MIKE 21 Flow Model FM

Particle Tracking Module

User Guide

PLEASE NOTE

COPYRIGHT

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

LIMITED LIABILITY

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

CONTENTS

1 ABOUT THIS GUIDE

1.1 Purpose

The main purpose of this User Guide is to enable you to use the MIKE 21 Flow Model FM, Particle Tracking Module, 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 there is a navigation tree showing the structure of the model setup file, and it is used to navigate through the separate sections of the file. By selecting an item in this tree, the corresponding editor is shown in the central pane of the setup editor.

1.3.2 Editor window

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

For sections containing spatial data - e.g. sources, boundaries and output - a geographic view showing the location of the relevant items will be available. The current navigation mode is selected in the bottom of this view, it can be zoomed in, zoomed out or recentered. A context menu is available from which the user can select to show the bathymetry or the mesh, to show the optional GIS background layer and to show the legend. From this context menu it is also possible to navigate to the previous and next zoom extent and to zoom to full extent. If the context menu is opened on an item - e.g. a source - it is also possible to jump to this item's editor.

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

1.3.3 Validation window

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

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

1.4 Online Help

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

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

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

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

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

2 INTRODUCTION

2.1 General Description

The Particle Tracking module of MIKE 21 Flow Model FM is used for modelling 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 nonconservative, 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 2 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 detail 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.

There is no feedback to the hydrodynamic module, and therefore the particle model may with advantage be run in decoupled mode.

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 (EIA)

- Monitoring of dredging works

3   GETTING STARTED

The hydrodynamic basis for the Particle Tracking module must be calculated using the Hydrodynamic module of the MIKE 21 Flow Model FM modelling system.

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

> MIKE 21 Flow Model, Hydrodynamic Module, User Guide

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

A comprehensive training guide covering the Particle Tracking Module of the MIKE 21 Flow Model FM modelling system is also provided with the DHI Software installation.The objective of this training guide is to set up a Particle Tracking model for the Great Belt area from scratch and to calibrate the model to a satisfactory level. The training guide (PDF-format) can be accessed from the MIKE 21 Documentation index:

> MIKE 21 Flow Model FM, Particle Tracking 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 Flow Model FM 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 module we have included four examples of simulations.

- Corner flow (see Corner Flow Example *(p. 13)*)
- Harbour (see Harbour Basin Example *(p. 19)*)
- Oresound (Oresound Example *(p. 23)*)
- Great Belt (Dredging spill in Great Belt *(p. 31)*)

In a standard installation you will find the files needed to run the examples in .\MIKE_21\FlowModel_FM\PT.

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 that is moving. 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.
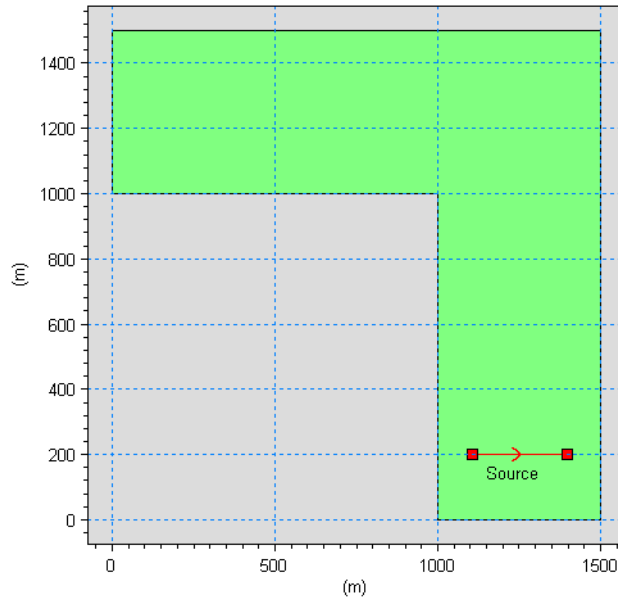
Figure 4.1   Computational domain and bathymetry.
              Position of particle source is indicated

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

- The particle source is moving from position (1100, 200) to position (1400, 200) at the water surface in the domain, see Figure 4.1. The discharge from the particle source is constant 10 kg/s.

4.2.3   Parameters

The main condition defining the hydrodynamic problem is:

- An unstructured mesh with 778 elements and 443 nodes is used. The mesh is shown in Figure 4.2.

- An overall time step of 0.5 seconds is selected and the duration of the simulation is 3 hours (21600 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 soft start for this boundary is defined as by a sinus variation with a time interval of 900 s.
  The west boundary is defined by a constant water level held at datum.

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 (1100, 200) to position (1400, 200) at the water surface.

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

- The dispersion is set horizontally at a constant level of 0.1 m/s. No dispersion in the vertical direction.

- Output is saved from the entire area as well as for three output points P1, P2 and P3, respectively, see Figure 4.2.
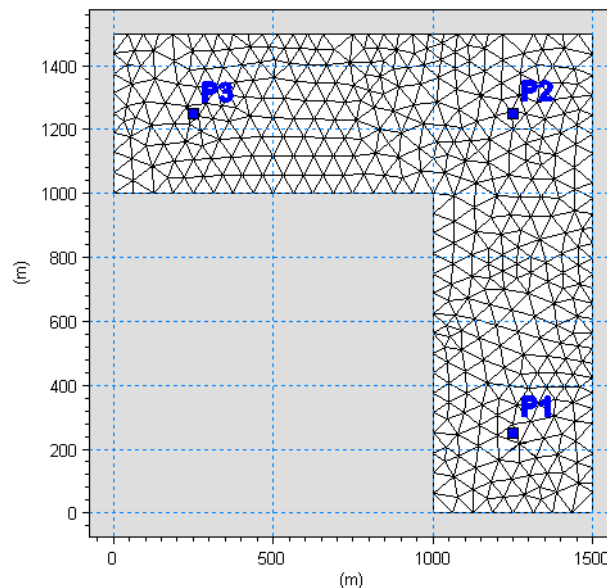
Figure 4.2   Computational mesh and output points.
P1: output at position (1250, 250)
P2: output at position (1250, 1250)
P3: output at position (250, 1250)

4.2.4   Results

The hydrodynamic flow field is slowly being built up from zero velocity to the maximum (Figure 4.3). 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 at the beginning of the simulation, see Figure 4.4. Contour plots of the concentration field is shown in Figure 4.5.

Note that due to the horizontal dispersion, two consecutive runs will not produce the exact same result.
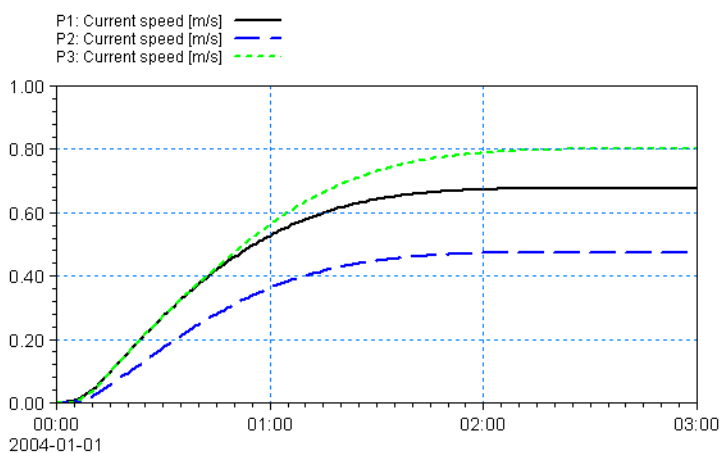
Figure 4.3   Time series of current speed
Solid black line : P1
Stippled blue line: P2
Dotted green line: P3
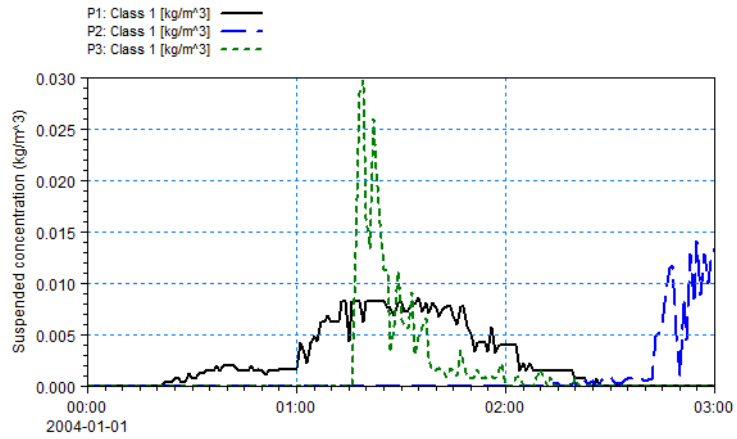
Figure 4.4   Time series of suspended sediment concentration
              Solid black line : P1
              Stippled blue line: P2
              Dotted green line: P3
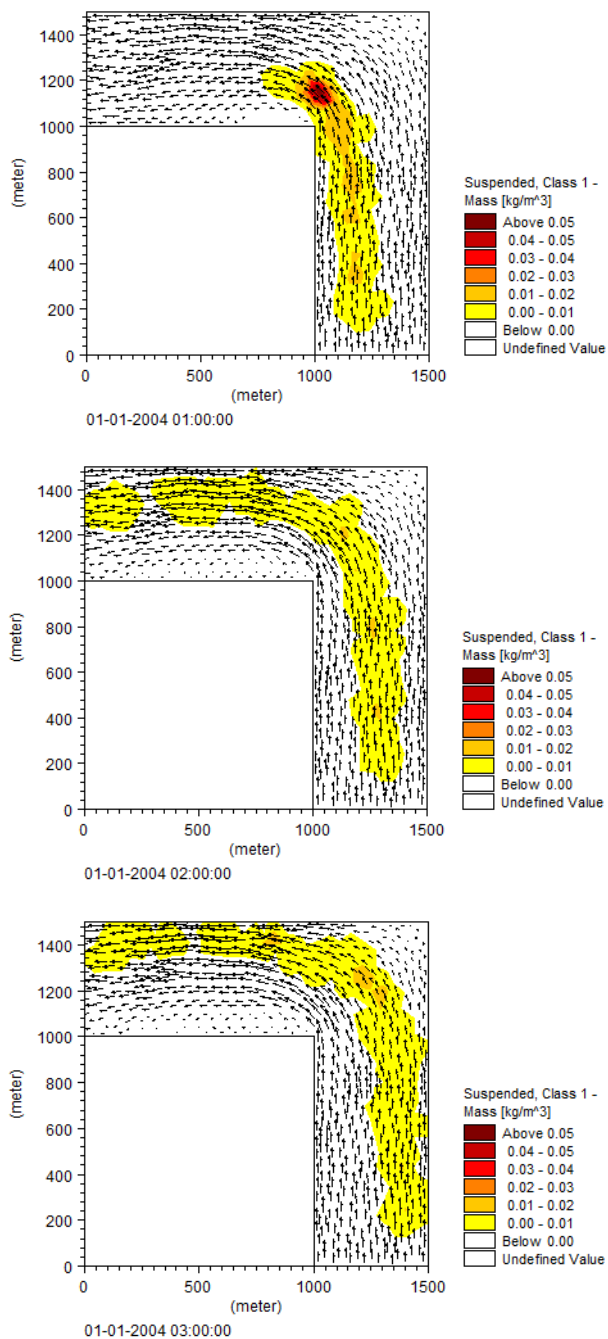
Figure 4.5   Concentration of suspended material (arrows indicate currents)
Upper: after 1 hour
Middle: after 2 hours
Lower: after 3 hours

4.2.5 Files

The following data files (included in the PT\Corner folder) are supplied with MIKE 21 Flow Model FM, PT module:

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

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

File name: Source_Location.dfs0
Description: Defines the location of the moving source

4.3 Harbour Basin Example

4.3.1 Purpose

This simplified example has been chosen to describe a case of particle tracking where a constant sediment source outlet disperse two different kinds of particles.
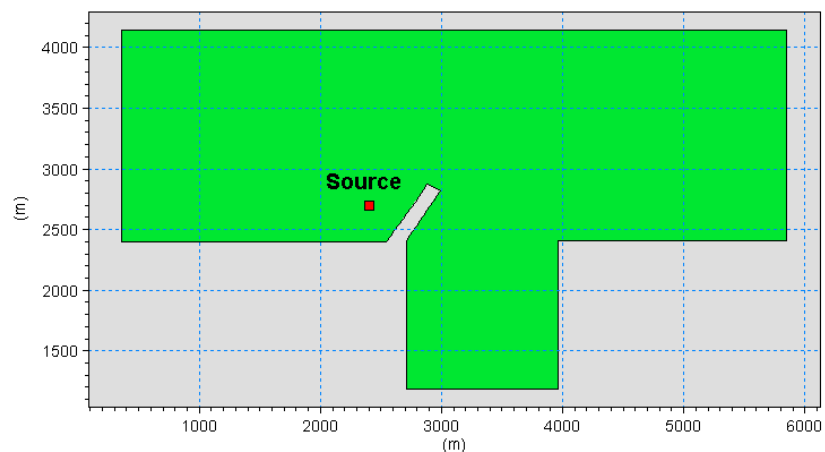
Figure 4.6   Computational domain and bathymetry.
Position of particle source is indicated

4.3.2 Scenario

The problem is to determine the fate of the suspended material from an outfall and show the effect of settling.

The test conditions are:

- The bottom of the harbour is 5 m below datum. A jetty extend out into the flow.

- The flow goes from left to the right.

- The particle source consist of two types of classes: floaters and divers, respectively. The particle source is placed at the water surface in position (2400, 2700) in the domain, see Figure 4.6.

- The discharge of floaters is constant 10 kg/s.

- The discharge of divers is constant 100 kg/s, but the divers has a maximum particle age of 1800 s and they settle with a constant velocity of 0.005 m/s.

4.3.3   Parameters

The main condition defining the hydrodynamic problem is:

- A unstructured mesh with 907 elements and 519 nodes is used. The mesh is shown in Figure 4.11.

- An overall time step of 60 seconds is selected and the duration of the simulation is 18 hours (1080 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.

- A constant wind is blowing from 270 deg.N with the speed 10 m/s. A constant friction factor of 0.001255 is applied.

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

- The upstream (east) boundary is defined by a constant water level of 0.2 m above datum. The soft start for this boundary is defined as by a linear variation with a time interval of 3600 s.
  The downstream (west) boundary is defined by a constant water level of 0.0.

The main condition defining the particle tracking problem is:

- The particle tracking simulations are performed for two particle classes, floaters and divers. The divers has a maximum particle age of 1800 s.

- The source location is fixed at position (2400,2700).

- The strength of the source of floaters is 10 m^3/s.
  One particle is released per time step.

- The strength of the source of divers is 100 m^3/s.
  10 particles is released per time step.

- The divers have a constant settling velocity of 0.005 m/s

- The horizontal dispersion for the floaters is defined by the scaled eddy viscosity formulation with a scaling factor of 1. The vertical dispersion is defined by a constant value of 0.01 m^2/s.

- No dispersion is defined for the divers.

- The drift profile is defined by the surface wind acceleation with a wind weight of 0.1, a wind drift angle of 28 and a kinematic viscosity of 1.14·10^{-6}.

- The bed roughness and wind forcing are both taken from the hydrody-namic model.

- Output is saved both from the entire area for the two classes and in a particle track file the full tracks of a few of the particles are stored.
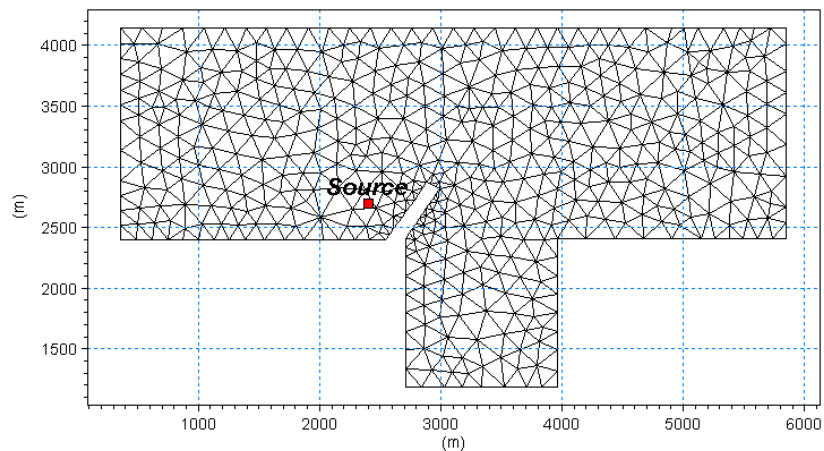
Figure 4.7   Computational mesh (source location indicated).

4.3.4   Results

The hydrodynamic flow field 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.

The float tracks for the Floaters class shows the majority of the tracks to be near the source point, but a number of the floaters is shown to travel farther away, see Figure 4.8. None of the floaters is being caught behind the jetty.

The float tracks for the Divers shows that the particles settle very quickly near the source point, see Figure 4.9. This indicate that in time the particles of class "Divers" will accumulate locally.
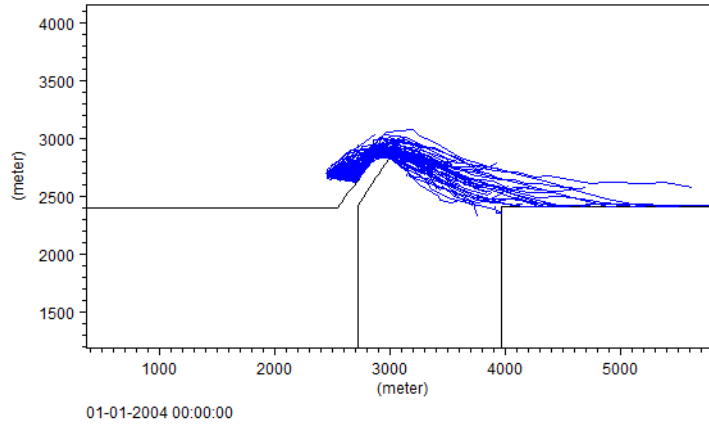
Figure 4.8   Float tracks for particle class "Floaters"
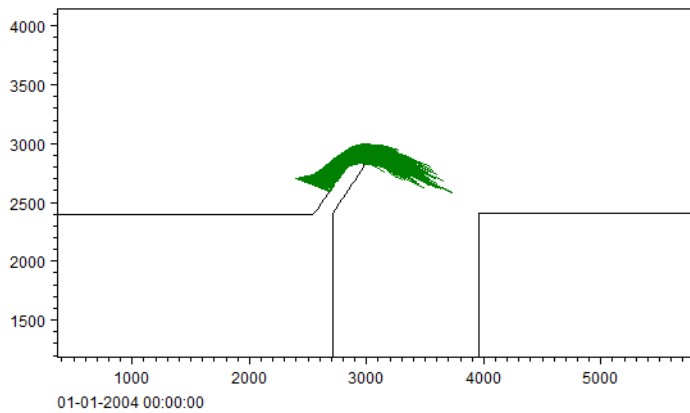
Figure 4.9   Float tracks for particle class "Divers"

4.3.5   Files

The following data files (included in the PT\HarbourBasin folder) are supplied with MIKE 21 Flow Model FM, PT module:

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

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

4.4   Oresound Example

4.4.1   Purpose

The construction of the Oresound link between Denmark and Sweden included compensation dredging of the sea bottom. This equalized the flow resistance from the bridge piers and thereby the construction didn't reduce the supply of saline and oxygen rich water from the North sea into the Baltic sea. 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 module is setup to simulate the transport of inorganic material from one source in the Oresound.

This example also illustrates how to use the Decoupling mode in MIKE 21 Flow Model FM simulations: In a first simulation (see 4.4.3 Parameters - HD simulation *(p. 24)*) the basic flow information from the hydrodynamic simulation is saved in a number of special decoupling output files. The second simulation (see 4.4.5 Parameters - PT simulation *(p. 27)*) re-run the first simulation in a decoupled mode, where instead of calculating the flow information, it is read from these files. The particle tracks from one single source are computed in this second simulation. Please note that running a simulation in decoupled mode can significantly reduce the computational time and is especially relevant in the case of Particle Tracking simulations.

4.4.2   Scenario

The hydrodynamic set-up correspond to the setup described in the step-by-step training guide covering the Hydrodynamic Module of MIKE 21 Flow Model FM. The training guide (PDF-format) can be accessed from the MIKE 21 Documentation Index:

> MIKE 21 & MIKE 3 Flow Model FM, Hydrodynamic Module, Step-by-Step Training Guide
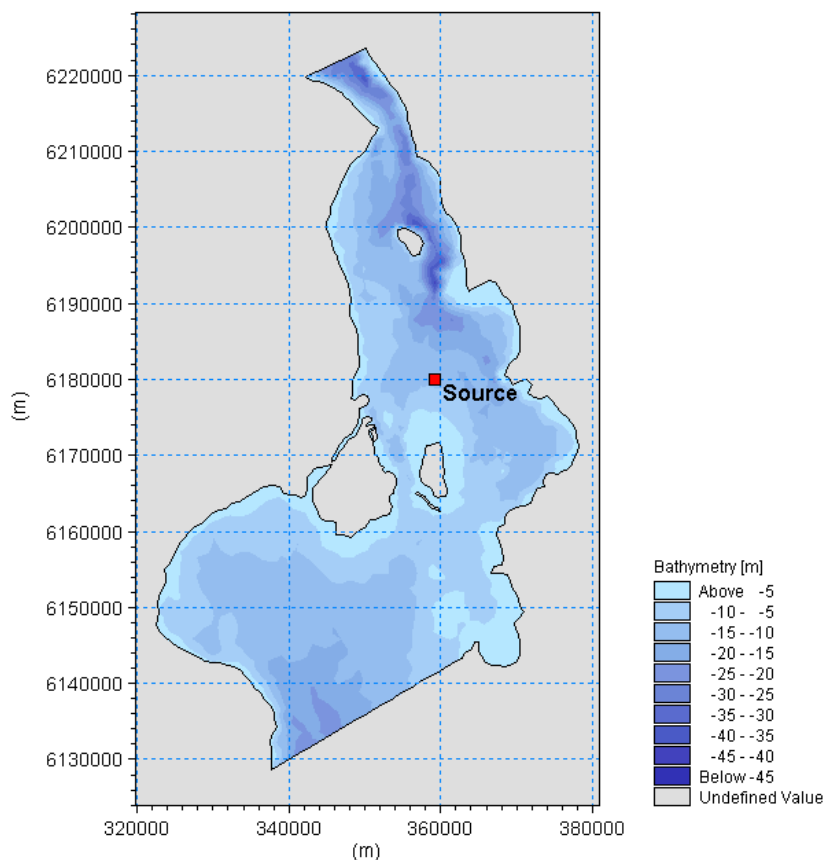
Figure 4.10   Model area, the sound between Denmark and Sweden
             Red point indicate source position.

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.

4.4.3   Parameters - HD simulation

The main condition defining the hydrodynamic problem is:

- A unstructured mesh with 2057 elements and 3636 nodes is used. The mesh is shown in Figure 4.11.

- An overall time step of 300.0 seconds is selected and the duration of the simulation is 6.75 days (1800 time steps) starting on 06/12/1993 at 18:00. The flow shifts almost every 12 hours between north and south.

- The south and north boundaries are each defined by a time and space varying water level.

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

- The Coriolis type is set to varying in the domain

- For this example the initial conditions are the result from a previous simulation (file: hot_start.dfsu) which act as a hot start of the flow field.

- The horizontal eddy viscosity type has been chosen to Smagorinsky formulation with a constant value of 0.24 m^2/s.

- The bed resistance type has been chosen to Manning number, with a constant value of 45 $m^{1/3}$/s.

- The Decoupling mode is set using a 'Time step frequency' of 1 meaning that Decoupled HD results will be saved every time step = 300 seconds in two files, Flux: **out_HD_DecouplingFlux.dfsu** and Area: **out_HD_-DecouplingArea.dfsu**.
The file: **Setup1b_Oresound_HD_decoupled.m21fm** will be created by this first setup and can be used later on to set up a new run using the HD decoupled results.
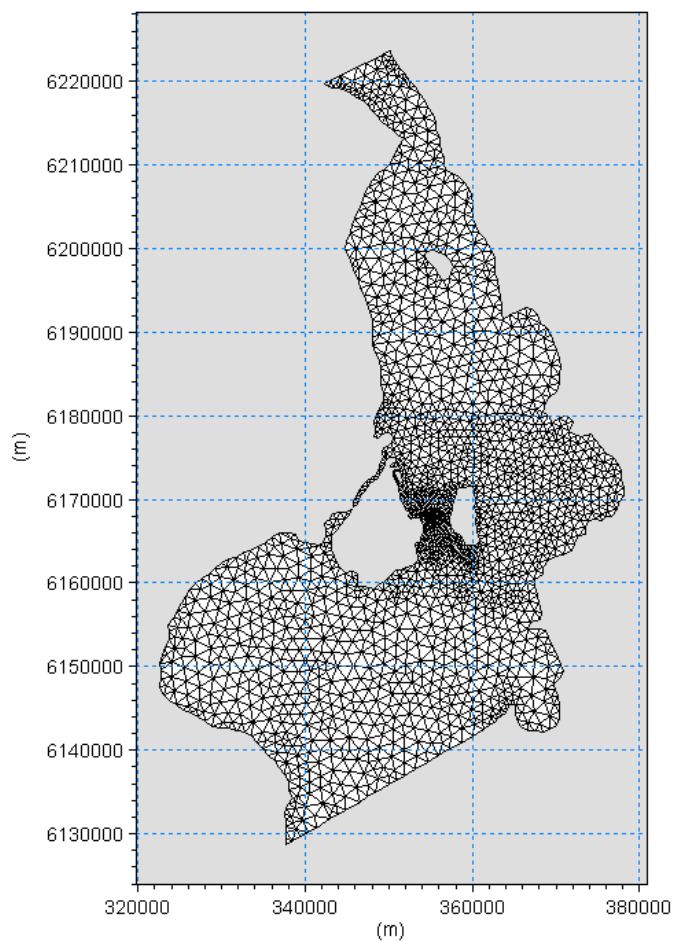
Figure 4.11   Computational mesh

4.4.4   Calibration - HD simulation

The hydrodynamic model is calibrated by comparing to measurements inside the domain. The simulated and measured water level and current speed is shown in Figure 4.12.
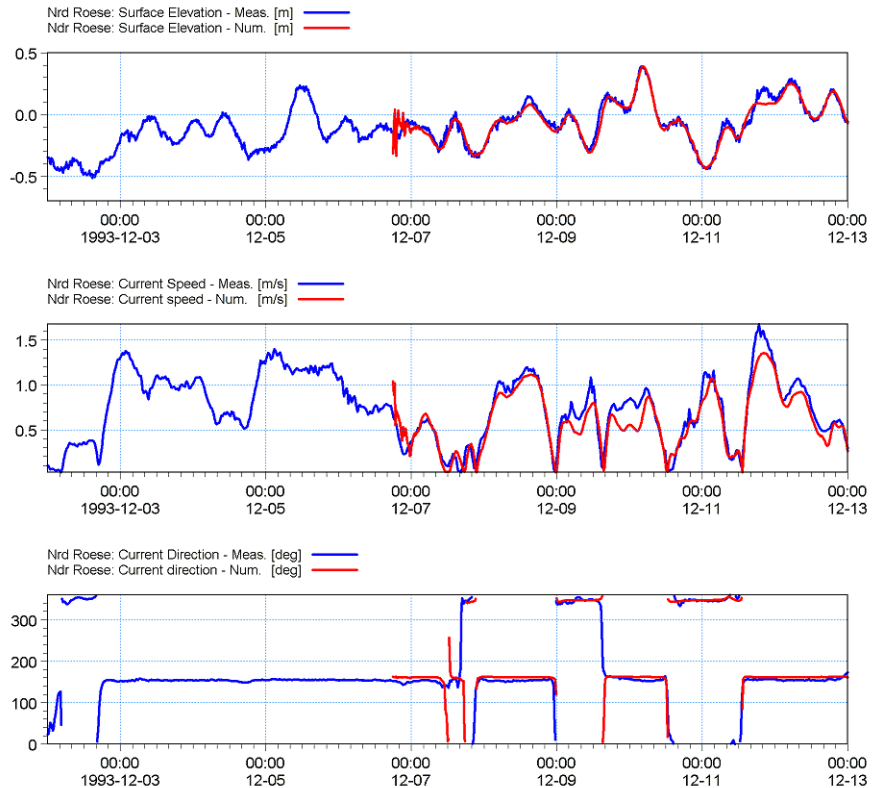
Figure 4.12   Measured values versus simulated values at Ndr. Roese
Upper: Water level
Middle: Current speed
Lower: Current direction

4.4.5   Parameters - PT simulation

The file **Setup2_Oresound_PT_using_HD_Decoupled.m21fm** has been created using the output specification file from the HD simulation, **Setup1b_Oresound_HD_decoupled.m21fm,** and enabling the 'Particle Tracking' module in the Module Selection.

The main settings defining the particle tracking problem are:

- A single class 'My 1st class' is set with default settings

- The source spill of this class is simulated for one constant source called 'My 1st source' located at E: 359250 N: 6180000 (see Figure 4.10). The flux at 'My 1st source' for 'My 1st class' is set to 10 kg/s and 5 particles are released per time step.

- Decay, Settling and Erosion are not included for the single class 'My 1st class'

- The horizontal dispersion of 'My 1st class' is set using a dispersion coefficient formulation with a constant value of 3 m^2/s whereas no dispersion is set with respect to the vertical direction.

- The Drift profile is set to 'Use raw data from hydrodynamics'

- The first output '2D area' is an Area series where basic parameters ('My 1st class' concentration and relevant HD parameters) are saved every hour (frequency = 12) in a 2D dfsu file called 'out_PT_2D_Area.dfsu'. This file can be loaded into the 'Data Viewer' at the end of the simulation (see Figure 4.13)

- The second output 'Tracks' is specific to the Particle Tracking module (Field type is set to 'Particle track' in the Output specification) and will contain the tracks of every 50th particle at each time step in a xml file called 'out_PT_Particles_Tracks.xml'.
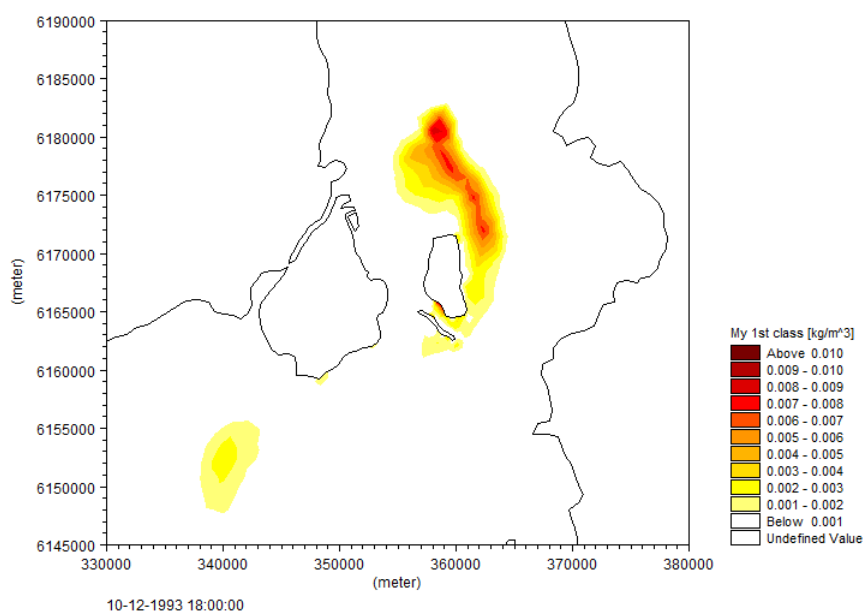
Figure 4.13   Mean concentration of suspended sediment 4 days after the start of simulation.

4.4.6   Visualising the particle tracks

The individual particle tracks saved in the second output file can be plotted over the results of the first file within the Data Viewer using the 'Add/Remove layers' option in the Data menu (or by right clicking and selecting 'Add/remove Layers' in the contextual menu), see Figure 4.14 to Figure 4.16.
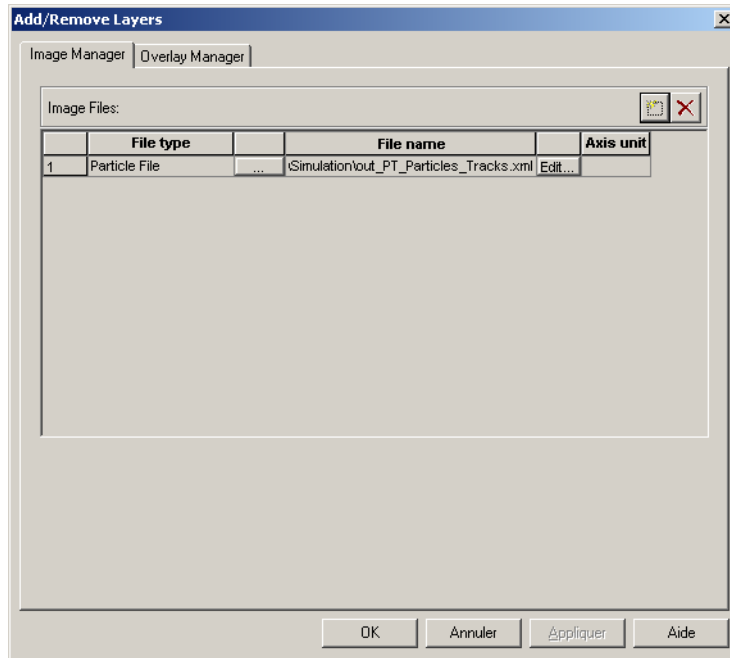
Figure 4.14   Add/Remove Layers - Selecting the Particle File.
Please use the 'Edit…' button to modify the particle styles setting
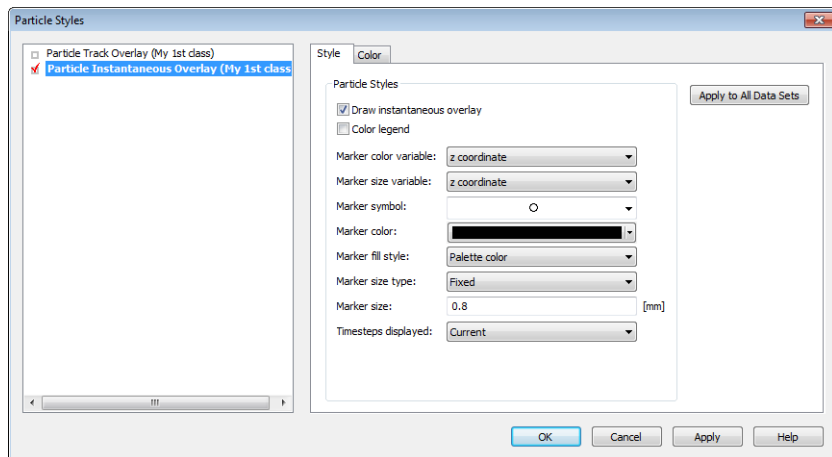
Figure 4.15   Particle Styles - Defining the style for the Particle Instantaneous Over-
lay
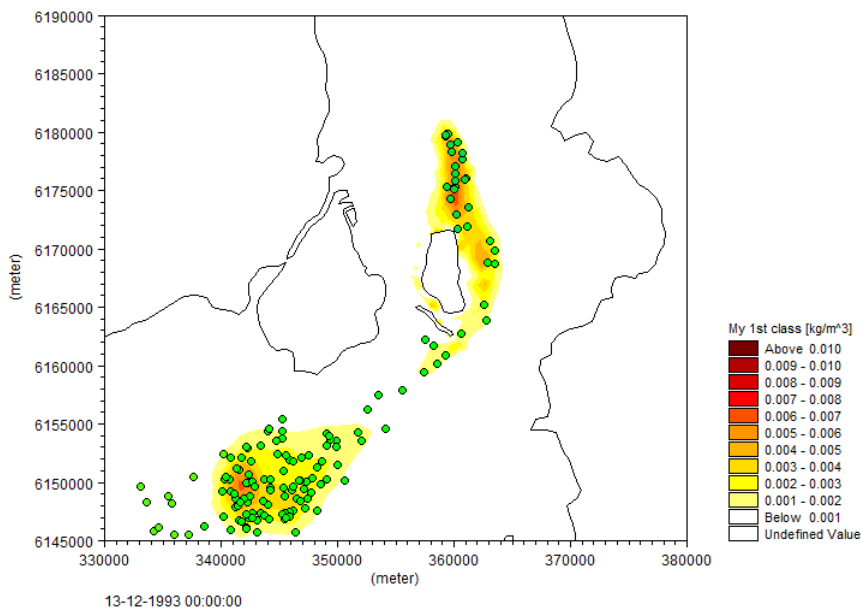
Figure 4.16   Particles and concentration at end of the simulation

4.4.7   Files

The following data files (included in the PT\Oresund folder) are supplied with MIKE 21 Flow Model FM, PT module:

Folder: Data\Boundary Conditions
File name: WaterLevel_South.dfs1
Description: Waterl level variation at south boundary

File name: WaterLevel_South.dfs1
Description: Water level variation at south boundary

Folder: Data\Domain
File name: Oresound.mesh
Description: Mesh file including the mesh and bathymetry

Folder: Data\Image Layers
File name: Map_Oresound.gif/gifw
Description: Rectified Image of the Oresound region

Folder: Data\Initial Conditions
File name: Hot_start.dfsu
Description: hot start file for simulation

Folder: Data\Measurements
File name: WaterLevel_Ndr_Roese.dfs0
Description: Measured water level at Ndr. Roese

File name: Currents_Ndr_Roese.dfs0
Description: Measured currents at Ndr. Roese

Folder: Data\Sources
File name: Source1_Discharge.dfs0
Description: Discharge volume from source

File name: Source1_Location.dfs0
Description: Location of source in domain

Folder: Data\Wind
File name: Wind_Kastrup.dfs0
Description: Wind speed and direction for area

Folder: Simulation
File name: Setup1_Oresound_Decoupling_HD.m21fm
Description: MIKE 21 Flow Model FM specification file, first run

File name: Setup2_Oresound_PT_using_HD_Decoupled.m21fm
Description: MIKE 21 Flow Model FM specification file, second run

4.5   Dredging spill in Great Belt

The objective of this example is to setup a MIKE 21 Flow Model FM, Particle Tracking module, for the Great Belt area, Denmark, from scratch and to calibrate the model to a satisfactory level.

An easy to follow Step-by-step guide is available from MIKE 21 Documentation Index:

MIKE 21 Flow Model FM, Particle Tracking Module,
Step-by-step Training Guide

5 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.

5.1 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.

5.1.1 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.

5.2 Sources

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

By switching property page you can see a Geographic View or a List View of the sources. New sources can be added to both these views.

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.

In the Geographic View it is also possible to create a new source by selecting "Add new..." from the context menu. The geographcal position of the source is set to the mouse pointer coordinates and depending on the source type the source location can then be edited by dragging the source symbols on the map. The additional information for the source is made subsequently.

5.2.1 Particle source specification

Two general types of source layouts can be specified:

- Point source
- Area source

Each of the source layouts has several sub-types. In general point sources specify a single, defined point in space whereas area sources describe a spatial domain where particles are released. 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)

An Area source may further spawn over a range in the water column. In this case a vertical layer thickness has to be given. The vertical position of a released particle is then randomly selected within the defined layer extending from the reference point plus the layer thickness. All data on vertical position refer to the length unit. If the source definition causes a particle to be released above surface/below bed level, the position is set to the surface/bed respectively.

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.

Area source

The location of an area source must be defined using one of two sub-types

- Point collection
- Contour/Shape

An area source describes either a collection of individual (fixed) points or a geometric shape. A point collection can be an irregular placed group of individual points or a regular grid within a defined rectangle. When declaring a contour/shape source particles are released in the space covered by a geometric figure. In all cases the specifications of the released particles are valid for any location within the source.

Particle casting

Particles are placed within the area covered by the area source definition. It may be that the described area includes land elements (e.g. a small island, land borders etc.). In this case a particle potentially can be placed outside the domain. If this happens the particle may be re-casted to find a valid location in the domain a given number of times. If this finally does not succeed the particle may be ignored (skip to the next particle) or the simulation will be stopped with an error. Note that the amount of released substances will probably not match your expectations if particles are ignored as the particle associated items will not enter the simulation!

Point collection

Point collections describe a number of individual, fixed source locations in the horizontal plane belonging to the same source. Such a collection is defined by one of two area definitions

- Regular grid
  (points located at the cell centers in a given rectangular grid)
- Irregular point collection
  (irregular placed, individual points)

The advantage over individual fixed Point sources is that the item definitions refer all release points.

Particle placing

Depending on the grid type the release locations will be used sequentially (row by column, column by row) or randomly permuted. Note that all positions inside a point collection have to be used before a particle can be released at the same position again, e.g. if there are 15 points in total and 15 particle per time step are released, every location will receive one particle. Thus even if random filling is selected all release locations will be used before a new permutation is selected.

Regular grid

In case of a regular grid the anchor point describes the lower left corner of the rectangle covering the source area. The source rectangle extends a given width along the X-axis and a height along the Y-axis and may be rotated around the anchor point. The area is divided into a regular grid of N x M columns and rows. The actual particle release positions are located at the crossing/centres of the columns and rows.

Irregular point collection

An irregular grid consists of a number of individual, irregular placed points in the horizontal plane.

Contour/Shape

A contour/shape source is defined by a set of points in the horizontal plane and a number of elements made of these points to describe the actual release area. Every release element must be characterised by three points. Depending on the selected points the release location will be a single location (all points of the element are the same), a line (2 points are the same) or a triangle (three different points). The final release location is a random point on a line element or inside a triangle element. Both the points and elements have to be created manually and the first point must be placed inside the domain (water point). Subsequent points may be placed outside water elements.

Particle placing

Normally particle placing is based on the total area. In this way particles cover the total source area (covered by all elements) uniformly. Please note that by definition elements describing single locations (all points in the element are the same) or elements describing a line (two points are the same) do not have an area. If particles are placed based on the total area then such elements will not receive any particles! In this case the particles should be placed based on the sub-element count. By this every sub element has the same probability to receive a particle.

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.

5.2.2 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. 33)* 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.

To create a uniform particle distribution covering the entire setup domain you need to define a contour/area source covering the complete domain. This can often be done by declaring the 4 corner points and two triangle elements (points (1, 2, 3) and (1, 3, 4)), but ensure that the first point is in the water. Set the number of recast depending on the proportion of covered land elements, start with 10 and distribute the particles according to the total area. You might want to ignore particles finally placed outside the domain but this may change the initial released mass.

5.3 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.

5.3.1 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

$$\frac{dm}{dt} = -k \cdot m \qquad (5.1)$$

or

$$m = m_0 \cdot e^{-kt} \qquad (5.2)$$

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. (5.1) 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).

5.3.2 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.

5.3.3 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.

5.4 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.

5.4.1 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:

$$w_s = \frac{(\rho_s - \rho)gd^2}{18 \cdot \nu} \qquad (5.3)$$

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.

$$w_s = w_0 \cdot factor_{floc} \tag{5.4}$$

$$C_{total} \le C_{floc} \qquad : factor_{floc} = 1$$

$$C_{floc} < C_{total} < C_{hindered} : factor_{floc} = 1 + \alpha\left(\frac{C_{total}}{C_{floc}} - 1\right) \tag{5.5}$$

$$C_{total} \ge C_{hindered} \qquad : factor_{floc} = 1 + \alpha\left(\frac{C_{hindered}}{C_{floc}} - 1\right)$$

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.

$$w_s = w_0 \cdot factor_{floc} \tag{5.6}$$

$$C_{total} \le C_{floc} : factor_{floc} = 1$$

$$C_{total} > C_{floc} : factor_{floc} = 1 + \alpha\left(\frac{C_{total}}{C_{floc}} - 1\right) \cdot \left(1 - \min\left(1, \frac{C_{total}}{C_{gel}}\right)\right)^5 \tag{5.7}$$

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.

$$w_{s,\,sali} = w_s(1 - C_1 \cdot e^{SC_2}) \tag{5.8}$$

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

5.4.2 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.

5.5   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.

5.5.1 Dispersion specification

The dispersion can be formulated three different ways:

- No dispersion
- Dispersion coefficient formulation
- Scaled eddy viscosity formulation
  (not available for vertical dispersion in 2D model)

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

Using the scaled eddy viscosity formulation the dispersion coefficient is calculated as the eddy viscosity used in solution of the flow equations multiplied by a scaling factor. For specification of the eddy viscosity see Eddy Viscosity in the manual for the Hydrodynamic module. The scaling factor can be specified as a constant factor.

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 unstructured data file (dfsu) or a 2D grid file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, a piecewise constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data.

Selecting Scaled eddy viscosity option the format of the scaling factor can be specified as

- Constant
- Varying in domain

For the case with values varying in domain you have to prepare a data file containing the scaling factor before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D data grid file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, a piecewice constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data.

5.5.2 Recommended values

When more sophisticated eddy viscosity models are used, as the Smagorinsky model, the scaled eddy formulation should be used.

The scaling factor can be estimated by $1/\sigma_T$, where σ_T is the Prandtl number. The default value for the Prandtl number is 0.9 corresponding to a scaling factor of 1.1.

The dispersion coefficient is usually one of the key calibration parameters for the 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).

5.6 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.

5.6.1 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.

5.6.2 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 unstructured data file (dfsu) or a 2D grid data file (dfs2).

The area in the data file must cover the model area. If a dfsu-file is used, a piecewice constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data.

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

5.7 Drift Profile

5.7.1 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. For MIKE 21 FM for instance it is depth averages values that are the output of the hydrodynamic setup, and even for MIKE 3 FM the drift conditions near the bed are normally not resolved adequately for describing sediment particles. 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). For MIKE 21 FM the bed shear profile will be applied in the whole water column.

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. In MIKE 3 FM this effect can be included in the hydrodynamic output, but the depth averaged MIKE 21 FM is not able to do that. So if this flow regime should be described in MIKE 21 FM PT, the wind induced profile must be applied, that will distribute the depth averaged flow in the water column.

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

5.7.2 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
- Use bed shear profile, wind induced profile and surface wind acceleration
- 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)

$$u = \frac{2{,}3}{k} \cdot u^* \cdot \log_{10} \frac{x}{z_o} \tag{5.9}$$

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 trans-

ferred 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.

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

(5.10)

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

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

(5.11)

where

α:	$-0.3 \cdot 10^{-8}$
β:	28° 38'
U_w:	Wind speed [m/s]
γ_w:	Kinematic viscosity [m^2/s]
g:	Gravity [m/s^2]

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

$$c_w(z) = c_w^* \left(1 - 3\frac{z}{h}\right)\left(1 - \frac{z}{h}\right) \tag{5.12}$$

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

$$c_w(z) = c_w^* \exp(-k_0 z) \tag{5.13}$$

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)

5.8 Hydrodynamic conditions

If the Hydrodynamic Module is not included in the simulation you have to pro-
vide the flow information using these settings. The Particle Tracking module
will then run in a so called stand-alone mode, using an external flow field. The
flow information can be read from both unstructured (dfsu) or gridded
(dfs2/dfs3) data files. The data files need to cover the model area and span
over the entire simulation period. The flow information will be interpolated into
the domain mesh, i.e. it can be based on a different (coarser) resolution than
the actual domain mesh. If a dfsu-file is used, a piecewise constant interpola-
tion is used to map the data. If a dfs2-file is used, a bilinear interpolation is
used to map the data

2D setup (Mike21)
For a 2D setup you have to provide a 2D/area data file with the information on
either the surface level or the total water depth and the U- and V-components
of the flow field. If the bathymetry is not constant during the simulation period
(i.e. morphological changes) the file also has to provide the still water depth.

5.9 Temperature

The temperature can be specified as

- Not included
  The model assumes a default value for the whole model domain

- User specified temperature

- From hydrodynamic model
  The data calculated in the hydrodynamic module is used directly
  (This option is not available when running a stand-alone setup)

5.9.1 User specified temperature

With a user specified setting, the data can be:

- 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 information before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, a piecewise constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data

5.10 Salinity

The salinity can be specified as

- Not included
  The model assumes a default value for the whole model domain

- User specified salinity

- From hydrodynamic model
  The data calculated in the hydrodynamic module is used directly
  (This option is not available when running a stand-alone setup)

5.10.1 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 unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, a piece-wise constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data.

5.11 Density

The density can be specified as

- Not included
  The model assumes a default value for the whole model domain

- User specified density

- From hydrodynamic model
  The data calculated in the hydrodynamic module is used directly
  (This option is not available when running a stand-alone setup)

5.11.1 User specified density

With a user specified setting, the data can be:

- 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 information before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, a piecewise constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data

5.12 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

- Bed roughness from hydrodynamic model
- User specified bed roughness

5.12.1 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:

$$k_s = \frac{12h}{10^{(C/18)}} \qquad (5.14)$$

where

h: Water depth
C: Chezy number

Manning number conversion to Nikuradse roughness *ks*:

$$k_s = \left(\frac{25,4}{M}\right)^6 \qquad (5.15)$$

where M is the Manning number.

5.12.2 Bed roughness from hydrodynamic model

The bed resistance is often already specified in the hydrodynamic setup.

In MIKE 21 FM bed resistance can be specified as Manning number, or Chezy number. If bed roughness from hydrodynamic model is selected, the Manning number, or Chezy number will be automatically converted to Nikuradse roughness k_s (see 5.12.1 General description).

5.12.3 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 unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, a piecewice constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data.

5.13 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

- Wind from hydrodynamic model
- User specified wind

5.13.1 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 unstructured data file (dfsu) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, a piece-wice constant interpolation is used to map the data. If a dfs2-file is used, a bilinear interpolation is used to map the data.

Furthermore a soft start interval must be speficied.

5.14 Outputs

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

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

5.14.1 Output specification

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

Field type

For a 2D simulation 2D field 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.

Output format

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

Particle track

For particle track the file format is xml. Variables for each particle will be written in the file: x, y, z, mass, age at each timestep.

The xml file can get very large, and therefore it is possible not to write particle variables for all particles, but only a fraction of the total number of particles, by selecting a frequency higher than 1 (see Particle numbers *(p. 56)*).

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

Field variables

For 2D field variables the following formats can be selected

- Point series. Selected field data in geographical defined points.

- Lines series. Selected field data along geographical defined lines.

- Area series. Selected field data in geographical defined areas.

Table 5.1

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

Output file

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

Particle format

The particle data in the XML file can be saved using uncompressed or compressed format. The different formats 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 5.1   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 5.2   Example of data in XML file using compressed particle format

Treatment of flood and dry

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

- Whole area
- Only wet area
- Only real wet area

Selecting the Only wet area option the output file will contain delete values for land points. The land points are defined as the points where the water depth is less than a drying depth. Selecting the Only real wet area option the output file will contain delete values for points where the water depth is less than the wetting depth. The drying depth and the wetting depth are specified on the Flood and Dry dialog. If flooding and drying is not included, then both the flooding depth and the wetting depth are set to zero.

Time step

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

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.

Point series

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

The geographical coordinates of the points are either taken from the dialog or from a file. The file format is an ascii file with four space-separated items for each point on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. For 2D field data the third item is unused (but must be specified). The last item (the remaining of the line) is the name specification for each point.

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

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

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

Line series

You must specify the first and the last point on the line and the number of discrete points on the line. The geographical coordinates are taken from the dialog or from a file. The file format is an ascii file with three space-separated items for each of the two points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. For 2D field data the third item is unused (but must be specified). If the file contains information for more than two points (more than two lines) the information for the first two points will be used.

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

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

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

Area series

The discrete field data within a polygon can be selected. The closed region is bounded by a number of line segments. You must specify the coordinates of the vertex points of the polygon. Two successive points are the endpoints of a line that is a side of the polygon. The first and final point is joined by a line segment that closes the polygon. The geographical coordinates of the polygon points are taken from the dialog or from a file. The file format is an ascii file with three space-separated items for each of the two points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. For 2D field data the third item is unused (but must be specified).

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

5.14.2 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 33).

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

3D field variables

3D field variables are only available for 3D simulations.

INDEX