MIKE 21/3 Oil Spill

Oil Spill 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 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 get you started in the use of MIKE 21/3 Oil Spill Model(OS), for applications involving the simulation of transport and fate of oil discharged or accidentally spilled in lakes, estuaries and coastal areas or at the open sea.

1.2 Assumed User Background

Although the OS Model 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 oil spill investigations, 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.

It is assumed that you are familiar with the basic elements of MIKE Zero: file types and file editors, the Plot Composer and the MIKE Zero Toolbox. 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.

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 On-line 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. See Figure 1.1.

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

   Open the On-line Help system by selecting "Help Topics" in the main menu bar.
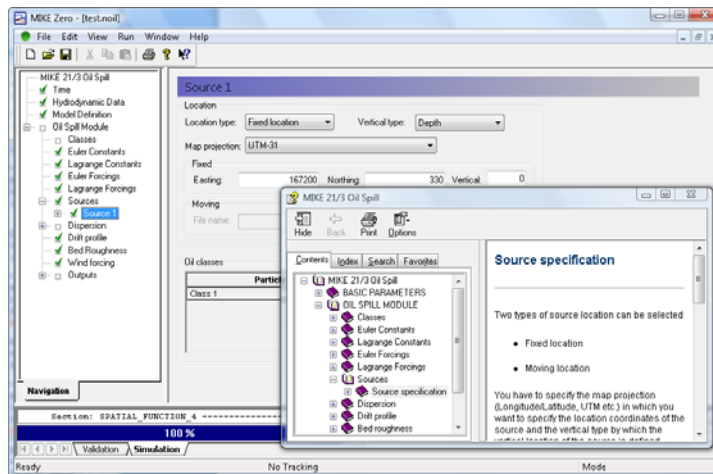
Figure 1.1   On-line Help system for MIKE 21/3 Oil Spill Model

2 Introduction

2.1 General Description

The MIKE 21/3 Oil Spill Model is used for modelling the fate of oil discharged or accidentally spilled in lakes, estuaries, and coastal areas or to the open sea.

The oil is represented as Lagrangian particles being advected with the surrounding water body and exposed to weathering processes. The weathering of the oil is modelled using a template. The default weathering model Oilspill_A3 superseeds older models, however weathering models from previous releases can be used and are available on request or can be copied from an older release.

The following processes can be included in the simulation:

- Spreading
- Evaporation
- Dissolution
- Vertical dispersion
- Settling
- Biodegradation
- Dissolution
- Photooxidation

Other facilities include:

- Moving sources (leaking ships)
- Wind surface acceleration of the oil
- Output as either lists of particles (XML files), or as concentration maps (DFS2 or DFS3 files) with particles interpolated into computational grid as concentrations.

2.1.1 Application areas

The Oil spill Model can be applied to the study of engineering problems such as:

- Risk analysis and accidental spillage

- Environmental Impact Assessment

3   Getting Started

The hydrodynamic basis for the Oil Spill 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 User Guide for the Hydrodynamic Module and the comprehensive step-by-step training guide covering the MIKE 21 Hydrodynamic Model. The user guide and the training guide (PDF-format) can be found in the installation folder:

.\MIKE Zero\Manuals\MIKE_21\FlowModel\HD\M21HD.pdf

.\MIKE Zero\Manuals\MIKE_3\FlowModel\HD\M3HD.pdf

.\MIKE Zero\Manuals\MIKE_21\
FlowModel\HD\MIKE_21_HD_Step_by_Step.pdf

4 Examples

4.1 General

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, if you like, to see what happens if one or other parameter is changed.

To demonstrate the capabilities of the Oil Spill model we have included some examples of simulations. A directory is provided for each example. The directory names are as follows:

- Corner flow (2D & 3D):
  .\Examples\MIKE_21-3_Integrated_Models\OS\Corner
- Oil refinery (2D):
  .\Examples\MIKE_21-3_Integrated_Models\OS\Refinery

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 oil distributed in a in a channel, given the material originates from a single point constant outfall. 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.
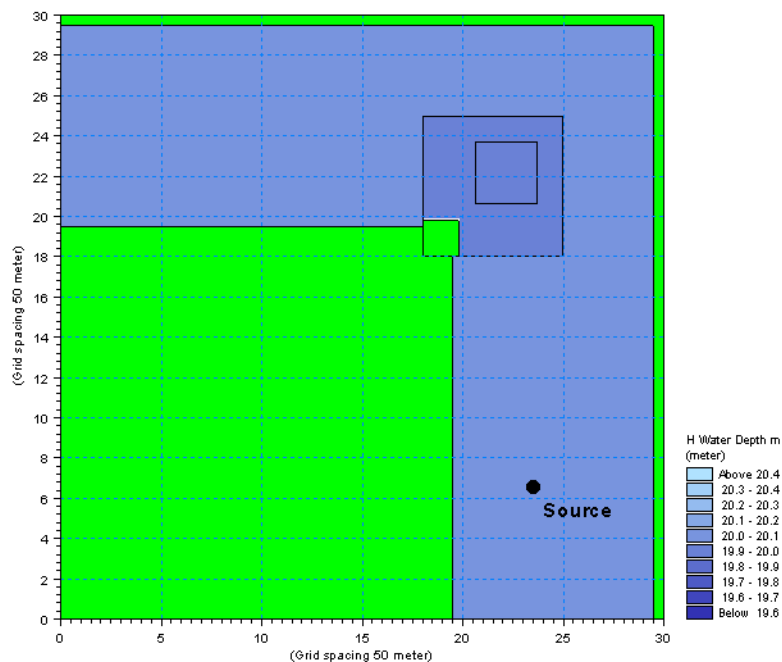
Figure 4.1   Computational domains and bathymetries for area 1, 2 and 3.
Area 1 has a gridspacing of 50 m, Area 2 has a gridspacing of 16.67 m
and finally Area 3 has a gridspacing of 5.58 m

The problem is to determine the fate of the oil spill 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 water temperature is set to 10 deg.C.

- 2D Only: The source is positioned at position (167200, 330) at the water surface in the domain, see Figure 4.1.

- 3D Only: The source is positioned at position (167200, 330) one meter above bed, see Figure 4.1.

- The oil can be described as a type STATFJORD oil, with similar characteristics as described in the Scientific Documentation, chapter 8: The oil consist of 50% volatile fractions, 40% heavy fractions, 8% wax and 2% asphaltene.

- The discharge from the particle source is constantly 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 4.1.
- 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.

Once the hydrodynamic flow field is obtained the Oil Spill module can be setup and executed.

The main conditions defining the oil spill problem are:

- A time step of 15 seconds is selected and the duration of the simulation is 3 hours (720 time steps).
- The oil spill simulation is performed using the OilSpill_A3 model.
- The source location is fixed at position (167200, 330) at the water surface.
- The properties of the source is specified per oil particle: 5 particles are released per time step and each particle contains 30 kg in total, the sum of the four fractions 'volatile', 'heavy', 'wax' and 'asfalthene'.
- A terminal thickness of 0.01 mm is specified to include the spreading process.
- 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 for the heavy fraction is shown in Figure 4.2. Notice the effect the fine nesting does to the interpolated results.
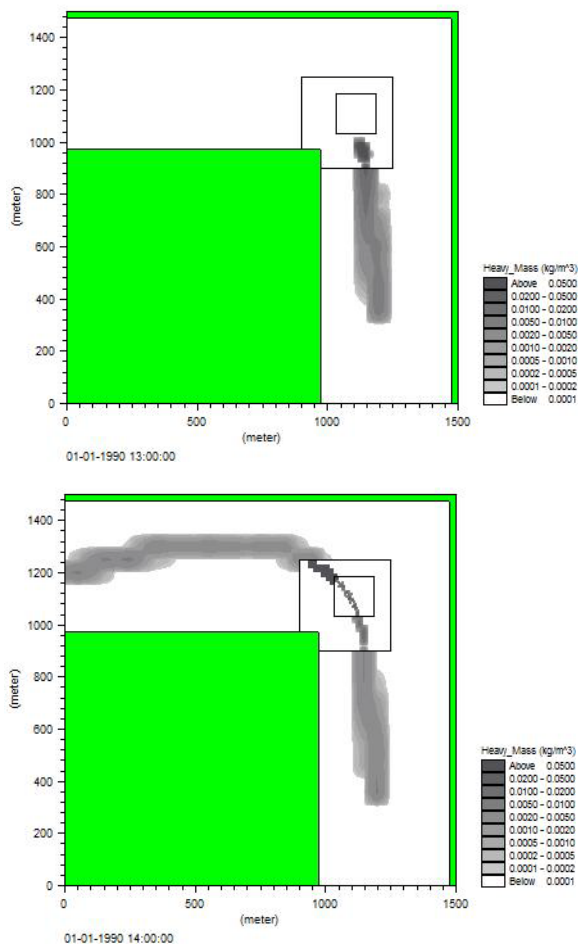
Figure 4.2   Interpolated concentration of total oil mass.
             Upper: After 1 hour.
             Lower: After 2 hours

A plot of the particle track showing the instantaneous location and mass of the volatile oil particles is shown in Figure 4.3. Notice that the volatile fractions of the oil is reduced a distance from the source because of evaporation.
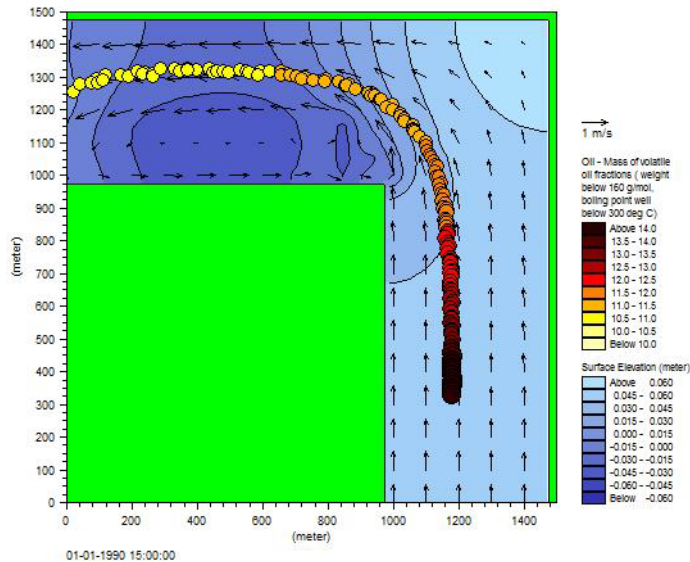
Figure 4.3   Visualization of the particle result file: mass of volatile oil fraction parti-
cles together with flow vectors and surface level

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 4.1 (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 Oil Spill Module can be setup and executed.

The main conditions defining the oil spill are:

- A time step of 15 seconds is selected and the duration of the simulation is 1½ hours (360 time steps).

- The oil spill simulation is performed using the OilSpill_A3 model.

- The source location is a fixed source at position (167200, 330) 1 m above bed.

- The properties of an oil particle in the source is 30 kg/particle. 5 particles are released per time step

- 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.

A plot of the particle track showing the instantaneous location and total mass of the particles is shown in Figure 4.4. It can be seen that the total mass decrease with time (i.e. the distance from the source), which is to be expected due to evaporation of the volatile mass.
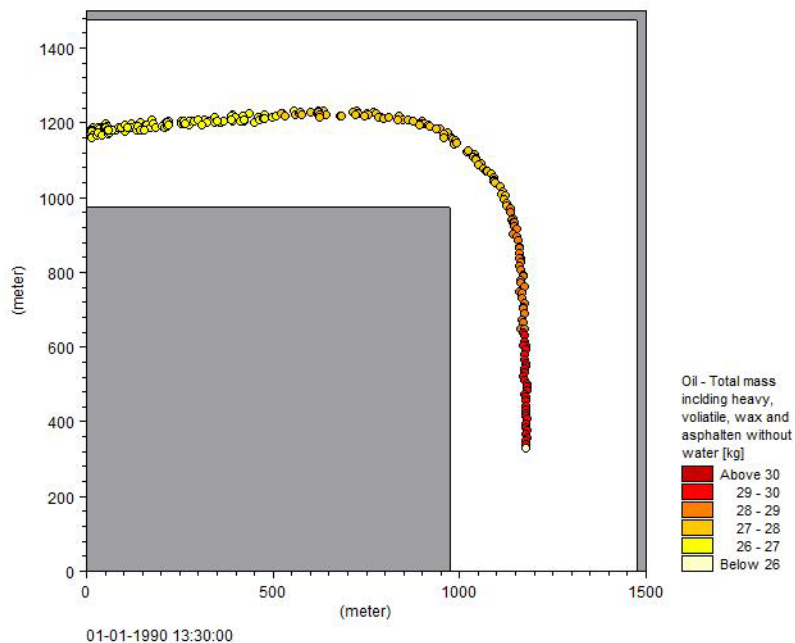
Figure 4.4   Horizontal visualization of oil particle file

Because of buoyancy effects the oil surfaces a distance from the source after the release and only then the evaporation processes starts. This is illustrated in Figure 4.5 that shows the change in time of particle 1.
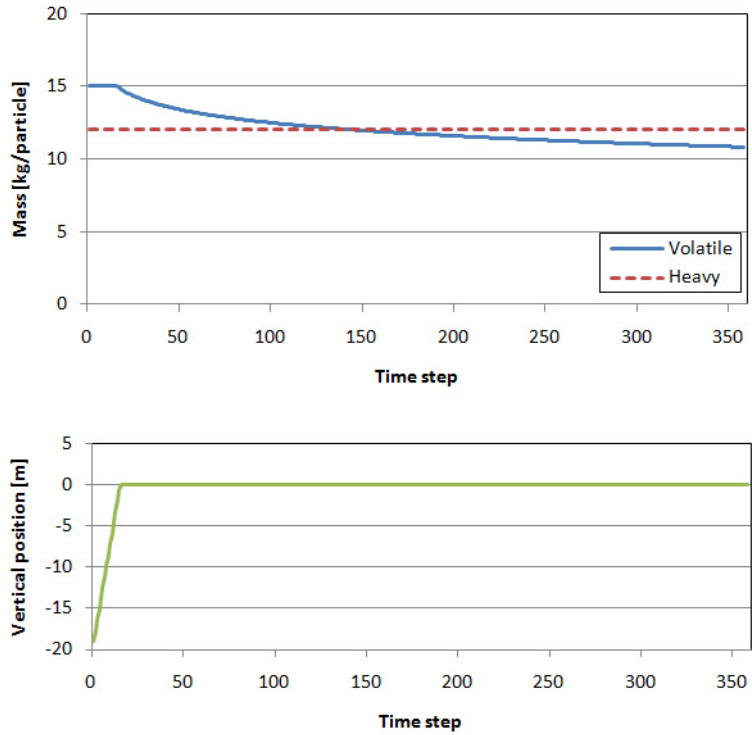
Figure 4.5   Evolution in time of volatile and heavy mass fraction for particle 1.
              Lower figure: position of particle in water column

During the same time the oil will slowly begin to form an emulsion with the water. A plot of the fraction of water in the oil mass at the water surface is shown in Figure 4.6.
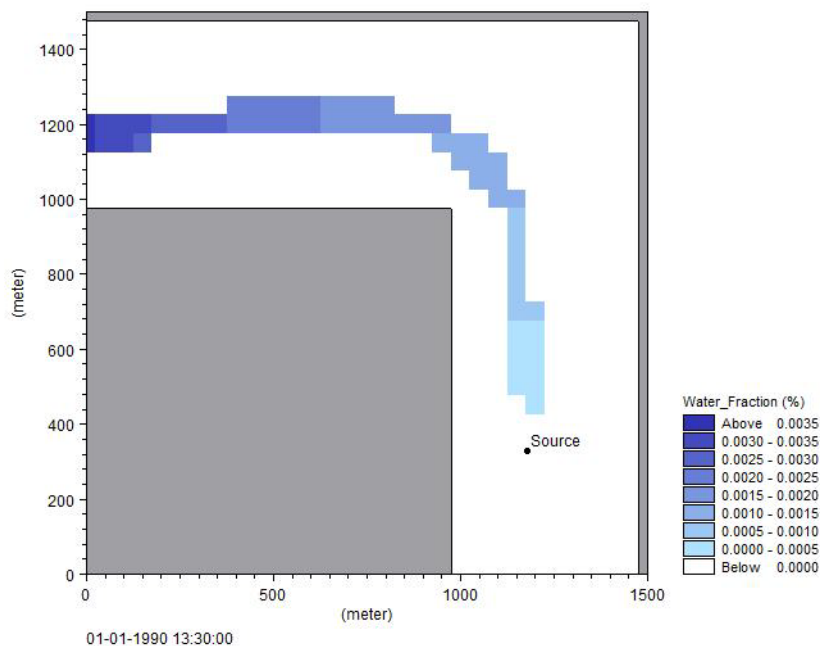
Figure 4.6   Fraction of water in the oil. Information given as concentration map.

4.2.7   Files

The following data files (included in the OS\Corner folder) are supplied with MIKE 21/3 Oil Spill 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: OS2D.noil
Description: MIKE´21/3 OS specification file, 2D simulation

File name: OS3D.noil
Description: MIKE´21/3 OS specification file, 3D simulation

4.3 Oil Refinery Example

4.3.1 Purpose

Prøvestenen is an artificial island that was originally created for the location of a fortress, whose mission was to protect the entrance to Copenhagen. In 1922 the area was converted to store oil and petrol. Today Prøvestenen offer the Copenhagen oil port for the reception of tankers. Prøvestenen harbour has a tank storage capacity of 1,000,000 m^3 for inflammable liquids.

In january 2008 a significant oil spill from Prøvestenen occured where several hundred thousand liters of Vacuum Gas oil ran out into the Sound. The accident happened when there was a hole in the hose to an oil tanker. The oil flowed north past Hellerup, Skovshoved up to Ven. Luckily, due to the low water temperature and the properties of the oil a large part of the oil could be picked up in clumps by environmental ships during the first 24 hours after the spill, but some of the oil drifted to settle on the coast between Helsingborg and Landskrona in Sweden.

With this accident in mind the Oil Spill model is setup to simulate the fate of different types of oil from an accident and to investigate the influence of water temperature on the weathering processes.The matrix of simulations is given in Table 4.1

Table 4.1   Combinations of climatic conditions and oil type

Oil type	Winter	Summer
Heavy fuel	SimA_1	SimA_2
Crude oil	SimB_1	SimB_2
Gasoline	SimC_1	SimC_2

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 in the installation folder:

.\MIKE Zero\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 simulation period for the oil spill is selected to 24 hours from mid-night december 11th to midnight december 12th, during which the flow direction shifts after 12 hours. At the same time the wind shifts from south-westernly wind to northwesternly wind (see Figure 4.7).
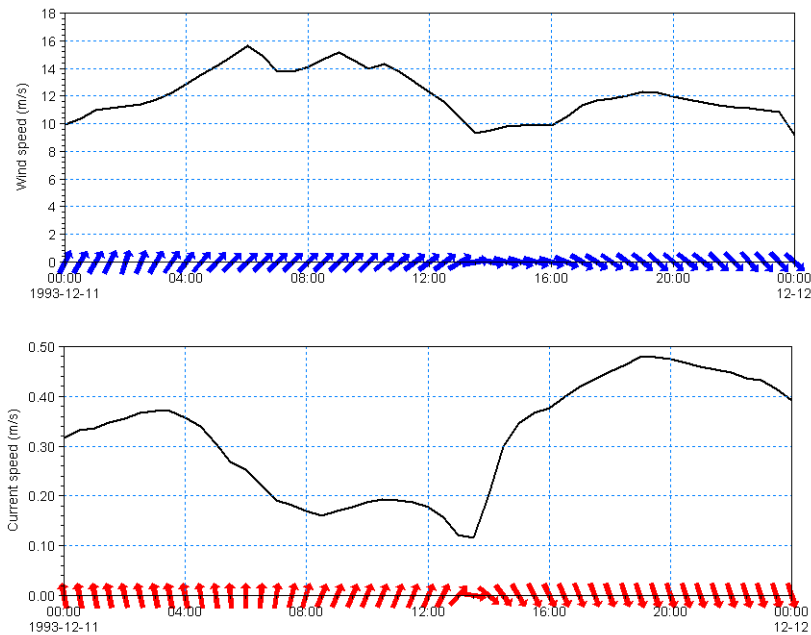
Figure 4.7　Descriptive conditions for oil spill simulation
Upper: Wind speed and direction
Lower: Simulated current speed and direction next to oil refinery

- The time step for the oil spill simulation is set to 300 s.

- The source spill event is simulated by a time-varying source positioned at the location of the oil refinery: 200 m^3 of oil was spilled during a 20 minutes period in the beginning of the simulation.

- Two water temperatures are investigated: winter conditions at 5 degrees Celcius and summer conditions at 15 degrees Celcius.

- Three types of oil are investigated: Heavy fuel oil, Crude oil (Statfjord) and Gasoline. The characteristics of each oil type are derived by the information given in the ESTC Spill Technology Databases[1].

4.3.3   HD simulation

The main condition defining the hydrodynamic problem is:

---

1   http://www.etc-cte.ec.gc.ca/databases/OilProperties/oil_prop_e.html

- The model area is resolved with a structured computational grid with a horizontal gridspacing of 900 m. The grid is shown in Figure 4.8.

- An overall time step of 300 seconds is used and the duration of the HD 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

- The output data can be limited to the period from 11/12/1993 00:00:00 to 12/12/1993 00:00:00 considering the OS model only, however saving the total period will be necessary for calibration of HD.
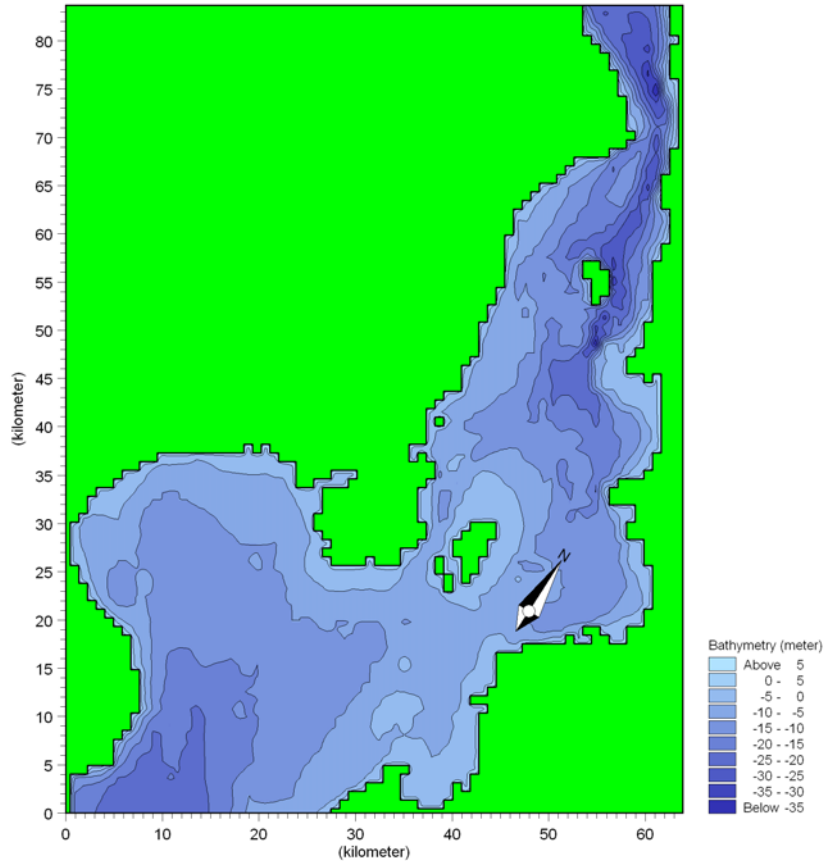
Figure 4.8   Model area, the sound between Denmark and Sweden

4.3.4   Parameters - PT simulation

The main settings defining the oil spill problem are:

- The model area is defined by the hydrodynamic result file.
- An overall time step of 300 seconds is used and the duration of the OS simulation is 24 hours starting at 11/12/1993 00:00:00, see Figure 4.7.
- The processes for the oil spill are descibed by the Oilspill_A3, DHI Oil Weathering model, template.

- The derived oil properties for the different oil types are shown in Table 4.2 below..

Table 4.2   Oil properties for the three oil types in the simulations

	Heavy Fuel oil	Crude oil	Gasoline
Distribution in (%):			
Volatile oil	25	50	100
Heavy oil	47	40	0
Wax	2	8	0
Asphaltene	26	2	0
Distillation percentage at 180 deg.C	1	26	95
1st oil specific constant	-0.12	2.67	13.2
2nd oil specific constant	0.013	0.06	0.21
Simple evaporation type	1 (quad)	0 (log)	0 (log)
Density (kg/m^3)			
Average	1099	831	753
Volatile	755	755	753
Heavy	1094	907	-
Dynamic Viscosity (cP)	13460	6	1
Viscosity coefficient temperature dependency	-0.136	-0.111	0

- Other processes are defined by default values.

- The dynamic viscosity values are given at reference temperature 15 degrees Celcius.

- The source spill of this class is simulated for an event located at (E: 351600 N: 6173500) The position of the source is shown in Figure 4.9.
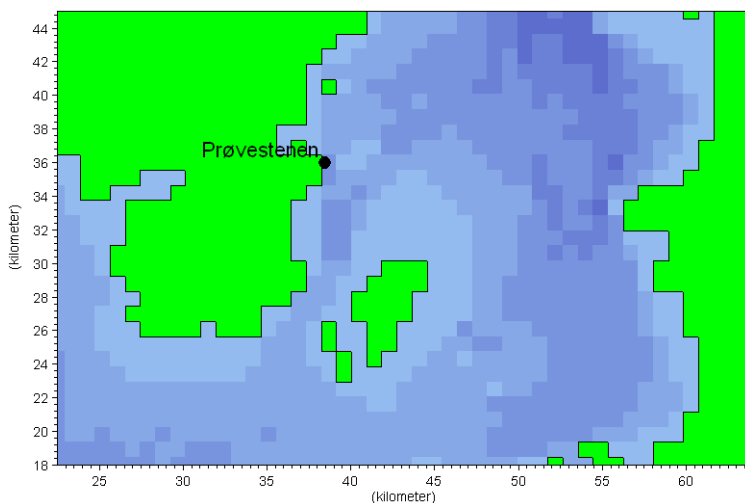
Figure 4.9   Location of Prøvenstenen shown in detail of model area

● The 200 m³ of released oil corresponds to a total mass of each oil component as given in Table 4.3...

Table 4.3   Total mass for each oil component in the simulations

	Heavy Fuel oil	Crude oil	Gasoline
Distribution in (kg):			
Volatile oil	37750	75500	150600
Heavy oil	102836	72560	0
Wax	4376	14512	0
Asphaltene	56888	3628	0

● 60 particles are released per time step, corresponding to 25% of the oil particles. The mass values are interpreted as flux.

● The horizontal dispersion is included using a constant value in the dispersion coefficient formulation of 2.0.

● Vertical dispersion is not included

● The drift profile is defined by Surface Wind Acceleration with a wind weight of 0.02 and a wind drift angle of 15 degrees.

● The wind forcing is defined as in the HD simulation

● The influence of waves is omitted from the calculation

● The first output file is a dfs2 file containing the state variables. The dfs2 file is saved every 3 time step (30 minutes).

● The second output file is a Particle file (xml file) and will contain every 6th particle saved every 3 time step

4.3.5   Results

The spill is released during the first 20 minutes of the simulation (4 time steps). The currents combined with the drag force from the westernly wind at the water surface may transport the oil away from the coast and out into Oresund in the beginning of the simulation. When the wind and current change direction the oil slick flows towards south. Figure 4.10 to Figure 4.12 shows the resulting particle tracks from the three simulations.

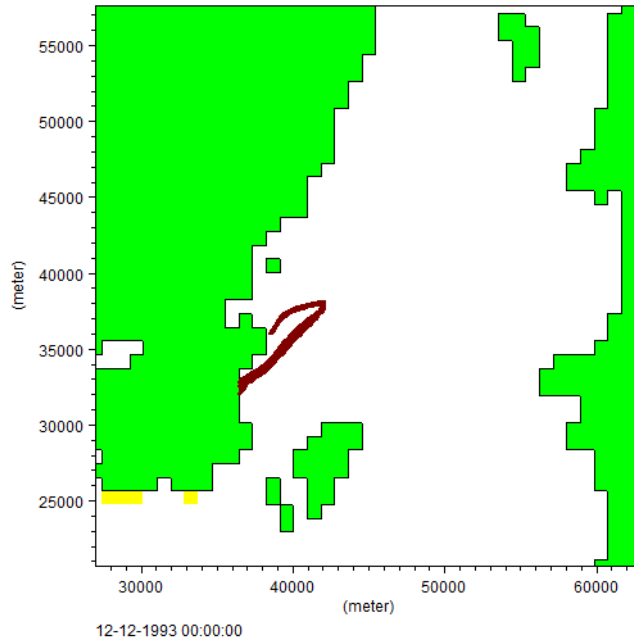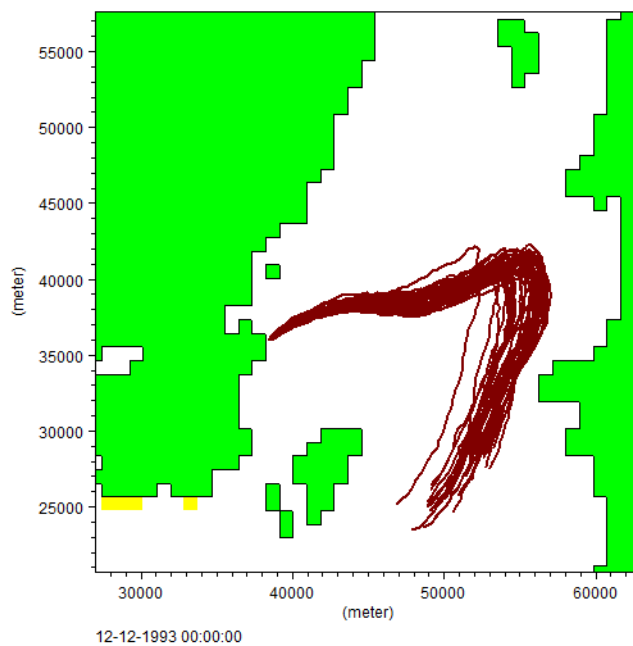Figure 4.10   Particle tracks during simulation, Heavy Fuel oil type

12-12-1993 00:00:00

Figure 4.11   Particle tracks during simulation, Crude oil type

12-12-1993 00:00:00
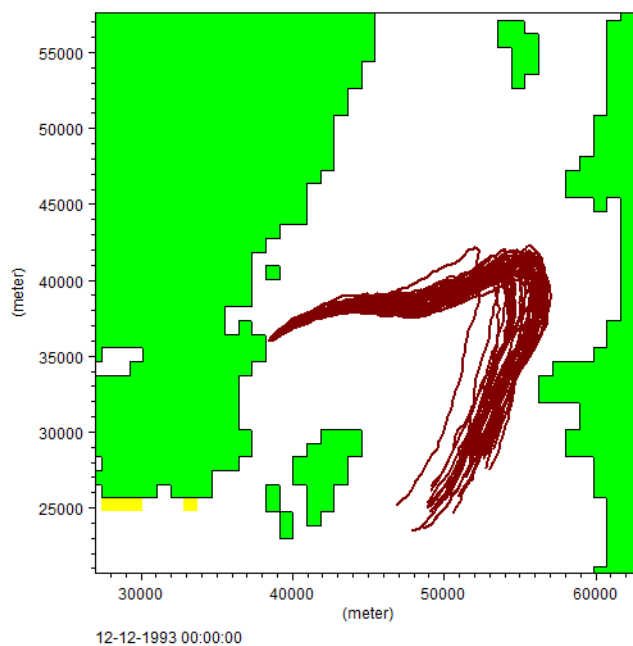
Figure 4.12   Particle tracks during simulation, Gasoline oil type

As seen from the three figures above the track is not the same for all the oil types. Some of the oil particles reach the bottom during the 24 hour simulation while others stay out into the water. This is because of the different composition of the oil types and the related difference in the weathering processes.The horisontal dispersion adds to the spreading of the results.

The evaporation of volatile material typically takes place during the first hours of an oil spill. Figure 4.13 shows the mass of the volatile component of particle number 61 during the simulation for the three oil types.
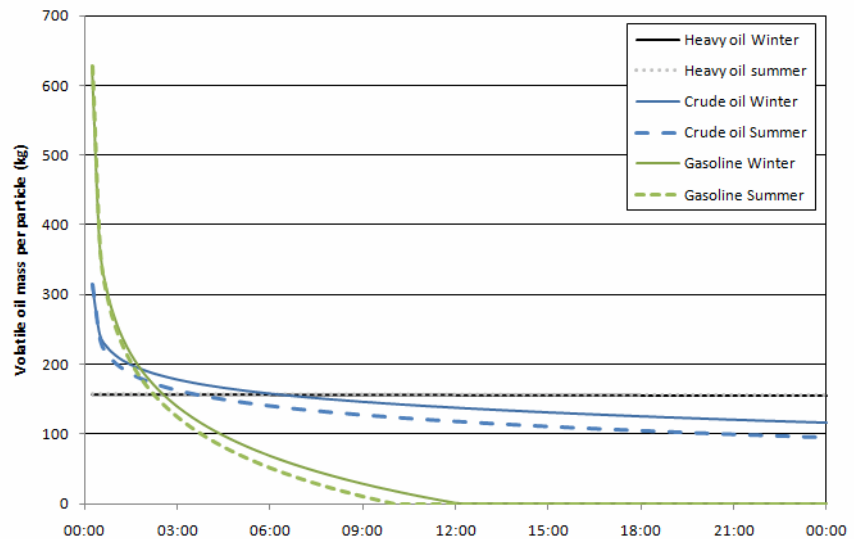
Figure 4.13   Mass of volatile component of oil particle no. 61 during simulation.

As seen from Figure 4.13 the volatile component of the oil quickly decrease during the first part of the simulation for the lighter oils. The oil evaporates more quickly in the summer than in the winter due to the higher temperature. The volatile component in Gasoline evaporates entirely during the simulation, whereas the evaporation rate for the volatile component of the Crude oil seem to decrease slightly after the first hour. This behavior can be explained by weathering processes: as the oil density increase due to evaporation the oil becomes more viscous and evaporation is slower. The settling velocity changes and subsequently influence the movement of the particles. In the case of heavy oil the evaporation is very slow but this also relates to the fact that the heavy oil settles on the bottom very quickly, see Figure 4.14 and Figure 4.15.
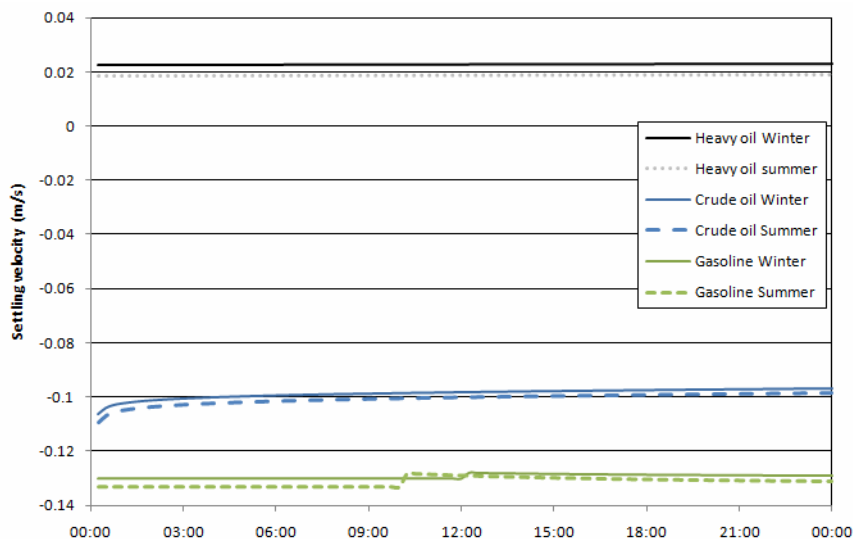
Figure 4.14   Settling velocity of oil particle number 61 during simulation.
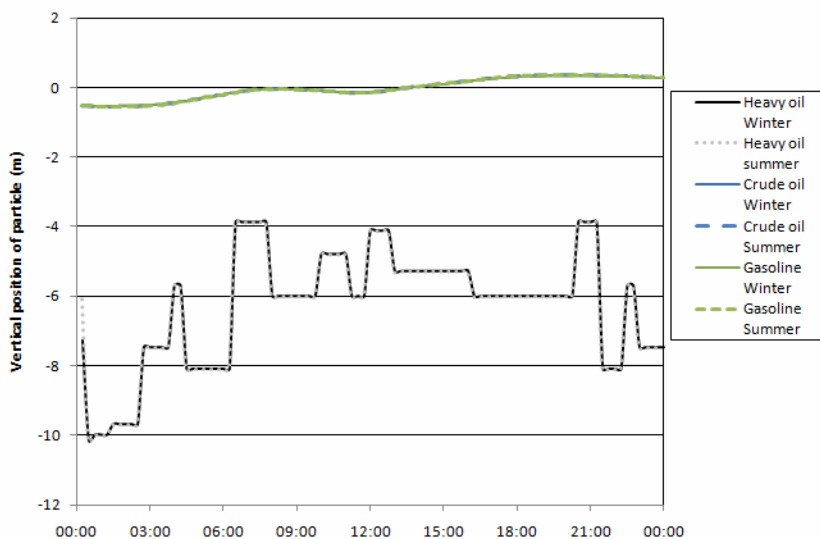
Figure 4.15   Vertical position of oil particle number 61 during simulation.

The lighter oil types remain buoyant during the simulation. As seen by Figure 4.15 the oil particles are released and remain at the water surface. Oppositely the heavy oil type sink quickly towards the bed and after reaching the bed the particles are moved along the bed in the remaining part of the simulation by the currents alone. When the oil particles are not at the water surface the evaporation process is hindered and hence the volatile component of the oil particle remains constant, as seen in Figure 4.13.

4.3.6   Files

The following data files (included in the OS\Refinery folder) are supplied with MIKE 21/3 Oil Spill Module:

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

Filename: SimA_1.noil, SimA_2.noil
Description: specification file contains the OS setup for Heavy oil, winter conditions and summer conditions, respectively.

Filename: SimB_1.noil, SimB_2.noil
Description: specification file contains the OS setup for Crude oil (type Statfjord), winter conditions and summer conditions, respectively.

Filename: SimC_1.noil, SimC_2.noil
Description: specification file contains the OS setup for Gasoline, winter conditions and summer conditions, respectively.

Filename: OilSource.dfs0
Description: Contains state variables for oil source

Filename: bathymetry.dfs2
Description: Contains the bathymetry

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

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.

5.3 Model Definition

Here you must specify the applied weathering model to use for the oil spill calculations. In Release 2017 you can select one of the following templates:

- Default weathering model Oilspill_A3
- Simplified Particle Assessment Model

For a detailed description of the models you should refer to the scientific documentation for the Oil Spill templates.

In previous releases other weathering models may have been included as well. All previous weathering models are available on request or can be copied from an older release.

Note: The scientific documentation for the two Oil Spill templates is located in the folder .\DHI\2017\MIKE Zero\Manuals\MIKE_Zero\ECOLab_OilSpill in the files DHI_OilSpill_Model.pdf and Simple_Particle_Assessment.pdf, respectively.

5.3.1 Oilspill_A3

This is the current default oil spill weathering model. The movement of the oil is mainly driven by the hydrodynamics, gravitational spreading, and the wind. This model splits the oil into 2 fractions: light and heavy, including wax and asphaltene components. The weathering processes included are::

- Spreading
- Buoyancy
- Biodegradation
- Photooxidation
- Evaporation
- Vertical dispersion
- Emulsification
- Dissolution

The release 2017 weathering model "Oilspill_A3" is a slightly extended version of the previous Oilspill_A2. No general weathering processes have been changed. In comparison to the previous Oilspill_A2 some item units have been adopted (e.g. terminal thickness "hterm") and an additional layer output has been added. Further, oil particles below a minimum mass threshold can be removed from the simulation automatically.

6 OIL SPILL MODULE

The oil spill module calculates the fate of oil spills in lakes, estuaries, and coastal areas, or at the open sea. The number and type of input parameters is determined by the Model definition.

To see a description of the different model definitions, please refer to the scientific documentation for the Oil Spill Model.

6.1 Classes

The Classes dialog shows a summary of the state variables defined in the selected model definition. Each class will have a number of internal state variables

The Description and Unit of each state variable are given.

6.1.1 General Description

Each oil particle can contain several states, referred to as state variables. So you can say that the state variables in the OS Model represent the particles internal states, and they are also the variables that the user wants to predict the state of. So they are the main result of the OS model. Typical state variables in the OS Model would be the masses of different oil fractions.

6.2 Constants

The Euler constants are defined as input parameters in the model definition and they have an Eulerian representation.

6.2.1 General description

Euler constants are constant in time but can vary in space. Usually they are used in the model definition to describe physical constants, or variables in the surrounding environment of the oil particles. An example could be density of water or light extinction coefficient..

6.2.2 Remarks and hints

Some parameters can be found in litterature, and others must be measured.

6.3 Class Constants

The class constants are defined as input parameters in the model definition, and they have a Lagrangian representation.

6.3.1 General description

Class constants are constant in time, and are parameters internally in the particles. Usually they are used to specify process rates, halfsaturation concentrations and other input parameters for the equations describing the processes in the particles.

6.3.2 Remarks and hints

Some coefficients and transformation rates can be found in the litterature, but some have not been documented in litterature. So this possible lack of information is wise to investigate, before a project is initiated.

6.4 Forcings

The Forcings are defined as any input parameter in the Oil Spill model, which has an Eulerian representation and can be varying in time. The Forcings are essentially divided into two groups:

- Built-in Forcings
- User-specified Forcings.

The built-in Forcings are automatically provided by the model system during execution as they have already been specified and calculated in the MIKE system.

The user-specified Forcings have to be specified in the present Dialog. Depending on the spatial variation of the Forcing, as defined in the model definition, it can be specified as a "Constant value", a "Type 0 data file" or a "Type 2 data file". The data files have to cover the entire simulation period and in case a type 2 file is allowed for, it must cover the spatial extent and coordinates of the bathymetry file.

6.4.1 General description

Forcings are used as arguments in the mathematical expressions of processes in the model definition. They can vary in time and space. They represent variables of an external nature that affect the oil slick. Typical examples of euler forcings are current conditions, solar radiation and wind.

6.5 Class Forcings

The Class Forcings are defined as input parameters in the model definition, which has a Lagrange representation, e.g. they are timevariables inside the oil. They represent variables of an internal nature within the oil that is driving

the processes in the oil. A typical example of a Class Forcing could be oil temperature.

The Class Forcings can be specified as a fixed value or as a timeseries that will be applied all the particles.

6.6 Particle Sources

The effects of oil spill from an outlet or a moving ship is included in the simulation using Particle Sources.

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

6.6.1 Source specification

Two types of source location can be selected

- Fixed location
- Moving location

You have to specify the map projection (Longitude/Latitude, UTM etc.) in which you want to specify the location coordinates of the source and the vertical type by which the vertical location of the source is defined. Three definitions are available for the vertical type

- Depth (positive downwards from the moving water surface)
- Above bed (positive upwards relative to the sea bed)
- Datum (positive downwards relative to the water level 0.0)

For the Fixed location source option you must specify the horizontal coordinates of the source point and its vertical position.

For the Moving location source option 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.

You may choose to include or exclude the source in the simulation.

Data

A source may include one or many different types of oil components. In the present oil spill model an oil particle is defined by the mass of volatile components, the mass of heavy components, the mass of wax and the mass of asphaltene as well as information about the water fraction, buoyancy and active particle area.

The discharge rate of oil released into the environment is defined per overall time step and can be interpreted in one of two ways:

- Interpreted as mass per particle
  The released mass is defined by the product of the given value (i.e. mass of a single oil particle) and the number of particles released into the flow per overall time step.

- Interpreted as flux
  The released mass is the given value

The format of the oil component information within the source can be specified as

- Constant in time
- Varying in time

For the case with the oil component source information varying in time you have to prepare a data file containing the oil component information. 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.

NOTE: A particle with all state variables defined as 0 will be omitted from the calculations.

6.7 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.

6.7.1 Horizontal dispersion

The horizontal dispersion can be formulated two different ways

- No dispersion
- Dispersion coefficient formulation

Selecting the dispersion coefficient formulation you must specify the dispersion coefficient (in m^2/s).

Data

Selecting dispersion coefficient option the format of the dispersion coefficient (in m^2/s) 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 (in m^2/s) 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.

6.7.2 Vertical dispersion

The vertical dispersion can be formulated in two different ways:

- No dispersion
- Dispersion coefficient formulation

The parameters for specifying the vertical dispersion is similar to those for the Horizontal dispersion.

6.7.3 Recommended values

It is 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.

6.8 Drift profile

6.8.1 General description

The drift profile is a description of the drift regime that the oil 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. In MIKE 3 this effect can be included in the hydrodynamic output.

If the oil is in the water surface it can be influenced directly by the wind additionally to the influence from the flow. This will result in an additional acceleration of the oil in a direction relative to the wind turned with a wind drift angle caused by coriolis forces.

6.8.2 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

Dependent on the choice of drift profile you 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 experiment. 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)

$$u = \frac{2,3}{k} \cdot u^* \cdot \log_{10} \frac{x}{z_o} \tag{6.1}$$

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 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 are 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 that is added to the particle speed.

$$U_{particle} = U_{current} + C_w \cdot W \cdot \sin(winddirection - \pi + \theta_w)$$
$$V_{particle} = V_{current} + C_w \cdot W \cdot \cos(winddirection - \pi + \theta_w) \tag{6.2}$$

where

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

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 Hemi-

sphere and to the left on the Southern Hemisphere. From Al-Rabeh (1994), it is assumed tha[t

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

(6.3)

where

a:	-0.3·10⁻⁸
b:	28° 38'
gw:	Kinematic viscosity [kg/m/s]
g:	Gravity [m/s²]

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 (in metres) and measured from the free water surface.

The vertical distribution of the parabolic onshore profile is given by

$$c_w(z) = c_w^*\left(1 - 3\frac{z}{h}\right)\left(1 - \frac{z}{h}\right) \tag{6.4}$$

where

h:	local water depth in meter
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

$$c_w(z) = c_w^*\exp(-k_0 z) \tag{6.5}$$

where

k_0:	$3/h_w$ [m^{-1}]
h_w:	depth of wind influence [m] (input)
z:	vertical co-ordinate measured from sea surface [m]
c_w^*:	wind drift factor [-] (input)

6.9   Bed roughness

The bed roughness is used in the Oil Spill 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 *ks*:

$$k_s = \frac{12h}{10^{(C/18)}} \tag{6.6}$$

where

h:	Water depth
C:	Chezy number

Manning number conversion to Nikuradse roughness *ks*:

$$k_s = \left(\frac{25,4}{M}\right)^6 \tag{6.7}$$

where *M* is the Manning number.

6.9.2 User specified bed roughness

The bed roughness must be specified as Nikuradse roughness [m] 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 (m) 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

For the case with values varying in domain you have to prepare a data file containing the wind forcing (speed (in m/s) and direction (in degrees from true North) or velocity components (in m/s)) 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 is made subsequently. You can go to the dialog for specification by clicking on the "Go to .." button. Finally, you can view the results using the relevant MIKE Zero viewing/editing tool by clicking on the "View" button during and after the simulation.

6.11.1   Output specification

For each selected output file the output type, 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.

Two types of output is available:

- Particles
- Concentrations (Field variables)

Particles
The particle file format is xml. Variables for each particle will be written in the file: x, y, z and state variable values. Furthermore the value of various derived output will be saved. The derived items are defined by the type of oil spill model.

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..."). You may also import the file to EXCEL as an XML table which makes it easy to select and review the properties for an individual particle in the time domain.

Please note that particle track output can become very large in size because it writes out positions every timestep. The intended use is to follow only a few of the particles. This is done by selecting a frequency higher than 1 when specifying the range of Particle numbers.

Concentrations

The field variables are saved in the modelled area in a dfs2/dfs3 file. If several nested areas have been used in the simulation, it is possible to select which area to apply.

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.

6.11.2 Output items

You can select basic output variables and additional output variables. The available output variables are determined by the selected oil spill model.

The basic variables consist of derived state variables of the oil spill model. All the particles of the oil spill are distributed according to the computational mesh, and the summarized mass of particles in each element are divided with the volumen of the element cell to estimate the concentration in the cell.

The basic variables are defined by the

- *State variable 1*
- *State variable 2...*

The additional variables are

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

INDEX