

MIKE 21 Flow Model

Advection/Dispersion 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	9
1.1	Purpose	9
1.2	Assumed User Background	9
2	Introduction	11
2.1	General Description	11
2.1.1	Application Areas	11
3	Dialog Overview	13
3.1	Starting Conditions	13
3.2	Component	13
3.3	Initial Concentrations	13
3.4	Boundary	14
3.5	Source and Sink	14
3.6	Decay	14
3.7	Heat Dissipation	14
3.8	Heat Exchange	15
3.9	Dispersion Coefficients	15
3.10	Precipitation	16
3.11	Evaporation	16
3.12	Infiltration	16
3.13	Feedback	17
3.14	Deposition Concentrations	17
3.15	Line Discharge	17
3.16	Box Summation	18
3.17	Mass Budget	18
3.18	Results	18
4	Reference Manual	21
4.1	AD Feed Back on HD	21
4.1.1	General Description	21
4.1.2	Equations	21
4.2	Area Summation	22
4.2.1	General Description	22
4.2.2	Example	23
4.2.3	Output	23
4.3	Blow-Up	24
4.4	Boundary Conditions	24



4.4.1	General description	24
4.4.2	Hint	25
4.5	Component Types	25
4.5.1	General description	25
4.6	Courant Number	25
4.6.1	General description	25
4.6.2	Recommended value	26
4.7	CPU Time	26
4.7.1	Factors influencing the CPU time	26
4.8	Deposition Concentration	27
4.8.1	General Description	27
4.8.2	Surface Deposition	27
4.8.3	Soil Deposition	27
4.9	Disk Space	27
4.9.1	Small files	27
4.9.2	Large files	28
4.9.3	Remarks and hints	28
4.10	Dispersion Coefficients	28
4.10.1	General description	28
4.10.2	Specifying the dispersion coefficients	29
4.10.3	Recommended selection and values	30
4.10.4	Remarks and hints	30
4.11	Evaporation Concentrations	31
4.12	Heat Dissipation	31
4.12.1	General Description	31
4.12.2	Remarks and hints	32
4.13	Heat Exchange	32
4.13.1	General Description	32
4.13.2	Convection	34
4.13.3	Vaporisation	34
4.13.4	Short Wave Radiation	36
4.13.5	Long Wave Radiation	39
4.13.6	Specifying the Heat Exchange	40
4.14	Hot Data	40
4.14.1	General description	40
4.14.2	Specifying the hot data	41
4.15	Initial Concentrations	41
4.16	Line Transport	41
4.16.1	General Description	41
4.16.2	Output	42
4.17	Linear Decay	43
4.18	Mass Budget	43
4.19	Output Area	44
4.19.1	General description	44
4.19.2	Specifying the output area	44
4.19.3	Remarks and hints	45



4.20	Precipitation Concentrations	45
4.21	Simulation Type	45
4.21.1	General description	45
4.21.2	Remarks and hints	46
4.22	Sources and Sinks	46
4.22.1	General description	46
4.23	Standard vs nested AD module	47
4.23.1	General Description	47
4.23.2	Nested Model Specifications	47
4.24	Time Step	48
4.24.1	Selecting the time step	48
Index	51





1 About This Guide

1.1 Purpose

The main purpose of this User Guide is to get you started in the use of MIKE 21 Flow Model, Advection/Dispersion Module (AD), for applications of spreading of dissolved substances subject to advection and dispersion processes in lakes, estuaries, bays, coastal areas and seas. It may be applied wherever stratification can be neglected.

This User Guide is complemented by the Online Help.

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 21. 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 21 Flow Model is a modelling system for 2D free-surface flows. MIKE 21 Flow Model is applicable to the simulation of hydraulic and environmental phenomena in lakes, estuaries, bays, coastal areas and seas. It may be applied wherever stratification can be neglected.

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

The advection/dispersion module simulates the spreading of dissolved substances subject to advection and dispersion processes in lakes, estuaries and coastal regions. The effects and facilities include:

- linear decay
- heat dissipation to the atmosphere

2.1.1 Application Areas

The Advection/Dispersion (AD) module can be applied to a wide range of hydraulic and related phenomena. This includes the spreading of dissolved substances such as:

- salt
- heat
- coliform bacteria
- xenobiotic compounds

Typical applications include cooling water recirculation and water quality studies.





3 Dialog Overview

3.1 Starting Conditions

Here you must specify the AD activation period. This is given in time steps relative to the HD simulation.

If your simulation is to be a continuation of a previous one, then you must select the 'Hot Start' option. The hot start facility requires a 'hot file', i.e. a type 2 data file containing hot start data, for each model area. These must all have originated from the same simulation. The hot start files contain all necessary information to continue a simulation. In this way simulation time can be reduced if for instance a number of scenarios are to be simulated, all requiring identical starting conditions based on the same warm up period.

NOTE: You can let the AD part of the calculation be hot started irrespective of the starting conditions of the HD part.

3.2 Component

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

You should identify each component with a unique 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 terms of general decay characteristics. You can choose between no decay (conservative matter), linear decay, decay according to a heat dissipation formula, and as heat exchange component. See also Component Types (p. 25).

For hot-started simulations, the number of components and the name and type of each component are obtained from the hot-data file(s) and they cannot be changed.

3.3 Initial Concentrations

For each component you should specify its initial concentration within each of computational area. 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 2 data file with values given at each grid point

For hot-started simulations, the initial concentrations are read from the hot-data file(s).



Note: The type 2 data file must only contain one timestep

3.4 Boundary

At each open boundary in the model you should specify the concentration for each component.

The concentration at the open boundary can be specified in one of three ways:

1. As a constant value, which will be used at all times at all grid points along the open boundary
2. From a type 0 data file with time-variant values, which will be used at all grid points along the open boundary
3. From a type 1 data file where time-variant values are defined for each grid point along the open boundary

NOTE: At one open boundary you cannot mix the different types of boundary data. Also any possible data file (type 0 or 1) must have the same number of items as there are components in the simulation.

3.5 Source and Sink

For each component, here you should specify the component concentrations at each defined source in the model.

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

NOTE: 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.

3.6 Decay

You can specify the decay factors in one of two ways:

- as a constant value applied at all time steps
- by way of a type 0 data file, which will be interpolated to match simulation time steps

3.7 Heat Dissipation

You can specify the reference temperature in one of two ways:



- as a constant value applied at all time steps
- by way of a type 0 data file, which will be interpolated to match simulation time steps

3.8 Heat Exchange

The heat exchange with the atmosphere is calculated on basis of the four physical processes

- the long wave radiation
- the sensible heat flux (convection)
- the short wave radiation
- the latent heat flux (evaporation)

The heat exchange affects the temperature (if varying). The effect of Precipitation and Evaporation will also be taken into account if this is included in the simulation.

The dialog prompts for the parameters which enter the heat exchange formulation. Please see the Heat Exchange (p. 32) for further details.

See also Precipitation Concentrations (p. 45) and Evaporation Concentrations (p. 31).

3.9 Dispersion Coefficients

On this dialog you specify the AD dispersion coefficients. The dispersion coefficients can be set either as independent of the current, or scaled in accordance with the calculated fluxes proportional to the current.

If you have selected "Independent on the current", you should specify the dispersion coefficients. You must specify a value for both model directions. For each area, you can specify the coefficients either as a constant value applied to all points in an area or as read from a type 2 data file.

If you have selected "Proportional to current", you should specify a proportionality factor for each direction. With this selection, the dispersion coefficients are at all times scaled in accordance with the calculated fluxes. To avoid values that would make the computation unstable, reasonable cut-off limits should be specified.

NOTE: Remember that dispersion coefficients in a mathematical model are dependent on the grid size and time step, in addition to the physics of the problem.



3.10 Precipitation

If you have included precipitation in your simulation 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. 45*), Evaporation Concentrations (*p. 31*) and Heat Exchange (*p. 32*).

3.11 Evaporation

If you have included evaporation in your simulation you should specify the concentration of each component in the evaporated water.

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. 45*), Evaporation Concentrations (*p. 31*) and Heat Exchange (*p. 32*).

3.12 Infiltration

If you have included infiltration in your simulation you need to specify how the component concentrations should change due to infiltration. There are two different ways to do this:

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



3.13 Feedback

By including the AD feedback, density gradients become a driving force in the HD module. The density is calculated on the basis of both salinity and temperature by application of the UNESCO formula.

You may specify either salinity or temperature as constant, but at least one of the two must vary.

You should set an AD component to be treated as the varying salinity, or/and temperature.

See also AD Feed Back on HD (*p. 21*).

3.14 Deposition Concentrations

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. 27*).

3.15 Line Discharge

The line discharge facility is used to calculate the transport of a substance through a user specified cross section of the model area.

Both the instantaneous and the accumulated transports are calculated.

You must set the number of lines, and then for each line specify:

- the position of the line
- the associated area number
- the period you want the line transport calculations to be active
- the name and (optionally) the title of the type 0 data file where the line transports are to be written
- which components to be included in the line transport calculations



NOTE: The calculated transports are not projected onto the line normal in order to integrate the flow across the line. The program transforms the specified line into little pieces of line that follow the grid instead, see the manual.

3.16 Box Summation

The box summation facility is used to calculate the amount of a substance, and decayed substance, within a user specified square box.

The instantaneous mass, and the instantaneous decayed and accumulated decayed masses are calculated.

You must set the number of square boxes, and then for each box specify

- the position of the box. This is done by selecting the corner coordinates of the box
- the associated area number
- the period you want the box summation calculations to be active
- the name and (optionally) the title of the type 0 data file where the line transports are to be written
- which components to be included in the box summation calculations

NOTE: The summation can be used together with the line transport option for mass budget calculations. When doing that, particular attention has to be taken in specifying the line coordinates, see the manual.

3.17 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 Reference Manual/Mass Budget (*p.* 43).

3.18 Results

You can obtain two types of result data from an Advection-Dispersion simulation. The detailed specifications of the result files are set by editing settings for each individual output area.



1. Up to 18 normal output data files (type 2) containing the computed concentration fields. You specify the spatial and temporal output range and the associated area number, and you specify a name and (optionally) a title for the output data file.
2. Additionally, special hot-start data can be created. These data can be used for a later continuation of the simulation. You have to specify the name of the hot-output file for each area. You can specify a title too, if you want to.

NOTE: Result files tends to become very big, so have that in mind when specifying the area and period.

See also Output Area (p. 44).



NOTE: In case of applying nested model areas, then only items which are calculated on fine grid points which are common to coarse grid points will be inherited and displayed in the result file for the coarser grid. The magnitude of the item inherited from the fine grid to the coarse grid will be the same on both grids (no resolution related scaling!).





4 Reference Manual

4.1 AD Feed Back on HD

4.1.1 General Description

By including the AD feedback the density gradients become a driving force in the HD module. The density is calculated on the basis of the salinity and the temperature by use of the UNESCO formula. Either the salinity or the temperature may be specified as a constant value and thereby left out of the AD calculations.

4.1.2 Equations

The pressure term in the general Navier-Stokes equation is effected by the density gradients. An analysis of this term reads

$$\begin{aligned}
 -\frac{1}{\rho} \cdot \frac{\partial p}{\partial x} &= -\frac{1}{\rho} \cdot \frac{\partial}{\partial x} (\rho \cdot g \cdot z) \\
 &= -\frac{1}{\rho} \cdot g \left[z \cdot \frac{\partial \rho}{\partial x} + \rho \cdot \frac{\partial z}{\partial x} \right] = -g \cdot \frac{\partial z}{\partial x} - \frac{1}{\rho} g \cdot z \frac{\partial \rho}{\partial x}
 \end{aligned}$$

Integration over the vertical

$$\begin{aligned}
 -\frac{1}{\rho} \int_0^h \frac{\partial \rho}{\partial x} dz &= -g \int_0^h \frac{\partial z}{\partial x} dz - \frac{1}{\rho} \cdot g \cdot \frac{\partial \rho}{\partial x} \int_0^h z \cdot dz \\
 &= -g \cdot h \cdot \frac{\partial z}{\partial x} - \frac{1}{\rho} \cdot g \cdot \frac{\partial \rho}{\partial x} \cdot 0.50 \cdot h^2
 \end{aligned}$$

Discretizing of the additional term



$$\begin{aligned} & \left[-\frac{1}{\rho} \cdot g \cdot \frac{\partial \rho}{\partial x} \cdot \frac{1}{2} \cdot h^2 \right] \\ &= -\frac{1}{2} \cdot \frac{2}{\rho_{j+1}^{old} + \rho_j^{old}} \cdot g \cdot \frac{\rho_{j+1}^{old} - \rho_j^{old}}{\Delta x} \cdot h^{*2} \\ &= -\frac{\rho_{j+1}^{old} - \rho_j^{old}}{\rho_{j+1}^{old} + \rho_j^{old}} \cdot g \cdot \frac{1}{\Delta x} \cdot h^{*2} \quad , \quad h^* = \frac{1}{2} (h_j + h_{j+1}) \end{aligned}$$

It is advisable to analyse the driving force on beforehand and only to include the AD feedback on HD whenever there is a necessity for modelling the effect of the density gradients.

4.2 Area Summation

4.2.1 General Description

The AD area summation option is used to calculate the amount of substance and decayed substance within a user-specified square.

When specifying the area where the summation should take place, it is important to realise which area or which grid points are involved in the summation. See Figure 4.1 for a definition.

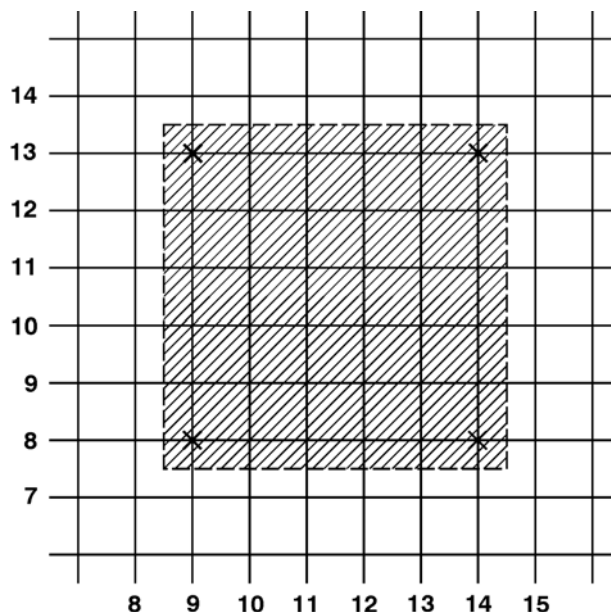


Figure 4.1 Area Definition

The summation can be used together with the line transport option for mass budget calculations. When doing that, particular attention has to be taken when specifying the line coordinates, see the Line Transport (*p. 41*) description.

4.2.2 Example

To calculate the mass transport in and out of the square shown in Figure 4.1, the following four lines should be defined

- 1: (8,8) - (8,13)
- 2: (9,13) - (14,13)
- 3: (14,8) - (14,13)
- 4: (9,7) - (14,7)

4.2.3 Output

The instantaneous mass, the instantaneous and accumulated decay are calculated and written in a type 0 data file. If the component used in the summation is conservative, the instantaneous and accumulated decay will be zero.



4.3 Blow-Up

Please inspect carefully the description in the **log** file.

4.4 Boundary Conditions

4.4.1 General description

Just as with the hydrodynamic set-up, the concentrations at the open boundaries are of great importance. Because of the different type of differential equation the advection-dispersion 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, 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.

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. The data file must have the same number of items, as there are components in the simulation. This means, that if you want to use a type 0 data file for your boundary concentrations, the values for all the components have to be in that file.

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 grid points along the open boundary. In addition the data file must have the same number of items, as there are components in the simulation. This means, that if you want to use a type 1 data file for your boundary concentrations, the values for all the components have to be in that file.

The file can be generated by the type 1 data editor (the Profile Editor) or by the advection-dispersion transfer data program in the "Environment" section of the MIKE 21 Toolbox.

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 via input under the Boundary section under Basic Parameters Dialog Overview.



4.4.2 Hint

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.

4.5 Component Types

4.5.1 General description

You can work with four different types of substances using the advection dispersion module.

- The first is a simple conservative substance. If no current or dispersion, sources concentrations or boundary conditions, different from the background values, are present, then the concentration will remain constant in time.
- The two next types of substances are both decaying substances; i.e. they disappear even without current or dispersion. They are distinguished by the manner in which they decay. One decays linearly, see Linear Decay (p. 43), and the other decays according to an empirical formula developed for heat dissipation studies, see Heat Dissipation (p. 31).
- The last type of substance is a substance (i.e. temperature) that reacts according to the heat exchange with the atmosphere, see Heat Exchange (p. 32). Only maximum one component can be specified as a substance that reacts on the heat exchange with the atmosphere.

4.6 Courant Number

4.6.1 General description

The transport Courant number in the x-direction is defined as follows:

$$C_{rx} = U \cdot \frac{\Delta t}{\Delta x}$$

where U is the current speed, Δt the time step and Δx the grid spacing.

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



4.6.2 Recommended value

The final Courant number is calculated as the sum of the Courant number in the x-direction and the Courant number in the y-direction. The sum of these two must be below 1.0. See further under Time Step (p. 48).

4.7 CPU Time

4.7.1 Factors influencing the CPU time

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.

If you wish to estimate how a change in your model specifications for the hydrodynamic part changes the CPU time required without specifying the model setup and doing a verification, please refer to the HD Reference manual, CPU Time (p. 71).

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.
- the CPU time for a hydrodynamic simulation alone is approximately the same as an advection-dispersion simulation alone (same time step and simulation period), including two components. Adding on more components to be calculated, the CPU time increases approximately linearly.

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

$$\frac{(\text{Number of time steps}) \times (\text{Number of water points}) \times (\frac{1}{2}(\text{no. of components}) + 1)}{\text{BCS}}$$

where BCS (basic computational speed) is the number of water points which your computer processes in one CPU second.



4.8 Deposition Concentration

4.8.1 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.* 27), e.g. pollution from air.
- Soil Deposition (*p.* 27), e.g. leaching from highly contaminated sea bed.

4.8.2 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 to 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".

4.8.3 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".

4.9 Disk Space

4.9.1 Small files

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

- the specification file containing the simulation specifications. This file will be placed in your present working directory and have a file extension of **m21**. It will only take up approximately 6 Kbytes.



- the **log** file describing the model setup, the statistics of the files used and created during the simulation and a message for each time step completed. The file extension of this file, which will also be placed in your present working directory, is **log** and it will take up to 300 Kbytes on the disk.
- a file for the continuation of a simulation, the hot file. This file only takes up disk space equivalent to 'the number of components' times the space taken up by the bathymetry file. This means that the hot file will not take up more space than 100 - 200 Kbytes (depending on your setup).

4.9.2 Large 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:

$$4 \cdot \left[3 \cdot \left(\frac{N_{last} - N_{first}}{N_{frequency}} \right) + 1 \right] \cdot \frac{J_{last} - J_{first} + 1}{J_{frequency}} \cdot \frac{K_{last} - K_{first} + 1}{K_{frequency}} + 1052$$

where N denotes time steps, J denotes points in the x-direction and K points in the y-direction.

4.9.3 Remarks and hints

Please note that MIKE 21 does not check whether or not you have enough free disk space for your requested output files.

4.10 Dispersion Coefficients

4.10.1 General description

Physical flow processes occur simultaneously at different spatial and temporal scales with a continuous spectrum ranging from molecular agitation to tidal, barotropic and baroclinic residual flows. The need to express conservation laws by means of mathematical methods imposes the treatment of fluid motions as part of continuum mechanics. This implies the use of spatial



scales much larger than the intermolecular spacings associated with molecular agitation processes, in order to define the properties such as concentrations. Furthermore, for the study of practical problems only the flow processes above a certain minimum scale are of interest, hence the direct modelling of smaller scales giving rise to unnecessary and in practice unaffordable computational costs. By using convenient filtering procedures it is possible to avoid the explicit inclusion of those scales in the dependent variables used in the mathematical formulations. These non-resolved scales can not be ignored, as part of the energy of the flow is absorbed by them. Therefore they have to be empirically related to the dependent variables (that are representative of all the resolved scales) through specific terms that naturally appear as a consequence of the filtering procedures. The relationships to be used and the respective empirical coefficients will then reflect the existence of non-resolved scales, the main effects of which can be classified as follows:

- Dissipative, which account for the amount of energy present in the non-resolved scales or/and the decay/growth of flow properties. In two-dimensional hydrodynamic models for example, bottom friction explains not only the energy losses due to shear stresses, but also the total amount of energy contained in the non-resolved scales along the vertical.
- Dispersive, which account for the transfer of energy or properties between resolved scales, due to the existence of non-resolved scales. These effects are represented by the so-called viscosity/diffusion coefficients.

The wide range of time and length scales involved, as well as the variability of flow conditions, makes it impossible to develop a unified approach with universally applicable expressions for the evaluation of the referred empirical coefficients. Therefore one has to resort to calibration values through the use of reliable and sufficient data. However the values of those coefficients are subject to large variations, thus raising the need for good first estimates, that can be obtained both by theoretical analysis of the physical processes and through the use of experience acquired in similar applications.

For further reading, see "Dispersion Coefficients" in the "Scientific Documentation".

4.10.2 Specifying the dispersion coefficients

Dispersion coefficients are required, whenever there are moving components.

The dispersion coefficients can be:

- specified as constant in space, as a coefficient for the x-direction and a coefficient for the y-direction.



- specified as two-dimensional maps of coefficients, similar to the bathymetry map or a resistance map.
- calculated every time step, proportional to the hydrodynamic current densities. If you choose this formulation, proportionality constants, minimum and maximum values are also required.

4.10.3 Recommended selection and values

Selecting dispersion coefficients is very difficult. The optimum method is to obtain the coefficients from a calibration process, where simulated and measured concentration fields are compared. Very seldom are measurements available for calibration of transport processes. These processes have very long response times (months or years), and therefore, empirical estimates have to be utilised.

Many different formulations can be applied e.g.

$$D = K_1 \Delta x^2 / \Delta t,$$

$$D = K_2 \Delta x u$$

or

$$D = K_3 \Delta t u^2$$

Δx is the grid spacing,
 Δt is the time step and
 u is the local current speed.

4.10.4 Remarks and hints

Using too high values of the dispersion coefficients, will cause instabilities. The stability is a function of the local transport Courant number and the dispersion coefficient. For practical applications it is not possible prior to a simulation, to judge whether the upper limit of the dispersion coefficients have been reached. If you insist on using a specific value of the dispersion coefficient, even though you get instabilities, you must decrease the time step.

When running the MIKE 21 Flow Model FM, Advection/Dispersion Module, the program does not accept all values, since an upper limit can be reached:

$$\left(\frac{D_x}{\Delta x^2} + \frac{D_y}{\Delta y^2} \right) \cdot \Delta t < 0.5$$

This is based on the stability criteria for a pure diffusive problem, where the upper limit is 0.5.



4.11 Evaporation Concentrations

When evaporation occurs or when the net-precipitation is negative, you need to specify how the component concentrations should change due to evaporation. There are two different ways to do this:

- You may choose the format "Ambient water" concentration, in which case the concentration of the evaporated water mass is set equal to the concentration of the ambient sea water.
- You may specify the value of the the concentration in the evaporated 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 "Evaporation concentrations" will be used when the specified net-precipitation is negative. When the net-precipitation is positive, an precipitation concentration as specified under Precipitation Concentrations (*p. 45*) is applied.

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 (*p. 32*)). Thus you should be careful not to specify evaporation from both the heat exchange option and the evaporation option.

4.12 Heat Dissipation

4.12.1 General Description

The most common use for a heat dissipating substance is that of modelling power plant cooling water recirculations. Here the advected and dispersed component is heat (or rather temperature).

The heat dissipation follows a prescribed function, where the decay factor F may take the form:

$$F = 0.2388 / (\rho C_p H)(4.6 - 0.09(T_{ref} + T) + 4.06W) \exp(0.033(T_{ref} + T))$$

which is suitable for heat dissipation to the atmosphere. W is the wind speed in m/s, T_{ref} is the reference temperature in degrees Celsius (same units as T), ρ is the density of water, C_p is the specific heat and H is the water depth. W may be given constant or varying in time, following a prescribed function. In units consistent with former ones, ρC_p is given the value 10^6 (Cal/m³ °C) in the program.

Further reading about the basis for the heat dissipation formula is available in the Scientific Documentation.



4.12.2 Remarks and hints

There is an important relationship between the initial temperature, the source, the boundary temperature and the reference temperature in the above equation.

The simulation can be performed in either absolute or excess temperatures. The excess temperature is the temperature, which is higher than what it would have been without the power plant.

If you choose to work with excess temperature, which is the most common, the initial temperature should be set at zero. The source temperature should be set at the absolute outlet temperature minus the water temperature, and the reference temperature should be set at the water temperature, which it would have been without the plant.

With absolute temperatures you should set the initial temperature at the background temperature. Here the source temperature should be set at the absolute temperature at the outlet and the reference temperature should be set at zero.

4.13 Heat Exchange

4.13.1 General Description

In some types of investigations it is necessary to take the heat exchange with the atmosphere into account. You can do that by choosing to include **heat exchange on the Heat Exchange dialog**. The heat exchange is calculated on basis of the four physical processes:

- sensible heat flux (or the heat flux due to Convection (*p. 34*))
- latent heat flux (or the heat loss due to Vaporisation (*p. 34*))
- net Short Wave Radiation (*p. 36*)
- net Long Wave Radiation (*p. 39*)

The heat exchange facility, however, requires that the temperature is a prognostic variable.

The heat exchange module is used for simulation of heat transfer between the water body and the atmosphere. The heat balance can be expressed as:

$$\Delta q = q_{io} + q_{ss} + q_p - q_c + q_s - q_{sr} - q_{su} + q_1 - q_{lr} - q_{lu} + q_g + q_{sed} - q_v \quad (4.1)$$

where



Δq is the change in the heat capacity of the water body

q_{io} is heat transfer from inflow/outflow

q_{ss} is the heat transfer from sources/sinks

q_p is the heat transfer from precipitation

q_v is loss of energy due to vaporation

q_c is the convective heat transfer

q_s is the short wave radiation

q_{sr} is the reflected short wave radiation

q_{su} is the short wave radiation emitted from the water body

q_l is the long wave radiation above the sea surface

q_{lr} is the reflected long wave radiation

q_{lu} is the emittance of long wave energy from the water body

q_g is the heat exchange between ground and water body

q_{sed} is the heat exchange between sediment and water body

Some of these terms are not included in the heat exchange module, since they are taken into account elsewhere, e.g. as for heat transfer from sources/sinks and inflow/outflow. Furthermore, some of the terms are lumped into a single term, e.g. as for the long wave radiation and short wave radiation. The following terms are set to zero in the heat balance module since they have minor influence on the total heat budget:

$$q_g = 0 \text{ and } q_{sed} = 0.$$

The heat exchange module calculates the four quantities sensible heat flux, latent heat flux, net short wave radiation, and net long wave radiation as described in the following sections.



4.13.2 Convection

The sensible heat flux, q_c (or the heat flux due to convection) depends on the type of boundary layer between the sea surface and the atmosphere. Generally this boundary layer is turbulent implying the following relationship

$$q_c = \begin{cases} \rho_{air} C_{air} C_c W_{10m} (T_{water} - T_{air}) & \text{for } T_{air} > T_{water} \\ \rho_{air} C_{air} C_c W_{10m} (T_{water} - T_{air}) & \text{for } T_{air} \leq T_{water} \end{cases} \quad (4.2)$$

where

ρ_{air} is the air density (kg/m^3)

C_{air} is the specific heat of air, $1007 \text{ J} (\text{kg} \cdot ^\circ\text{K})$

C_w is the specific heat of water, $4186 \text{ J} (\text{kg} \cdot ^\circ\text{K})$

W_{10m} is the wind speed 10 m above the sea surface

T_w is the absolute temperature of the sea

T_{air} is the absolute temperature of the air

C_c is the sensible transfer coefficient, given as $1.41 \cdot 10^{-3}$

The convective heat flux typically varies between $0 - 100 \text{ W}/\text{m}^2$.

4.13.3 Vaporisation

Dalton's law yields the following relationship for the vaporative heat loss (or latent flux):

$$q_v = LC_e(a_1 + b_1 W_{2m})(Q_{water} - Q_{air}) \quad (4.3)$$

where

L is the latent heat of vaporisation, $2.5 \cdot 10^6 \text{ J}/\text{kg}$

C_e is the moisture coefficient, $1.32 \cdot 10^3$

W_{2m} is the wind speed 2 m above the sea surface

Q_{water} is the water vapour density close to the surface

Q_{air} is the water vapour density in the atmosphere



a_1 and b_1 are user specified coefficients

Measurements of Q_{water} and Q_{air} are not directly available but the vapour density can be related to the vapour pressure as

$$Q_i = \frac{0.2167}{T_i} e_i \quad (4.4)$$

in which subscript i refers to both water and air. The vapour pressure close to the sea surface, e_w can be expressed in terms of the water temperature assuming that the air close to the surface is saturated and has the same temperature as the water

$$e_{water} = 6.11 e^{K\left(\frac{1}{T_k} - \frac{1}{T_{water}}\right)} \quad (4.5)$$

where

K_i is a constant being 5418 °K

T_{ki} is the temperature at 0 °C being 273.15 °K

Similarly the vapour pressure of the air can be expressed in terms of the air temperature and the relative humidity, R

$$e_{water} = 6.11 e^{K\left(\frac{1}{T_k} - \frac{1}{T_{water}}\right)} \quad (4.6)$$

Replacing Q_{water} and Q_{air} with these expressions the latent heat can be rewritten as

$$q_v = P_v(a_1 + b_1 W_{2m}) \left(\frac{\exp\left\{K\left(\frac{1}{T_k} - \frac{1}{T_{water}}\right)\right\}}{T_{water}} - \frac{R \exp\left\{K\left(\frac{1}{T_k} - \frac{1}{T_{air}}\right)\right\}}{T_{air}} \right) \quad (4.7)$$

where all constants have been included in a new latent constant, P_v being $4370 \text{ J} \cdot \text{°K/m}^3$. During cooling of the surface the latent heat loss has a major effect with typical values up to 100 W/m^2 .



Remark: Please note, if you have not selected the net-precipitation option on the Source and Sink (p. 31) dialog, the evaporation rate will be calculated from the latent heat flux as

$$\Delta\eta_e = -\frac{q_v}{L\rho_{water}}$$

when q_v is positive.

4.13.4 Short Wave Radiation

Radiation from the sun consists of electromagnetic waves with wave lengths varying from 1,000 to 30,000 Å. Most of this is absorbed in the ozone layer, leaving only a fraction of the energy to reach the surface of the Earth. Furthermore the spectrum changes when sunrays pass through the atmosphere. Most of the infrared and ultraviolet compound is absorbed such that the solar radiation on the Earth mainly consists of light with wave lengths between 4,000 and 9,000 Å. This radiation is normally termed short wave radiation. The intensity depends on the distance to the sun, declination angle and latitude, extraterrestrial radiation and the cloudiness and amount of water vapour in the atmosphere.

Distance Between the Earth and the Sun

The ratio between the mean distance, r_0 to the Sun and the actual distance, r is given by

$$E_0 = \left(\frac{r_0}{r}\right)^2 = 1.000110 + 0.034221 \cos(\Gamma) + 0.001280 \sin(\Gamma) + 0.000719 \cos(2\Gamma) + 0.000077 \sin(2\Gamma) \quad (4.8)$$

in which Γ is defined by

$$\Gamma = \frac{2\pi(d_n - 1)}{365} \quad (4.9)$$

and d_n is the Julian day of the year.



Solar Declination and Day Length

The daily rotation of the Earth around the polar axes contributes to changes in the solar radiation. The seasonal radiation is governed by the declination angle, which can be expressed by

$$\begin{aligned} \delta = & 0.006918 - 0.399912 \cos(\Gamma) + 0.07257 \sin(\Gamma) \\ & - 0.006758 \cos(2\Gamma) + 0.000907 \sin(2\Gamma) \\ & - 0.002697 \cos(3\Gamma) + 0.00148 \sin(3\Gamma) \end{aligned} \quad (4.10)$$

The day length, N_d varies with δ . For a given latitude (positive on the northern hemisphere) the day length is given by

$$N_d = \frac{24}{\pi} \arccos \{ -\tan(\phi) \tan(\delta) \} \quad (4.11)$$

and the sunrise angle, ω_{sr} is

$$\omega_{sr} = \arccos \{ -\tan(\phi) \tan(\delta) \} \quad (4.12)$$

Extraterrestrial Radiation

The intensity of short wave radiation on the surface parallel to the surface of the Earth changes with the angle of incident. The highest intensity is in zenith and the lowest during sunrise and sunset. Integrated over one day the extra-terrestrial intensity in short wave radiation on the surface can be derived as

$$H_0 = \frac{24}{\pi} q_{sc} E_0 \cos(\phi) \cos(\delta) (\sin(\omega_{sr}) - \omega_{sr} \cos(\omega_{sr})) \quad (4.13)$$

where q_{sc} is a solar constant.

Radiation Under Cloudy Skies

For determination of daily radiation under cloudy skies, H , the following relation is used

$$\frac{H}{H_0} = a_2 + b_2 \frac{n}{N_d} \quad (4.14)$$

in which n is the number of bright sunshine hours, N_d the length of the day and a_2 and b_2 are user specified constants. The default values are $a_2=0.295$ and $b_2=0.371$. The user-specified clearness coefficient corresponds to (n/N_d) , where 100% specifies a clear sky and 0% specifies cloudy weather.



The average hourly short wave radiation, q_s can be expressed as

$$q_s = \left(\frac{H}{H_0}\right) q_0 (a_3 + b_3 \cos(\omega_i)) \quad (4.15)$$

where

$$a_3 = 0.4090 + 0.5016 \sin\left(\omega_{sr} - \frac{\pi}{3}\right) \quad (4.16)$$

$$b_3 = 0.6609 + 0.4767 \sin\left(\omega_{sr} - \frac{\pi}{3}\right)$$

The extraterrestrial intensity, q_0 and the hour angle ω_i is given by

$$q_0 = q_{se} E_0 \left(\sin(\phi) \sin(\delta) + \frac{24}{\pi} \sin\left(\frac{24}{\pi}\right) \cos(\phi) \cos(\delta) \cos(\omega_1) \right) \quad (4.17)$$

$$\omega_i = \frac{\pi}{12} \left(12 + \text{correction for summer time} - \frac{E_t}{60} \frac{4}{60} - (L_s - L_E) - \text{local time} \right) \quad (4.18)$$

Correction for summer time should be specified as +1 hour. We although recommend using standard time and then convert the timing when presenting the results if needed.

E_t is varying during the year and is called the equation of time given by

$$E_t = (0.000075 + 0.001868 \cos(\Gamma) - 0.032077 \sin(\Gamma) - 0.014615 \cos(2\Gamma) - 0.04089 \sin(2\Gamma)) \cdot 229.18 \quad (4.19)$$

The time meridian L_s is the standard longitude for the time zone and L_E is the local longitude. The local standard meridian L_s should be given corresponding to the actual time zone. E.g for a simulation at the US East cost (UTC-8) you would normally use -120 Degrees ($15 \cdot 8$). The definition is negative westward and positive eastward.



Albedo

Solar radiation that impinges on the sea surface does not all penetrate the water surface. Parts are reflected back and are lost unless they are backscattered from the atmosphere of the surrounding topography. This reflection of solar energy is termed the albedo. The amount of energy, which is lost due to albedo, depends on the angle of incident and angle of refraction. For a smooth sea the reflection can be expressed as

$$\alpha = \frac{1}{2} \left(\frac{\sin^2(i-r)}{\sin^2(i+r)} + \frac{\tan^2(i-r)}{\tan^2(i+r)} \right) \quad (4.20)$$

where i is the angle of incident, r the refraction angle and α the reflection coefficient, which typically varies from 5 to 40 %.

Thus the net short wave radiation, $q_{s,\text{net}}$ can eventually be expressed as

$$q_{s,\text{net}} = (1 - \alpha)q_s \quad (4.21)$$

4.13.5 Long Wave Radiation

A body or a surface emits electromagnetic energy at all wavelengths of the spectrum. The long wave radiation consists of waves with wavelengths between 9,000 and 25,000 Å. The radiation in this interval is termed infrared radiation and is emitted from the atmosphere and the sea surface. The long wave emittance from the surface to the atmosphere minus the long wave radiation from the atmosphere to the sea surface is called the net long wave radiation and is dependent on the cloudiness, air temperature, vapour pressure in the air and relative humidity. In MIKE 3 the net outgoing long wave radiation is given by

$$q_{\text{lr},\text{net}} = \sigma_{sb} T_{\text{air}}^4 (a - b \sqrt{e_d}) \left(c + d \frac{n}{n_d} \right) \quad (4.22)$$

where a , b , c and d are coefficients given as:

$$a = 0.56; b = 0.77 \text{ mb}^{-1/2}; c = 0.10; d = 0.90$$

e_d is the vapour pressure at dew point temperature measured in mb

n is the number of sunshine hours

N_d is the number of possible sunshine hours

σ_{sb} is Stefan Boltzman's given as $5.6697 \cdot 10^{-8} \text{ W}/(\text{m}^2 \cdot \text{K}^4)$



T_{air} is the air temperature

4.13.6 Specifying the Heat Exchange

As many as possible of the parameters in the heat exchange description have been hardcoded in MIKE 21. However, some need to be specified as part of your simulation set-up.

First of all you should specify if the evaporation and precipitation temperature should be calculated from the latent heat flux. Alternatively, the evaporation and precipitation temperature is specified on the corresponding Evaporation Concentrations and Precipitation Concentrations dialogs.

Then, you need to specify the air temperature variation (given in Celcius), which either can be a constant, a type 0 data file or a type 2 data file.

For the latent heat flux you need to specify the constants a_1 and b_1 , called **Constant in Dalton's law** and **Wind coefficient in Dalton's law**, respectively. You also need to specify a relative humidity and cloudiness for the short wave and long wave calculations. These can either be specified as constants or as read from type 0 or type 2 data files. Furthermore you should specify the constants a_2 and b_2 known as **Sun constants in Angstrom's law**, the time displacement (if summer time is effective) and the local standard meridian for the time zone.

Finally, you may select the time-integration method. Usually, the simple 1st order Euler method is sufficient, but cases with large heat fluxes may benefit from a more sophisticated 2nd or 4th order Runge-Kutta method. Please note that even though increasing the order of integration might be useful in some situations, the user should in first hand suspect that the global simulation time step might possibly be too large.

4.14 Hot Data

4.14.1 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 (p. 45)). In the latter case you need to save information about the simulation you wish to continue. These data are called "hot data".

The hot data file contains the following model information:

- model dimensions and grid spacings, time step and time at end of simulation, the latitude of the model and its orientation (relative to true north), and the value above which a point is always considered to be land,
- all component arrays.



Thus, if you use a hot start, you need not (and cannot) specify the data listed above.

4.14.2 Specifying the hot data

You specify that you wish to be able to continue the simulation you are about to execute by enabling the "General Hot Start" on the Results (*p. 18*) dialog, and then writing the **name** of the hot data file.

You specify that you wish to do a simulation as a continuation of a previous one by selecting "hot Start" on the Starting Conditions (*p. 13*) dialog, and then writing the **name** of the hot data file created earlier.

4.15 Initial Concentrations

A description of the starting concentrations/temperatures 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 2 Data File**.

4.16 Line Transport

4.16.1 General Description

The AD line transport option is used to calculate the transport of substance through a user-specified line in the model area.

The advection-dispersion module calculates the transport of substance in the x- and y-direction at each grid point. Due to the staggered grid used by the hydrodynamic module these transports are placed in-between the grid points, see Figure 4.2.

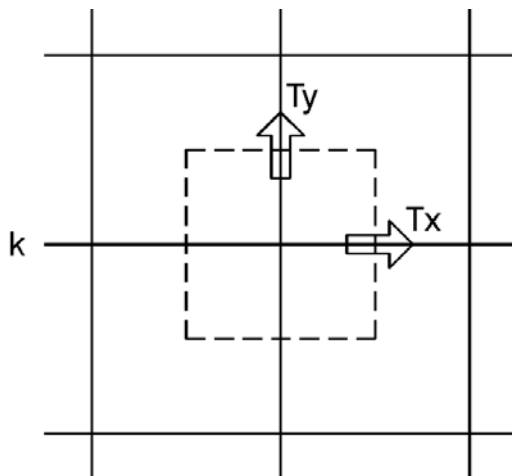


Figure 4.2 Position of Transports

The calculated transports are directly used by the AD line transport option to calculate the amount of substance passing the line. It is important to realise that the transports are **NOT** projected onto the line normal in order to integrate the flow along the line. Instead, the program transforms the specified line into line pieces that follow the grid, see Figure 4.3.

Two special cases are the vertical and horizontal lines; from Figure 4.2 it can be seen that the transports are actually taken half a grid point away from the specified line.

The points that the program has used to sum the transport through the line can be found in the generated **log** file. If used in connection with the mass summation particular care must be taken when specifying the line.

4.16.2 Output

The instantaneous and accumulated transports are written to a type 0 data file.



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.
- 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

This means that for every model component selected, five 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 m³. 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.

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).

4.19 Output Area

4.19.1 General description

Computers are not so powerful that a simulation can be run each time a plot of, for example, the concentration field 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 up to 18 selected parts of the output.

4.19.2 Specifying the output area

On the Results dialog 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:



- you specify the data name and data title,
- 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 and you have finished the calibration, you could specify the area of interest only.
- you specify the range of time steps to be saved and if every time step should be included or only every second, third or whatever.

4.19.3 Remarks and hints

One way of following the progress of your simulation is to follow the number of time steps written in your output file (or one of them). In most postprocessing tasks you start by specifying the data file 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.

4.20 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 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 concentrations" will be used when the specified net-precipitation is positive. When the net-precipitation is negative, an evaporation concentration as specified under Evaporation Concentrations (*p.* 31) is applied.

In simulations with temperature variations the precipitation is calculated as part of the latent heat flux if the heat exchange option is selected (see Heat Exchange (*p.* 32)). Thus you should be careful not to specify precipitation from both the heat exchange option and the precipitation option.

4.21 Simulation Type

4.21.1 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 on the Results (p. 18) dialog.

4.21.2 Remarks and hints

Normally all simulations will be "cold started". However, if you have very long simulations, your computer system often stops (planned or unplanned), or if you create so much output that the disk needs to be emptied, it is wise to use the "hot start" facility.

4.22 Sources and Sinks

4.22.1 General description

The effects of rivers, intakes and outlets from power stations, etc. can be included in a simulation. These sources and sinks are included in both the hydrodynamic and the advection-dispersion models. The source/sink positions and discharges are specified in the Basic Parameters Dialog Overview and Dialog Overview part, respectively, whereas the source concentrations are specified in the Dialog Overview.

There are three different ways of specifying sources and sinks:

1:Isolated Source.

Here you specify the source concentration, constant or varying in time for each component.

2:Isolated Sink.

Isolated sinks are only processed in the hydrodynamic part.

3:Source - Sink Pair.

Here it is possible to deal with 'connected sources', several sources connected to one sink (or 'connected sinks'), several sinks connected to one source. This is done in the way that the concentration at the sink point is added to the user defined source(s) concentration. In the hydrodynamic part you ensure that the sum of the connected source discharges are the same as the sink discharges.

This Source-Sink Pair feature is normally used in cooling water investigations for power plants. For example, the 'heated water source' is connected to a number of sinks in the power plant area.



By using a Source-Sink Pair it is possible to investigate the recirculation of heated water; is it likely that the water temperature in the environment increases continuously?

You can specify the source concentrations in one of the following ways:

- A constant value used throughout the whole simulation.
- A time series (type 0 data file) which automatically is interpolated to match the simulation time step. The data file must have the same number of items, as there are components in the simulation. This means that if you want to use a type 0 data file for your source concentration, the values for all the components have to be in that file.

4.23 Standard vs nested AD module

4.23.1 General Description

The purpose of this section is to enable the user to use the Nested Advection-Dispersion module (NAD) of the MIKE 21 Flow Model. As most features in the Nested Advection-Dispersion Module (NAD) are identical to the features in the standard Advection-Dispersion module (AD), this manual only describes the nested facilities.

The nested advection-dispersion module solves the advection-dispersion equations simultaneously in a user defined number of dynamically nested grids, and runs coupled with the nested hydrodynamic module.

The advantage of applying the nested grid facility compared to the standard 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.

4.23.2 Nested Model Specifications

As the NAD module runs coupled with the Nested Hydrodynamic Module (NAD), the rules laid out in Standard vs. nested HD module should be followed.

Most advection-dispersion model specifications are identical in AD and NAD, but a few comments regarding the differences are listed below:



- Initial concentration fields should be specified for each area.
- Dispersion coefficients, which are specified as constants, are common for all model areas.
- Dispersion coefficients, which are specified as two-dimensional maps should be specified for each model area.
- Dispersion coefficients, which are proportional to the current, are common for all model areas.
- Decay factors (linear decay and heat dissipation) are common for all model areas.
- As in the 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 input/output files must be specified for each model area.
- For Line Discharge and Box Summation calculations, each line and box must be related to one specific associated model area.

Time step

The time step to be used in a NAD simulation is determined in the same way as in AD (based on the maximum transport Courant number). In the nested version though, it is necessary to calculate the transport Courant number within each model area 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. 28)).

CPU time

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 NAD is about 10 % lower than for AD, which is because of an overhead due to the handling of the nesting.

4.24 Time Step

4.24.1 Selecting the time step

The time step for the advection-dispersion simulation is the same as the one used for the hydrodynamic simulation, since the two models are fully coupled. Both the hydrodynamic model and the advection-dispersion model must be considered, when selecting the time step. The procedure is:



- first you determine the grid spacing, Δx , as described under Bathymetry (p. 49),
- then you decide on the maximum allowed Courant number, C_r , as described under Courant Number (p. 70),
- then you can determine the maximum time step, Δt_{\max} , which can be used in the model from the definition of the Courant number:

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

where c is the wave celerity (see Courant Number (p. 70) for a description). The time step to be used in the hydrodynamic model, Δt , can then be chosen as a "convenient" number not greater than Δt_{\max} .

- finally you have to check that the Courant number based on the current speed, (the transport courant number) $C_{r,tr}$, instead of the wave celerity, is less than 1 for the time step chosen. If not, you must reduce the chosen time step. $C_{r,tr}$ is defined as

$$C_{r,tr} = \Delta t \cdot \left(\frac{U}{\Delta x} + \frac{V}{\Delta y} \right)$$

where U, V is the current speed in the x -direction and y -direction. For all points within the model U and V must follow the above formula. As you have not yet carried out the simulation you will have to make an estimate and then check this after the simulation.

Please note that formally U and V in the above expression should be replaced by P/H and Q/H , respectively, where P and Q are the magnitudes of the fluxes in the x - and y - directions and H is the local water depth. Often the two expressions – based on current speeds or on the flux magnitudes – yield almost the same value, but care should be taken in models with large surface and/or bottom gradients.





INDEX



C	
Convective heat flux	34
D	
Dalton's law	34
Declination angle	37
L	
Latent flux	34
N	
Nested model	19
S	
Sensible heat flux	34
Solar radiation	39
V	
Vaporative heat loss	34