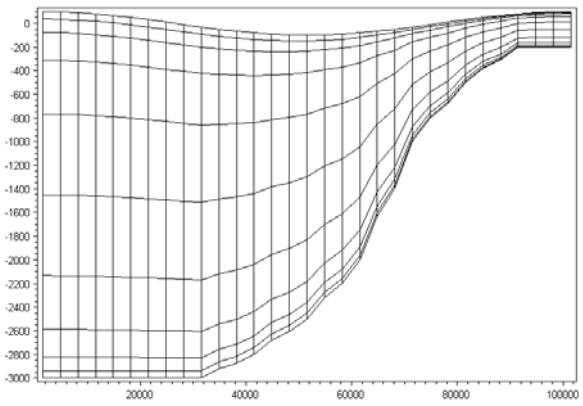

Figure 5.6 Example of vertical distribution using variable distribution. Number of layers: 10, sigma_c = 0.1, theta = 5, b = 1

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.

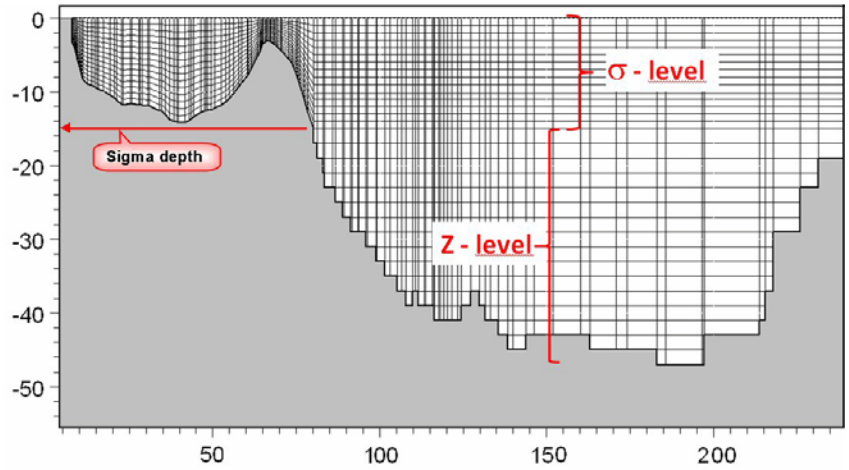

Figure 5.7 Example of vertical distribution using combined sigma/z-level

When flood and dry is included in the simulation, the flooding and drying (see p. 51) 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 than that sigma depth.

The specification of the mesh in the sigma domain is done as described in the previous section (see p. 43). 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 corresponds 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.

Note: 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 the mesh is specified using a mesh file generated by the MIKE Zero Mesh Generator you already have defined a code value for porosity zones and sponge layers.

Figure 5.8 shows the definition of codes in a simple application. In this case 2 boundary codes have been detected from the mesh file specified in the domain parameters; code 2 and code 3. Code 1 is interpreted as closed land boundaries.

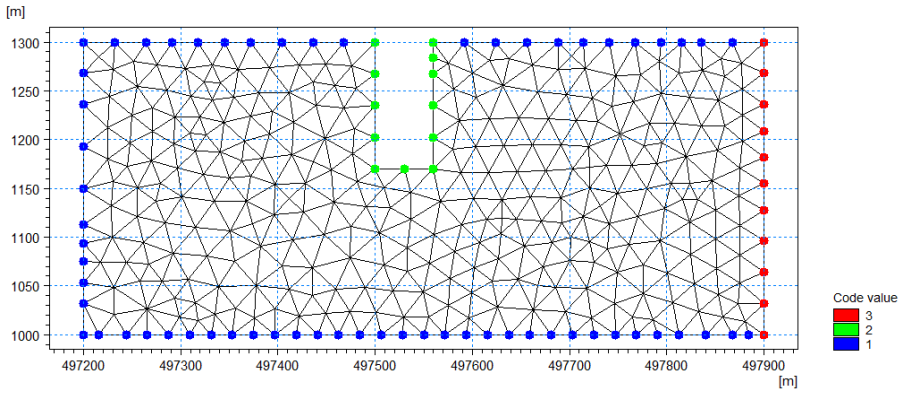

Figure 5.8 The definition of boundary codes in a mesh is made in the Mesh Generator

When the mesh is specified using a bathymetry data file generated by the MIKE Zero Bathymetry Editor the boundary code is defined as described on the Domain dialog.

In the main Boundary names dialog you can re-name the code values to more appropriate names, see Figure 5.9.

Boundary	Name
Code 2	Code 2
Code 3	Code 3

Boundary	Name
Code 2	Porosity
Code 3	Sponge layer

Figure 5.9 Change of default boundary code names 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.

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

The time steps for the hydrodynamic calculations are dynamic and determined to satisfy stability criteria. All time steps within the simulation are synchronized at the overall discrete time step.

6 HYDRODYNAMIC MODULE

The hydrodynamic module calculates the resulting flow subject to a variety of forcing and wave conditions. Turbulence is considered as subordinated to the HD module and are set up here.

6.1 Solution Technique

The time integration of the Navier-Stokes equations and the transport (advection-dispersion) equations is performed using an explicit scheme (second order Runga-Kutta). 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 Navier-Stokes equations and the transport equations determined so that the CFL number is less than a critical CFL number in all computational nodes.

The type of Riemann solver can be specified in two ways

- No Riemann solver
- HLLC

Selecting the Riemann solver you have to specify the Riemann factor.

The Navier-Stokes equations are discretized using a finite volume method with a shock-capturing scheme where the interface fluxes are calculated using an approximate Riemann solver. In MIKE 3 Wave Model FM the HLLC Riemann solver (Toro et al. (1994)) is applied. The shock-capturing scheme enables robust and stable simulation of flows involving shocks or discontinuities such as bores and hydraulic jumps. This is essential for modelling of waves in the breaking zone or porous structures. The numerical dissipation accounts for the dissipation in the breaking waves.

For fast varying flows with a coarse resolution the HLLC solver can be too diffusive. The HLLC solver includes four stages. For two stages simple upwinding is applied for calculating the interface fluxes. For the two additional stages the energy dissipation is a function of the difference between the left and right Riemann states (flow variables). To be able to control the numerical dissipation a Riemann factor is applied to the calculated difference of the Riemann states. The factor should be >= 0 and <= 1. The value 1 corresponds to the full HLLC solver.

MIKE 3 Wave Model FM is using a cell-centered finite volume approach. For simulations using triangular elements or distorted quadrilateral elements spurious noise can occur in the horizontal velocity field. To remove this noise it is recommended to apply the HLLC solver with a Riemann factor of 0.05 or higher.

6.1.1 CFL number

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

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

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

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

6.1.2 Remarks and hints

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. 48), the time integration will be performed with constant time step. In this case the time step interval should be selected so the CFL number is smaller than 1.

For both the time integration of the Navier-Stokes 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 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 piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data.

In case the 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.

6.2.1 General description

The corrected still water depth, $d_{\text{corrected}}$, is defined as

$$d_{corrected} = d_{initial} - \Delta z \tag{6.3}$$

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.

6.3 Flood and Dry

The flood and dry can be specified in two different ways

- No flood and dry
- Flood and dry

If your model is located in an area where flooding and drying occur, you must enable the flood and dry facility by selecting "Flood and dry". In this case you have to specify a drying 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 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.

6.3.1 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 when the water depth at one side of face is less than a tolerance depth, h_{dry} , and the water depth at the other side of the face larger than a tolerance depth, h_{wet} .
- 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.

A non-physical flow across the face will be introduced for a flooded face when the surface elevation in the wet element on one side of the face is lower than the bed level in the partially wet element on the other side. To overcome this problem the face will be treated as a closed face.

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. 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. 100 iterations).

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 flooded again.

6.3.2 Recommended values

The default values are: drying depth h_{dry} = 0.005m and wetting depth h_{wet} = 0.1m. The wetting depth, h_{wet} , must be larger than the drying depth, h_{dry}.

6.3.3 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.

6.4 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).

6.4.1 Horizontal eddy viscosity

The horizontal eddy viscosity can be specified in four different ways

- No eddy
- Constant eddy formulation
- Smagorinsky formulation
- k-ε formulation

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

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 specifi-

cation of the setup for the solution of the additional transport is described in Section 7 TURBULENCE MODULE.

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

Eddy parameters

You must specify a minimum and maximum value for the eddy viscosity.

6.4.2 Vertical Eddy Viscosity

The vertical eddy viscosity can be specified in three different ways

- No eddy
- Constant eddy 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 Section 7 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 piecewise constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data.

Eddy parameters

You must specify a minimum and maximum value for the eddy viscosity.

6.4.3 General description

Smagorinsky formulation

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 in the horizontal direction is given by

$$\nu_t^h = c_s^2 l^2 \sqrt{2 S_{ij} S_{ij}} \tag{6.4}$$

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

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \quad (i, j = 1,2) \tag{6.5}$$

For more details on this formulation, the reader is referred to Lilly (1966), Leonard (1974), Aupoix (1984), and Horiuti (1987).

k-ε formulation

The *k*-ε formulation can be used for both the horizontal eddy viscosity, ν_t^h, and the vertical eddy viscosity, ν_t^v. Here it is assumed that $\nu_t = \nu_t^h = \nu_t^v$.

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

$$\nu_t = c_\mu \frac{k^2}{\varepsilon} \tag{6.6}$$

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

6.4.4 Recommended values

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}=1.0·10^{-7} m^2/s^2 and ε_{min}=5·10^{-10} m^2/s^3, yielding =1.8·10^{-6} m^2/s.

6.4.5 Remarks and hints

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.

6.5 Bed Resistance

The bed resistance can be specified in three different ways

- No bed resistance
- Quadratic drag coefficient
- Roughness height

If including bed resistance you must specify the quadratic drag coefficient or the roughness height, respectively.

Data

The drag coefficient and the roughness height, k_s, 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 piecewise 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.

6.5.1 General description

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

$$\frac{\overline{\tau_b}}{\rho_0} = c_f \, \overline{u_b} \left| \overline{u_b} \right| \tag{6.7}$$

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 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 seabed

$$c_f = \frac{1}{\left(\frac{1}{\kappa} \ln \left(\frac{\Delta z_b}{z_0} \right) \right)^2} \tag{6.8}$$

where κ 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,

$$z_0 = m k_s \tag{6.9}$$

where m is approximately 1/30.

6.5.2 Recommended values

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.

6.5.3 Remarks and hints

Short periodic waves

Usually the effects of bottom friction are relatively unimportant in simulation of short waves. This is because the area covered by most short wave models is relatively small (typically less than a few square km's), and with the exception of high waves and/or very shallow water there is usually not a sufficient distance for the bed resistance to attain any significant effect on short wave propagation. In these applications the bed resistance can usually be excluded, without the need for detailed evaluations.

Long periodic waves

For modelling of long wave transformation (e.g. harbour resonance and seiching) the effect of bottom friction may be important.

Wave-induced currents

As opposed to the shore-normal case steady solutions for the wave-induced mean flow can only exists when the forcing by radiation stress is balanced by bottom friction and mixing processes. Therefore you need to include bottom friction in simulations of wave-induced flow fields.

6.6 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 geographical information given in the mesh file.

6.6.1 Remarks and hints

Usually the effect of Coriolis force is not important in simulation of short waves and can be excluded. This is because the area covered by most short wave models is relatively small.

6.7 Porosity

Porosity can be used to model transmission of wave energy at porous structures such as rubble mound breakwaters. Porosity can also be used to partial reflection.

The porosity can be specified in five different ways

- No porosity
- Porosity zones (boundary information)
- Porosity zones (2D map)
- Porosity zones (3D map)
- Porosity (2D map)

When "Porosity zones (boundary information)" is selected, the porosity zones are determined based on the boundary information in the mesh file. For changing the boundary information for a mesh file see “Editing mesh boundary code values”, page 60. In the list view you can add a new zone and remove a zone. For each zone you can specify whether the zone should be active or not, the boundary name and the width of the zone. The options for the boundary name are the boundary names specified on the domain page. The following porous flow parameters also have to be specified for each zone:

- Porosity, n
- Grain diameter, d_{50}
- Laminar resistance coefficient, α
- Turbulent resistance coefficient, β
- Oscillating period, T

For more details on these parameters see the General description.

When "Porosity zones (2D map)" or "Porosity zones (3D map)" is selected, the user has to supply a map identifying the location of the different porosity zones. Each zone is identified by a integer number larger than zero. The value at open water should be set to zero. If a 2D map is specified the porosity is assumed to cover the whole water column.

When "Porosity (2D map) is selected the user has to supply a 2D map containing the porosity. The porosity is assumed to cover the whole water column. The porosity should only be set to less than unity in areas where you want to include the dissipation effect of porous flow. The porosity values at open water elements should be set to unity (i.e. porosity = 1.0). Here the number of zones is assumed to be 1 and the specified additional parameters listed above (excluding the porosity) are applied in all areas where the porosity is below 1.0.

Note: When "Porosity zones (boundary information) is selected, it is possible to create an area or volume output file containing a map with porosity zone information. If no zone is selected, the map will contain the value zero in the whole domain. These maps can be edited using the Data Manager and used

as input to the model by selecting "Porosity zones (2D map)" or "Porosity zones (3D map)".

Data

When "Porosity zones (2D map)" or "Porosity zones (3D map)" are selected you have to prepare a data file containing the zone map before you set up the wave simulation. If the zones are specified using a 2D map, the file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). If a dfsu-file is used piecewise constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data. If the zones are specified using a 3D map, the file must be a 3D unstructured data file (dfsu). The area in the data file must cover the model area.

When "Porosity (2D map)" is selected you have to prepare a data file containing the porosity map before you set up the wave simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). If a dfsu-file is used piecewise constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data. The area in the data file must cover the model area.

Editing mesh boundary code values

The boundary code values of an existing mesh file can be edited using the following work flow.

Open the mesh file using the MIKE Zero Data Viewer and select the "Code value" item. Using the "Position of node selection polygon" tool from the toolbar, you can select a set of node points for which you would like to edit the boundary code.

Notice, for the "Code value" item the "Position of node selection polygon" tool will only list the boundary nodes in your selection, since it is not possible to change the code value for an interior node point, only for boundary node points.

6.7.1 General description

For wave simulations, the equations in MIKE 3 Wave Model FM have been modified to include porosity and the effects of non-Darcy flow through porous media.

The main effects of porosity are introduced by additional laminar and turbulent friction terms for describing losses due to flow through a porous structure. In most practical cases the pore sizes are relatively large (typically 0.1 m to 1.0 m), and losses due to turbulent friction will dominate. The laminar friction term has been included to allow the simulation of small scale physical model tests.

The flow resistance inside the porous structure is described by the linear and non-linear resistance forces expressed as

$$F_i = \rho a u_i + \rho b |u| u_i \tag{6.10}$$

where a and b are resistance coefficients accounting for the laminar and turbulent friction loss, respectively, u_i is the velocity components and the magnitude of the flow velocity is defined by $|u| = \Sigma_i u_i$. a and b are determined by the empirical expressions formulated by van Gent (1995) and Liu et al. (1999):

$$a = \alpha \frac{(1-n)^2}{n^3} \frac{\upsilon}{(d_{50})^2} \tag{6.11}$$

$$b = \beta \left(1 + \frac{7{,}5}{KC}\right) \frac{(1-n)}{n^2} \frac{1}{d_{50}} \tag{6.12}$$

Where n is the porosity, α and β is user specified coefficients, υ is the kinematic viscosity and d_{50} is the grain diameter of the porous materials. KC is the Keulegan-Carpenter number defined as

$$KC = \frac{u_m T}{n d_{50}} \tag{6.13}$$

where u_m is the maximum oscillating velocity and T is the period of the oscillation. u_m is approximated by the magnitude of the flow velocity. T is often approximated by the characteristic wave period for the simulation.

In the momentum equations the time derivative terms is multiplied by a factor $(1+C_m)$ where C_m is the added mass coefficient to take transient interaction between grains and water into account. van Gent (1995) gave C_m as

$$C_m = \gamma \cdot \frac{1-n}{n} \tag{6.14}$$

where γ is an empirical coefficient, which takes the value 0.34.

6.8 Sponge Layer

Sponge (or absorbing) layers can be used as efficient numerical wave absorbers in wave simulations. These could be set up along model boundaries to provide radiation boundary conditions, which absorb wave energy propagating out of the model area.

The sponge layer can be specified in three different ways

- No sponge layer
- Sponge zones (boundary information)
- Sponge coefficient (2D map)

When "Sponge zones (boundary information)" is selected, the sponge zones are determined based on the boundary information in the mesh file. For changing the boundary information for a mesh file see “Editing mesh boundary code values”, page 63. In the list view you can add a new zone and remove a zone. For each zone you can specify whether the zone should be active or not, the boundary name, the width of the zone, a characteristic spacing and the reference level. The options for the boundary name are the boundary names specified on the domain page. The sponge coefficient is then calculated based on the approach described in General description. You can chose if the base and power value should be calculated as function of the width and the spacing or should be user-defined. In the first case the base and power value will be calculated using linear interpolation based on the values in Table 6.1. In the second case you have to specify the base and power value for each zone.

When "Sponge coefficient" is selected the user has to supply a 2D map containing sponge layer coefficients. You can also specify the reference level.

General guidelines for the preparation of this map are as follows:

- To minimise reflections, the values of the sponge layer coefficients should be close to unity along the front edge of the sponge layer, and should increase smoothly towards the closed/land boundary.
- Very good absorbing characteristics are obtained for a sponge layer width of one to two times the wave length corresponding to the most energetic waves.
- For typical short-wave studies (resolution of 3-10 m) the width of the sponge layers should be at least 20 times the characteristic size of the elements in the sponge layer area.
- For studies involving longer waves (e.g. harbour resonance studies) the width of the sponge layers may need to be 50 or even more times the characteristic size of the elements in the sponge layer area.
- In most wave transformation applications (say, resolution < 1 m) the width of the sponge layers is usually within the range 50-200 times the characteristic size of the elements in the sponge layer area.
- The sponge layers should normally be backed by a closed/land boundary.
- The sponge layer coefficients at open water elements should always be set to unity (i.e. sponge coefficient = 1.0).

Given the guidelines above, you are free to select your own values for the sponge layer (damping) coefficients.

Data

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

Editing mesh boundary code values

The boundary code values of an existing mesh file can be edited using the following work flow.

Open the mesh file using the MIKE Zero Data Viewer and select the "Code value" item. Using the "Position of node selection polygon" tool from the toolbar, you can select a set of node points for which you would like to edit the boundary code.

Notice, for the "Code value" item the "Position of node selection polygon" tool will only list the boundary nodes in your selection, since it is not possible to change the code value for an interior node point, only for boundary node points.

6.8.1 General description

The implemented method is based on the sponge layer technique introduced by Larsen and Dancy (1983). In the sponge layer the surface elevation, s, and the velocities u, v and w, is corrected at every time step as

$$\begin{aligned} s &= \frac{s - s^{ref}}{c} + s^{ref} \\ u &= \frac{u}{c} \\ v &= \frac{v}{c} \\ w &= \frac{w}{c} \end{aligned} \tag{6.15}$$

where c is the sponge coefficient and s^{ref} is the reference level.

When selecting the sponge layer coefficients, c, the following formula has been found to work well:

$$c = a^k \quad \text{where} \quad k = r^{s/\Delta s}, \quad 0 \le s \le w \tag{6.16}$$

where w is the width of the sponge layer and a and r are assigned constant values. s is the distance from the closed boundary and Δs is the characteristic size of the elements in the sponge layer area. Depending on ratio $w/\Delta s$ you may use the values listed in Table 6.1 below.

Table 6.1 Recommended values for sponge layer coefficients

w/Δs	a	r
10	5	0.5
20	7	0.7
50	10	0.85
100	10	0.92
200	10	0.95

6.9 Internal Wave Generation

The relaxation zone technique is applied for wave generation and absorption. The location in the horizontal domain of the generation line should be given by a number of geo-referenced points which together make up a polyline. The relaxation zone is defined as the area to the right of the polyline when positioned at the starting point and looking forward along the line. In addition the width of the ramp-up zone has to be specified. The generation line is normally placed near a closed/land boundary, but the distance to the boundary must be larger than the specified width of the relaxation zone.

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

There are two different methods for specification of the relaxation zone.

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

In the Geographic View it is also possible to create a new zone by double-clicking inside the domain or by selecting "New zone" from the context menu. Click once to add a point on the polyline and twice to add the end point of the polyline. You may edit the individual polyline using the Relaxation data tab. The additional information for the zone is entered subsequently.

6.9.1 Relaxation data

The coordinate grid allows you to specify or edit the location of the individual points defining the polyline. Using the "Add" and "Remove" buttons it is possible to change the number of geographical coordinates.

It is also possible to import the geographical coordinates 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.

You must also select the map projection (Longitude/Latitude, UTM, etc.) in which you want to specify the geographical coordinates defining the polyline.

The relaxation zone is defined as the area to the right of the polyline when positioned at the starting point and looking forward along the line. The width parameter specifies the width of the ramp-up zone, see Figure 6.1.

By defining a soft start interval, during which the amplitude of the generated wave is continuously increased until reaching the specified amplitude, it is possible to avoid shock waves being generated in the model. The increase can either be linear or follow a sinusoidal curve.

Width of ramp-up zone

To be able to absorb the wave propagation out of the domain, the width of the ramp-up zone must be approximately the wave length of the most energetic waves.

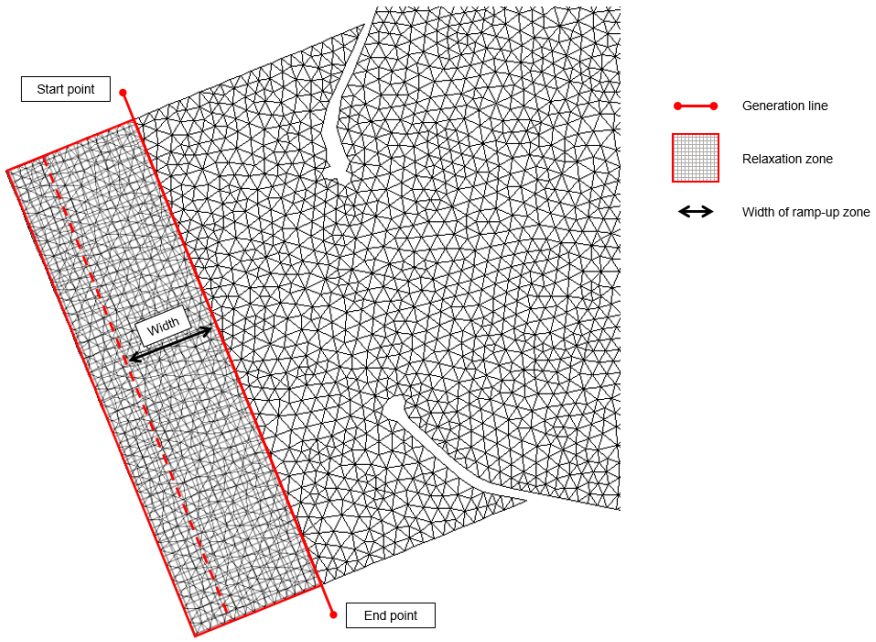

Figure 6.1 The relaxation zone is the area to the right of the generation line when looking forward along the line from the starting point. The width of the ramp-up zone is specified by the width parameter

6.9.2 Wave data

For the calculation of the wave conditions in the relaxation zone you should specify a representative value for the water depth in the zone and the reference level. The main surface elevation for the generated waves will be the specified reference level, and the effective water depth used in the calculation is the sum of specified water depth and reference level.

The wave conditions can be specified in four ways

- Regular wave parameters
- Irregular wave parameters
- Wave spectrum from file
- Wave components from file

Regular wave parameters

You have to select a theory from among the three different theories available

- Stokes 1st order
- Stokes 5th order
- Stream function

For the Stokes theory, you have to specify the wave conditions trough

- Height, H
- Period, T
- Direction, θ

where θ is the meteorological direction of the wave as specified in Convention of angles (page 70).

For the Stream function theory you have to specify if the wave condition should be defined by the wave period or wave length and through

- Height, H
- Period, T
- Length, L
- Direction, θ
- Number of components

You need to specify if the meteorological or scientific convention is used for defining the wave direction (see Convention of angles (page 70)).
The number of components refers to the number of Fourier components used in the wave generation (see Fenton (1988)).

Irregular wave parameters

For this type of waves you have to specify both a frequency spectrum and a directional distribution.

You can choose among the following frequency spectra, where you have to specify the listed parameters

- Pierson-Moskowitz
  - Significant wave height, H_{mo}
  - Peak wave period, T_p
- JONSWAP
  - Significant wave height, H_{mo}
  - Peak wave period, T_p
  - Non-dimensional peak shape parameter, gamma
  - Spectral width parameter, alpha
  - Spectral width parameter, beta
- TMA
  - Significant wave height, H_{mo}
  - Peak wave period, T_p
  - Non-dimensional peak shape parameter, gamma
  - Spectral width parameter, alpha
  - Spectral width parameter, beta

For specifying the directional distribution you have the following options

- Unidirectional
- Cosine
  A $\cos^n(\theta-\theta_{main})$ distribution where θ_{main} is the main wave direction and n is the directional spreading index. This distribution is specified through:
  - Main wave direction, θ_{main}
  - Maximum deviation from main wave direction, $\Delta\theta_{max}$
  - Spreading index, n
- Normal
  The normal (i.e. Gaussuian) distribution is specified through:
  - Main wave direction, θ_{main}
  - Maximum deviation from main direction, $\Delta\theta_{max}$
  - Directional variance, s^2
- Uniform
  The distribution is specified through:
  - Main wave direction, θ_{main}
  - Maximum deviation from main direction $\Delta\theta_{max}$

  In this case, in principle, the waves do not have a main wave direction. However, θ_{main} has to be given as the maximum deviation specified relative to it.
- Frequency dependent
  A $\cos^{2s}[\frac{1}{2}(\theta-\theta_{main})]$ distribution where s is a frequency dependent shape parameter. This distribution is specified through:
  - Main wave direction, θ_{main}
  - Maximum deviation from main direction, $\Delta\theta_{max}$

Furthermore, you have to specify the minimum and maximum cut-off frequency for the frequency spectrum, and the number of Fourier components. The specification for the frequency discretization determines the repetition period (see page 70) for the generated wave input.

You also have to specify the initial random number (seed) used in the wave generation calculations. The seed is used for the calculation of the random phases in the Fourier description, and for the calculation of the random wave directions. Thus, you can reproduce the random computations by specifying the same seed.

Finally, you have the option to rescale the truncated spectrum as described in Rescale truncated spectrum (page 71).

You need to specify if the meteorological or scientific convention is used for defining the wave direction (see Convention of angles (page 70)).

Wave spectrum from file

This way of specifying wave data is used when you have a data file containing the energy density spectrum. The data file should cover the entire simulation period. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2).

Furthermore, you have to specify the minimum and maximum cut-off frequency for the frequency spectrum, the main wave direction, θ_{main}, the maximum deviation from main direction, $\Delta\theta_{max}$, for the directional distribution and the number of Fourier components. The specification for the frequency discretization determines the repetition period (see page 70) for the generated wave input.

You also have to specify the initial random number (seed) used in the wave generation calculations. The seed is used for the calculation of the random phases in the Fourier description, and for the calculation of the random wave directions. Thus, you can reproduce the random computations by specifying the same seed.

Finally, you have the option to rescale the truncated spectrum as described in Rescale truncated spectrum (page 71).

You need to specify if the meteorological convention or scientific convention is used for defining the wave direction (see Convention of angles (page 70)).

Wave components from file

The wave components data should be provided using an ASCII file containing one line for each spectral component. Each line should contain either 4 or 5 items holding either the first 4 or all 5 of the following parameters

- Frequency (1/s)
- Direction (degree)
- Amplitude (m)
- Phase (degree)
- Wave number

The items must be separated by either space, comma or tab.

You also have to specify the minimum and maximum cut-off frequency for the frequency spectrum, and the main wave direction, θ_{main}, and the maximum deviation from main direction, $\Delta\theta_{max}$, for the directional distribution.

You need to specify if the meteorological or scientific convention is used for defining the directions in the ASCII file (see Convention of angles (page 70)).

You have the option to rescale the truncated spectrum as described in Rescale truncated spectrum. The gain factor is a scaling factor used for scaling the amplitude of the specified wave.

Using the offset parameters, it is possible to make both a temporal and spatial correction of the input data.

Convention of angles

The direction convention for the waves can be one of the following angle conventions

- Meteorological
- Scientific

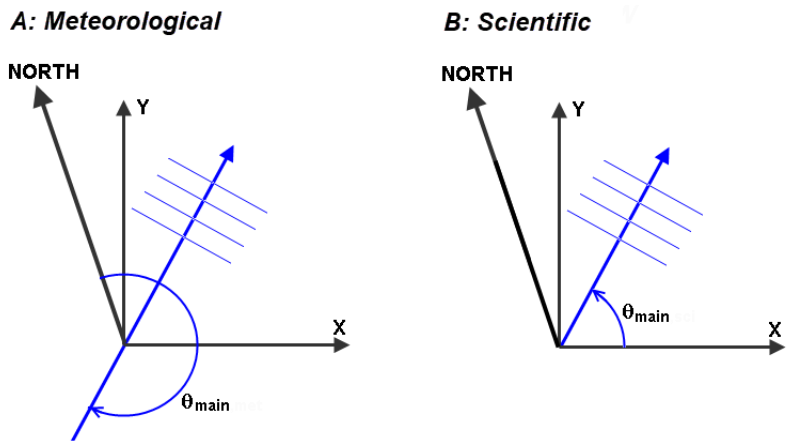

Figure 6.2 Definition of the two types of wave angle conventions

The Meteorological convention relates the wave direction to North and express the wave direction as 'coming from' in a clockwise direction relative to North. Waves travelling from North to South have therefore a direction of 0 degrees. This is shown to the left in Figure 6.2.

The Scientific convention relates the wave direction to the general (or global) x-axis of your mesh file and express the direction the waves are going towards. This is shown to the right in Figure 6.2.

The default angle convention is the meteorological convention.

Repetition period

The repetition period is determined as $1/\Delta f$, where Δf is the frequency interval of the discrete frequency spectrum. The frequency interval is calculated as

$$\Delta f = \frac{f_{max} - f_{min}}{n - 1} \tag{6.17}$$

Here f_{max} is the specified maximum frequency, f_{min} is the minimum frequency and n is the number of components.

A good estimate is that the repetition period should be the simulation period minus the time it takes the incoming wave to reach the area of interest.

Rescale truncated spectrum

As the low frequency part ($f < f_{min}$) and the high frequency part ($f > f_{max}$) of the original spectrum is removed, the resulting integral parameters such as the significant wave height and mean wave periods are altered relative to the input specifications.

If you would like to maintain the significant wave height as specified, then you can choose to re-scale the spectrum. Thus, the total energy in the truncated spectrum is the same as in the original specified spectrum, see Figure 6.3.

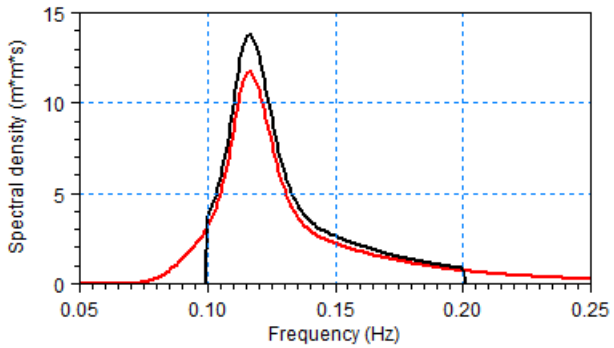

Figure 6.3 Example of the original and rescaled, truncated spectrum. The spectrum is a standard JONSWAP spectrum (i.e. having the shape parameters s_a= 0.07, s_b= 0.09, and g= 3.3). H_{m0}= 2.65 m and T_p= 8.6 s. The minimum and maximum cut-off frequency is 0.1 Hz and 0.2 Hz, respectively.

6.9.3 Interpolation overlay

To reduce the computational time it is possible to perform the internal wave generation calculations using a coarser resolution both in space and/or time, and then interpolate the result of these coarser calculations onto the actual resolution.

For the temporal interpolation, you have to specify the coarse time interval and the interpolation type. The temporal interpolation type could be either a linear or a cubic spline approach.

When using spatial interpolation, a structured quadrangular grid overlay is applied in the relaxation zone. The structured overlay should have a coarser

resolution than the resolution in the relaxation zone, see Figure 6.4. Instead of performing the internal wave generation calculations for every element in the relaxation zone, the calculations will only be performed for the smallest number of elements in the structured overlay that covers the relaxation zone. This is indicated by the blue dots in Figure 6.4.

The resulting values will be interpolated onto the elements in the relaxation zone.

For the spatial interpolation, you have to specify an orientation of the structured grid overlay. This orientation is using the scientific convention as described in Convention of angles. Furthermore, you have to specify the resolution of the structured grid by giving the grid spacings Δx and Δy.

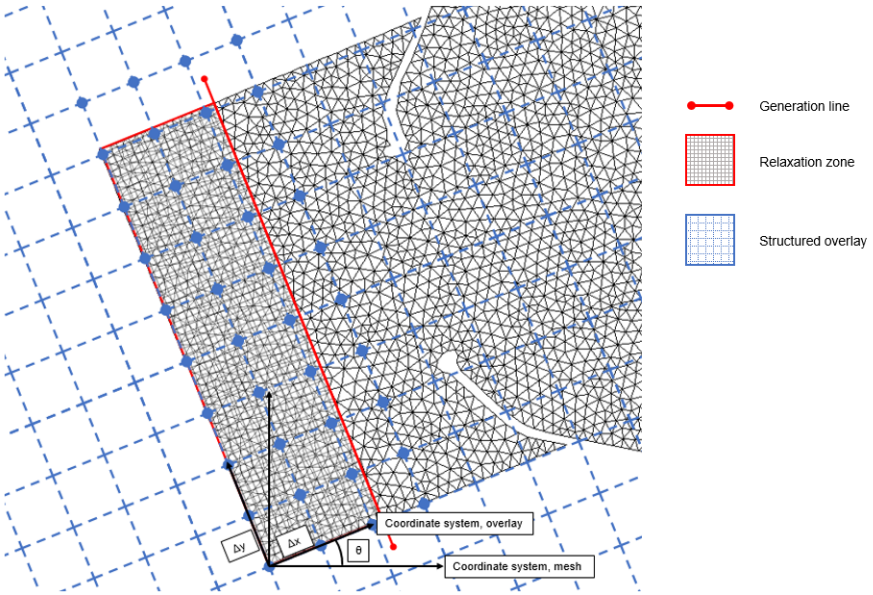

Figure 6.4 The coarse structured overlay is specified using an orientation relative to the mesh coordinate system and grid spacings in the x- and y-direction. The grid spacings are defined in the rotated coordinate system for the structured overlay.

6.9.4 General description

Relaxation approach

The target and the computed solution are weighted in the relaxation zone

$$\theta = \alpha\theta_{computed} + (1-\alpha)\theta_{target} \tag{6.18}$$

where θ is a velocity component (u, v or w) and the ramp up factor, α, can be calculated as

$$\begin{aligned} \alpha &= 1 - \frac{\exp(s^f) - 1}{\exp(1) - 1} \qquad 0 \le s \le 1 \\ \alpha &= 0 \qquad\qquad\qquad\qquad s > 1 \end{aligned} \tag{6.19}$$

Here s is the distance from the generation line divided by the width of the ramp up zone and f is the ramp up factor. Usually, f=3.5 is applied. For the pressure, q, the ramp up factor is given by

$$\begin{aligned} \alpha &= 1 \qquad 0 \le s \le 1 \\ \alpha &= 0 \qquad s > 1 \end{aligned} \tag{6.20}$$

The target value for the non-hydrostatic pressure is applied as Dirichlet condition for s > 1 solving the Poisson pressure equation.

Wave theory

The theoretical background for the different types of waves can be found in numerous textbooks and papers on wave theories. A number of suggestions for further reading are: Fenton (1988), Mei (1983) and Svendsen et al (1980).

6.10 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.

The water level in the simulation can be changed by specifying a constant surface elevation and by setting the reference level for the sponge layer and internal wave generation to the same constant value.

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

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

One containing the total water depth and one containing the velocity components in the x- and y-directions and the vertical velocity 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 piecewise 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.

6.11 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 p.47).

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.

6.11.1 Boundary specification

For hydrodynamic boundaries there are two types:

- Land (zero normal velocity)
- Land (zero velocity)

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.

6.12 Turbulence Module

The turbulence module is invoked from the specification of horizontal or vertical eddy viscosity, provided k-ε model is selected (see Section 6.4 Eddy Viscosity).

A more detailed description of the Turbulence module can be found in section 7 TURBULENCE MODULE.

6.13 Outputs

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

In the main Outputs dialog you can add a new output file by clicking on the "New output" button. By selecting a file in the Output list and clicking on the "Delete output" 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.

6.13.1 Geographical view

This dialog shows the geographical position of the output data.

6.13.2 Output specification

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

Field type

For a 3D wave simulation the following field types can be selected:

- 2D flow parameters
- 3D flow parameters
- Mass budget
- Discharge through a cross section
- Cumulative statistics 2D
- Subseries statistics 2D
- Cumulative statistics 3D
- Subseries statistics 3D

Output format

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

For 2D flow parameters, cumulative statistics 2D and subseries statistics 2D 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 flow parameters, cumulative statistics 3D and subseries statistics 3D the following formats can be selected:

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

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

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

Data type

You must specify the data type. You can select discrete values or interpolated values. For point series, area series, vertical plane series and volume

series both options are available. For line series only interpolated values is available.

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

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

For cumulative statistics 3D and subseries statistics 3D the data type is always “interpolated values”.

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

Output file

A name and location of the output file must be specified along with the file type. The file type depends on the Output format and the Data type as shown in Table 6.2.

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

Table 6.2 List of file types for the output files

Output format	Data type	File type
Point series	Discrete values	dfs0
	Interpolated values	dfs0
Line series	Interpolated values	dfs1
Area series	Discrete values	dfsu, dfs2*
	Interpolated values	dfs2
Vertical plane series	Discrete values	dfsu
	Interpolated values	dfs2
Volume series	Discrete values	dfsu
	Interpolated values	dfs3

Table 6.2 List of file types for the output files

Output format	Data type	File type
Cross-section	Not relevant	dfs0
Domain series	Not relevant	dfs0

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

Table 6.3 List of tools for viewing, editing and plotting results

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

Flood and dry

For 2D and 3D flow 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.

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

If you have selected "Cumulative statistics 2D" or "Cumulative statistics 3D" your specified output items will be updated regularly corresponding to the specified update interval. In case of "Subseries statistics 2D" or "Subseries statistics 3D" your output items will be set to zero at the update interval. In most cases you would select "Cumulative wave statistics". The update interval should cover a reasonable time period (at least one wave period or more) in order to provide meaningful statistics.

Point series

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

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

Layer number

The layer number selected for discrete values in the point output is defined from the lowest active layer (=1) increasing upwards. In case the mesh is a type sigma mesh the number of active layers in the water column will always be the same in any point in the domain. In case the mesh is a combined sigma-z level mesh the number of active layers may vary in the domain. An example is shown in Figure 6.5.

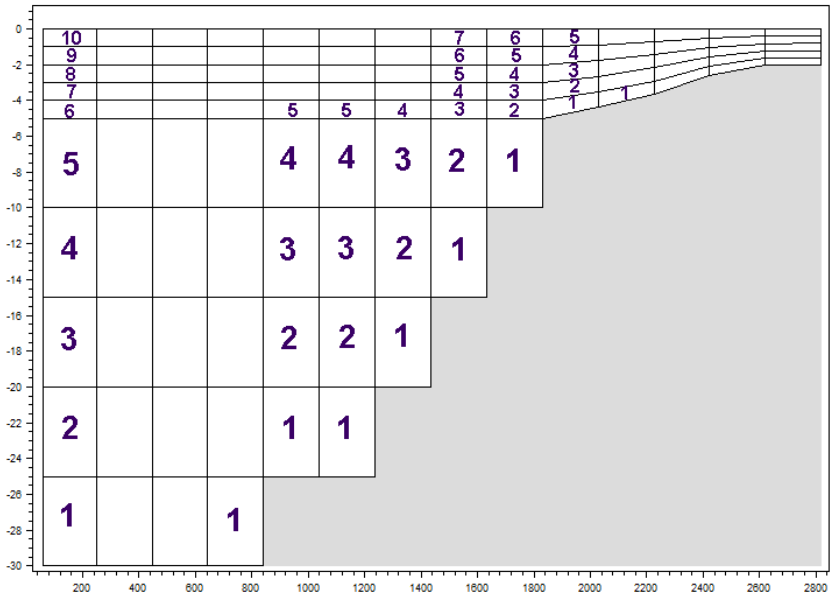

Figure 6.5 Example of layer numbers in point output specification in case of combined sigma-z level mesh.

Line series

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

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

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

Area series

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

If discrete values is selected for the data type the discrete field data within a polygon can be selected. The closed region is bounded by a number of line segments. You must specify the coordinates of the vertex points of the poly-

gon. Two successive points are the endpoints of a line that is a side of the polygon. The first and final point is joined by a line segment that closes the polygon. The geographical coordinates of the polygon points are taken from the dialog or from a file. The file format is an ascii file with three space separated items for each of the two points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. The third item is unused (but must be specified).

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

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

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

Vertical plane series

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

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

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

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

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

Layer number

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

Volume series

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

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

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

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

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

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

Layer number

The layer number(s) selected for the volume output refer to the vertical discretization specified on the Vertical mesh page. Here the layers are numbered 1 at the lowest layer and increase upwards. In case of a combined sigma-z level mesh only the elements containing water are saved in the output. An example is shown in Figure 6.6.

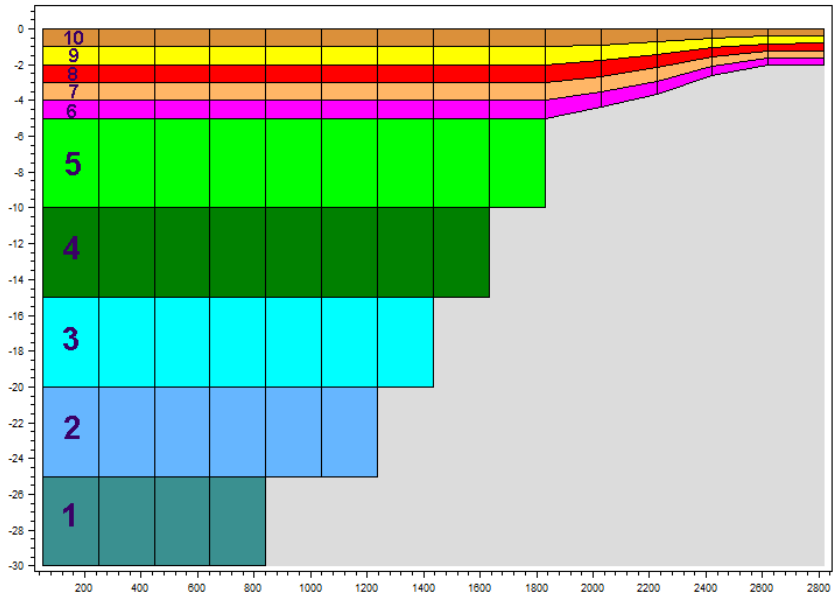

Figure 6.6 Example of layer numbers in volume output specification in case of combined sigma-z level mesh.

Cross section series

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

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

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.

6.13.3 Output items

2D and 3D flow parameters

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

The following items are included in the output file:

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

- Error - accumulated volume 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 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.

Discharge

You can select between two types of output items:

- Basic
- Extended

The basic output items is as follows:

- Discharge - volume flux through the cross section
- Acc. discharge - accumulated volume 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. If porosity is included in the simulation you can specify if the whole cross section should be used or only the cross section excluding porosity.

Cumulative wave statistics and subseries wave statistics

The following items may be selected for the output file of 2D statistics:

- Significant wave height, H_{m0}
- Wave disturbance coefficient, $H_{m0}/H_{m0,incoming}$
- Maximum wave height, H_{max}
- Maximum surface elevation, s_{max}
- Minimum surface elevation, s_{min}
- Mean surface elevation, s_{mean}
- Mean velocity in x-direction, u_{mean}

- Mean velocity in y-direction, v_{mean}
- Skewness, S
- Kurtosis, K
- Atiltness, A
- Time of arrival of the first wave that is used in the calculation, $t_{arrival}$

If wave disturbance coefficient is selected, you must specify whether the scaling of the coefficients should be relative to a user defined incoming wave height, or relative to the wave height in a specific reference point. In the first case you must provide the incoming significant wave height, $H_{m0,incoming}$, and in the second case the coordinates for the reference point.

The following items may be selected for the output file of 3D statistics:

- Mean velocity in x-direction, u_{mean}
- Mean velocity in y-direction, v_{mean}
- Mean velocity in z-direction, w_{mean}
- Time of arrival of first wave that is used in the calculation, $t_{arrival}$

The "Start at wave No" is the number of the first where the computation is initiated. The first wave is counted from when the water level crosses (uncrossing) the still water level first time. Most often you should keep the default value.

If flood and dry is included or 3D statistics is selected you should also specify whether the phase-averaging should include partly wet domain or not. If partly wet domain is included the averaging is performed for the duration where the domain is wet.

Significant wave height

The significant wave height Hm0 is defined by

$$H_{m0} = 4\sigma \tag{6.21}$$

where σ is the standard deviation of the surface elevation. Usually you will include this output item in your model specifications.

Wave disturbance coefficient

The wave disturbance coefficient is defined as the significant wave height divided by the user defined incoming wave height

Maximum wave height

The maximum wave height H_{max} is defined as the maximum surface elevation minus the minimum surface elevation within a specified time period.

Maximum surface elevation

The maximum surface elevation η_{max} is defined as the maximum value of the surface elevation within a specified time period.

Minimum surface elevation

The minimum surface elevation η_{min} is defined as the minimum value of the surface elevation within a specified time period.

Mean surface elevation

The mean surface elevation η_{mean} is defined as the mean value of the surface elevation taken over a specified time period. Usually you will include this output item in your model specifications when wave breaking and moving shoreline is included. The wave set-down and wave setup can be calculated from this item.

Mean velocity in x-direction

The mean velocity u_{mean} is defined as time mean of the depth-averaged velocity in x-direction. You should include this parameter in your output when studying wave-induced circulation.

Mean velocity in y-direction

The mean velocity v_{mean} is defined as time mean of the depth-averaged velocity in y-direction. You should include this parameter in your output when studying wave-induced circulation.

Skewness

The skewness S is defined by

$$S = \frac{E[(\eta - \bar{\eta})^3]}{(E[(\eta - \bar{\eta})^2])^{3/2}} \tag{6.22}$$

where E denotes the mean operator. Thus, the skewness may be examined as the mean cube of the time series of the surface elevation normalised by the mean square of the time series of surface elevation to the power 3/2. The skewness is a higher-order integral measure of the lack of horizontal wave profile symmetry. For further information on this output type please see Kofoed-Hansen and Rasmussen (1998) p. 213ff. Usually you will not include this item in your model output specifications.

Kurtosis

The excess kurtosis K is defined by

$$K = \frac{E[(\eta - \bar{\eta})^4]}{(E[(\eta - \bar{\eta})^2])^2} - 3 \tag{6.23}$$

where E denotes the mean operator. The kurtosis is a higher-order measure to identify non-Gaussian characteristics of a wave field. For a Gaussian sea

state (i.e. linear and random waves) the kurtosis is 0. Usually you will not include this quantity in your model output specifications.

Atiltness

The atiltness A is defined by

$$A = \frac{E\left[\left(\frac{\partial \eta}{\partial t} - \overline{\frac{\partial \eta}{\partial t}}\right)^3\right]}{\left(E\left[\left(\frac{\partial \eta}{\partial t} - \overline{\frac{\partial \eta}{\partial t}}\right)^2\right]\right)^{3/2}} \tag{6.24}$$

where E denotes the mean operator. The atiltness is a higher-order integral measure of the lack of vertical wave profile symmetry. The parameter takes a positive value when the wave profile are tilted forward (like in the shoaling zone) and takes a negative value when tilted backward. For further information on this output type please see Goda and Morinobu (1998) p. 314ff. Usually you will not include this quantity in your model output specifications.

7 TURBULENCE MODULE

The turbulence module is invoked from the specification of horizontal or vertical eddy viscosity, provided a k-ε model is selected (see section 6.4, Eddy Viscosity).

The turbulence model is based on a standard k-ε model, with buoyancy extension, see e.g. Rodi (1980). This model uses transport equations for the turbulent kinetic energy (TKE), k, and the dissipation of TKE, ε, to describe the turbulence.

7.1 Equation

The turbulence model is based on a standard k-ε model, with buoyancy extension, see e.g. Rodi (1980). This model uses transport equations for the turbulent kinetic energy (TKE), k, and the dissipation of TKE, ε, to describe the turbulence.

In the k-ε closure model, the Prandtl number, which appears in the transport equations for k and ε, can be modified explicitly following the empirical expression of Munk and Anderson (1948). This correction will introduce a damping of the buoyancy production in case of stable stratification. This damping is included by default.

Empirical constants

Four empirical constants can be specified for the standard k-ε model: c1e, c2e, c3e are related to the equation for the dissipation of TKE and the Prandtl number (σ_T) is used in the buoyancy production term. Cmy (c_μ) is an empirical constant used in the expression for determination of the eddy viscosity (see section 6.4, Eddy Viscosity).

Several carefully calibrated empirical coefficients enter the k-ε turbulence model. Therefore great care should be taken if you decide to alter any of these coefficients. The empirical constants are listed in Table 7.1.

Table 7.1 Default empirical constants in the k-ε turbulence model (see Rodi, 1980)

c1e	c2e	c3e	Prandtl	cmy
1.44	1.92	0	0.9	0.09

Maximum and minimum values

Here you specify maximum and minimum values for the TKE, k, and the dissipation of TKE, ε.

7.2 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.

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

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

7.2.1 Remarks and hints

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

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

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

time step. In this case the time step interval should be selected so the the CFL number is smaller than 1.

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

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

7.3 Dispersion

The diffusion coefficient in the turbulence module is estimated based on the eddy viscosity used in solution of the flow equations divided by at scaling factor sigma. For specification of the eddy viscosity (see section 6.4, Eddy Viscosity). The diffusion coefficient is calculated as the eddy viscosity coefficient divided by a Prandtl number.

You can specify the horizontal and vertical Prandtl number used in the transport equations for the turbulent kinetic energy, k, and the dissipation of TKE, ε, repectively.

7.4 Initial Conditions

The initial conditions are the spatial distribution of the turbulent kinetic energy, *k*, and the dissipation of TKE, ε, throughout the computational domain at the beginning of the simulation. Initial conditions must always be provided. The initial conditions can be the result from a previous simulation in which case the initial conditions effectively act as a hot start of the field for the turbulent kinetic energy and the dissipation of TKE.

The initial conditions are specified individually for the turbulent kinetic energy, k, and the dissipation of TKE, ε.

Data

The initial condition for the turbulent kinetic energy, *k*, and the dissipation of TKE, ε , can be specified in two ways

- Constant (in domain)
- Varying in domain

For the case with varying in domain you have to prepare a data file containing the initial condition (turbulent kinetic energy or the dissipation of TKE) before you set up the hydrodynamic simulation. The file must be a 3D unstructured data file (dfsu) or a 3D grid data file (dfs3).

The area in the data file must cover the model area. If a dfsu-file is used, the mesh in the data file must match exactly the mesh in the simulation. If a dfs3-file is used, bilinear interpolation is used to map the data. In case the input data file contains a single time step, this field is used. In case the file contains several time steps, e.g. from the results of a previous simulation, the actual starting time of the simulation is used to interpolate the field in time. Therefore the starting time must be between the start and end time of the file.

7.4.1 Recommended values

Generally, it is very difficult to estimate realistic initial conditions for the turbulent kinetic energy and the dissipation of TKE. Therefore, it is recommended to apply the minimum values. k_{min} =1.0·10^{-7} m^2/s^2 and ε_{min} =5·10^{-10} m^2/s^3, yielding ν_T =1.8·10^{-6} m^2/s (see section 6.4, Eddy Viscosity).

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.

Bruce, T., van der Meer, J. W., Franco, L., & Pearson, J. M. (2009). Overtopping performance of different armour units for rubble mound breakwaters. *Coastal Engineering*, 56(2), 166-179.

Fenton, J. D. (1988): The numerical solution of steady water wave problems, *Computers & Geosciences*, Vol. 14, No. 3, pp 357-368.

Goda, Y., Morinobu, K. (1988): Breaking wave heights on horizontal bed affected by approach slope. *Coastal Eng. Journal*, 40, 307-326.

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.

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.

Kofoed-Hansen, H. & Rasmussen, J. H. (1998), Modelling of nonlinear shoaling based on stochastic evolution equations, *Coastal Engineering*, 33, 203-232.

Larsen, J., & Dancy, H. (1983), Open Boundaries in Short-wave Simulations - A New Approach, *Coastal Engineering*, 7, 285-297.

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.

Liu, P. L. F., Lin, P., Chang, K. A., Sakakiyama, T. (1999) Numerical modeling of wave interaction with porous structures, *Journal of Waterway, Port, Coastal and Ocean Engineering*, 125, 322-330

Mei, C. C. (1983): The applied dynamics of ocean surface waves, John Wiley & Sons, New York.

Munk, W., Anderson, E. (1948), Notes on the theory of the thermocline, *Journal of Marine Research*, 7, 276-295.

Rodi, W. (1980), Turbulence Models and Their Application in Hydraulics - A State of the Art Review, *Special IAHR Publication*.

Smagorinsky (1963), J. General Circulation Experiment with the Primitive Equations, *Monthly Weather Review*, 91, No. 3, pp 99-164.

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.

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.

Svendsen, I. A., Jonsson, I. G. (1980): Hydrodynamics of Coastal Regions, Technical University of Denmark.

Ting, F.C.K. and Kirby, J.T., (1994), Observation of undertow and turbulence in a laboratory surf zone, *Coastal Eng*., 24, 51-80.

Toro, E.F., Spruce M., Speares W. (1994), Restoration of the contact surface in the HLL Riemann solver, *Shock Waves*, 4(1), 25-34.

van Gent, M. R. A. (1995), Wave Interaction with Permeable Coastal Structures, (Ph.D. thesis), Delft University.

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