MIKE
Powered by DHI

MIKE 3 Flow Model

Advection/Dispersion Module

User Guide

PLEASE NOTE

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 3 Flow Model, Advection/Dispersion Module (AD), for applications of spreading of dissolved substances subject to advection and dispersion processes in oceans, coastal regions, estuaries and lakes.

This User Guide is complemented by the User Guide for the MIKE 3 Flow Model, Hydrodynamic Module.

1.2 Assumed User Background

Although the Advection/Dispersion 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. It is assumed that you are familiar with the Hydrodynamic module of MIKE 3. This User Guide is not intended as a substitute for a basic knowledge of the area in which you are working: mathematical modelling of advection/dispersion phenomena.

2 Introduction

2.1 General Description

MIKE 3 Flow Model is a modelling system for 3D free-surface flows. MIKE 3 Flow Model is applicable to the simulation of hydraulic and environmental phenomena in oceans, coastal regions, estuaries and lakes.

The Advection/Dispersion (AD) module provides the advection/dispersion basis for the computations performed in the Mud Transport module and ECO Lab module (ecological modelling).

The AD module simulates the spreading and fate of dissolved or suspended substances when provided with the flow field from the hydrodynamic module. The AD module also supports dynamically nesting. The AD module specifications includes:

- conservative substances
- linear decay
- dispersion coefficients
- sources/sinks

2.2 Application Areas

The Advection/Dispersion module can be applied to a wide range of hydraulic and related phenomena. Typical substances which are modelled using the AD module are:

- tracers
- coliform bacteria
- xenobiotic compounds
- suspended sediment

3 Dialog Overview

The Advection-Dispersion (AD) module of MIKE 3 solves the three-dimensional transport equation of the advection-dispersion type for a user-specified number of dissolved or suspended substances when provided with the flow field from the HD module as set up under Dialog Overview.

In the model setup dialogs of this module you specify the advection and dispersion related parameters for your MIKE 3 Flow Model simulation. The AD specifications comprise

Component Specifications
Initial Concentrations
Boundary Conditions
Decay Specifications
Dispersion Specifications
Source and Sink
Precipitation Concentration *(p. 17)*
Deposition Concentration *(p. 17)*
Mass Budget *(p. 18)*
Results

3.1 Component Specifications

The component specification dialog is shown in Figure 3.1.

On this dialog you can specify a number of components (substances) you want to include in your simulation.

You should identify each component with a unique component name for later recognition. The name will be used later when prompting for component specific input.

In addition to the name you should also describe the components in term of decay type. You can choose between no decay (a conservative substance) and a linear decay formula.

The background value is used to minimise numerical inaccuracies in the applied advection-dispersion scheme. The background value is subtracted from the concentration field prior to the transport calculations and added back afterwards.

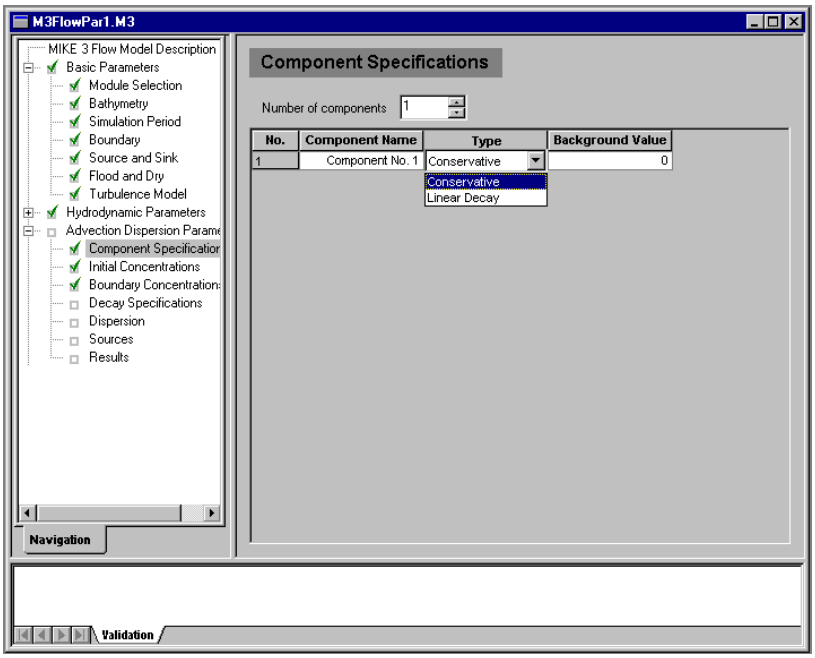

Figure 3.1 The component specification dialog

3.2 Initial Concentrations

The initial concentrations dialog is shown in Figure 3.2.

For each component you should specify its initial concentration.

You can specify the initial concentration in one of two ways:

- As a constant value applied to all points in the area.
- From a type 3 data file with values given at each grid point.

You have to specify initial concentrations for each model area separately.

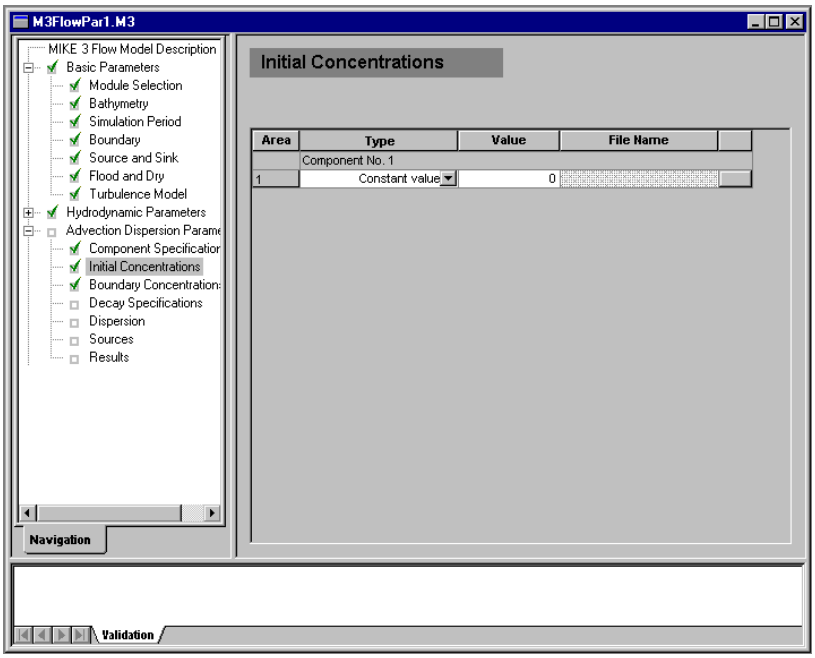

Figure 3.2 The initial concentrations dialog

3.3 Boundary Conditions

The boundary concentrations dialog is shown in Figure 3.3.

You should specify the concentration at each open boundary for each component.

The concentration at the open boundary can be specified four ways

1. as a constant value
2. as a time series (type 0 data file), the instantaneous value of which will be used at all grid points along the open boundary
3. from a type 1 data file where each horizontal grid point along the open boundary is defined
4. from a type 2 data file where all grid point along the open boundary are defined

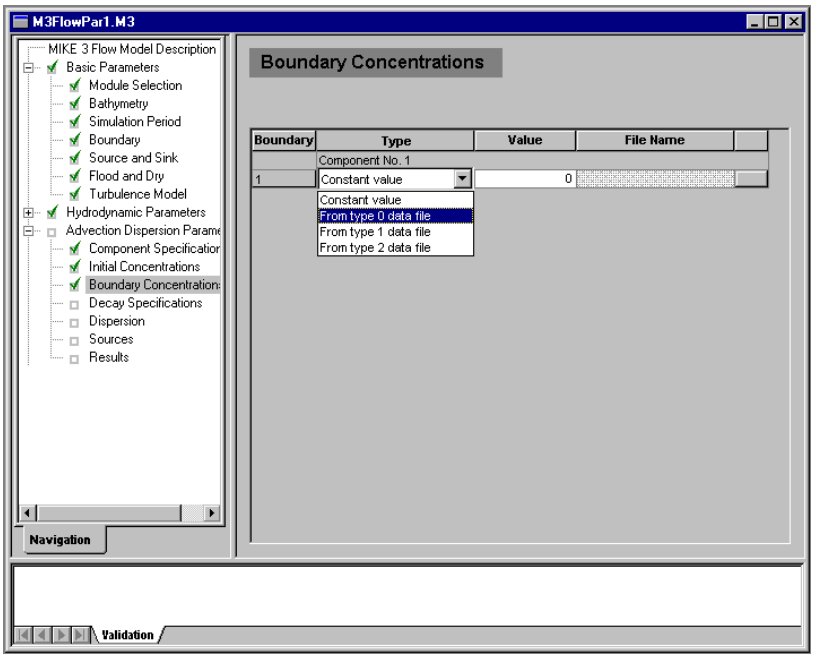

Figure 3.3 The boundary concentrations dialog

3.4 Decay Specifications

The decay specifications dialog is shown in Figure 3.4.

For each component you specify decay as either a constant value or a time varying parameter. In the latter case values should be found separately, in a specified file.

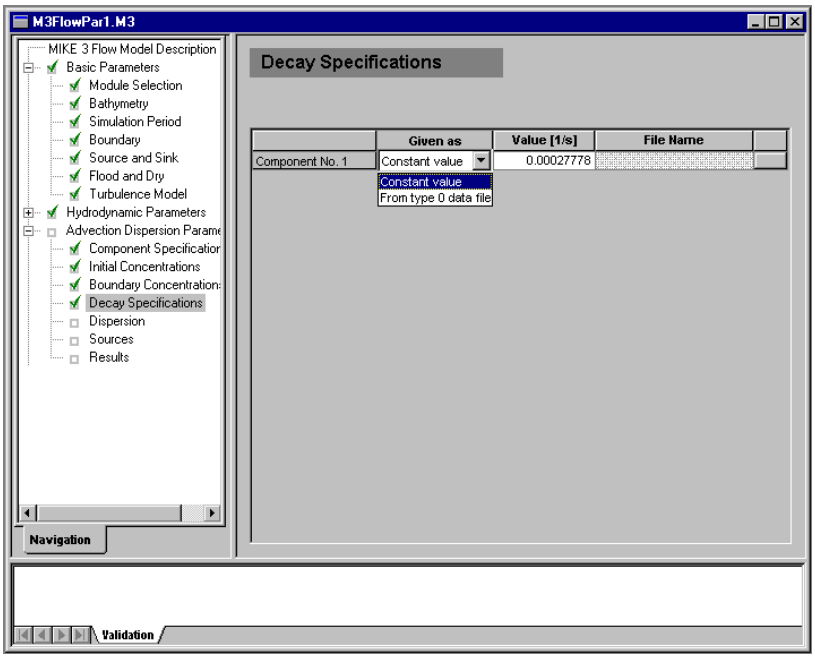

Figure 3.4 The decay specifications dialog

3.5 Dispersion Specifications

The dispersion specification dialog is shown in Figure 3.5.

On this menu you specify the dispersion variation. You should specify the dispersion proportionality factors and set limits on the dispersion coefficients in all three directions.

You have the option of defining the dispersion relation of each component: You may select the dispersion to be proportional to the local Eddy Viscosity, the local velocity or the local current speed (tensor dispersion).

A linear relationship is assumed between the dispersion of the components and the eddy viscosity, the velocity or the current speed, respectively. Dispersion factors define this linear relationship. To control dispersion the various parameters can be set differently for different areas.

To control the dispersion, a lower and upper dispersion limit in each direction must be specified. Hereby you can avoid unreasonable dispersion coefficients which could make the computation unstable. If you want to use a constant dispersion coefficient, simply set the maximum value equal to the minimum value.

Remember that dispersion coefficients in a mathematical model are dependent on the grid size and time step in addition the physics.

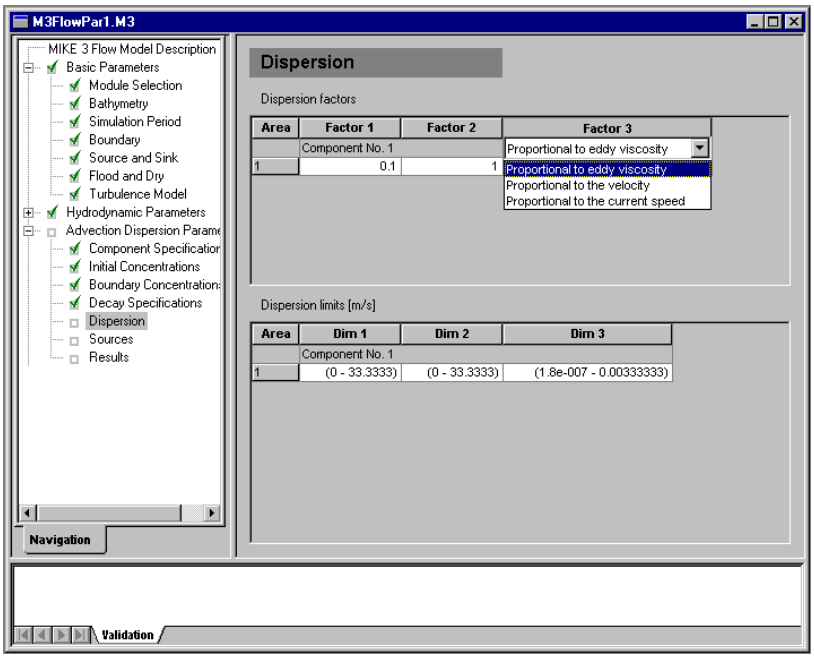

Figure 3.5 The dispersion specification dialog

3.6 Source and Sink

The source and sink dialog is shown in Figure 3.6.

You should specify the concentration at each defined source in the model. You must specify the concentration for each component in the simulation.

You can either specify the source concentration as a constant value or read and interpolate the concentration from a type 0 data file.

At one source you cannot mix the two different types. Also any possible data file must have the same number of items as there are components in the simulation.

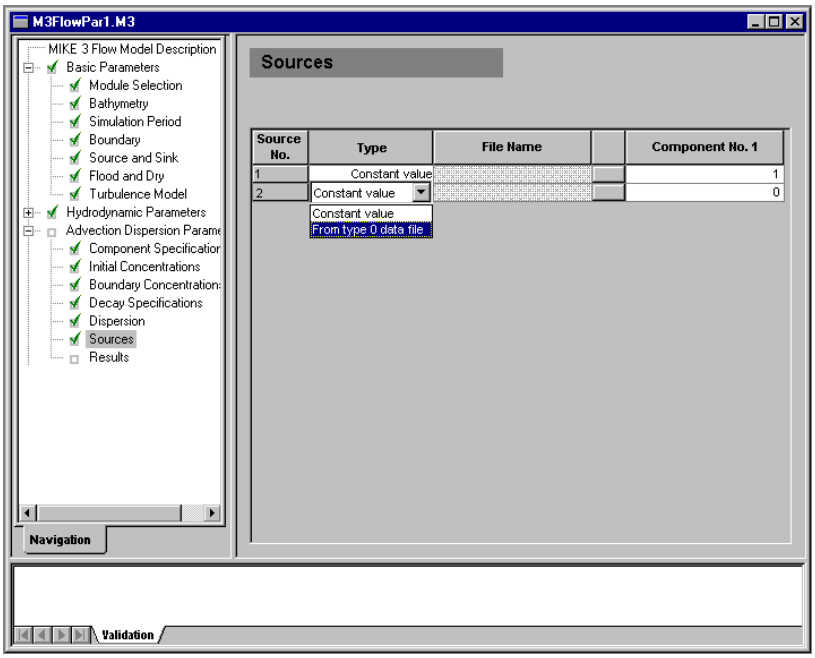

Figure 3.6 The source and sink dialog

3.7 Precipitation Concentration

If you have included precipitation in your simulation you should you should specify the concentration of each component in the rain water, or snow.

You can set the concentration as obtained from the ambient water, as a constant value or as read from a type 0 data file. The constant value will be applied to all points in the model, at all time steps. The type 0 and type 2 data file must have the same number of items as there are components in the simulation. The type 2 data file must match the spacing extent of the main area.

See also Precipitation Concentrations *(p. 31)*, Precipitation, Evaporation (HD) and Heat Exchange (HD).

3.8 Deposition Concentration

You may choose to include dry deposition in your AD simulation. If you do so, you must also specify how the component concentrations should change due to deposition through the water surface or from the seabed.

There are two different types of dry deposition

- Surface Deposition, e.g. pollution from air.
- Soil Deposition, e.g. leaching from highly contaminated sea bed.

For both types of dry deposition, you may specify the deposition rate (in units of component's unit times cubic metres per second) as "Constant value", "Type 0 data file" or "Type 2 data file".

See also Deposition Concentration *(p. 24)*.

3.9 Mass Budget

Initially the number of mass budget files is specified.

Subsequently each mass budget file is defined by an associated mass budget polygon, information on which time steps to store, filename and title, and selected model components.

Notice that it is not possible to specify any mass budget files before one or more polygons have been specified under the Basic Parameters Dialog (see Mass Budget).

See also Mass Budget (AD).

3.10 Results

The output specification dialog is shown in Figure 3.7. AD result data files can contain computed component concentrations and the velocities which are used for the AD calculations.

All output is optional, i.e. the user is free to select among the computed data.

The output data from the advection-dispersion simulation is stored in separate files. These files may be of type 0 (time series), type 1 (line series), type 2 (area series) or type 3 (volume series) depending on the spatial range settings.

The spatial ranges refer to coordinates in the associated area. The actual spatial bounds determine the type of the output data.

The temporal range refers to the time steps specified under Simulation Period. By selecting time averaged output the specified output items are averaged over the specified interval between output and stored at the specified time steps.

Because result files tend to become huge, it is normally not possible to save the computed data in all grid points at all time steps. Sub-areas and sub-sets must be selected.

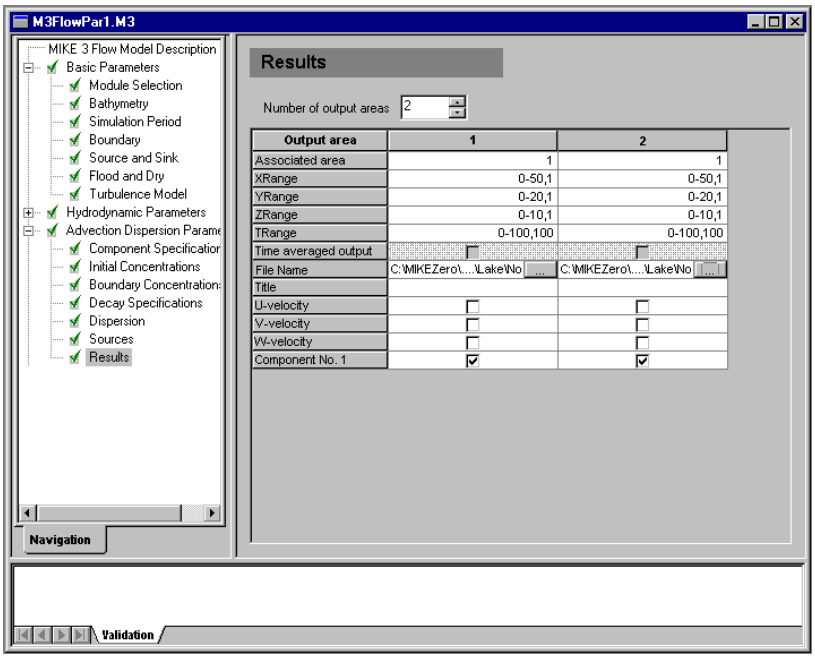

Figure 3.7 The output specification dialog

4 Reference Manual

4.1 Boundary Conditions

4.1.1 General Description

Just as with the hydrodynamic set-up, the concentrations at the open boundaries are of great importance. Because of the type of differential equation used in the Advection-Dispersion module, the module is, in some situations, able to calculate its own boundary values. This is in the situation where the flow is out of the open boundary.

Because you very seldom are able to tell beforehand whether the flow is running out of the model, boundary concentrations have to be specified at all time steps. When the flow is directed out of the model, the internally calculated values are used instead of the specified boundary values.

For each component you can specify the boundary concentrations in one of the following ways:

- A constant value used throughout the whole simulation. The value is applied at all points along the boundary.
- A time series (type 0 data file) which automatically is interpolated to match the simulation time step. Again the value at a time step is applied at all points along the boundary.
- A time series of line values (type 1 data file) which automatically is interpolated to match the simulation time step. The number of grid points in the data file must match the number of horizontal grid points along the open boundary, i.e. the boundary concentration will be vertically constant. The file may be generated by several means, e.g.: using the type 1 data editor (**ProfileEditor**), using the MIKE 21 advection-dispersion transfer data program with results from a previous MIKE 21 AD run, or using the tools in the MIKE Zero Toolbox to extract line series from 2D and 3D data.
- A time series of values at a vertical section (type 2 data file) which automatically is interpolated to match the simulation time step. The number of grid points, both horizontally and vertically, in the data file must match the number of grid points at the open boundary. There are several ways to generate such a file of boundary values, e.g.: You may use the type 2 data editor (**GridEditor**) or the tool in the MIKE Zero Toolbox to extract 2D data from a type 3 data file.

For all the different ways of specifying boundary values, it is necessary to specify them for all open boundaries and for all components.

- Note that the number and the position of open boundaries are given in the Boundary dialog in the Basic Parameters Dialog Overview section.

4.1.2 Remarks and Hints

Normally you should, if possible, place the open boundaries away from both the areas of interest and areas of influence. Doing this, the boundary concentrations can be given as the background concentration.

4.2 Component Type

You can work with two different types of substances using the advection-dispersion module:

- The first is a simple conservative substance. If no current or dispersion is present, then the concentration will remain constant in time.
- The other is a linearly decaying substance, i.e. it disappears even without current or dispersion. See Linear Decay *(p. 29)*.

4.3 Courant Number

4.3.1 General Description

The transport Courant (Peclet) number is defined as

$$C_{rU} = \frac{U\Delta t}{\Delta s} \tag{4.1}$$

where Δt is the time step, U the current speed in one direction and Δs the grid spacing in the direction corresponding to U.

The transport Courant number actually expresses how many grid points the given matter moves in one time step.

Please Note: A more restrictive condition is that the Courant number based on the flux, C_{rF} , is less than 1. C_{rF} is defined for the horizontal directions as C_{rU} with U replaced by the quantity F/h, where F is the local horizontal flux and h is the height of the considered grid cell. Often – but not always - C_{rF} is nearly equal to C_{rU} . Care should be taken in models with large gradients. If criteria

$C_{rF} < 1$

is violated during a simulation, a warning is given in the log file.

4.3.2 Remarks and Hints

The final transport Courant number is calculated as the sum of the Courant number in all three directions. This sum must theoretically be below 1.0. However, this criterion may in many cases be violated for shorter or longer periods during a simulation. In order to allow the execution to proceed, MIKE 3 automatically subdivides the time step used in the advection-dispersion module into smaller parts as

$$\Delta t_{NEW} = \frac{\Delta t}{N_{AD}} \tag{4.2}$$

This subdivision is permitted until N_{AD} equals 15. See also Time Step.

4.4 CPU Time

4.4.1 General Description

The CPU time required by a hydrodynamic and advection-dispersion simulation depends on the size of your model, on the number of time steps in your simulation, on the number of components included, on which features you have specified for the simulation and on the general computational speed of your computer.

4.4.2 Factors Influencing the CPU Time

If you wish to estimate how a change in your specifications for the hydrodynamic part changes the CPU time required without specifying the model set-up, please refer to your *Hydrodynamic Module, Reference Manual and Hydrodynamic Module, Examples* manuals.

However, if you wish to estimate how changes in your advection-dispersion set-up changes the CPU time required, the following guidelines can be used:

- The CPU time varies linearly with the number of water points (or computational points) in the model
- The CPU time also varies linearly with the number of time steps if flooding and drying is not selected. If this feature is selected the variation as a function of the number of time steps is only approximately linear

- The CPU time varies with the number of equations to be solved. In addition to the advection-dispersion equation for the selected component(s), the hydrodynamic mass and momentum equations are solved. In stratified areas also equation(s) for salinity or/and temperature variations are solved. Similarly, *k* and *k*-ε turbulence formulations imply extra equations to be solved

If you wish to estimate the CPU time required by a simulation (in real CPU seconds, not elapsed seconds) the following formula can be used:

(Number of time steps) · (Number of water points) · (1 + 0.2 · (Number of extra equations)) / BCS

where BCS (basic computational speed) is the number of water points which your computer processes in one CPU second. The number of extra equations includes the advection-dispersion equations for the selected components as well as those for possible salinity/temperature and *k*-ε variations.

4.4.3 Remarks and Hints

In cases with more than one component it might be advantageous to choose an internal component loop in order to reduce CPU time, see also Module Selection. The QUICKEST/ULTIMATE and the simple UPWIND advection-dispersion schemes in MIKE 3 Flow Model have been implemented in such a way that the CPU time consumption for solution of advection-dispersion equations is practically independent of the number of components. With the default QUICKEST/SHARP or the 3D UPWIND advection-dispersion schemes, extra memory is needed to perform the internal component loop and the user may choose to do so and thereby reduce the CPU time. Using the default QUICKEST/SHARP or the 3D UPWIND advection-dispersion schemes without internal component loop the CPU time consumption increases linearly with the number of components.

4.5 Deposition Concentration

4.5.1 General Description

Deposition of component concentrations is implemented as so-called dry deposition. Dry deposition is as a source term in the AD without a corresponding HD term. We distinguish between two different types of dry deposition

- Surface Deposition *(p. 25)*, e.g. pollution from air.
- Soil Deposition *(p. 25)*, e.g. leaching from highly contaminated sea bed.

4.5.2 Surface Deposition

You may choose to include surface deposition in your AD simulation. If you do so, you must also specify how the component concentrations should change due deposition through the water surface.

You may specify the deposition rate (in units of component's unit times cubic metres per second) as "Constant value", "Type 0 data file" or "Type 2 data file".

4.5.3 Soil Deposition

You may choose to include soil deposition in your AD simulation. If you do so, you must also specify how the component concentrations should change due to deposition from the seabed.

You may specify the deposition rate (in units of component's unit times cubic metres per second) as "Constant value", "Type 0 data file" or "Type 2 data file".

4.6 Disk Space

4.6.1 General Description

The disk space required for your simulation mainly depends on the amount of results that you request to be saved. During a simulation only a few other files, in addition to the data files containing the results, are created.

4.6.2 System Generated Files

- A specification file in the pfs system (Parameter File Standard) containing the simulation specifications will be generated by the menu system. This ASCII file will be placed in your present working directory and have a file extension of **M3**. It will only take up approximately 2 Kbytes.
- A log file describing the model set-up, the statistics of the files used and created during the simulation and a message for each time step completed will be generated by the computational module. The file extension of this ASCII file, which will also be placed in your present working directory, is **log** and it will usually take up less than 1 Mbytes on the disk. It is recommended always to carefully inspect the **log** file after termination, abnormal or successful, of a simulation.

4.6.3 User Defined Output Files

The amount of data generated by a simulation is very large. Therefore you should only save as much data as is needed for your further work. Nevertheless, very large files will often be generated.

If you wish to calculate the disk space required for a single output data file the following formula can be used. The result is in bytes:

$$4 \cdot NCPTS^{P} \cdot \left(NVAR \cdot \left(\frac{N_{last} - N_{first}}{N_{frequency}} + 1 \right) + NSKP \right) \cdot \left(\frac{J_{last} - J_{first} + 1}{J_{frequency}} \cdot \frac{K_{last} - K_{first} + 1}{K_{frequency}} \cdot \frac{L_{last} - L_{first} + 1}{L_{frequency}} \right)^{1-p} + NHEADER \tag{4.3}$$

where NVAR is the number of output variables, N denotes time steps, J denotes points in the x-direction, K points in the y-direction and L points in the z-direction. NCPTS is the number of computational (water) points in the output domain. This number is normally not known prior to a simulation, but in the **Bathymetry dialog,** the total number of computational (water) points per time step can be seen. The power p is 1 for type 3 data files, and zero otherwise. NSKP is the number prefix records in the data file and equals zero for type 0 data files, 1 for type 1 and type 2 data files, and 5 for type 3 data files.

NHEADER is 1052 for files written in the older ctx/dtx format, and may vary slightly from 1052 for files in the dfs (Data File Standard) system.

4.6.4 Remarks and Hints

Please note that MIKE 3 does not check whether or not you have enough free disk space for your requested output files. If the required disk space is too large, you should sub-divide your simulation period into smaller parts using the "hot start" facility, see Hot Data and Simulation Type.

4.7 Dispersion Coefficients

4.7.1 General Description

The transport equation for a component concentration c is formulated as:

$$\frac{Dc}{Dt} = \frac{\partial}{\partial x_j} \left(\delta_j \frac{\partial c}{\partial x_j} \right) + \mathrm{SS} \tag{4.4}$$

where SS is the source/sink term and δ_j is the dispersion coefficient in the j-direction. You have three options:

- The dispersion can vary proportionally to the local effective eddy viscosity with the factor of proportionality being $1/\sigma_T$, the dispersion factor. σ_T is the Prandtl number. Values of σ_T greater than one imply that diffusive transport is weaker for the concentration c than for momentum. See also the section Dispersion Coefficients.
- The dispersion coefficients can be specified as being proportional to the local velocity components in each grid direction, e.g. in the x-direction δ_x is proportional |u|.
- The dispersion coefficients can vary proportionally to the local current vector, i.e. there are separate dispersions in the longitudinal, the transverse and the orthogonal directions.

4.7.2 Specifying Dispersion Factors

In the **Dispersion Specifications dialog** you choose the dispersion variation for each of the components from the combo-box selection placed to the right of each component's name.

In the **Dispersion Specifications dialog** you also give the factor of proportionality. You must also specify the dispersion limits, i.e. the limits within which the dispersion coefficients are allowed to vary, for each grid direction: Defining the dimensionless dispersion coefficient for direction j as

$$D_j = \frac{\delta_j \Delta t}{(\Delta s_j)^2} \tag{4.5}$$

where Δs_j is the grid spacing in the j-direction, it is for stability required that the sum of all three D_j is less than 0.5.

4.7.3 Recommended Values

A wide range of values for σ_T occurs in the literature. In many cases a value of 10 can be applied, corresponding to a dispersion factor of 0.1.

When the mixed Smagorinsky/*k*-ε turbulence model is applied, the temporally and spatially varying values of σ_T are calculated as an integrated part of the turbulence model, and a dispersion factor of 1 is recommended.

4.7.4 Remarks and Hints

A constant dispersion coefficient may be obtained by entering a positive value for the proportionality factor and setting all dispersion limits equal to the desired value (in m^2/s).

The above mentioned dispersion stability criterion often leads to a restriction, which yields a small time step due to a very fine vertical resolution, see Time Step *(p. 35)*. Therefore, it might advantageous to select an implicit scheme for the vertical dispersion. With an implicit dispersion scheme in the vertical direction and an explicit scheme in the horizontal direction the above mentioned dispersion stability criterion is replaced by the less restrictive criteria

$D_x + D_y < 0.5$

$D_z < 10$

4.8 Hot Data

4.8.1 General Description

You can start your simulation either from scratch (a "cold start") or on the basis of a previous simulation (a "hot start"), see Simulation Type. In the latter case you need to save information about the simulation you wish to continue. These data are called "hot data".

In addition to the information described in the Hot Data section in the Reference Manual the hot data includes all component concentrations.

4.8.2 Specifying the Hot Data

You specify that you wish to be able to continue the simulation you are about to execute from the Hot Start dialog in the Hydrodynamic Parameters section, see Hot Data section in the Reference Manual. In other words, you cannot specify separate hydrodynamic and advection-dispersion hot data.

You specify that you wish to do a simulation as a continuation of a previous one by selecting **Hot start** in the **Bathymetry dialog**, and then writing the **name** of the hot data created earlier.

4.9 Initial Concentrations

4.9.1 General Description

A description of the starting concentrations is required for all the components in a simulation. There are two possibilities: A constant value used everywhere

in the model or the starting concentrations can be read from type 3 data file. A type 3 data file covering the model area as defined through the bathymetry may be created from an *xyzv*-file (ASCII file containing positions and values) with the **Digitizing** tool in the MIKE 3 Toolbox.

4.10 Linear Decay

4.10.1 General Description

The contribution from decay is calculated as:

$$dc/dt = -Fc \tag{4.6}$$

or

$$c = c_0 \exp\{-(t - t_0)\} \tag{4.7}$$

where c is the concentration and t the time. The T_{90} value (the time it takes to reduce matter by 90%), is used to calculate c from

$$c = c_0 \cdot 10^{-\left(\frac{\tau}{T_{90}}\right)} \tag{4.8}$$

4.10.2 Specifying Decay Factors

The decay factor F (in s^{-1}), one for each decaying component, is specified as being constant in time, or as a type 0 data file.

4.11 Mass Budget

The mass budget facility provides the user with a possibility to establish the mass budget of one or more model components within a certain area of the model domain. The specification of a mass budget comprises two steps: Firstly the area (or polygon) corresponding to the mass budget has to be defined and secondly the mass budget contents and output file have to be defined. The former is performed in the Basic Parameters Dialog whereas the latter is performed in the Advection-Dispersion Parameters Dialog.

At first the number of mass budget files is specified. A mass budget file contains the mass budget of one or more model components. A mass budget of a model component consists of time series of:

- Mass within polygon

- Accumulated mass transported over lateral limits of polygon
- Accumulated mass added/removed by sources/sinks within polygon. (This item also includes an AD Solver mass corrections.)
- Accumulated mass added/removed by sources/sinks within polygon. (This item also include any AD Solver mass corrections.)
- Accumulated mass added/removed by “internal” processes such as decay within polygon
- Accumulated mass deviation (error) within polygon determined as the difference between the mass change and the transported, added and removed mass
- If the ‘Section transports’ switch is enabled in the Basic Parameters Dialog, one or more additional time series will be provided in the mass budget. These correspond to the transports through each lateral section of the mass budget polygon. Notice that the sum of the section transports equals the total transport over the lateral limits of the polygon

This means that for every model component selected, five or more items will be included in the corresponding mass budget file.

The mass budget file is thus defined by an associated polygon, information on which time steps to store, filename and title, and selected model components. Notice that the mass budget file is a Type 0 data file, since it contains simple time series.

The units of the masses in the mass budget file are defined as the equivalent of the unit of the corresponding model component multiplied by 10^6 m^3. This means that if the unit of the model component is mg/l, the mass unit will be 10^9 mg, i.e. metric tons.

For information on mass budget polygons see Mass Budget under Basic Parameters Dialog.

4.12 Output Area

4.12.1 General Description

Computers are not yet so powerful that a simulation can be run each time a plot of (say) the concentration field of one of the components is needed. Therefore it is necessary to store the basic results from the simulations. On the other hand, the amount of output produced by a single simulation is often so large that it is necessary to limit the amount of output saved. You therefore have the option of saving multiple parts of output in the advection-dispersion module.

4.12.2 Specifying the Output Area

On the Results menu you specify how many output data files you wish to have produced from the simulation. You can then specify the contents of each data file by writing the number of the output area you wish to edit and pressing return:

- First you specify the data file name and data title.
- Secondly you specify the area to be included in the data file. By default the whole area is chosen but, if you are only interested in a part of the model area, you could specify the area of interest only. The type of your output data file(s) depends on the selected output area: If you specify a grid point a type 0 data file will be generated, if you specify a grid line a type 1 data file will be generated, etc.
- You specify the range of time steps to be saved and if every time step should be included or only every second, third etc.
- Finally, you choose the desired output items. You may include the velocity components as well as any of the component concentrations.

4.12.3 Remarks and Hints

One way of following the progression of your simulation is by following the number of time steps written in your output data files (or one of them). In most post-processing tasks you start by specifying the data name and after having done so, you are presented with the description of the data. This description includes the number of time steps already written and thus finished.

4.13 Precipitation Concentrations

4.13.1 General Description

In applications where the rainfall is important for the flow, you can include precipitation in your simulation. Precipitation is included into the simulation on the Precipitation dialog under Hydrodynamic Parameters. You may describe the precipitation either as a constant value or as a time series (type 0 data file), which then is applied to the entire model area, or as a time series of maps (type 2 data file) in which case each grid point is assigned its own value. The precipitation rate is specified in mm/day. You can use the **Time Series Editor** or the **Grid Editor** tool to create your precipitation data.

You can also use the precipitation facility to include evaporation in your simulation. This is simply done by specifying a negative precipitation. However, in simulations with temperature variations the evaporation is calculated as part of the latent heat flux if the heat exchange option is selected (see Heat

Exchange). Thus you should be careful not to specify evaporation from both options.

4.13.2 Precipitation Concentrations

If your simulation includes precipitation, you need to specify how the component concentrations should change due to precipitation. There are two different ways to do this:

- You may choose the format "Ambient water" concentration, in which case the concentration of the precipitated water mass is set equal to the concentration of the ambient sea water.
- You may specify the value of the concentration in the precipitated water mass through the formats "Constant value", "Type 0 data file" or "Type 2 data file".

If you have chosen the net-precipitation option, then the selected format for "Precipitation concentration" will be used when the specified net-precipitation is positive. When the net-precipitation is negative, an evaporation concentration of value zero is applied.

4.13.3 Evaporation Concentrations

When evaporation occurs or when the net-precipitation is negative, the evaporation concentration is always zero.

4.14 Simulation Type

4.14.1 General Description

There are two ways of starting your simulation:

- From scratch, also called a "cold start", which means that you have to specify the model bathymetry as well as all other model parameters.
- As the continuation of a previous simulation, also called a "hot start", in which case you must prepare "hot data" when doing the previous simulation. This is done by requesting that a file containing "hot data" be prepared in the **Hot Start dialog** in the Hydrodynamic Parameters section.

When your simulation is a continuation of a previous one the bathymetry, flooding and drying information, time step, turbulence and density variation selection, etc., as well as the component concentrations are reused from the previous simulation and cannot be changed. The rest of the model parameters you specify as for a "cold start".

4.14.2 Remarks and Hints

In most applications all simulations will be "cold started". However, it is wise to use the "hot start" facility if you have very long simulations, if your computer system often stops (intentionally or unintentionally) or if output from your simulation fill up your disk space.

4.15 Source and Sink

4.15.1 General Description

The effects of rivers, intakes and outlets from power stations, outlets from sewers, etc., can be included in a simulation. MIKE 3 distinguishes between three different kinds of sources:

- isolated source, a point where a certain amount of water is discharged into the model with a certain velocity, affecting both momentum and continuity equations,
- isolated sink, a point where a certain amount of water is discharged out of the model, affecting only the continuity equation, and
- connected source–sink pair, used for recirculation studies, the amount of water removed at the sink is re-entered at the source point with a specified velocity.

The sources and sinks are included in the hydrodynamic equations as described in *Source and Sink.*

4.15.2 Specifying Sources and Sinks

In your model you can have up to a total of 300 sources and sinks, the location and strength of which you must specify in the **Source and Sink** and the **Source and Sink dialogs** of the Basic Parameters section and Hydrodynamic Parameters section, respectively.

You specify the source component concentrations in the **Source and Sink dialog** of the Advection-Dispersion section. These may be given as constants or as included in a type 0 data file. You run through all sources one by one. For isolated sources the absolute value of the component concentration is specified whereas for connected source-sink pairs the excess source concentration is specified. At sinks, the intake component concentration equals that of the ambient water.

4.16 Standard vs. Nested Advection-Dispersion Modules

4.16.1 General Description

The purpose of this section is to enable the user to use the Nested Advection-Dispersion module of the MIKE 3 Flow Model. As most features in the nested Advection-Dispersion module are identical to the features in the standard Advection-Dispersion module, this section only describes the nested facilities, and the user is referred to other sections for guidance on common aspects.

The nested advection-dispersion module solves the advection-dispersion equations simultaneously in a user defined number of dynamically nested grids, and runs concurrently with the nested hydrodynamic module (see Standard vs. Nested HD Modules).

The advantage of applying the nested grid facility compared to the standard method of using only one grid is mainly the reduced CPU time requirements. Typical applications of the advection-dispersion modules have a limited physical area of main interest, which is only a smaller part of the total modelling area. To obtain a satisfactory spatial resolution of the model within this area of interest, the standard advection-dispersion module can be used, but will often result in a very large number of computational grid points (and often within areas of only limited interest for the application), and accordingly will require much CPU time. Applying the nested module, the spatial resolution can be optimised to save computer time.

4.16.2 Nested Model Specifications

As the Nested Advection-Dispersion module runs concurrently with the Nested Hydrodynamic Module, the rules laid out in Nested Bathymetries should be followed.

Most advection-dispersion model specifications are identical in standard Advection-Dispersion module and Nested Advection-Dispersion module, but a few comments regarding the differences are listed below:

- Initial concentration fields should be specified for each area.
- Deposition concentration fields should be specified for each area.
- Precipitation concentrations are specified for the main area only and are interpolated to the relevant sub-areas.
- Dispersion coefficients should be specified for each model area.
- Decay factors are common for all model areas.

- As in Nested Hydrodynamic Module, sources and sinks are specified in the model area of the finest resolution, which covers the actual geographical position of the source/sink.
- In the output specifications, each output area must be related to an associated model area.
- Hot start data input/output for/from a Nested Advection-Dispersion simulation must be specified for each model area. The relevant hot start data is found in/written to the corresponding hot start data files from the Nested Hydrodynamic simulation, i.e. HD and AD hot start data are in the same files.

Time step

The time step to be used in a Nested Advection-Dispersion simulation is determined in the same way as in standard Advection-Dispersion (based on the maximum transport Courant number). However, in the nested version it is necessary to calculate the transport Courant number within each model area (or bathymetry) based on the grid spacing, the current speed and the time step (which is the same in all grids) and then base the time step on the maximum of these transport Courant numbers. (Remember also to check the dispersion stability criteria for each model area, cf. Dispersion Coefficients *(p. 26)*).

The CPU time is proportional to the number of computational water points in all model areas (neglecting the "hidden" water points due to nesting). The computational speed (points per second) of a Nested Advection-Dispersion module is about 10% lower than for standard AD module, which is because of an overhead due to the handling of the nesting.

NOTE: The Nested Advection-Dispersion module allows for automatic subdivision of the time step separately in each model area, see Courant Number *(p. 22)*.

4.17 Time Step

4.17.1 Selecting the Time Step

The time step for your simulation is selected as follows:

- First you determine the horizontal grid spacing, Δx, as described under Bathymetry.
- Secondly you decide on the maximum allowed Courant number, C_r, as described under *Courant Number.*

- Then you can determine the maximum time step, Δt_{max} , which can be used in the model from the definition of the Courant number:

$$\Delta t_{max} = \frac{\Delta x C_r}{c} \tag{4.9}$$

  where c is the celerity (see Courant Number for a description). The time step to be used in the model, Δt, can then be chosen as a "convenient" number not greater than Δt_{max}.

- Furthermore, you have to check that the criterion on the final transport Courant number is not violated, see Courant Number *(p. 22)*. As you have not yet carried out the simulation you will have to make an estimate of the maximum current speeds occurring during the simulation. Note that even for small vertical velocities C_{rU} could be large due to the often very small vertical grid spacing.

- Finally, you should check that the dispersion criteria described under Dispersion Coefficients *(p. 26)* as well as in Dispersion Coefficients and Eddy Viscosity are fulfilled.

Please Note: A more restrictive condition is that the Courant number based on the flux, C_{rF} , is less than 1. C_{rF} is defined for the horizontal directions as C_{rU} with U replaced by the quantity F/h, where F is the local horizontal flux and h is the height of the considered grid cell. Often – but not always - C_{rF} is nearly equal to C_{rU} . Care should be taken in models with large gradients. If criteria

$C_{rF} < 1$

is violated during a simulation, a warning is given in the log file.

INDEX