MIKE 21/3 Particle Tracking

Particle Tracking Model

User Guide

PLEASE NOTE

COPYRIGHT

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

LIMITED LIABILITY

The liability of DHI is limited as specified in 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 the MIKE 21/3 Particle Tracking Model, for applications involving the simulation of transport and fate of dissolved and suspended substances discharged or accidently spilled in lakes, estuaries and coastal areas or at the open sea.

1.2 Assumed User Background

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

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

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

1.3 General Editor Layout

The MIKE Zero setup editor consists of three separate panes.

1.3.1 Navigation tree

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

For sections containing spatial data - e.g. sources, boundaries and output - a geographic view showing the location of the relevant items will be available. The current navigation mode is selected in the bottom of this view, it can be zoomed in, zoomed out or recentered. A context menu is available from which the user can select to show the bathymetry or the grid 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 opend on an item - e.g. a source - it is also possible to jump to this item's editor.

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

1.3.3 Validation window

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

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

1.4 Online Help

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

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

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

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

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

2 INTRODUCTION

2.1 General Description

The MIKE 21/3 Particle Tracking Model is used for modelling of the transport and determine the fate of dissolved, suspended and sedimented substances discharged or accidently spilled in lakes, estuaries and coastal areas or at the open sea.

The particles/substances may be a pollutant of any kind, conservative or non-conservative, for example suspended sediment particles, inorganic phosphorus, nitrogen, bacteria or chemicals.

The pollutant is considered as particles being advected with the surrounding water body and dispersed as a result of random processes in dimensions. The particles may settle with a constant settling velocity and settled particles may be resuspended. A corresponding mass is attached to each particle, which may be reduced during the simulation due to decay.

The following processes may be attached to individual particle classes:

- Settling in various details including flocculation
- Erosion
- Decay
- Moving sources
- Wind forcing
- Dispersion

The model calculates the path of each particle and outputs the instantaneous concentrations of individual classes in 2D or 3D depending on the hydrodynamic input.

2.1.1 Application areas

The Particle Tracking Module can be applied to the study of engineering problems such as:

- Sedimentation
- Planning, design and monitoring of outfalls
- Risk analysis and accidental spillage
- Environmental Impact Assessment
- Monitoring of dredging works

3   Getting Started

The hydrodynamic basis for the Particle Tracking model must be calculated by the MIKE 21 or MIKE 3 Hydrodynamic model.

If you are not familiar with setting up a hydrodynamic model you should refer to the User Guide for the Hydrodynamic Module and the comprehensive step-by-step training guide covering the MIKE 21 Hydrodynamic Model. The user guides and the training guide (PDF-format) can be accessed from the MIKE 21 and MIKE 3 Documentation index:

MIKE 21 Flow Model, Hydrodynamic Module, User Guide

MIKE 3 Flow Model, Hydrodynamic Module, User Guide

MIKE 21 Flow Model, Hydrodynamic Module, Step-by-step Training Guide

4 Examples

4.1 Introduction

One of the best ways to get to know how to use a modelling system such as MIKE 21 or MIKE 3 is through practical applications. Therefore examples are included which you can go through yourself and which you can modify to see the effects if one or other parameter is changed.

To demonstrate the capabilities of the Particle Tracking model we have included two examples of simulations. A directory is provided for each example. The directory names are as follows (default installation):

- Corner flow (2D & 3D)
  .\Examples\MIKE_21-3_Integrated_Models\PT\Corner
- Sound (2D)
  .\Examples\MIKE_21-3_Integrated_Models\PT\Sound

4.2 Corner Flow Example

4.2.1 Purpose

This simplified example has been chosen to describe a simple case of simulating the concentration and amount of material sedimented in a channel, given the material originates from a single point constant outfall. In the 2D simulation the outfall is moving, in the 3D example the position is fixed.Emphasis is made on the interpretation and presentation of the results.

4.2.2 Scenario

The model is a channel with a L-shaped corner, initially with a constant depth. The computational domain and bathymetry is shown in Figure 4.1.
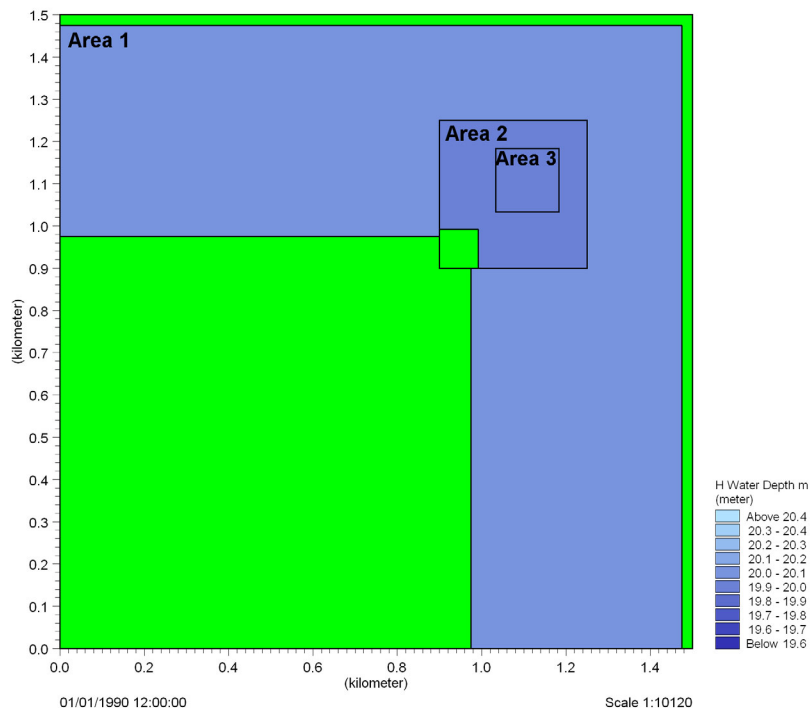
Figure 4.1   Computational domains and bathymetries for area 1, area 2 and area 3

The problem is to determine the fate of the suspended material from the outfall.

The test conditions are:

- The channel is about 500 m wide, with 1 km stretch on either side of the corner. The bottom of the channel is 20 m below datum

- The flow goes from south to west with a current speed of approximately 0.8 m/s caused by a water level difference of 0.05 m between the two open boundaries

- 2D Only: The particle source is moving from position (167100, 200) to position (167400, 200) at the water surface in the domain, see Figure 4.1. The discharge from the particle source is constant 10 kg/s.

- 3D Only: The particle source is positioned at position (167250, 200) at depth -1, see Figure 4.1.

- The discharge from the particle source is constant 10 kg/s.

4.2.3   Parameters for 2D simulation

The hydrodynamic flow field is calculated by MIKE 21 HD using the conditions below.

- A structured grid with 3 nested areas. The grid is shown in Figure 4.2.

- A time step of 5 seconds is selected and the duration of the simulation is 3 hours (2160 time steps).

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

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

- Initially the water surface is 0.0 m in the entire domain.

- The upstream (south) boundary is defined by a constant water level of 0.05 m above datum. The west boundary is defined by a constant water level held at datum.
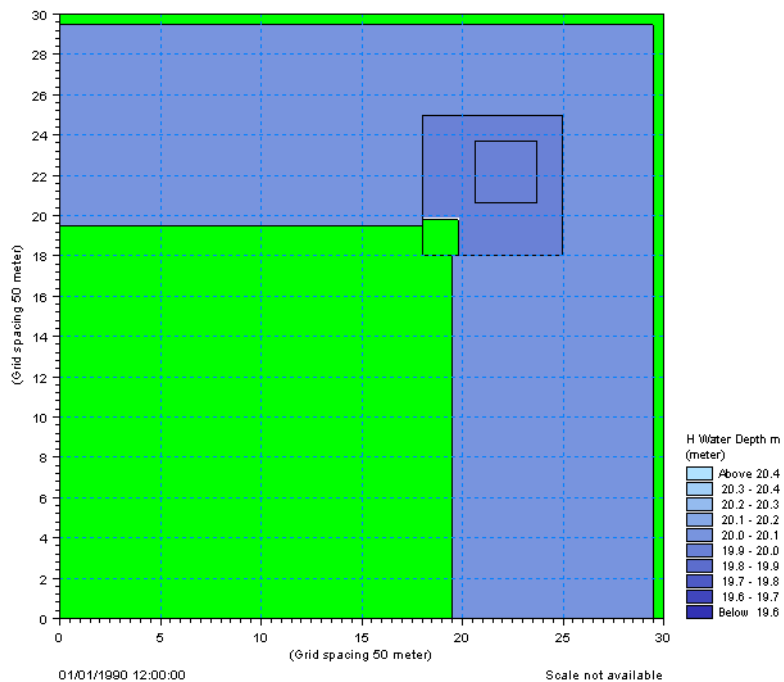

Figure 4.2   Computational nested grids and showing the model coordinates. Area 1 has a gridspacing of 50 m, Area 2 has a gridspacing of 16.67 m and finally Area 3 has a gridspacing of 5.58 m

Once the hydrodynamic flow field is obtained, the Particle Tracking module can be setup and executed.

The main condition defining the particle tracking problem is:

- The particle tracking simulations are performed for one particle class.

- The source location is moving from position (167100, 200) to position (167400, 200) 1 m below the water surface.

- The strength of the source is 10 kg/s. 10 particles are released per time step.

- The horizontal dispersion is set at a constant level of 0.1 m^2/s.

- The settling velocity is specified directly as a constant value of 0.005 m/s.

- Output is saved from the entire area as well as for the two nested sub areas. Also a particle file is saved.

4.2.4   Results for 2D simulation

The hydrodynamic flow is slowly being built up from zero velocity to the maximum. However, the emission of material from the source is constant through the whole simulation. This is the reason for the very high concentrations reached at the source point in the beginning of the simulation.

Contour plots of the concentration field is shown in Figure 4.3. A plot of the particle tracks along with the instantaneous location of particles is shown in Figure 4.4.

Due to the settling parameter, some particles will reach the bed and sedimentate before leaving the domain. Therefore you will see some particles get stuck in the channel.

Note that due to the horizontal dispersion, two consecutive runs will not produce the exact same result.
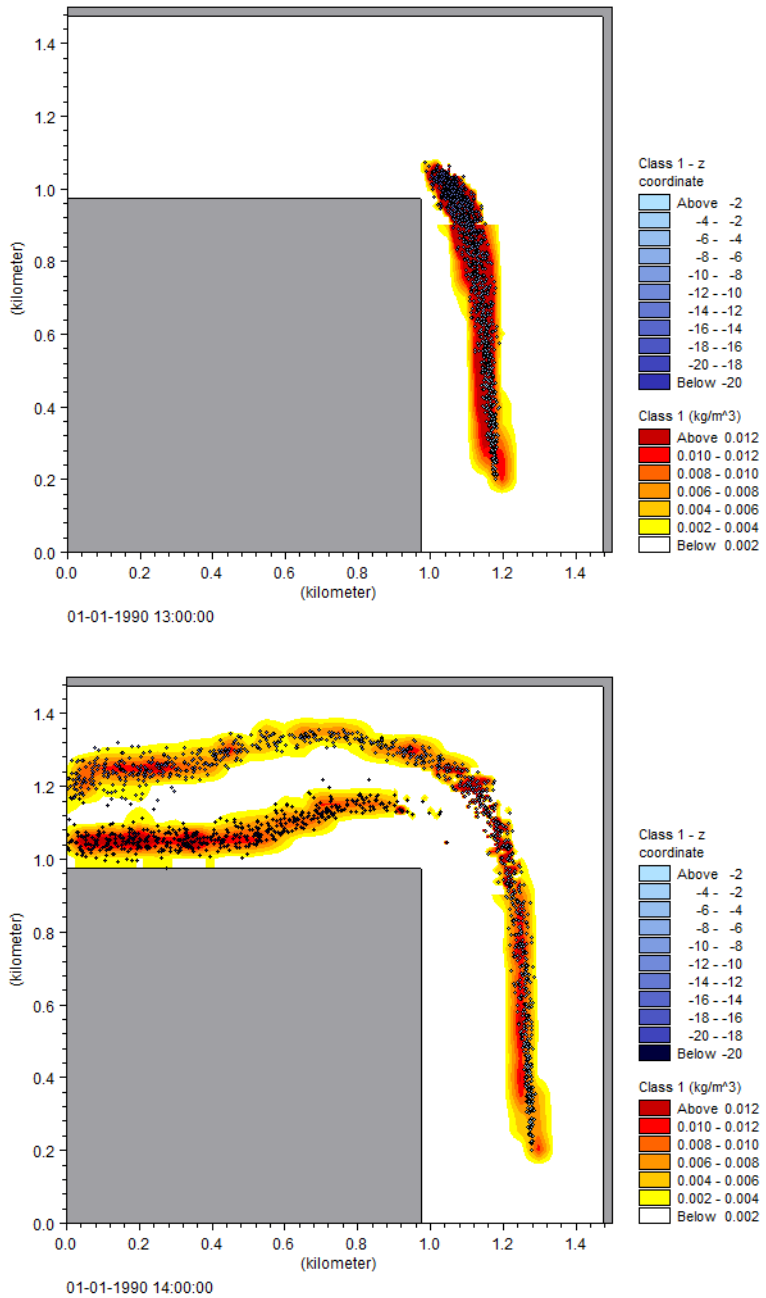
Figure 4.3   Total concentration of material + cloud of particles snapshot.
             Upper: After 1 hour.
             Lower: After 2 hours

01-01-1990 15:00:00
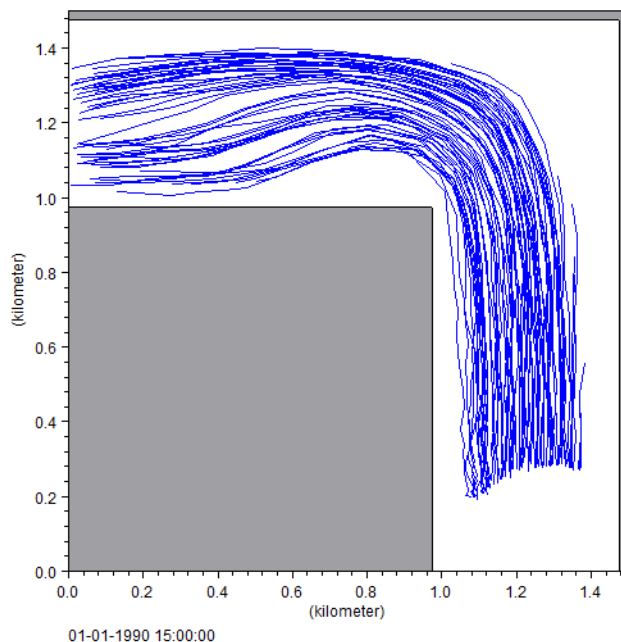
Figure 4.4   Particle tracks covering every 50th particle. The particle tracks illustrate
the path of particles at all time steps

4.2.5   Parameters for 3D simulation

The hydrodynamic flow field is calculated by MIKE 3 HD using the conditions below.

- A structured grid without nested areas. The grid is shown in Figure 4.2 (without nested areas). The horizontal gridspacing is 50 m. The computational grid has 20 layers with a vertical gridspacing of 1 m

- A time step of 5 s is selected and the duration of the simulation is 1½ hours (1080 time steps).

- The Eddy viscosity type has been chosen with a constant value of 0.5

- The bed resistance has been chosen as a roughness height of 0.05 m

- Initially the water surface is 0.0 m in the entire domain

- The upstream (south) boundary is defined by a constant water level of 0.05 m above datum. The west boundary is defined by a constant water level at datum.

Once the hydrodynamic flow field is obtained the Particle Tracking Module can be setup and executed.

The main conditions defining the particle tracking problem is:

- The particle tracking simulations are performed for one particle class.
- The source location is a fixed source at position (167250, 200) 5 m below the water surface
- The strength of the source is 10 kg/s. 10 particles are released per time step
- The horizontal dispersion is set at a constant level of 0.1 m^2/s, and no dispersion in the vertical direction
- Settling is not included
- Output is saved for the entire area covering the same areas as the hydrodynamic file. Also a particle file is saved.

4.2.6 Results for 3D simulation

The hydrodynamic flow is slowly being built up from zero to the maximum. However the emission from the source is constant through the whole simulation.

Contour plots of the concentration field in layer 16 (corresponds a depth of 5 m) is shown in Figure 4.5.

As no processes for settling, decay, erosion are included the sediment spreading reaches a steady situation after a while. The particles stay at the depth where they were released. Concentration variations can be explained by variations in the hydrodynamic flow caused by the warming up of the hydrodynamic model.
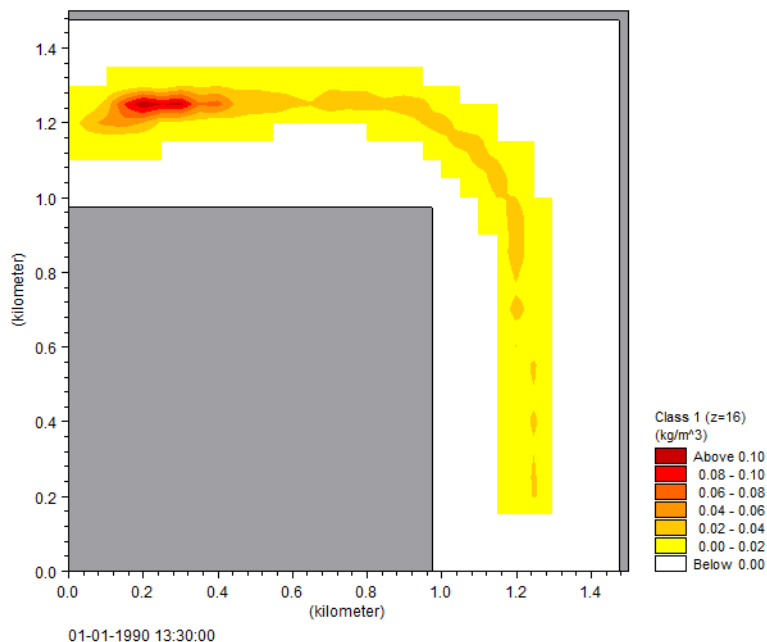
Figure 4.5   Concentration map of suspended material after 1½ hour. The map shows the concentrations as they appear in layer 16

4.2.7   Files

The following data files (included in the PT\Corner folder) are supplied with MIKE 21/3 Particle Tracking Module

File name: bathy1.dfs2
Description: bathymetry for Area 1

File name: bathy2.dfs2
Description: bathymetry for Area 2

File name: bathy3.dfs2
Description: bathymetry for Area 3

File name: hd.m21
Description: MIKE´21 HD specification file

File name: hd.m3
Description: MIKE´3 HD specification file

File name: PT2D.npt
Description: MIKE´21/3 PT specification file, 2D simulation

File name: PT3D.npt
Description: MIKE´21/3 PT specification file, 3D simulation

4.3 Oresound Example

4.3.1 Purpose

The construction of the Oresund link between Denmark and Sweden included compensation dredging of the sea bottom to obtain the socalled zero solution for the flow through Oresund, and also dredging operations were carried out for the construction work itself. The spill from the dredging operations resulted in plumes of sediment that spread with the currents and settled in Oresund.

The particle tracking model was used for planning and monitoring the various dredging operations in connection with the link construction work.

In this example the Particle tracking model is setup to simulate the transport of inorganic material from one source in Oresund.

4.3.2 Scenario

The hydrodynamic setup correspond to the setup described in the step-by-step training guide covering the MIKE 21 Hydrodynamic Model. This training guide (PDF-format) can be found (applies for default installation):

.\Manuals\MIKE_21\FlowModel\HD\MIKE21_HD_Step_By_Step.pdf

The test conditions are:

- The sound area covers about 60 km x 100 km
- The flow direction shifts almost every 12 hours
- The source spill is simulated for one constant source with a single class
- The discharge from the particle source is 10 kg/s
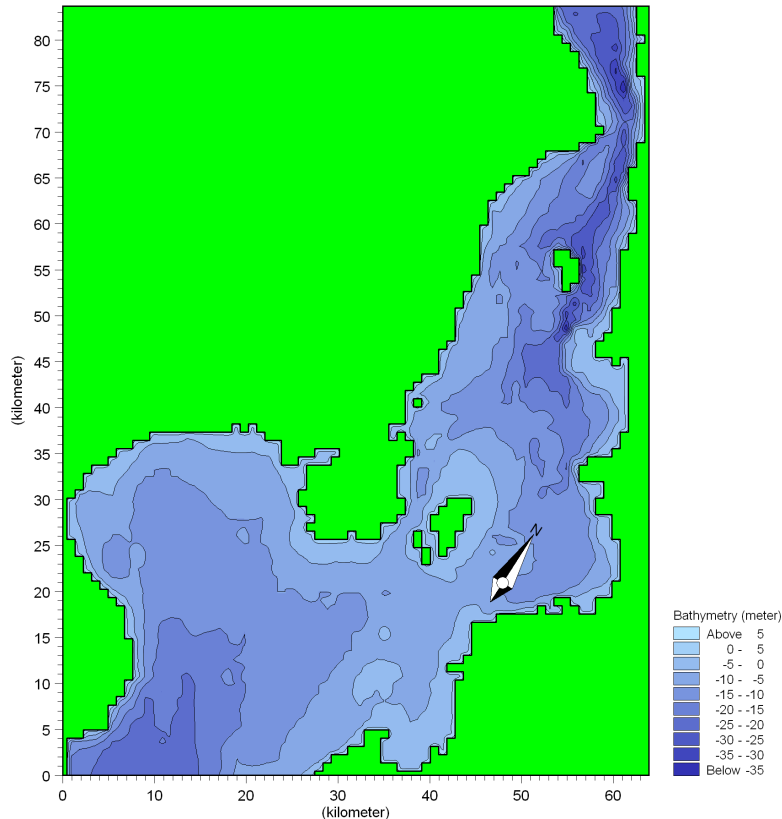
Figure 4.6   Model area, the sound between Denmark and Sweden

4.3.3   HD simulation

The main condition defining the hydrodynamic problem is:

- The model area is resolved with a structured computational grid with a horizontal gridspacing of 900 m. The grid is shown in Figure 4.7

- An overall time step of 300 seconds is used and the duration of the simulation is 12 days starting at 02/12/1993 00:00:00

- The south and north boundaries are each defined with water levels specified as line series

- The wind forcing is included as varying in time but constant in domain. The wind conditions are measured in Kastrup. The wind friction varies with the wind speed using default values.

- The eddy viscosity has been chosen to Smagorinsky formulation with a constant value of 0.5

- The bed resistance type has been chosen as Manning number, with a constant value of 44 $m^{1/3}$/s
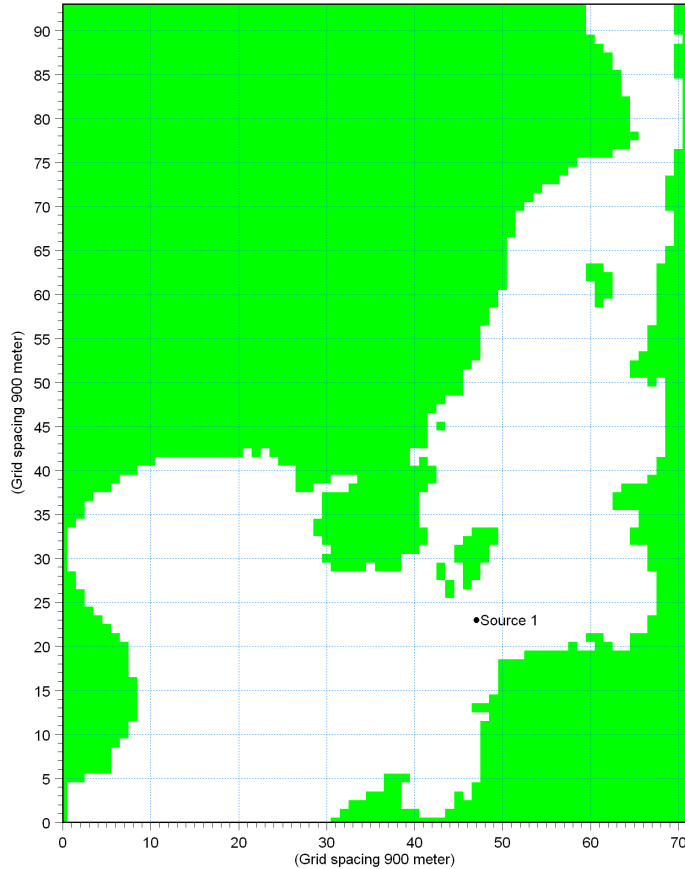
Figure 4.7   Computational grid and model coordinates, and position of the source

4.3.4   Parameters - PT simulation

The main settings defining the particle tracking problem are:

- A single class is used with default settings.

- The source spill of this class is simulated for one constant source located at E: 364500 N: 6162000. The position of the source is shown in Figure 4.7. It is not possible to use model coordinates, but only real coordinates. Hint: Real coordinates can be found from the used bathymetry, if the bathymetry file is opened in for example the Result viewer tool.

- Decay and Erosion properties are not included

- Settling is specified directly as settling velocity. A settling velocity of 0.00008 m/s is used to represent sediments with a size in the order of 10 micron.

- The horizontal dispersion is set as a constant dispersion coefficient of 3 m^2/s

- Vertical dispersion is not included

- The drift profile is set to bed shear profile using a bed roughness of 0.001 m

- The first output file is a dfs2 file containing class concentration, surface elevation, still water depth, total water depth, depth integrated U-velocity, depth integrated V-velocity. The dfs2 file is saved every 12 time steps.

- The second output file is a Particle file (XML file) and will contain all particles saved every 12 time steps

4.3.5 Results

A continous spill is included in the simulation. The currents in Oresund vary a lot and therefore the spreading of the sediments never reaches a steady situation. The released sediments settle on the sea bottom throughout Oresund controlled by the currents, and the applied settling velocity.

It can be seen that the particles are stuck once they reach the sea bed, because erosion is not included.
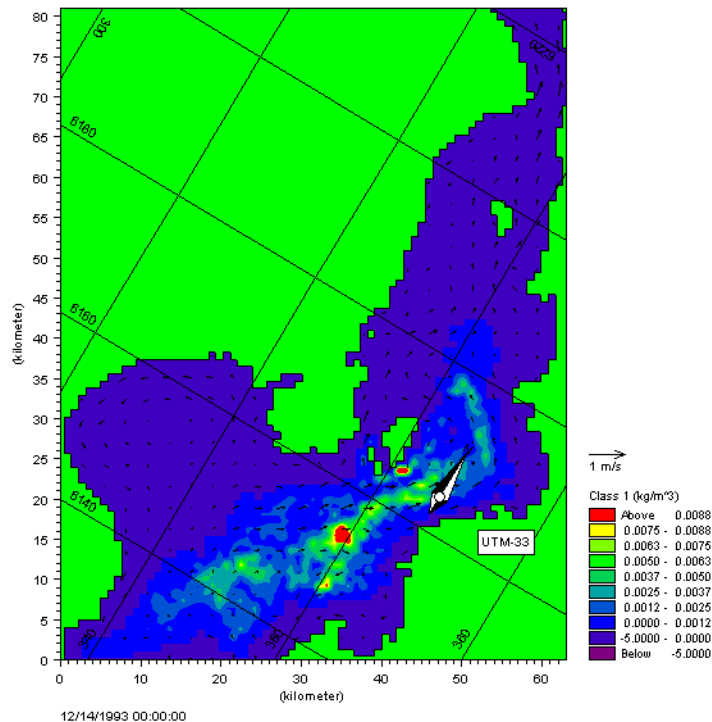
Figure 4.8   Concentration field 12 days after the start of simulation.

4.3.6   Visualising the particle tracks

The individual particles saved in the second file can be plotted as an overlay in the dfs2 result file. The particle file overlay function for dfs2 files works from within Plot Composer, and Grid Editor.

The following plots have been made with Plot Composer. Open a Plot composer window and add a grid series. Afterwards right click the plot and pick Add/Remove layers. Pick the file type as a particle file type.
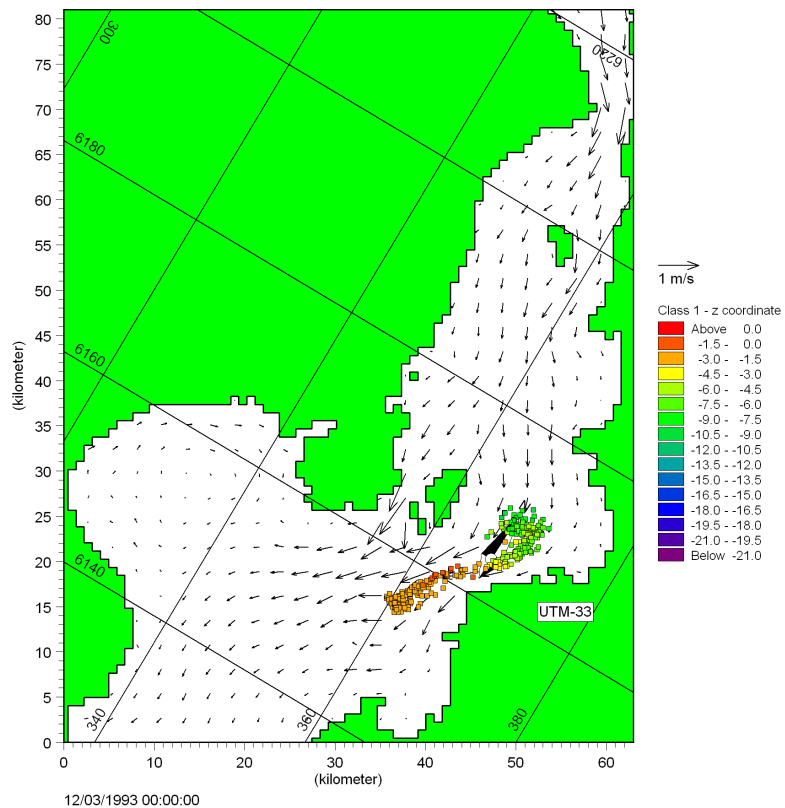
Figure 4.9   Particle cloud after 1 day.
The colour of the particle refers to z coordinate of the particle, so it can be seen that the particles are settling, and their vertical position depends on when the particle was released from the source
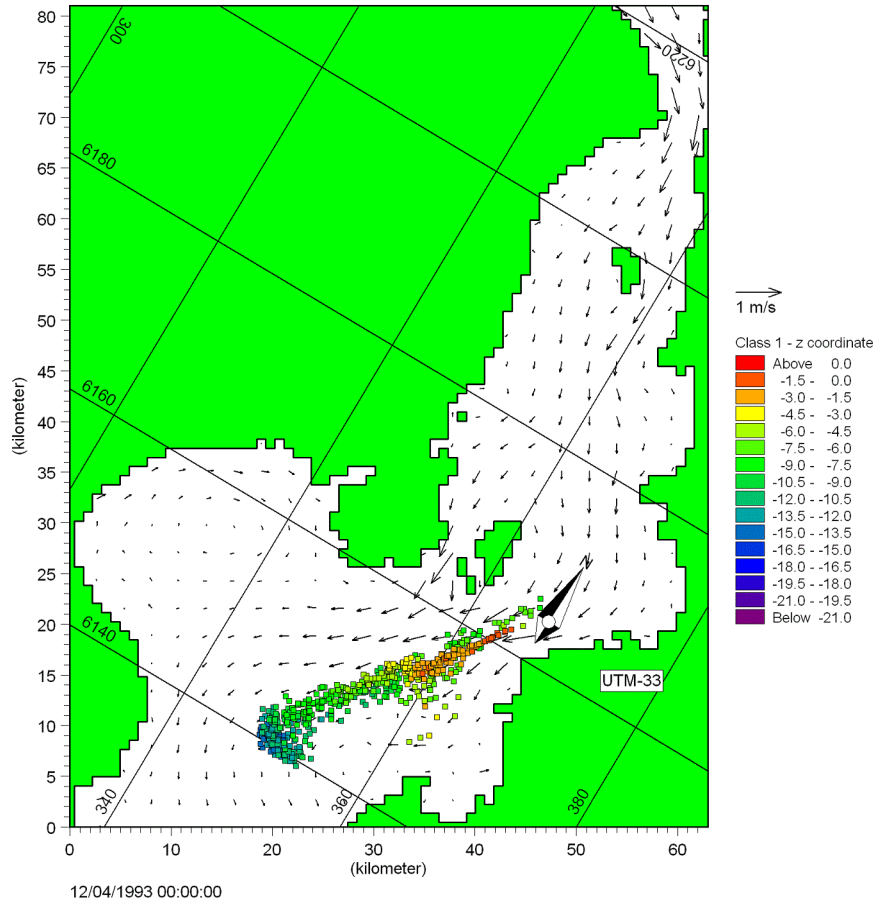
Figure 4.10   Particle cloud after 2 days.
The colour of the particle refers to z coordinate of the particle, so it can
be seen that the particles are settling, and their vertical position
depends on when the particle was released from the source
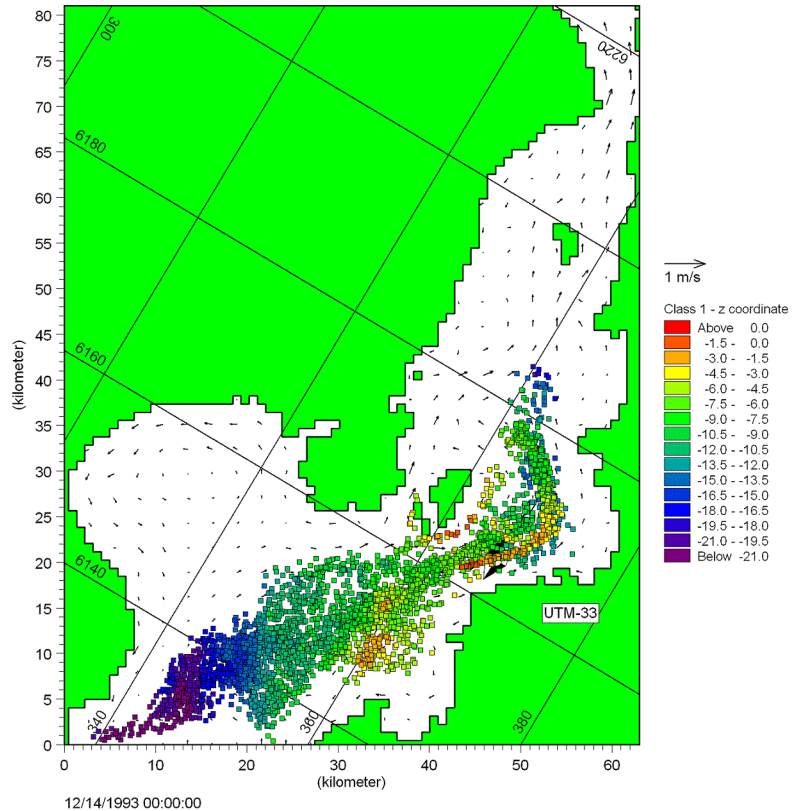
Figure 4.11   Particle cloud after 12 days.
The colour of the particle refers to the z coordinate of the particle, so it can be seen that some of the particles have settled on the sea bed

4.3.7   Files

The following data files (included in the PT\Sound folder) are supplied with MIKE 21/3 Particle Tracking Moule:

Filename: hd.m21
Description: Specification file contains the HD setup, open and run this file first to generate hydrodynamic result files for the PT simulation

Filename: pt.npt
Description: Specification file contains the PT setup, open this file to run the simulation

Filename: bathymetry.dfs2
Description: Contains the bathymetry

Filename: pt.plc
Description: Contains plot setup showing particle cloud

Filename: pt2.plc
Description: Contains plot setup showing concentration

Filename: wind_kastrup.dfs0
Description: Wind speed and direction for area

Filename: wln.dfs1
Description: Water level variation at north boundary

Filename: wls.dfs1
Description: Water level variation at south boundary

5   BASIC PARAMETERS

The hydrodynamic basis for the Particle Tracking simulation must be calculated beforehand using the MIKE 21 or MIKE 3 Hydrodynamic model for rectangular grid.

5.1   Time

You specify the period to be covered by the simulation in this dialog.

You have to specify the simulation start date, the overall number of time steps and the overall time step interval (in seconds).

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   Hydrodynamic Data

Here you must specify the file containing data from a hydrodynamic simulation made with MIKE 21 HD or MIKE 3 HD. You can type in the file path or click the browse button on the right to navigate to the file.

Once selected, the graphical user interface will validate that the file is actually a data file from MIKE 21 HD or MIKE 3 HD, and that the simulation period is represented within the data file.

Note:
For 2D simulations the input file must be a dfs2 file with Data type = 1 and the items (H,P,Q) only.
For 3D simulations the input file must be a dfs3 file with Data type = 3 and the items (U,V,W,pressure). The pressure item is necessary in order to calculate concentration in the upper layer and can only be output when using the non-hydrostatic version.

6 PARTICLE TRACKING MODULE

The Particle Tracking module calculates the transport and determine the fate of dissolved, suspended and sedimented substances discharged or accidently spilled in lakes, estuaries and coastal areas or at the open sea.

6.1 Classes

Particles are divided into different groups called classes. Each class has specific properties regarding decay, settling/buoyancy, erosion, and dispersion that has to be specified separately.

Typical examples of classes are different size fractions of sediment particles, organic pollutants with different decay rates, floating objects or coloured tracers.

6.1.1 Class description

You must specify a name and a description of the class.

You must also pick a EUM unit for the particle suitable for the particle in question. This choice determines the unit of the output, and also the source flux.

You furthermore specify the minimum mass and maximum age a particle in this class can have. These parameters are introduced in order to exclude particles that are old, or have decayed to insignificant masses. The computational simulation time is proportional to the number of particles in the simulation, so therefore it is often a good idea to exclude the particles that are dead or have insignificant masses. The particle age, as per default, is unlimited by setting age to zero.

6.2 Sources

The effects of spill, an outlet or a moving ship can be included in the simulation using particle sources.

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

6.2.1 Particle source specification

Only one source type can be specified for rectangular grid models:

- Point source

In general point sources specify a single, defined point in space. A normal source is active throughout the complete simulation time. An initial source is used to specify an initial particle distribution and does not release any particles during the rest of simulation.

A source may include one or many different types of particle classes. The additional information for each particle class is made subsequently. You may choose to include or exclude the defined classes in the simulation.

Coordinates
You have to specify the map projection (LONG/LAT, UTM etc.) in which you want to specify the location coordinates of the particle source.

Vertical source specification
Apart from the horizontal specification a source location must be specified vertically in the water column. Three definitions are available for the vertical positioning of a source. The vertical reference can be defined as:

- Depth (positive downwards from the moving water surface)
- Above bed (positive upwards relative to the sea bed)
- Datum (absolute level)

Point source

The location of a point source must be defined using one of two sub-types

- Fixed location
- Moving location

For the Fixed location source option, i.e. the source is stationary during simulation time, you must specify the horizontal coordinates of the source point and its vertical position.

For the Moving location source option, i.e. the source is moving along a defined path, you must specify a time series file (dfs0) that contains the horizontal coordinates and the vertical position of the source as a function of time. The unit of the vertical position is meters and the value is to be given according to the defined vertical type.

Released class terms

The format of the particle class information within the source can be specified as

- Constant in time
- Varying in time

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

You furthermore define the number of particles to be released into the flow per overall time step. This can be a constant number (same number every time step). Alternatively the number of released particles can be read from a time series data file (dfs0). Be aware that interpolation between data points may lead to a unexpected number of particles, thus it is best to provide a data series with the same time resolution as the simulation.

Further it can be declared that the specifications refer to the overall emitted amount, i.e. the amount is split among all released particles. This is especially useful if you change the number of particles as the released amount will be independent on the number of particles. On the other hand the released amount can be specified on a per-particle base. Similar the amount can be declared to be handled as flux or instantaneous release.

Note:
The source data will be checked against the allowed minimum/maximum particle mass. If the source value is outside this range, the value will be clipped to ensure that the particle value will not violate the minimum/ maximum.

6.2.2 Remarks and hints

To model a single particle/object several approaches can be followed:

Method 1:

Specify a time series with just one particle for the number of released particles.

Method 2:

1. Specify a time series file for the source. Set the source values to 0 apart from the time step where the particle is to be released.
2. Specify only one particle to be released per time step.
3. In the Class description *(p. 31)* set the minimum particle mass to a value greater than zero. This will cause all particles without mass to be eliminated from the calculation and only the relevant particle with mass to

remain.

6.3 Decay

The decay is specified individually for each particle class and you have the option to include or exclude the decay for the individual class.

6.3.1 General description

Many processes can be approximated by a simple first-order decay, such as die-off of E. Coli due to exposure to light, decay of the activity of radioactive substances or estimating the age of water bodies.

First order decay of a component is generally described by

$$\frac{dm}{dt} = -k \cdot m \qquad (6.1)$$

or

$$m = m_0 \cdot e^{-kt} \qquad (6.2)$$

where m is the specific mass of a particle, m_0 is the initial mass of the particle and k is the decay rate. In the model the decay term is calculated for all particles of a class that includes decay.

A decay where the decay rate k given in eq. (6.1) is time varying can also be calculated. This could for instance be the case if the decay has a temperature dependency, which means that k will vary over the season. An example could be decay of BOD (Biochemical oxygen demand).

6.3.2 Class decay

The format of the decay factor can be specified as

- Constant (in time)
- Varying in time

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

6.3.3 Remarks and hints

If the decay represents a very rapid process such that the product $k\Delta t > 1$, the decay term may cause negative masses of the individual particles. A solution is then to reduce the time step.

Also, the decay process might result in masses that are close to zero. In order to increase the simulation speed, it is possible to specify a minimum mass for each class' particles. If the decay reduces the particle mass below the specified value, the particle is removed from the simulation, and thereby increasing the computational performance.

6.4 Settling

The settling is specified individually for each particle class and you have the option to include or exclude the settling for the individual class.

6.4.1 General description

It is possible to include vertical movement of the particles other than the vertical hydrodynamic flow: Buoyancy and settling. Settling and buoyancy are processes seen for particles with different densities than the surrounding water.

An example, where the buoyancy process is used, could be release of oil below the water surface.

Typical examples of cases, where the settling process is used, are simulation of mud, and sand.

When settling or buoyancy is included the user specifies a vertical settling velocity (positive for settling and negative for buoyancy). The settling velocity is dependent on the size of the particles, and can be measured in the lab. Therefore when simulating for instance mud, it is normal to describe the different size fractions of mud, as a number of particle classes with different settling properties.

If measured settling velocities are not available, it is a possibility to give a rough estimate of the settling velocity of a single free particle through Stokes law, if the size of the particle is known:

$$w_s = \frac{(\rho_s - \rho)gd^2}{18 \cdot \nu} \tag{6.3}$$

where

w_s:	Particle fall velocity [m/s]
ρ_s:	Sediment density [kg/m^3] (Quartz = 2650 kg/m^3)
ρ:	Density of water [kg/m^3]
g:	Gravity [9.81 m/s^2]
d:	Diameter of particle [m]
ν:	Dynamic viscosity of water [m^2/s]

In case of fine grained cohesive particles (<0.006 mm) the settling velocity may change during the simulation.

The total concentration of sediment is derived in the PT model, by summarizing the number of particles found in each cell, and dividing with the cell volume. The cell uses a discretization outlined horizontally with the computational mesh element, and vertically with a fixed vertical resolution (not sigma coordinates).

With low concentrations of particles, the probability for collision between the particles is low and the settling velocity will be close to the settling velocity for a single grain. With increasing concentration, collision between particles will occur more frequently and eventual cohesiveness of the particles will result in formation of flocs. This leads to an increase in average particle/floc size and with that an increase in settling velocity. This process can be included with the flocculation process.

If particle concentration increases, further hindered settling occurs, which leads to a reduction in settling velocity. Further increase in sediment concentration will result in decreasing distance between the flocs, which leads to negligible settling velocity and the mixture will act as fluid mud. This process can be included with the hindered settling option.

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

Flocculation

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

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

With low concentrations of suspended sediment, the probability for collision between the cohesive particles is low and the settling velocity will be close to the settling velocity for a single grain. With increasing concentration, collision

between particles will occur more frequently and the cohesiveness of the particles will result in formation of flocs. This leads to an increase in average particle/floc size and with that an increase in settling velocity.

$$w_s = w_0 \cdot factor_{floc} \tag{6.4}$$

$$
\begin{aligned}
C_{total} \leq C_{floc} \quad &: factor_{floc} = 1 \\[8pt]
C_{floc} < C_{total} < C_{hindered} \quad &: factor_{floc} = 1 + \alpha\left(\frac{C_{total}}{C_{floc}} - 1\right) \\[8pt]
C_{total} \geq C_{hindered} \quad &: factor_{floc} = 1 + \alpha\left(\frac{C_{hindered}}{C_{floc}} - 1\right)
\end{aligned}
\tag{6.5}
$$

where

w_s:	Particle settling velocity
w_0:	Particle settling velocity unflocculated
C_{floc}:	Concentration at which flocculation begins
$C_{hindered}$	Concentration at which hindered settling begins
C_{total}:	Total concentration of sediment (sum of concentrations of all fractions)
α:	Gradient coefficient, for calibration

Hindered settling

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

$$w_s = w_0 \cdot factor_{floc} \tag{6.6}$$

$$
\begin{aligned}
C_{total} \leq C_{floc} \quad &: factor_{floc} = 1 \\[8pt]
C_{total} > C_{floc} \quad &: factor_{floc} = 1 + \alpha\left(\frac{C_{total}}{C_{floc}} - 1\right) \cdot \left(1 - \min\left(1, \frac{C_{total}}{C_{gel}}\right)\right)^5
\end{aligned}
\tag{6.7}
$$

where

w_s:	Particle settling velocity
w_0:	Particle settling velocity unflocculated
C_{floc}:	Concentration at which flocculation begins
C_{gel}:	Gelling point
C_{total}:	Total concentration of sediment (sum of concentrations of all fractions)
α:	Gradient coefficient, for calibration

Salinity

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

Salinity only plays a significant role on flocculation at salinities below 9 psu.

$$w_{s,sali} = w_s(1 - C_1 \cdot e^{SC_2}) \tag{6.8}$$

where

$w_{s,sali}$:	Particle settling velocity including salinity
w_s:	Particle settling velocity
C_1:	Calibration parameter, default C_1 = 0.5
C_2:	Calibration parameter, default C_2 = -0.33
S:	Salinity

6.4.2   Class settling

The settling for particles in suspension can be defined by five types:

- Specify settling velocity directly
- Include flocculation
- Include flocculation and hindered settling
- Include flocculation and salinity
- Include flocculation, hindered settling and salinity

Depending on the choice of settling type you must specify the settling velocity data and the settling parameters.

Settling velocity data

The format of the settling velocity coefficient for particles in suspension can be specified as:

- Constant (in time)
- Varying in time

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

For the case with time varying settling velocity coefficient you have to prepare a data file containing the velocity coefficients. The data file must be a time series file (dfs0). The data must cover the complete simulation period. The

time step of the input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

Settling parameters

Minimum concentration for flocculation
When this concentration is exceeded, flocculation occurs.

Maximum concentration for flocculation
When this concentration is exceeded, the settling velocities do not increase further

Gelling point
This parameter is used in the Richardson and Zaki formulation for hindered settling.

C1 and C2
These parameters define the influence from the salinity.

Alpha
Calibration parameter for changing the gradient between the relation between the settling velocity and the total concentration.

6.5 Dispersion

The dispersion describes the transport due to molecular diffusion and due to non-resolved turbulence or eddies. In coastal areas the dispersion due to non-resolved turbulence is normally by far the most important. In the horizontal directions the effects of non-resolved turbulence can be significant, in which case the dispersion coefficient should depend on the resolution.

In coastal areas it is important to distinguish between horizontal dispersion due to e.g. non-resolved eddies, and vertical dispersion due to e.g. bed generated turbulence. Hence, dispersion in horizontal and vertical directions is specified separately. Individual particles cannot break into pieces and therefore the process of dispersion is implemented as a likelihood to move in a random direction.

6.5.1 Dispersion specification

The dispersion can be formulated two different ways:

- No dispersion
- Dispersion coefficient formulation

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

Data

Selecting dispersion coefficient option the format of the dispersion coefficient can be specified as

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

For the case with dispersion coefficient varying in domain the values are constant in the vertical domain and only varying in the horizontal domain. You have to prepare a data file containing the dispersion coefficient before you set up the hydrodynamic simulation. The file must be a 2D grid file (dfs2) and the area in the data file must cover the model area. Bilinear interpolation is used to map the data.

6.5.2 Recommended values

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

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

6.6 Erosion

The erosion of a bed layer is the transfer of particles from the bed to the water column. Erosion takes place from the bed in areas where the bed shear stress (τ_b) is larger than the critical shear stress for erosion (τ_{ce}).

The erosion is specified individually for each particle class and you have the option to include or exclude the erosion for the individual class.

6.6.1 General description

Erosion is a process that describes how sediments like mud and sand (with densities larger than water) can get resuspended into the water column.

The flow velocities are normally zero at the bottom and particles in contact with the bed will therefore not move horizontally, unless erosion has been included for a given particle class. If erosion is included and the bed shear

stress exceeds the critical shear stress, the particle can perform a random walk through the dispersion term in the vertical direction. In this way the particle can get resuspended into the water column.

6.6.2   Class erosion

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

The format of the critical shear stress for each particle class can be specified as:

- Constant (in domain)
- Varying in domain

A typical shear stress value is 0.1 N/m^2.

For the case with varying in domain you have to prepare a data file containing the critical shear stress before you set up the hydrodynamic simulation. The file must be a 2D grid data file (dfs2).

The area in the data file must cover the model area. Bilinear interpolation is used to map the data.

You furthermore have to define the relative sediment density and mean diameter of the particles in the class.

6.7   Drift Profile

6.7.1   General description

The drift profile is a description of the drift regime that the particles are influenced by. It will normally be the currents and the wind that governs the drift regime. The currents including wind are already calculated in the hydrodynamic setup, but the hydrodynamic output does normally not have a fine discretization of the current profile near the bed. Therefore it can be a good idea to assume some shapes of the vertical drift profile.

It is possible to include the bed friction drag on the current profile by assuming a bed shear profile (or logarithmic profile).

The wind drag can also cause increased flow velocities in the upper part of the water column, and corresponding velocities in the opposite direction in the lower part.

If there are particles in the water surface, they can be influenced directly by the wind additionally to the influence from the flow. This will result in an addi-

tional acceleration of the particle in a direction relative to the wind turned with a wind drift angle caused by coriolis forces.

6.7.2 Class drift profile

The different drift profile types can be combined in the following ways:

- Use raw data from hydrodynamics
- Use bed shear profile
- Use surface wind acceleration
- Use bed shear profile and wind induced profile (2D only)
- Use bed shear profile, wind induced profile and surface wind acceleration (2D only)
- Use bed shear profile and surface wind acceleration

Depending on the choice of drift profile you must specify some additional parameters.

Bed shear profile

The shape of the velocity profile within a turbulent boundary layer is well established by both theory and experience. The profile has specific characteristics very close to the bed where viscosity controls the vertical transport of momentum, and different characteristics farther from the bed where turbulence controls the vertical transport of momentum. The region closest to the bed boundary is called the laminar sub-layer or viscous sub-layer, because within the region, turbulence is suppressed by viscosity. The laminar sublayer only plays a significant role for smooth flows, whereas for rough flows the flow is zero for z smaller than z_0.

Logarithmic layer (z >= δ_s smooth flow) (z >= z_0 rough flow)

$$u = \frac{2.3}{k} \cdot u^* \cdot \log_{10} \frac{x}{z_o} \tag{6.9}$$

where

u^*:	Friction velocity
k:	Karman constant = 0.4
z_0:	characteristic roughness

Surface wind acceleration

The user must specify wind weight, wind drift angle and the kinematic viscosity in order to calculate the surface wind acceleration.

Particles that are exposed to wind in the water surface (i.e. located in the upper 5 cm of the water column) are affected according to the wind regime in 2 ways: indirectly via the currents that include the wind, but also directly as an extra force directly on the particle. How much of the wind speed that is transferred to the particle speed depends on the nature of the particle, how much is the particle exposed, etc., so therefore it is a calibration factor that expresses how much of the windspeed is added to the particle speed.

$$U_{particle} = U_{current} + C_w \cdot W \cdot \sin(\alpha_w - \pi + \theta_w)$$
$$V_{particle} = V_{current} + C_w \cdot W \cdot \cos(\alpha_w - \pi + \theta_w)$$

(6.10)

where

C_w:	Calibration factor for wind drag on particle
θ_w:	Wind drift angle
α_w:	Wind direction

Wind drift angle

The coriolis force is normally included in the hydrodynamic currents, but also for the wind acceleration of surface particles the coriolis force must be considered.

Due to the influence from the Coriolis force, the direction of the wind drift vector is turned relatively to the wind direction. The angle *qw* of deviation is termed with the wind drift angle. It turns to the right on the Northern Hemisphere and to the left on the Southern Hemisphere. Thus wind drift angles for the Southern Hemisphere should be specified by negative values. From Al-Rabeh (1994), it is assumed that

$$\theta_w = \beta \exp\left(\frac{\alpha |U_w|^3}{g \gamma_w}\right)$$

(6.11)

where

α:	$-0.3 \cdot 10^{-8}$
β:	28° 38'
U_w:	Wind speed [m/s]
γ_w:	Kinematic viscosity [m²/s]
g:	Gravity [m/s²]

The recommended value of the kinematic viscosity is of the order $1 \cdot 10^{-6}$. The magnitude of the wind drift angle varies with the geographical location and wind speed and it is often estimated at 12-15 degrees in the North Sea.

Wind induced profile

The user must specify a wind drift factor, a depth of influence and an offshore limit in order to calculate a wind induced drift profile.

The direction and the magnitude of currents are varying with the depth. The wind may induce strong currents in the surface layer and has a significant effect on the surface drifting.

The depth-averaged currents must therefore be modified with respect to the influence from the wind. This has been done by calculating the tracking as a vectorial sum of the wind driven currents (the wind drift vector) and the current velocity vector.

The wind velocity component is added to the current velocity component vectorially to determine the total surface drift velocity. Once the drift velocity is known the movement of the parcels are computed by numerical integration.

The magnitude of the wind drift vector is commonly assumed to be proportional to the magnitude of the wind speed 10 m above the sea surface.

The vertical distribution of the wind drift vector consists of an offshore part and an onshore part. The onshore distribution is based on a parabolic vertical profile and is able to produce back flow at depth, where the offshore logarithmic profile does not.

The parabolic profile acts in shallow waters with a water depth less than a specified water depth, h_{sep}, which is a positive value and measured from the free water surface.

The vertical distribution of the parabolic onshore profile is given by

$$c_w(z) = c_w^*\left(1 - 3\frac{z}{h}\right)\left(1 - \frac{z}{h}\right) \tag{6.12}$$

where

h: local water depth
z: vertical particle co-ordinate, measured from sea surface
c_w^*: wind drift factor (input)

The parabolic profile causes the wind-generated flow in the upper third of the water column to be in the same direction as the current and the flow in the lower part to be in the opposite direction of the wind. There is no net depth averaged mass transport due to the wind.

The vertical distribution of the offshore wind drift vector is given by

$$c_w(z) = c_w^*\exp(-k_0 z) \tag{6.13}$$

where

k_0:	$3/h_w$
h_w:	depth of wind influence (input)
z:	vertical co-ordinate measured from sea surface
$c_w{}^*$:	wind drift factor [-] (input)

6.8 Salinity

The salinity can be specified as User specified salinity.

6.8.1 User specified salinity

The salinity can be specified as:

- Constant

- Varying in time, constant in domain

- Varying in time and domain

For the case with values varying in domain you have to prepare a data file containing the salinity before you set up the hydrodynamic simulation. The file must be a 2D grid data file (dfs2). The area in the data file must cover the model area. Bilinear interpolation is used to map the data.

6.9 Bed Roughness

The bed roughness is used in the Particle Tracking module if the Drift Profile is defined as a Bed shear profile (logarithmic profile). It is an important parameter for describing the near bed flow conditions.

The bed roughness can be specified as User specified bed roughness.

6.9.1 General description

The bed roughness in the Particle Tracking model is defined by the Nikuradse roughness *ks*. This roughness parameter may be converted from other roughness definitions as described below.

Chezy number conversion to Nikuradse roughness k_s:

$$k_s = \frac{12h}{10^{(C/18)}}$$ (6.14)

where

h:	Water depth
C:	Chezy number

Manning number conversion to Nikuradse roughness *ks*:

$$k_s = \left(\frac{25.4}{M}\right)^6 \tag{6.15}$$

where *M* is the Manning number.

6.9.2 User specified bed roughness

The bed roughness must be specified as Nikuradse roughness and it can be specified as:

- Constant
- Varying in domain

For the case with values varying in domain you have to prepare a data file containing the bed roughness before you set up the hydrodynamic simulation. The file must be a 2D grid data file (dfs2). The area in the data file must cover the model area. Bilinear interpolation is used to map the data.

6.10 Wind Forcing

The wind forcing is used in the Particle Tracking module if the Drift Profile is defined using the influence from wind.

The wind forcing can be specified as User specified wind.

6.10.1 User specified wind

The wind forcing can be specified as

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

Note that the first two options are only available when defining the wind forcing by speed and direction.

For the case with values varying in domain you have to prepare a data file containing the wind forcing (speed and direction (degrees from true North) or velocity components) before you set up the hydrodynamic simulation. The file

must be a 2D grid data file (dfs2). The area in the data file must cover the model area. Bilinear interpolation is used to map the data.

Furthermore a soft start interval must be speficied.

6.11   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 at all time steps. In practice, subsets must be selected.

In the main Outputs dialog you can add a new output file by clicking on the "New output" button. By selecting a file in the Output list and clicking on the "Delete output" you can remove this file. For each output file you can specify the name (title) of the file and whether the output file should be included or not. The specification of the individual output files are 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.

6.11.1   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 2D simulation 2D field variables can be selected. For a 3D simulation 3D field variables can be selected. The particle track can also be selected.

Please note that particle track output can become very large in size if all particles are saved every time step. The intended use is to follow only a few of the particles.

Two types of output is available:

- Particles
- Concentrations

Particles
For particle track the default file format is xml. Other track storage formats may be available (see File format page 48). Variables for each particle will be written in the file: x, y, z, mass, age at each time step.

The xml format is generally recognised by many readers and 3rd party soft-ware. Within MIKE Zero it can be used in the Plot Composer (Grid Plot), Grid Editor, Result Viewer and Data Viewer where you can add it as a layer to vis-ualize particle movements (right click "Add/Remove Layers...").

Please note that particle track output can become very large in size because it writes out positions every time step. The intended use is to follow only a few of the particles. This is done by selecting a frequency higher than 1 (see Par-ticle numbers page 49).

Concentrations

The concentrations are saved in the modelled area in a dfs2 file. If several nested areas have been used in the simulation, it is possible to select which area to apply.

Output file

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

File format

Particle tracks may be stored in various track storage formats. Binary formats can only be used by the MIKE software, whereas XML storage offers easy data access. Particle data can be saved as XML files using eithera com-pressed or uncompressed XML variant. The different XML variants are described below.

- In uncompressed format the values of the xyz-coordinates and each selected variable are saved on separate lines in the file

- In compressed format the values of the xyz-coordinates and each selected variable are saved in the same line in the file

```
<Particle Nr="51">
<x>1079.511942</x>
<y>1017.705602</y>
<z>-0.9754099093</z>
<age>3120.000000</age>
<Mass>60.00000000</Mass>
</Particle>
```

Figure 6.1   Example of data in XML file using uncompressed particle format

```
<Particle Nr="51">
<![CDATA[1079.511942,1017.705602,-0.9754099093,3120.000000,60.00000000]]>
</Particle>
```

Figure 6.2   Example of data in XML file using compressed particle format

Time step

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

Alternative spatial output

You have the option to output the concentration field in a different grid resolution than the input grid and based on only part of the water column.

Particle numbers

The range of the particle numbers refer to the particles ID's in each class. During the simulation all new particles are given an ID. The first particle number is 1 and the total number of particles are the multipla of the overall time step and the number of particles released per time step in each class.

6.11.2 Output items

2D field variables

You can select basic output variables and additional output variables.

The basic variables consist of derived concentrations of the included classes. For each class, all the particles of the class are distributed according to the computational mesh, and the summarized mass of particles in each element is divided with the volume of the element cell to estimate the concentration in the cell. The unit of the concentration output fields depends on the how the units for the class has been specified (see Classes, page 31).

The basic variables for each class are

- Total concentration
- Concentration of suspended material
- Amount of sedimented material
- Amount of material for a given Z-Range specification

The additional variables are

- Surface elevation
- Still water depth
- Total water depth
- Depth averaged U-velocity
- Depth averaged V-velocity

Z-Range specification

You can select to output the amount of material within a specific z-range of the water column. The Z-Range limits are defined in the dialog below the basic variables. The Z-Range can be specified in three different ways:

- Datum/absolute
  The Z-Range limits are defined as absolute values, related to datum

- Above bed
  The Z-Range is defined by the offset from the bed (positive upwards) and the layer thickness

- Below surface
  The Z-Range is defined by the offset from the water surface (positive downwards) and the layer thickness

If Alternative spatial output is selected the additional variables will not be an option.

3D field variables

You can select basic output variables and additional output variables.

The basic variables consist of derived concentrations of the included classes. For each class, all the particles of the class are distributed according to the computational mesh, and the summarized mass of particles in each element is divided with the volumen of the element cell to estimate the concentration in the cell. The unit of the concentration output fields depends on the how the units for the class has been specified.

The basic variables for each class are

- Total concentration
- Concentration of suspended material

Note that the two basic variables only differs in the bottom layer (layer 1) where some of the material may be sedimented.

The additional variables are

- U Velocity
- V Velocity
- W Velocity

Particle track

The particle track file contains the (x, y, z) location and mass for the individual particles.

INDEX