

MIKE 3 Flow Model FM

Transport 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 A/S (hereinafter referred to as "DHI"). For details please refer to your 'DHI Software Licence Agreement'.

LIMITED LIABILITY

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

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

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

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

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





CONTENTS



1	About This Guide	7
1.1	Purpose	7
1.2	Assumed User Background	7
1.3	General Editor Layout	7
1.3.1	Navigation tree	7
1.3.2	Editor window	7
1.3.3	Validation window	8
1.4	Online Help	8
2	Introduction	9
2.1	Application Areas	9
3	Getting Started	11
4	Examples	13
4.1	General	13
4.2	Funningsfjord	13
4.2.1	Purpose of the example	13
4.2.2	Defining the problem	14
4.2.3	Presenting and evaluating the results	15
4.2.4	List of data and specification files	17
5	TRANSPORT MODULE	19
5.1	Component Specification	19
5.2	Solution Technique	19
5.2.1	Remarks and hints	19
5.3	Dispersion	20
5.3.1	Dispersion specification	20
5.3.2	Recommended values	21
5.4	Decay	22
5.4.1	Remarks and hints	23
5.5	Precipitation-Evaporation	23
5.5.1	Recommended values	24
5.6	Infiltration	24
5.7	Sources	25
5.7.1	Source specification	25
5.7.2	Remarks and hints	26
5.8	Initial Conditions	27
5.9	Boundary Conditions	27
5.9.1	Boundary specification	27
5.10	Outputs	29
5.10.1	Output specification	29
5.10.2	Output items	38
6	LIST OF REFERENCES	41
	Index	43



1 About This Guide

1.1 Purpose

The main purpose of this User Guide is to enable you to use, MIKE 3 Flow Model FM, Transport Module, for applications of transport phenomena in lakes, estuaries, bays, coastal areas and seas.

The User Guide is complemented by the Online Help.

1.2 Assumed User Background

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

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

It is assumed that you are familiar with the basic elements of MIKE Zero: File types and file editors, the Plot Composer, the MIKE Zero Toolbox, the Data Viewer and the Mesh Generator. The documentation for these can 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 “zoom in”, “zoom out” or “recenter”. A context menu is available from which the user can select to show the bathymetry or the mesh and to show the legend. From this context menu it is also possible to navigate to the previous and next zoom extent and to zoom to full extent. If the context menu is opened on an item - e.g. a source - it is also possible to jump to the 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

MIKE 3 Flow Model FM is a new modelling system based on a flexible mesh approach. The modelling system has been developed for applications within oceanographic, coastal and estuarine environments.



Figure 2.1 Odense Estuary, Denmark. Computation mesh used in MIKE 3 Flow Model FM for studying hydrodynamics and ecosystem dynamics.

MIKE 3 Flow Model FM is composed of following modules:

- Hydrodynamic Module
- Transport Module
- MIKE ECO Lab / Oil Spill Module
- Mud Transport Module
- Particle Tracking Module
- Sand Transport Module (2D)

The Hydrodynamic Module is the basic computational component of the entire MIKE 3 Flow Model FM modelling system providing the hydrodynamic basis for the other modules.

2.1 Application Areas

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





3 Getting Started

The hydrodynamic basis for the Transport Module must be calculated using the Hydrodynamic Module of the MIKE 3 Flow Model FM modelling system.

If you are not familiar with setting up a hydrodynamic model you should refer to the comprehensive step-by-step training guide covering MIKE 3 Flow Model FM. This training guide (in PDF-format) can be accessed via the MIKE 3 Documentation index:

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





4 Examples

4.1 General

One of the best ways of learning how to use a modelling system such as MIKE 3 Flow Model FM is through practice. Therefore an example is included which you can go through yourself and which you can modify, if you like, in order to see what happens if one or other parameter is changed.

The specification files for the example is included with the installation of MIKEZero. A directory is provided for the example. The directory name are as follows (default installation):

- Funningsfjord example:
.\Examples\MIKE_3\FlowModel_FM\TR\Funningsfjord

4.2 Funningsfjord

4.2.1 Purpose of the example

Funningsfjord is a small fjord situated at the NE corner of the Faroe Islands. The computational domain and bathymetry is shown in Figure 4.1. Here is shown a typical example of calculation of the fjord circulation. The exchange of water between the fjord and the ocean generates a continuous dilution of the river water that enters the southernmost part of the fjord. One measure of the water exchange is the residence time, which for a well mixed water body can be given as $T=V/Q$. In the real world the picture is more complicated as the exchange depends on tides, wind circulations etc. The residence time is here estimated as the age (see Delhez et. at. (2003)). It should be noted that artificial forcings has been used to highlight the aspects of the test.

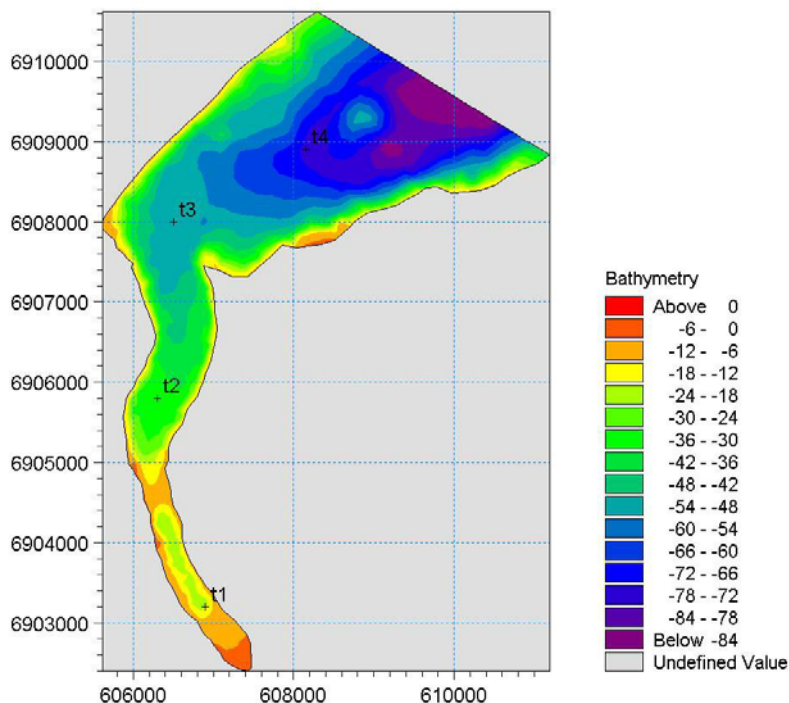


Figure 4.1 Computational domain and bathymetry.

4.2.2 Defining the problem

The main condition defining the hydrodynamic problem is:

- An unstructured mesh with 1802 elements and 1033 nodes is used. The mesh is shown in Figure 4.2. 10 layers are used for the vertical discretization.
- The starting time of the simulation is 2/8/1985 03:00:00. A *time step* of 10 seconds is selected and the duration time of the simulation is 5 days (43200 time steps).
- The horizontal eddy viscosity type has been chosen to Smagorinsky type and a constant value of $0.28 \text{ m}^{1/3}/\text{s}$ is applied for the Smagorinsky coefficient.
- The vertical eddy viscosity type has been chosen to the $k-\varepsilon$ model.
- The bed resistance type has been chosen to Roughness height and a constant value of 0.05 m is applied.
- The wind is specified as varying in time and constant in domain. A data file containing timeseries of measured wind speeds and directions are given. The length of *soft start interval* (warm-up period) for the wind has been chosen to 2 hours (7200 seconds) to avoid chock effects.
- A point source discharging $250 \text{ m}^3/\text{s}$ is applied at the southernmost point in the computational domain. The source is placed in the top layer.

- Tidal elevations, consisting of a M_2 component with amplitude 1.0 m, are applied at the open boundary along the NE section.

The main condition defining the hydrodynamic problem is:

- Transport calculations are performed for two components: One conservative and one decaying.
- The decaying component is decaying with a constant decay constant $k = 10^{-5}$.
- The vertical dispersion type has been chosen to *scaled eddy viscosity* with a scaling factor of 0.1.
- The source concentration of the two components are both set to 1.
- The initial conditions and boundary conditions for both components at constant values of 0.

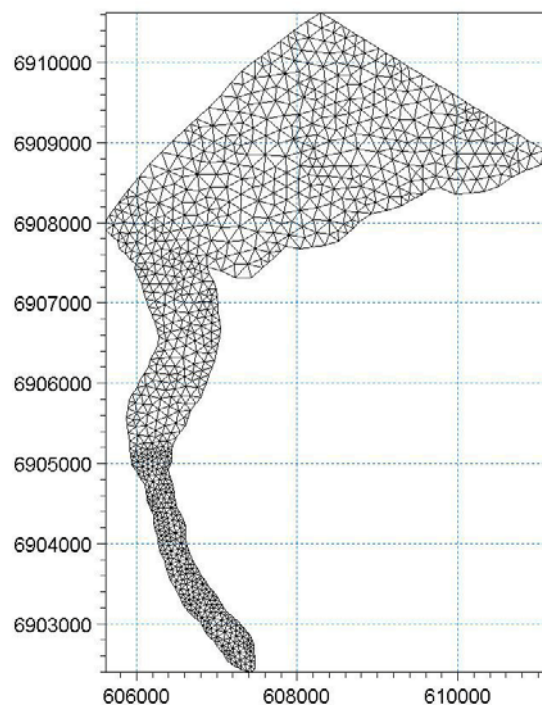


Figure 4.2 Computational mesh.

4.2.3 Presenting and evaluating the results

A contour plot of the concentration in the top layer for the two components is shown in Figure 4.3. A contour plot of the conservative component along a vertical cross-section profile for a certain time step is shown in Figure 4.4, lower figure. The horizontal location of the alignment of the cross-section are indicated in Figure 4.4, upper figure.

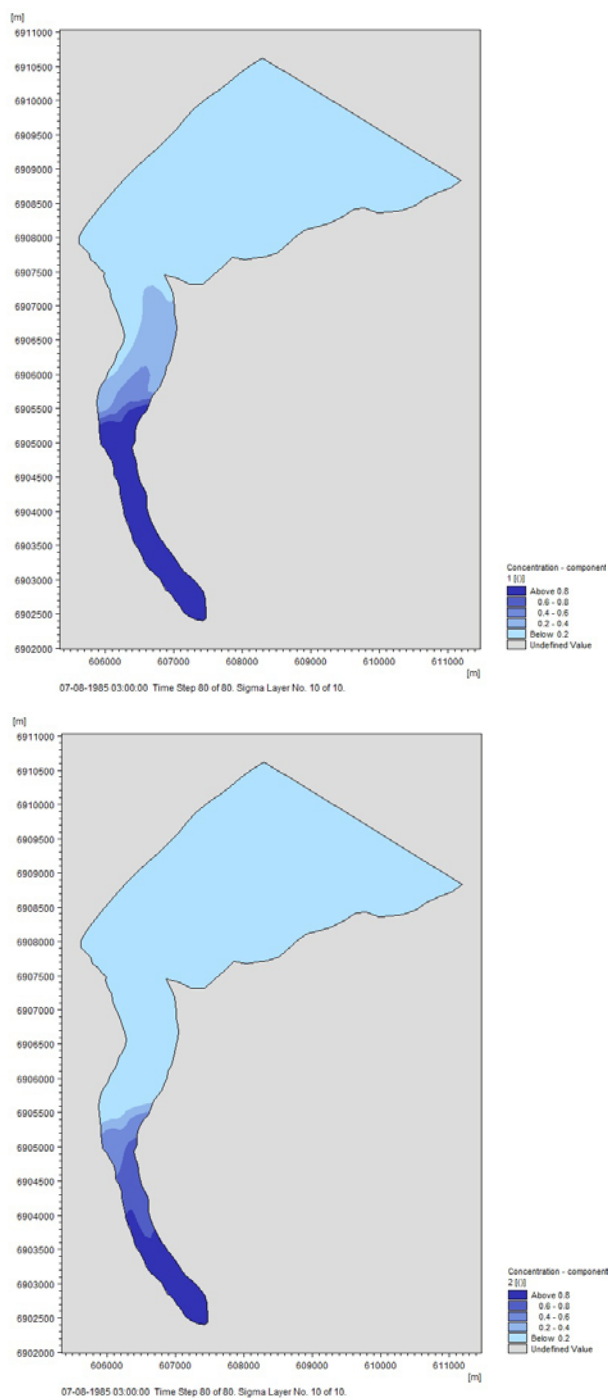


Figure 4.3 Contour plot of the concentration in the top layer for the conservative component (top) and the decaying component (bottom).

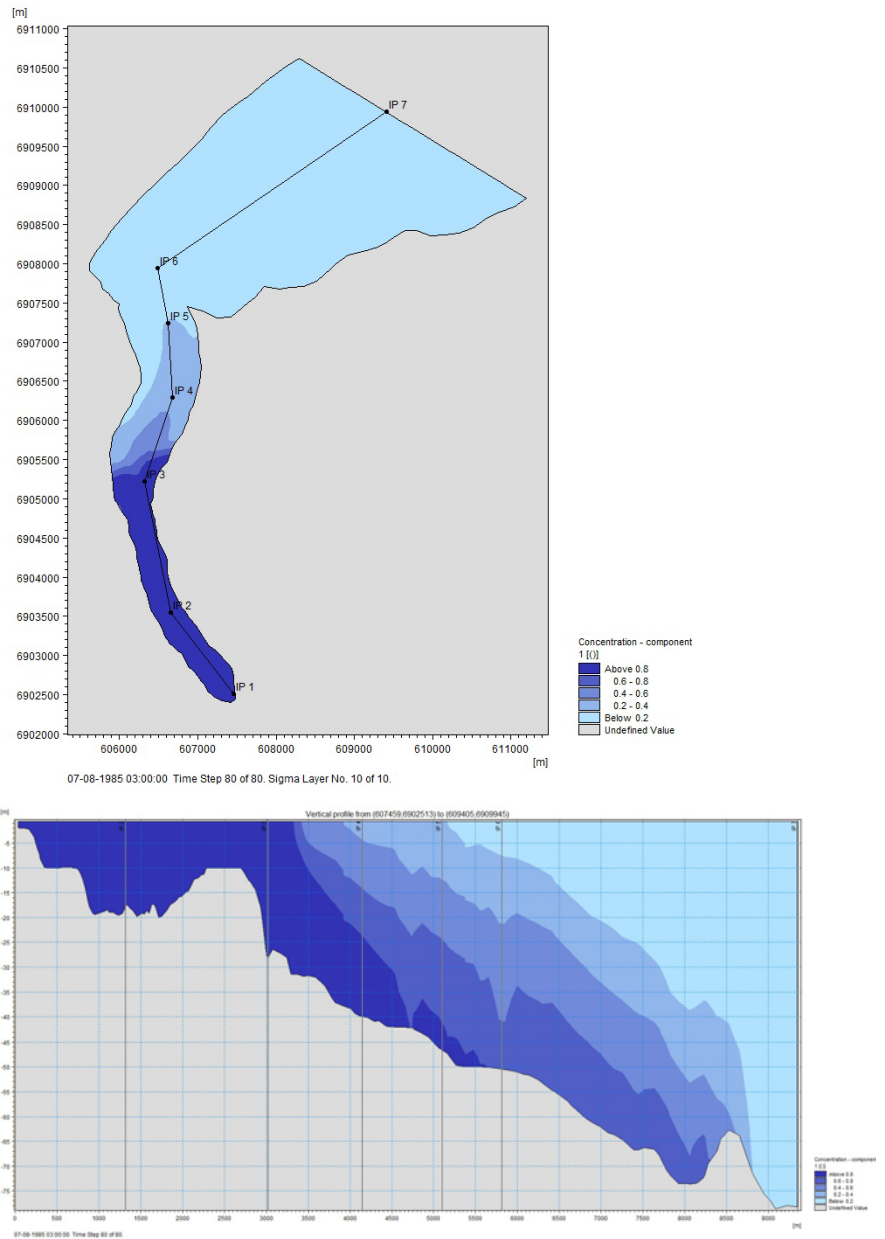


Figure 4.4 Vertical concentration of conservative component along the cross-section indicated above.

4.2.4 List of data and specification files

The following data files (included in the \TR\Funningsfjord folder) are supplied with MIKE 3 Flow Model HD FM:



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

File name: Funningsfjord.m3fm
Description: MIKE 3 Flow Model FM specification file

File name: Wind.dfs0
Description: Wind speed and direction

File name: Tide.dfs0
Description: Tidal elevations at open sea boundary



5 TRANSPORT MODULE

The transport module calculates the resulting transport of materials based on the flow conditions found in the hydrodynamic calculations.

5.1 Component Specification

On this dialog you specify the number of components (or species) and the name of the components that should be solved for. Each component defines a separate transport equation.

5.2 Solution Technique

The simulation time and accuracy can be controlled by specifying the order of the numerical schemes which are used in the numerical calculations. Both the scheme for time integration and for space discretization can be specified. You can select either a lower order scheme (first order) or a higher order scheme. The lower order scheme is faster, but less accurate. For more details on the numerical solution techniques, see the Scientific documentation.

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

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

5.2.1 Remarks and hints

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

Choosing the higher order scheme for time integration will increase the computing time by a factor of 2 compared to the lower order scheme. Choosing



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

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

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

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

5.3 Dispersion

In numerical models the dispersion usually describes transport due to non-resolved processes. In coastal areas it can be transport due to non-resolved turbulence or eddies. Especially in the horizontal directions the effects of non-resolved processes can be significant, in which case the dispersion coefficient formally should depend on the resolution.

In a 3D model 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.

The dispersion is specified individually for each component.

5.3.1 Dispersion specification

The dispersion can be formulated in three different ways

- No dispersion
- Dispersion coefficient formulation
- Scaled eddy viscosity formulation



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 section 6.6 Eddy Viscosity in the manual for the Hydrodynamic module.

Note: There is no dispersion across open boundaries as the gradient of the concentration here is assumed to be zero.

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

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 3D unstructured data file (dfs_u) or a 3D data grid file (dfs3). The area in the data file must cover the model area. If a dfs_u-file is used piecewise constant interpolation is used to map the data. If a dfs3-file is used bilinear interpolation is used to map the data.

5.3.2 Recommended values

When more sophisticated eddy viscosity models are used, as the Smagorinsky or k- ϵ models, the scaled eddy formulation should be used.

The scaling factor can be estimated by $1/\sigma_T$ where σ_T is the Prandtl number. To be consistent with the empirical constants for the k- ϵ turbulence model, the value of σ_T should be the same as the value for the Prandtl number specified



on the Equation dialog in the TURBULENCE MODULE. The default value here 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 Transport 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 the horizontal plane. For more information, see (Rodi, 1980).

The scaling factor for the vertical plane is usually much smaller, e.g. 0.00001. As it is a calibration factor this can sometimes even be 0.

5.4 Decay

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

$$\frac{\partial c}{\partial t} = -kc \quad (5.1)$$

or

$$c = c_0 \cdot e^{-kt} \quad (5.2)$$

where c is the specific concentration, c_0 is the initial concentration and k is the decay factor. In the model the decay term is added to the general transport equation.

The decay is specified individually for each component.

Data

The format of the decay factor can be specified as

- Constant (in time)
- Varying in time

For the case with time varying decay factors you have to prepare a data file containing the decay factors. 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.

5.4.1 Remarks and hints

The decay may affect the stability of the numerical solution, in a way similar to the advection or diffusion terms. If the decay represent a very rapid process such that the product $k\Delta t > 1$ the decay term may be the source of instability or at least occurrence of negative concentrations. A solution is then to reduce the time step.

5.5 Precipitation-Evaporation

If your simulation include precipitation and/or evaporation, you need to specify the concentration of each component in the precipitated and evaporated water mass. The precipitation and evaporation can be included in two ways

- **Ambient concentration**
The concentration of the precipitated/evaporated water mass is set equal to the concentration of the ambient sea water.
- **Specified concentration**
The concentration of the precipitated/evaporated water mass is specified explicitly.

If you have chosen the net precipitation option in the Precipitation-Evaporation dialog in the HYDRODYNAMIC MODULE, the precipitation concentration will be used when the specified net-precipitation is positive. When the net-precipitation is negative, the evaporation concentration is used.

The precipitation-evaporation is specified individually for each component.

Data

Selecting the specified concentration option the format of the concentration (in component unit) can be specified as

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

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



For the case with concentration varying both in time and domain you have to prepare a data file containing the concentration (in component units) before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfs_u) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfs_u-file is used piecewise constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data. The data must cover the complete simulation period. The time step of the input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

Soft start interval

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

5.5.1 Recommended values

Usually it is most sensible to set the concentration of the evaporated water mass to zero, in which case the component is conserved in the water.

5.6 Infiltration

If you have included infiltration in your simulation you need to specify how the concentration of each component is described in the infiltrated water mass. There are two different ways to do this:

- Ambient water temperature
The concentration of each component in the infiltrated water mass is set equal to the concentration in the ambient sea water.
- Specified concentration
The concentration of each component in the infiltrated water mass is specified explicitly.

Data

Selecting the specified concentration option the format of the concentration can be specified as:

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

For the case with concentration varying in time but constant in domain you have to prepare a data file containing the concentration (in component unit)



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

For the case with concentration varying both in time and domain you have to prepare a data file containing the concentration (in component units) before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfs1) or a 2D grid data file (dfs2). The area in the data file must cover the model area. If a dfs1-file is used piecewise constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data. The data must cover the complete simulation period. The time step of the input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

Soft start interval

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

5.7 Sources

Point sources of dissolved components are important in many applications as e.g. release of nutrients from rivers, intakes and outlets from cooling water or desalination plants.

In the Transport Module the source concentrations of each component in every sources point can be specified. The number of sources, their generic names, location and discharges magnitude are specified in the Sources dialog in the HYDRODYNAMIC MODULE.

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

The source concentrations are specified individually for each source and each component. From the list view you can go to the dialog for specification by clicking on the "Go to.." button.

5.7.1 Source specification

The type of sources can be specified in two ways

- Specified concentration
- Excess concentration



The source flux is calculated as the product of $Q_{\text{source}} * C_{\text{source}}$, where Q_{source} is the magnitude of the source and C_{source} is the component concentration of the source. The magnitude of the source is specified in the Sources dialog in the HYDRODYNAMIC MODULE.

Selecting the specified concentration option the source concentration is the specified concentration if the magnitude of the source is positive (water is discharge into the ambient water). The source concentration is the concentration at the source point if the magnitude of the source is negative (water is discharge out the ambient water). This option is pertinent to e.g. river outlets or other sources where the concentration is independent of the surrounding water.

When selecting the excess concentration option, the source concentration is the sum of the specified excess concentration and concentration at a point in the model if the magnitude of the source is positive (water is discharge into the ambient water). If it is an isolated source, the point is the location of the source. If it is a connected source, the point is the location where water is discharged out of the water. The source concentration is the concentration at the source point if the magnitude of the source is negative (water is discharge out the ambient water). This type can be used to describe e.g. a heat exchange or other processes where the temperature (heat) or salinity is added to the water by a diffusion process.

Data

The format of the source information can be specified as

- Constant (in time)
- Varying in time

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

5.7.2 Remarks and hints

Point sources are entered into elements, such that the inflowing mass of the component is initially distributed over the element where the source resides. Therefore the concentration seen in the results from the simulation usually is lower than the source concentration.



5.8 Initial Conditions

The initial conditions are the spatial distribution of the component concentration throughout the computational domain at the beginning of the simulation. Initial conditions must always be provided. The initial conditions can be the result from a previous simulation in which case the initial conditions effectively act as a hot start of the concentration field for each component.

The initial conditions are specified individually for each component.

Data

The format of the initial concentration (in component unit) for each component can be specified as

- Constant (in domain)
- Varying in domain

For the case with varying in domain you have to prepare a data file containing the concentration (in component unit) before you set up the hydrodynamic simulation. The file must be a 3D unstructured data file (dfs3) or a 3D grid data file (dfs3). The area in the data file must cover the model area. If a dfs3-file is used, the mesh in the data file must match exactly the mesh in the simulation. If a dfs3-file is used bilinear interpolation is used to map the data. In case the input data file contains a single time step, this field is used. In case the file contains several time steps, e.g. from the results of a previous simulation, the actual starting time of the simulation is used to interpolate the field in time. Therefore the starting time must be between the start and end time of the file.

5.9 Boundary Conditions

Initially, the set-up editor scans the mesh file for boundary codes (sections), displays the recognized codes and suggest a default name for each of them. You can re-name these names to more meaningful names in the Domain dialog (see Boundary names).

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

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

5.9.1 Boundary specification

You can choose between the following three boundary types



- Land
- Specified values (Dirichlet boundary condition)
- Zero gradient (Neumann boundary condition)

Data

If specified values (Dirichlet boundary condition) is selected the component concentration at the boundary (in component unit) can be specified in one of three ways

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

For the case with boundary data varying in time but constant along the boundary you have to prepare a data file containing the component concentration (in component unit) before you set up the hydrodynamic simulation. The data file must be a time series file (dfs0). The data must cover the complete simulation period. The time step of the input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. You can choose between different types of time interpolation.

For the case with boundary data varying both in time and along the boundary you have to prepare a data file containing the component concentration (in component unit) before you set up the hydrodynamic simulation. The data file must be a dfs2 file or a 2D dfsu file containing information from a vertical plane. The data must cover the complete simulation period. The time step of the input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. You can choose between different types of time interpolation.

Interpolation type

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

- Linear
- Piecewise cubic

In the case with values varying along the boundary two methods of mapping from the input data file to the boundary section are available:

- Normal
- Reverse order.

Using normal interpolation the first and last point of the line are mapped to the first and the last node along the boundary section and the intermediate boundary values are found by linear interpolation. By using reverse order



interpolation the last and first point of the line are mapped to the first and the last node along the boundary section and the intermediate boundary values are found by linear interpolation.

Soft start interval

You can specify a soft start interval (in sec) during which boundary values are increased from a specified reference value to the specified boundary value in order to avoid shock waves being generated in the model. The increase can either be linear or follow a sinusoidal curve.

5.10 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 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.

5.10.1 Output specification

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

Field type

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

Output format

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

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

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



- Horizontal plane series. Selected field data in geographical defined horizontal plane.
- Vertical plane series. Selected field data along geographical defined vertical plane.
- Volume series. Selected field data in geographical defined areas.

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

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

Data type

You must specify the data type. You can select discrete values or interpolated values. For point series, horizontal plane series, vertical plane series and volume series both options are available. For line series only interpolated values is available. See also Table 5.1.

If "discrete values" is selected for the data type for point series, vertical plane series and volume series, the values written in the data file are the cell-averaged values. For horizontal plane series this is also the case if "Layer number" or "Layer number above the bed" is selected for the vertical type. For the four remaining options (see Horizontal plane series), a linear interpolation in the vertical direction is applied to calculate the values at the specified z-level.

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



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

Output file

A name and location of the output file must be specified along with the file type. The file type depends on the Output format and the Data type as shown in Table 5.1.

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



Table 5.1 List of file types for the output files

Output format	Data type	File type
Point series	Discrete values	dfs0
	Interpolated values	dfs0
Line series	Interpolated values	dfs1
Horizontal plane series	Discrete values	dfsu, dfs2*
	Interpolated values	dfs2, dfsu
Vertical plane series	Discrete values	dfsu
	Interpolated values	dfs2
Volume series	Discrete values	dfsu
	Interpolated values	dfs3, dfsu
Cross-section	Not relevant	dfs0
Domain series	Not relevant	dfs0

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

Table 5.2 List of tools for viewing, editing and plotting results

File type	Viewing/editing tools	Plotting tools
dfs0	Time Series Editor	Plot Composer
dfs1	Profile Series Editor	Plot Composer Result Viewer
dfs2	Grid Series Editor Data Viewer	Plot Composer Result Viewer Data Viewer



Table 5.2 List of tools for viewing, editing and plotting results

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

Flood and dry

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

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

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

Time step

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

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

Point series

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

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



are numbered 1 at the bed and increasing upwards. The last item (the remaining of the line) is the name specification for each point.

Layer number

The layer number selected for discrete values in the point output is defined from the lowest active layer (=1) increasing upwards. In case the mesh is a type sigma mesh the number of active layers in the water column will always be the same in any point in the domain. In case the mesh is a combined sigma-z level mesh the number of active layers may vary in the domain. An example is shown in Figure 5.1.

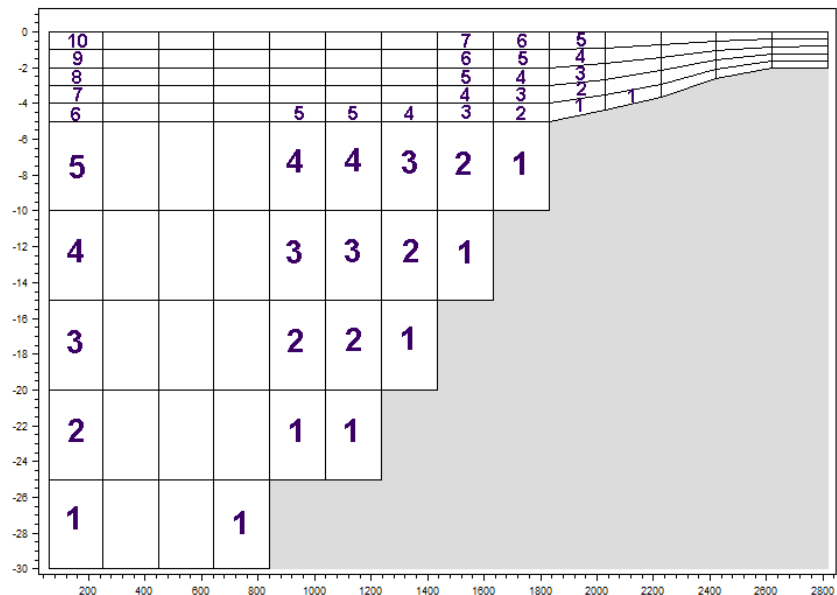


Figure 5.1 Example of layer numbers in point output specification in case of combined sigma-z level mesh.

Line series

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

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



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



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

Horizontal plane series

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

If discrete values is selected for the data type the discrete field data within a polygon can be selected. The closed region is bounded by a number of line segments. You must specify the coordinates of the vertex points of the polygon. Two successive points are the endpoints of a line that is a side of the polygon. The first and final point is joined by a line segment that closes the polygon. The geographical coordinates of the polygon points are taken from the dialog or from a file. The file format is an ascii file with three space separated items for each of the two points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. The third item is unused (but must be specified). For the vertical type the following 6 options are available

- Layer number
- Layer number above bed
- Distance above bed
- Distance below surface
- z-coordinate
- Percentage of water column

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

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

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

Layer specification

When "Layer number" is specified for the "Vertical type", the number refer to the vertical discretization specified on the Vertical Mesh page. Here the layers are numbered 1 at the lowest layer and increase upwards. In case of a combined sigma-z level mesh only the active elements are saved in the output.



An example is shown in Figure 5.1. When "Layer number above bed" is specified for the "Vertical type", the number is defined from the lowest active layer (=1) increasing upwards. In case the mesh is a type sigma mesh, the number of active layers in the water column will always be the same in any point in the domain. In case the mesh is a combined sigma-z level mesh, the number of active layers may vary in the domain. An example is shown in Figure 5.2.

Vertical plane series

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

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

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

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

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

Layer number

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

Volume series

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

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

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

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

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



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

Layer number

The layer number(s) selected for the volume output refer to the vertical discretization specified on the Vertical mesh page. Here the layers are numbered 1 at the lowest layer and increase upwards. In case of a combined sigma-z level mesh only the elements containing water are saved in the output. An example is shown in Figure 5.2.

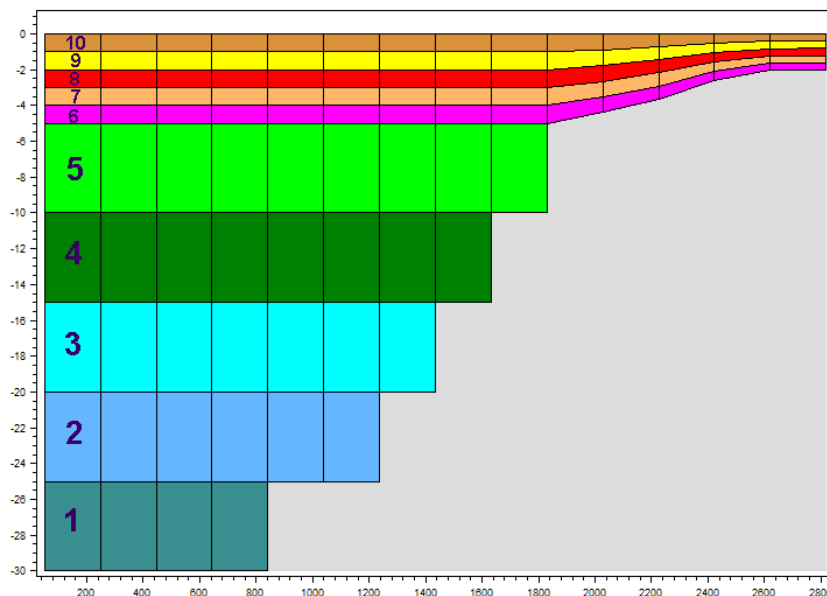


Figure 5.2 Example of layer numbers in volume output specification in case of combined sigma-z level mesh.

Cross section series

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

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

Domain series

The domain for which mass budget should be calculated is specified as a polygon in the horizontal domain. The closed region is bounded by a number of line segments. You must specify the coordinates of the vertex points of the polygon. Two successive points are the endpoints of a line that is a side of the polygon. The first and final point is joined by a line segment that closes the



polygon. The geographical coordinates of the polygon points are taken from the dialog or from a file. The file format is an ascii file with three space separated items for each of the two points on separate lines. The first two items must be floats (real numbers) for the x- and y-coordinate. The third item is unused (but must be specified).

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

5.10.2 Output items

Field variables

You can select basic output variables and additional output variables.

The basic variables are

- Component concentration [component unit] for each transported component

The additional variables are

- Velocity components
- CFL number

Mass Budget

You can select the mass budget calculation to be included for the flow and for the transported components. For each selected component the following items are included in the output file

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



- Error - accumulated volume/mass error within polygon determined as the difference between the total mass change and the accumulated mass due to transport, sources and processes

The accumulated volume/mass error contains the contribution due to correction of the transported component when the values become larger than the specified maximum value or lower than the specified minimum value. For the water volume the minimum value is 0, while there is no upper limit. For the component concentrations the minimum values are -1010, while the maximum values are 1010.

Discharge

You can select the discharge calculation to be included for the flow and for the transported components. Each selected component will result in a number of output items.

You can select between two types of output items:

- Basic
- Extended

The basic output items are as follows:

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

The extended output items that are included in the output file in addition to the basic output items are as follows:

- Positive discharge
- Accumulated positive discharge
- Negative discharge
- Accumulated negative discharge

By definition, discharge is positive for flow towards left when positioned at the first point and looking forward along the cross-section line. The transports are always integrated over the entire water depth.





6 LIST OF REFERENCES

- /1/ Delhez, E. J. M., Deelersijder, E., Maouchet, A., Beckers, J. M.(2003), On the age of radioactive tracers, *J. Marine Systems*, 38, pp. 277-286.
- /2/ Holmes, D. G. and Connell, S. D. (1989), Solution of the 2D Navier-Stokes on unstructured adaptive grids, AIAA Pap. 89-1932 in Proc. AIAA 9th CFD Conference.
- /3/ Rodi, W. (1980), Turbulence Models and Their Application in Hydraulics - A State of the Art Review, *Special IAHR Publication*.





INDEX



A	
About this guide	7
B	
Boundary conditions	27
D	
Decay	22
Dirichlet boundary condition	28
Dispersion	20
Dispersion coefficient	20
E	
Evaporation	23
Excess concentration	25
H	
Horizontal dispersion	20
I	
Initial conditions	27
L	
Layer number in output	33
Line series	33
N	
Neumann boundary condition	28
O	
Outputs	29
P	
Point series	32
Prandtl number	21
Precipitation	23
S	
Scaled eddy viscosity	20
Sources	25
T	
Tensor	30
U	
User background	7
V	
Vector	30
Vertical dispersion	21