MIKE 3 Flow Model

Hydrodynamic Module

User Guide

PLEASE NOTE

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

1.1 Purpose

The main purpose of this User Guide is to get you started in the use of MIKE 3 Flow Model, Hydrodynamic module (HD), for applications of hydraulic phenomena in oceans, coastal regions, estuaries and lakes.

This User Guide is complemented by the Online Help.

1.2 Assumed User Background

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

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

2 Introduction

2.1 General Description

The three dimensional, baroclinic model MIKE 3 is a general non-hydrostatic numerical modelling system developed for a wide range of applications in areas such as oceans, coastal regions, estuaries and lakes.

The hydrodynamic (HD) module is the basic module in the MIKE 3 Flow Model. It simulates unsteady three-dimensional flows, taking into account density variations, bathymetry and external forcings such as meteorology, tidal elevations, currents and other hydrographic conditions.

Hydrodynamic model features include

- flood and drying
- bed resistance
- density variations
- transport of salinity and temperature
- turbulence modelling including buoyancy effects
- wind friction ð level and/or velocity boundaries
- hot start facility
- isolated sources and sinks, connected source/sink pairs
- pier resistance
- heat exchange with atmosphere including evaporation/precipitation
- particle tracking
- discharge calculations
- dynamical nesting

2.1.1 Application Areas

MIKE 3 Flow Model, Hydrodynamic Module is applicable to the study of a wide range of phenomena related to hydraulics wherever the three-dimensional flow structure is important:

- tidal exchange and currents
- stratified flows
- oceanographic circulation
- heat and salt recurculation

As mentioned previously, the HD output results are also used as input for many of the other MIKE 3 modules such as the Advection-Dispersion module, the Mud Transport module and the MIKE ECO Lab module (environment).

3 Examples

3.1 General

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

The specification files for the examples are included with the installation of MIKE Zero. A directory is provided for each example. The directory names are as follows (these may have been changed at your installation; please ask your system administrator if you cannot find the directories):

- Wind set-up example:
  .\MIKE Zero\Examples\MIKE_3\FlowModel\HD\Lake
- Sound (Øresund), Denmark example:
  .\MIKE Zero\Examples\MIKE_3\FlowModel\HD\Sound
- Turtle Bay, Nested model example:
  .\MIKE Zero\Examples\MIKE_3\FlowModel\HD\TurtleBay

3.2 Wind set-up in a rectangular lake

3.2.1 Defining the Model

This example has been chosen as a fairly simple one, so that it is possible to check the results analytically. Also, the example is very similar to the lake example of MIKE 21 such that users familiar with MIKE 21 can easily gain experience with MIKE 3.

The problem is to determine the wind set-up in the lake shown below.

The test conditions are:

- The lake is rectangular in shape with a length of 5 km in the east/west direction, and a length of 2 km in the north/south direction, as shown in Figure 3.1. The lake has a uniform depth of 10 m.
- The lake is connected to the sea by a 100 m wide channel in the middle of the shore to the west. The sea is assumed to have a constant water level of 0.0 m. Thus, there is one open boundary.
- A westerly wind of 35 m/s is blowing.

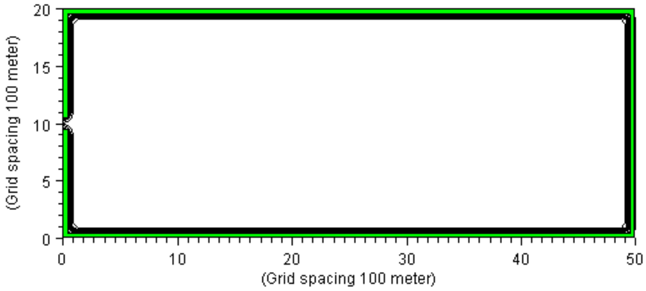


Figure 3.1 Wind setup in a small lake, model layout

Additional information required by MIKE 3 is:

- The horizontal grid spacing, which on the basis of the size of the lake is selected to be 100 metres.
- The *vertical grid spacing* and the *number of layers*. A single layer of 1 metre resolution has been chosen. Note: The so-called bottom-fitted approach used in MIKE 3, see the Scientific Documentation, assures that the water depth (10 m) is correct despite the small grid spacing (1 m).
- The *time step*, which on the basis of the grid spacing and the water depth is selected as 20 seconds (corresponding to a *Courant Number* of 2).
- The length of the *warm-up period* has been chosen to 20 minutes to allow for a soft start of the wind forcing and thereby avoid chock effects.
- The wind friction coefficient, which has been set to 0.0026.
- The turbulence model, for which a constant eddy viscosity of 100 m^2/s has been selected.

3.2.2 Extracting Data for Plotting

After execution you may extract a time series of surface elevations in order to see the effect of the wind. The time series extraction tools are located in the MIKE Zero Toolbox.

In Figure 3.2, a time series of the water level (or elevation) in point (49, 10) is plotted.

The resulting plot should correspond to Figure 3.2.

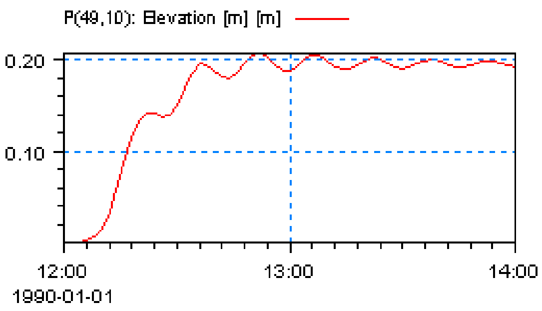


Figure 3.2 Time series of the wind set-up in a rectangular lake

3.2.3 Evaluating the Results

The equations on which the calculations are based are given in the Scientific Documentation. Assuming a static equilibrium between the wind set-up and the wind shear stress, the wind set-up can be determined from:

$$\frac{\Delta h}{L} = \frac{\tau_{wind}}{\rho_{water} g H}$$

Entering the parameters as you have given them above and assuming that the model has reached a steady solution, you get a set-up of $4 \cdot 10^{-5}$ m/m. Thus, as the lake is 4900 m long, the total set-up at the eastern end of the lake under steady state conditions should be slightly below 0.2 m. This corresponds closely to the final result given in Figure 3.2.

3.2.4 List of Data and Specification Files

The following data files are supplied with MIKE 3:

Name: bathy
Description: Lake bathymetry

The following specification files were used together with the specified tasks for running the simulation:

File: Lake.M3
Task: m3hd (Hydrodynamic Module)
Description: Flow simulation

3.3 The Sound

3.3.1 Purpose of the Study

This example is taken from the Oresund Link project, which is a study of a fixed link positioned in the sill area in the narrow sound between Denmark and Sweden involving an immersed tunnel, an artificial island, a high bridge, and compensation dredgings.

From different reports of the study we quote:

"The flow in Oresund is forced by a water level difference between the northern and southern boundary (pressure), the wind and the variations of water density through Oresund. The forcing is mainly balanced by friction. ... The flow close to the link is a complicated one-, two- and/or three-layer system. ... When the current changes from N to S flowing high saline water starts flowing to the south across the sill. ... When the current in the sill area changes from S to N flowing, most of the water with salinity higher than approx. 10 PSU which passed the sill during the southward flow continues to flow southward along the bottom. "

"The environmental impact assessment of the Oresund Link with respect to water flow and salt flux is addressed by means of numerical models. The modelling programme has been designed to meet the requirements of the different phases of the Oresund Link project. ... Based on recommendations put forward by the International Expert Panel ... a modelling strategy has been launched. ... The hydrographic investigations in Oresund comprise an ongoing measurement programme. ... Parts of these data have been used to prescribe the variations in water level and salinity at the boundaries. ... The remaining data has been used to calibrate the models. ..."

"The Oresund model based on MIKE 3 has been calibrated and validated against measurements of water level, current and salinity. ... The philosophy behind obtaining a calibrated and validated Oresund model may be explained as follows: If the Oresund model is calibrated and validated, then it is possible to include the link and the compensation dredgings in the model and evaluate the blocking. The link design may also be altered to optimize the layout considering the environmental and economical response."

It should be noted that only a minor part of the work, which was carried out, is described here, and that the full model set-up is not supplied with this release.

3.3.2 About the Model Set-up

The present model example has been set up as a simplified MIKE 3 model of the Oresund reference situation (i.e. corresponding to the situation without any marine works included). The simplifications made compared to the original model concern mainly the spatial model resolution, the temporal coverage of the simulation, and the degree of verification model results against measurements.

Whereas the original Oresund model consists of a three-level dynamical nesting, the present example includes only the 900m main area bathymetry, see Figure 3.3. The vertical grid spacing is 1 m, which is necessary to resolve the stratification in Oresund. The projection zone is the local DKS. Existing lighthouses, which are not resolved by the grid, are included as pier data.

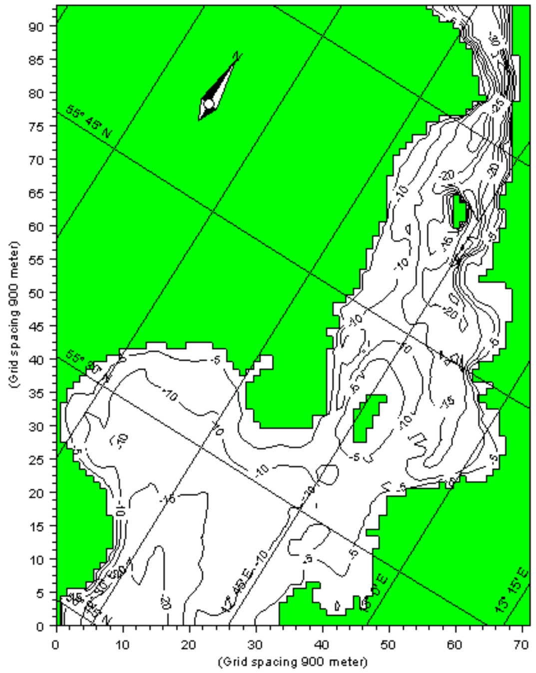


Figure 3.3 The Oresund 900 m bathymetry

The model forcings are based on measurements of water level, temperature and salinity variations near the open model boundaries and measurements of wind speed and direction in the central part of the model.

- Measurements and analysis of the local hydrography yielded line series of water level boundary variations of which some of the data applied at the northern boundary is shown in Figure 3.3: A non-linear variation along the boundary is applied. Since obtained from measurements the water level boundary variations include air pressure variations.

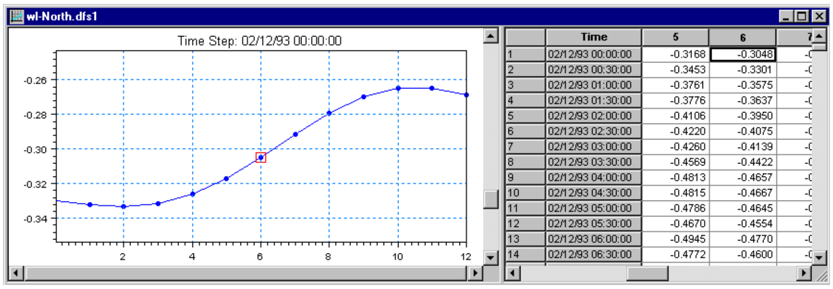


Figure 3.4 Water level at northern boundary

- Stratification and baroclinic forcing is obtained from analyses of measured CT data from which vertical grid series of temperature and salinity have been constructed and prescribed as boundary conditions during both inflow and outflow. An example of salinity at instant with a three-layer flow at the northern boundary is shown in Figure 3.4. The salinity at the southern boundary is kept constant at 7 PSU.

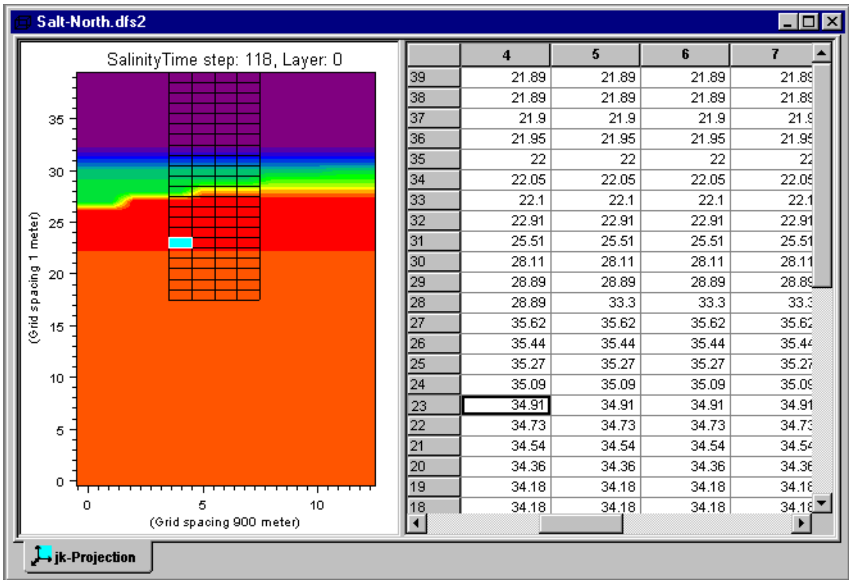


Figure 3.5 Salinity at northern boundary

- A time series data file constructed from the measured wind data is applied as the wind forcing at the free surface. Note, air pressure variations within the model area and air pressure corrections of the water level boundaries should not be applied in this case.

Other important model specifications include initial salinity, temperature and surface elevation fields, time step, bed roughness, and turbulence model. The reader is encouraged to open the supplied specification file from the

MIKE Zero shell (using File-->Open and browse for the Sound directory) and find these model settings.

3.3.3 Model Evaluation

If the simulation defined by the specifications in Sound.m3 is executed (select 'Run-->Start simulation' from the Main Menu bar), two result files will be generated:

- a type 2 data file, Wlresults, containing simulated surface elevations
- a type 3 data file, UVSTresults, containing simulated current, salinity and temperature

From the MIKE Zero Toolbox you may apply the data extraction specifications Wlextraction.tst and SaltExtraction.tst (both supplied under the Sound example folder) to extract time series of simulated water level and salinity. You may then compare these directly to time series of measurements (type 0 data files wl-Measured and MeasuredS-1 in the Sound folder), see Figure 3.6 and Figure 3.7 for some examples.

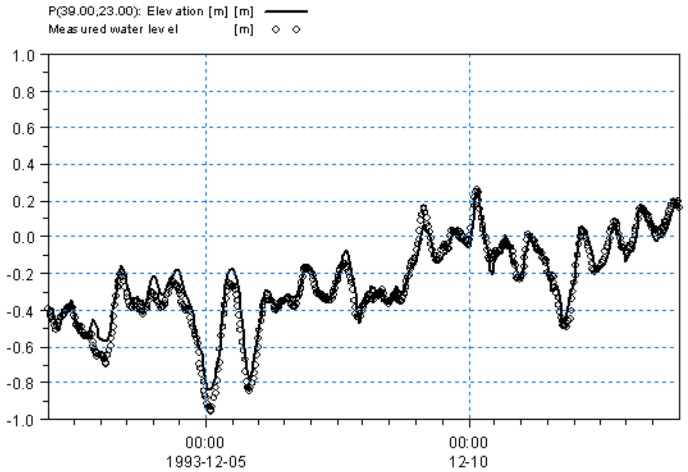


Figure 3.6 Comparison between measured and simulated water levels

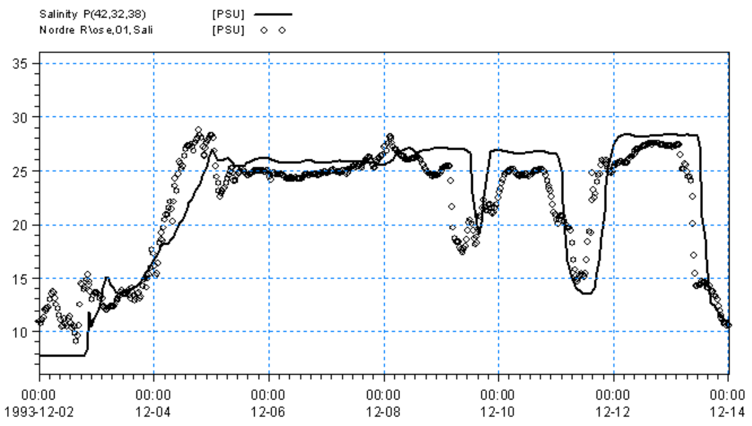


Figure 3.7 Comparison between measured and simulated salinities

3.3.4 List of Data and Specification Files

The following data files are supplied:

Name: bathy900m
Description: Oresund bathymetry in 900 metres resolution

Name: wl-North, wl-South
Description: water level variation at the northern and southern model boundaries, respectively (line series data)

Name: Salt-North
Description: Vertical section (type 2 data file) of salinity variations at the northern model boundary

Name: Temp-North, Temp-South
Description: Vertical sections (type 2 data files) of temperature variations at the northern and southern model boundaries:

Name: InitSalt900m, InitTemp900m
Description: Initial salinity and temperature (type 3 data files)

Name: InitElev900m
Description: Initial surface elevation (type 2 data file)

Name: Wind
Description: Wind data (type 0 data file)

Name: Piers
Description: Pier data (type 1 data file)

The following specification files were used together with the specified tasks for running the simulation:

File: Sound.M3
Task: m3hd (Hydrodynamic Module)
Description: Flow simulation

3.4 Turtle Bay

3.4.1 General Remarks

This example has been chosen as a fairly complex one, involving the nested hydrodynamic and the nested advection-dispersion modules of the MIKE 3 Flow Model.

The model set-up includes:

- two-level nesting with an outer 120 m grid and an inner 40 m grid,
- source specifications
- flooding and drying since the Turtle Bay is dominated by large tidal flats
- transfer data from a MIKE 21 simulation for HD boundaries in the MIKE 3 model
- conservative and decaying AD components
- constant, time series and line series AD boundary specifications.

You may load the specification (i.e. the **m3** file, see below) into the MIKE 3 Flow Model set-up editor and make your own modifications through the dialogs. However, to be able to run the actual model simulations requires license to the nested hydrodynamic and/or the nested advection-dispersion modules.

In case you have a license for the nested hydrodynamic module but not for the nested advection-dispersion module and want to run the corresponding HD model without the AD, then you may use the sample set-up provided in the HD-directory of the MIKE 3 Flow Model examples.

3.4.2 About the Model

A few comments are given to the model set-up which aims at demonstrating HD/AD modelling with two different components (conservative and decaying) in a rather complex area exhibiting extensive flooding and drying.

Time step

The time step has been chosen to be 5 sec yielding a maximum Courant number of approx. 2.1 in the fine grid area.

Bathymetry

Having generated the bathymetry files (using e.g. the **Bathymetry Editor**) it is necessary to adjust the regions surrounding the borders, see *Nested Bathymetries (p. 115)*). A tool to help the user doing these adjustments is found in the **Hydrodynamics** part of the **MIKE 3 Toolbox**. The resulting type 2 data files are supplied with this example and plotted in Figure 4.1. The origin of the 40 m fine grid is (31, 40) in coarse grid coordinates.

Vertical resolution

5 layers of 1 m have been chosen.

(Grid spacing 120 metres)

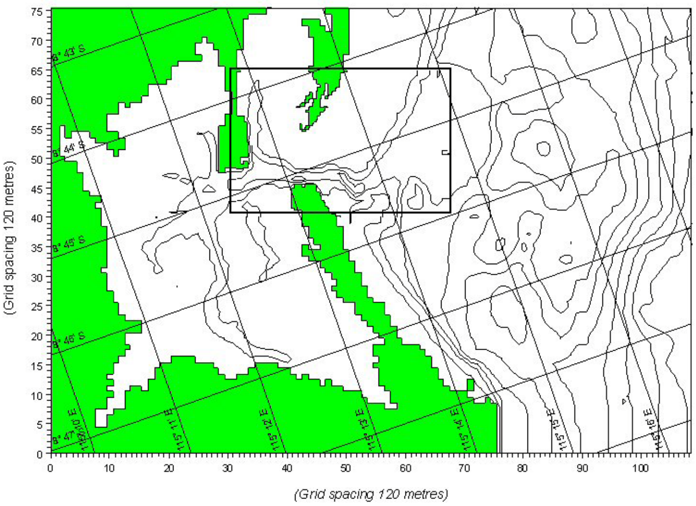


Figure 3.8 Nested bathymetries for the Turtle Bay example

HD Boundary conditions

Transfer data has been obtained from a MIKE 21 tidal simulation by a regional model covering the above show area by use of the **Transfer Boundary** tool in the **Hydrodynamics** part of the **MIKE 21 Toolbox**. The resulting type 1 data files are supplied with this example. The hydrodynamic MIKE 3 model is set-up with these data applying velocity as primary boundary variation for the north and east model boundaries and level as primary boundary variation on the south model boundary.

AD Boundary conditions

The AD components on the open boundaries are specified as constants on the north boundary, as read from a type 0 data file on the south boundary, and as read from a type 1 data file on the east boundary. These data files may be created by use of the **Time Series Editor** and the **Profile Series Editor**, respectively. Note, that both of these AD boundary data files contains three items, but the model is set-up for only two components - a conservative component and a linearly decaying component. This means that the user needs to specify which data file item corresponds to which model component.

Eddy viscosity

The velocity based Smagorinsky formulation has been applied with Smagorinsky coefficients of 0.088 in the horizontal direction and 0.176 in the vertical direction, which corresponds to the default values in both areas.

Source specifications

A single unconnected, constant source has been placed at grid point (20, 20, 5) in the fine grid.

3.4.3 Data and Specification Files

The following data files are supplied:

Name: coarse, fine
Description: Coarse and fine grid bathymetry, respectively (type 2 files)

Name: nBnd120lwx, ebnd120lwx, sbnd120lwx
Description: type 1 transfer data files for the north, east and south model boundaries

Name: bndts
Description: time series AD boundary condition

Name: bndls
Description: line series AD boundary condition

The following specification files were used together with the specified tasks for running the simulation:

File: TurtleBay.M3
Task: m3hd (Hydrodynamic Module)
Description: Flow simulation

File: TurtleBayAD.M3
Task: m3ad (Advection/Dispersion Module)

4 Basic Parameters Dialog Overview

In this section, you set up the basic parameters for your MIKE 3 Flow Model simulation. You need to specify parameters for

Module Selection
Bathymetry
Simulation Period
Boundary
Source and Sink
Flood and Dry
Turbulence Model
Mass Budget

Selections made here determine the structure of the rest of the model setup editor, ie the required entries for the hydrodynamic module and possible add-on modules. The tree-view and the number of available dialogues will expand according to the selections made for the basic parameters.

4.1 Module Selection

The module selection dialog is shown in Figure 4.1.

On this dialog you first choose between

- hydrostatic, or
- non-hydrostatic

hydrodynamic engine. The first one applies a hydrostatic (HS) pressure assumption, whereas the last corresponds to the “classic” MIKE 3 engine applying an artificial compressibility method (ACM).

On this dialog you also make your selection to explicitly include

- density variations
- environmental modules
- sediment

by ticking one, or more, of the corresponding check boxes.

In the MIKE 3 Flow Model,Density Variation is enabled by selecting salinity and/or temperature variations. The density of sea water is also affected by possible mud concentration. Thus, if mud transport is selected it is required that you also enable at least one of the density options, ie salinity and/or temperature variations.

The environmental modules available with the MIKE 3 Flow Model are

- the Advection-dispersion (AD) module for simulations of transport and fate of passive scalars
- the MIKE ECO Lab (EL) module for simulations of water quality, eutrophication, heavy metals and ecology using process oriented formulations

The sediment module available with the MIKE 3 Flow Model is

- the mud transport (MT) module for simulation of processes regarding transport of cohesive sediments

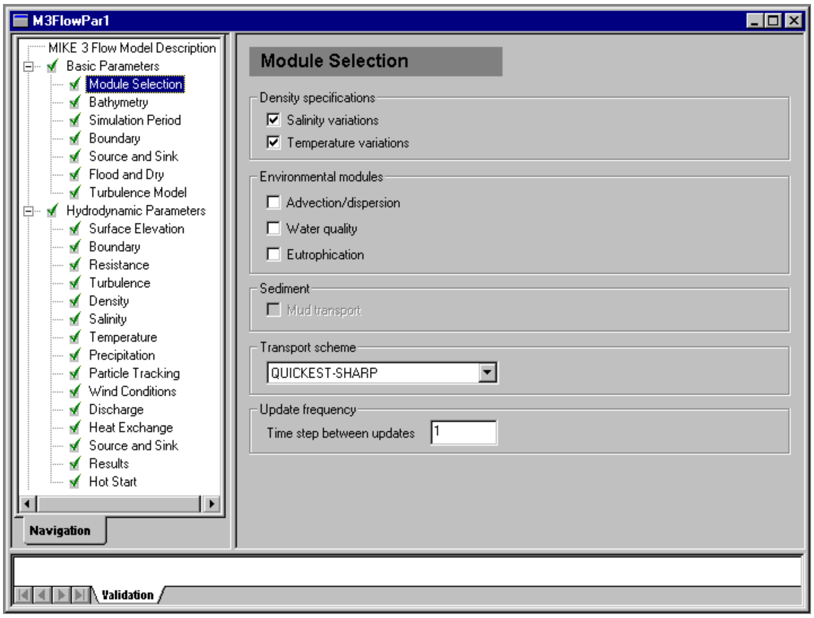


Figure 4.1 The module selection dialog

Having selected one or more of the above modules, you need to specify which transport (or advection-dispersion) scheme you wish to use. You have four options

- The fully 3D QUICKEST-SHARP scheme which is especially suitable for simulations with steep gradients
- The ULTIMATE-QUICKEST scheme with operator splitting and intermediate surface elevations calculated on basis of locally 1D continuity equations. The ULTIMATE-QUICKEST scheme is an alternative to the QUICKEST-SHARP scheme and it is designed to reduce computation time when more than one component has been selected
- The simple UPWIND scheme which is similar to the ULTIMATE-QUICKEST scheme except that upwinding is applied all over

- The fully 3D UPWIND scheme which is similar to the QUICKEST-SHARP scheme except that upwinding is applied all over

The ULTIMATE-QUICKEST and the simple UPWIND schemes have a build-in internal loop over components which reduces the computation time when more components have been selected. With the QUICKEST-SHARP and the 3D UPWIND schemes the user may choose to turn on the “internal component loop” by ticking the corresponding checkbox; this will increase the computational speed in cases with more than one component at the expense of requiring more memory during computations.

The schemes 3D QUICKEST-SHARP and the 3D UPWIND are so-called CWC schemes (for Consistency With Continuity), both designed to be consistent with the continuity equation (the mass equation) of the HD module.

Finally, you may choose to update the concentration(s) at every (say) 2nd time step in order to reduce the computation time compared to updating at each time step. This is done by changing "Time steps between updates" from the default 1 to 2. However, take care when increasing this update interval because it will also affect the updating of possible turbulence variables (Eddy Viscosity, turbulent kinetic energy and dissipation of turbulent kinetic energy), since the turbulence module applies the same update frequency.

4.2 Bathymetry

The bathymetry specification dialog is shown in Figure 4.2.

You can start a simulation in two different ways, see also Simulation Type *(p. 110)*.

- As a cold start where the initial velocity field is initialised to zero. Selecting this option you have to specify the number of nested areas and the data files containing the bathymetry for each area. A maximum of 9 nested areas is allowed. The outermost area, i.e. the one with the coarsest grid, is referred to as the main area and will always be area No. 1 in all dialogs. Except for the main area you should also specify the origin of each nested (embedded) area. This is done in terms of the horizontal grid coordinates referring to the (sub-) area in which it is enclosed
- As a hot start where the initial data is taken from a previous simulation. Selecting this option you must specify type 3 data files containing the 'hot start' information

A cold start requires information about the number of areas and a bathymetry for each of these areas. Two types of bathymetry data files are allowed

1. Type 2 data files containing bathymetric information. This data file must obey the standards for bathymetric files in MIKE 3. Pre-processing tools (e.g. the Bathymetry Editor or the Grid Editor) from MIKE Zero can be

applied to create these data files. Before doing a nested model simulation, make sure that your bathymetries adhere to the rules for nested bathymetries: You should run the Border Adjustment tool from the MIKE 3 Toolbox prior to any nested model simulation

2. Type 3 data files containing the bathymetric information. There is presently no preprocessing programs available for border adjustment of such bathymetries

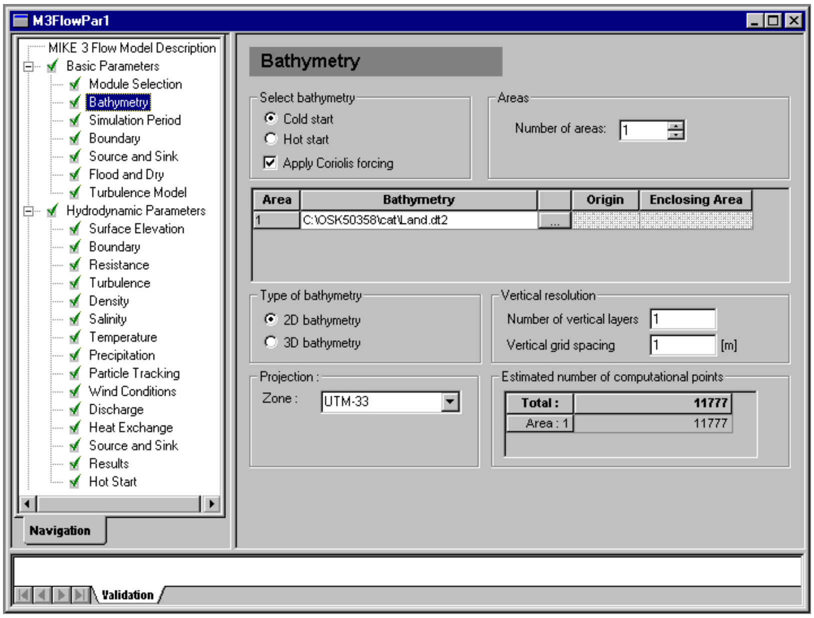


Figure 4.2 The bethymetry specification dialog

If the main area bathymetry data file contains geographic information, you may opt to include the Coriolis force in your simulation and you should specify the relevant projection zone for your MIKE 3 project. A map projection is suggested based on the information from the file.
Note: It is not possible to use a bathymetry specified for a Geographical Coordinate System such as e.g. LONG/LAT.

For type 2 bathymetry data files, the number of vertical levels and grid spacing must be specified. To allow for a 'cut-off' level, both parameters can be specified. Default values are a vertical spacing of 1 m and a number of levels corresponding to the depth. The vertical resolution is the same for all model areas.

For background details and recommendations in connection with bathymetry specification see Bathymetry *(p. 59)*, especially Additional Area Description *(p. 66)* and Standard vs. Nested HD Modules *(p. 114)*.

An example with a step-by-step description of how to use the Bathymetry Editor for creating a bathymetry data file is included with the installation. Please find this example in your installation folder under Examples\MIKEZero\BatEdit. If you apply nested models, the Border Adjustment tool should be applied to all the bathymetry data files, see the MIKE 21 Toolbox manual.

Before you leave the bathymetry specification dialog, the number of computational points per time step in each area will be shown.

4.3 Simulation Period

The simulation period specification dialog is shown in Figure 4.3.

On this dialog you must give simulation time information

- Time step range is the number of time steps the simulation should cover. Note, the simulation always starts with time step number 0 (which are the initial conditions)
- The time step interval (in seconds) is the time the historical time is incremented by at each time step. Note, this parameter is very important with respect to the numerical stability, see Time Step.
- The simulation start date is the historical date and time corresponding to time step 0
- The warm-up (or sort start) period, i.e. a number of time steps during which the is applied boundary variation is building up. This may be applied to avoid shock waves in the computations

Before you leave this dialog, the system will calculate and show you the maximum Courant Number. You should consider switching to a smaller time step if this value gets above 8 to 10.

Also, the end date of your simulation is presented to you for reference.

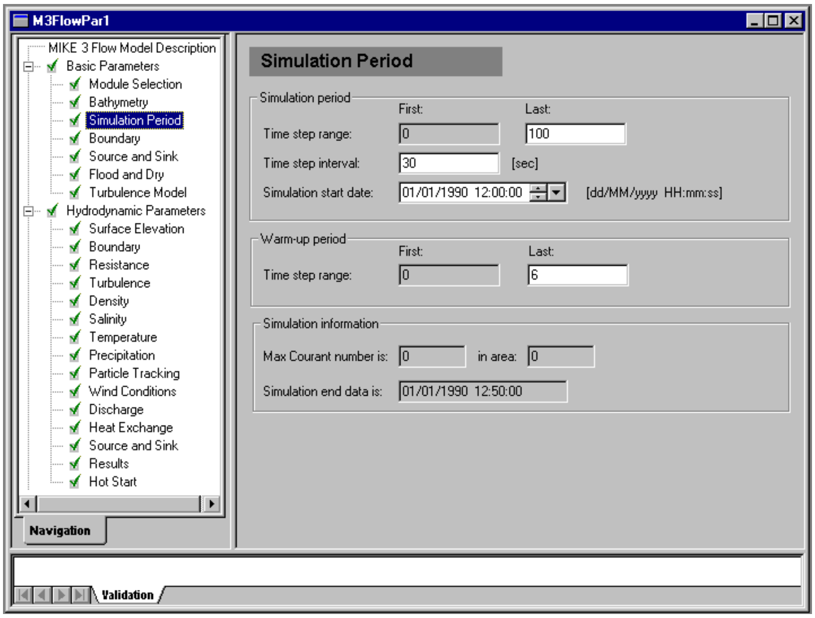


Figure 4.3 The simulation period specification dialog

4.4 Boundary

The boundary basic parameters dialog is shown in Figure 4.4.

On this dialog you specify where the open lateral boundaries of your model are positioned within your (outermost, main) bathymetry.

The open boundaries can either be detected by the model system itself or can be explicitly specified by the user. A maximum of 8 open boundaries is allowed.

For program detected boundaries, the system scans the extremes of your main area bathymetry for water points and make a corresponding suggestion as to the number and position of open boundaries.

In most cases boundary positions as detected by the program may be used; but in a few cases the user might want to define the positions himself. It can be due to a boundary streching over a series of small islands, due to so-called "internal" boundaries, etc.

You may also choose to enable the zero-gradient boundary condition.

For background details see Boundary Conditions *(p. 70)*.

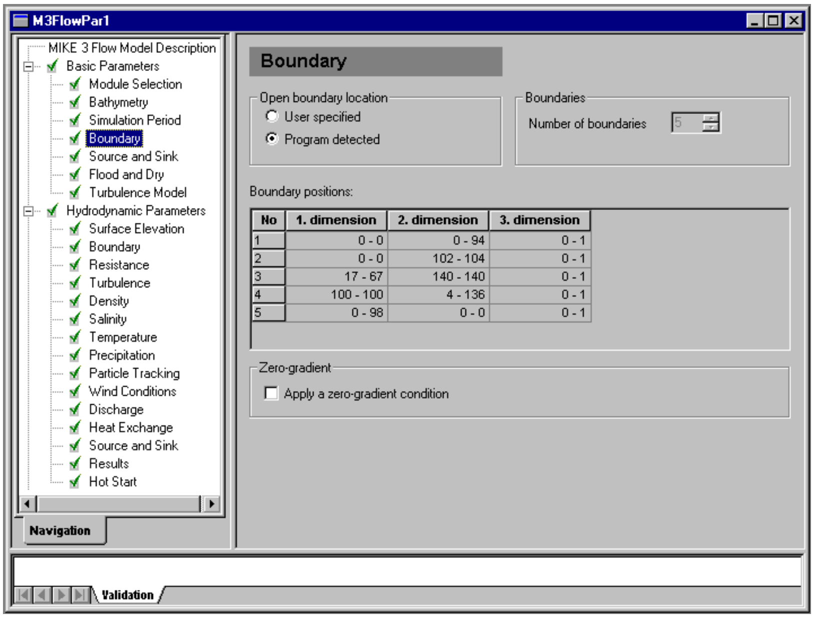


Figure 4.4 The boundary basic parameters dialog

4.5 Source and Sink

The source and sink basic parameters dialog is shown in Figure 4.5.

Point sources, and point sinks, can be included in the model. A total of 300 source/sink points is allowed.

The model distinguishes between two different kinds of sources/sinks:

- Unconnected (or isolated) sources/sinks: isolated sources where a certain amount of water is discharged into the model with a specified velocity. Thus, isolated sources affects both the continuity and momentum equations. At isolated sinks a certain amount of water is discharged out of the model, and only the continuity equation is affected.
- Connected source-sink pairs, used for recirculation studies. The amount of water removed at the sink point is reentered at the source point, with a specified velocity.

The source/sink points are defined by their coordinates in the area within which the source/sink is located.

Do not place the sources/sinks on land, and be careful when placing sources/sinks at locations that may occasionally dry out.

For background details see Source and Sink *(p. 112)*.

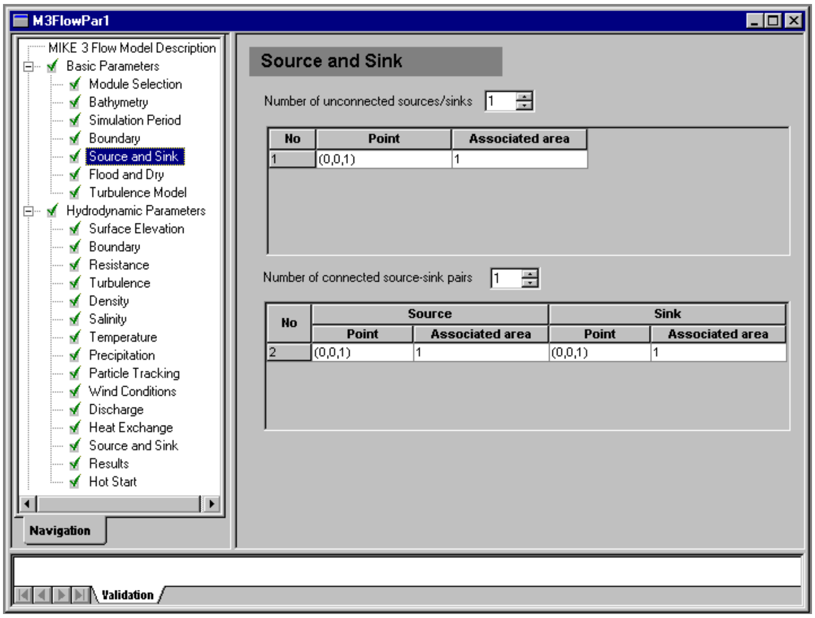


Figure 4.5 The source and sink basic parameters dialog

4.6 Flood and Dry

If your model is located in an area with tidal flats, you can enable the flood and dry facility. In this case you have to specify a drying water depth and a flooding water depth. These two depths are used to determine whether a point should be dried (and thus be taken out of the computations), or flooded (i.e. reentered into the calculations).

The flooding and drying parameter specification dialog is shown in Figure 4.6.

If flooding and drying is not enabled, you should specify a minimum water depth, i.e. the water depth above which computations are allowed to continue. If at a point in your model the water depth goes below this value, a blow-up is detected and the simulation halted.

Flooding and drying is not permitted at model open boundaries. This should be taken into account when the bathymetries are set up.

For background details see Flood and Dry *(p. 86)*.

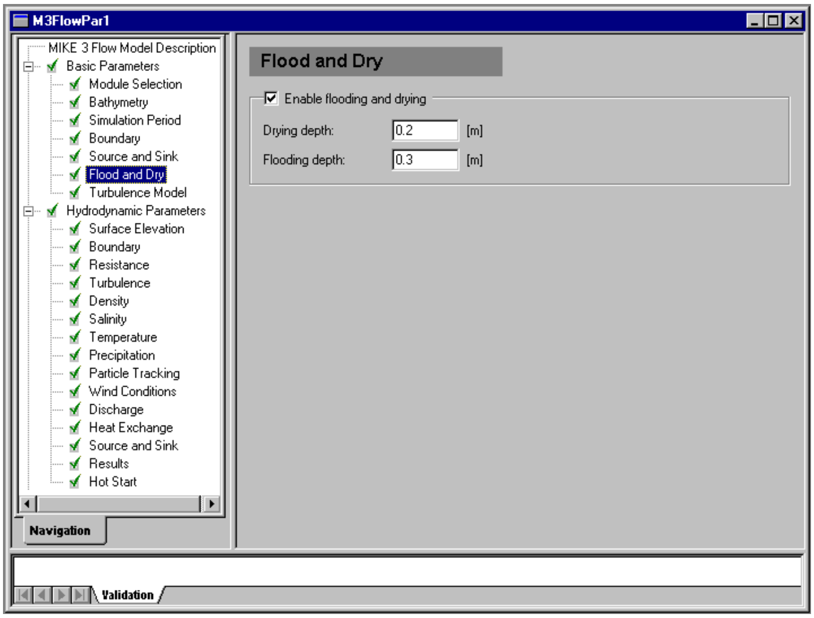


Figure 4.6 The flood and dry parameter selection dialog

4.7 Turbulence Model

The turbulence model dialog is shown in Figure 4.7.

The turbulence is modelled in terms of an Eddy Viscosity and a bed shear stress. The eddy viscosity can be specified in one of five different ways, yielding the choices

1. The terms in the equations can be omitted, i.e. no eddy viscosity.
2. A constant value is applied in the entire area.
3. Dynamically calculated by means of the Smagorinsky formulation.
4. Dynamically calculated by means of a standard k-model (k is the turbulent kinetic energy).
5. Dynamically calculated by means of a standard k-ε model (ε is the dissipation of turbulent kinetic energy).
6. Dynamically calculated by means of the mixed k-ε/Smagorinsky formulation with a standard k-ε model in the vertical and a Smagorinsky formulation in the horizontal.

The time interval between updates of dynamically calculated eddy viscosities is specified on the Module Selection dialog.

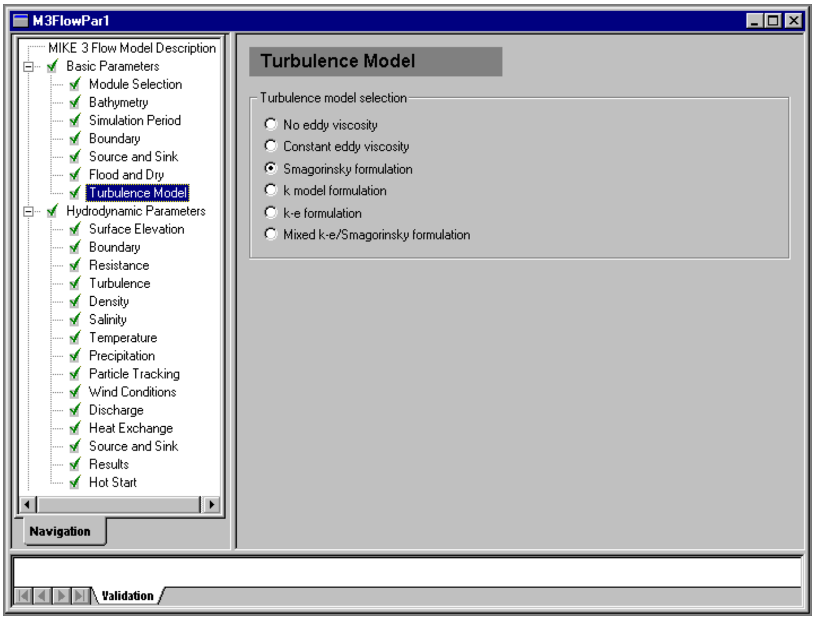


Figure 4.7 The turbulence model selection dialog

Please Note: If you have selected the hydrostatic engine on the Module Selection dialog, then it is recommended that you choose a turbulence model; you should not select the option “No eddy viscosity” if the hydrostatic model has more than one layer. This is because some vertical eddy is needed to “glue” the layers together due to not having a vertical momentum equation in the hydrostatic version.

4.8 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 corresponding to the mass budget has to be defined. Secondly the mass budget contents and output file have to be defined. The latter is performed in the Dialogs of the individual Modules whereas the former is performed in the Basic Parameters Dialog.

The area corresponding to the mass budget is represented as a polygon defined within the model domain. Notice that it is possible to specify several mass budget polygons allowing for several mass budgets. The following information has to be specified:

- For each polygon the associated computational area (main area/ sub-areas; only relevant for nested models) is specified.
- After that the 'ZRange', which defines the vertical extension of the mass budget area, is given.
- By enabling the calculation of section transports, the mass budget will provide the transport through each lateral section of the mass budget area.
- Finally the horizontal extension of the mass budget area is defined by specifying the number of corner points in the polygon and the grid coordinates of the corner points.

Please Note: A polygon can only contain grid points one grid point or more inside the associated computational area; i.e. grid points on boundaries or on borders of possible nested computational areas can not be included. Further a polygon can not traverse a possible nested computational area. A polygon can, on the other hand, contain land points; the model will simply exclude the land points, when calculating the mass budget.

5 Dialog Overview

The hydrodynamic (HD) module of MIKE 3 is a mathematical modelling system for calculation of the hydrodynamic behaviour of water in response to a variety of forcings, i.e. Wind Conditions and water level variations at the open boundaries.

In the dialogs under this Hydrodynamic Parameters section you specify the HD related parameters for your MIKE 3 Flow Model simulation. The number of available dialogs and the actual contents of the dialogs is in part determined from your choice of Basic Parameters Dialog Overview, i.e. the specified number of areas entered on the Bathymetry dialog determines the size of most dialog-grids under in this hydrodynamic parameters section. As another example, the pick of turbulence closure model (as opposed to "No Eddy Viscosity") under Turbulence Model will reveal a Turbulence dialog, the content of which corresponds to the chosen eddy viscosity formulation.

5.1 Surface Elevation

The surface elevation dialog is shown in Figure 5.1.

The initial surface elevations relative to the datum level can be specified in each area as either a constant value or taken from type 2 data files whereby each grid point is assigned its own value.

To avoid generation of shock waves, it is recommended that your initial surface elevation roughly matches the boundary conditions at the start of the simulation.

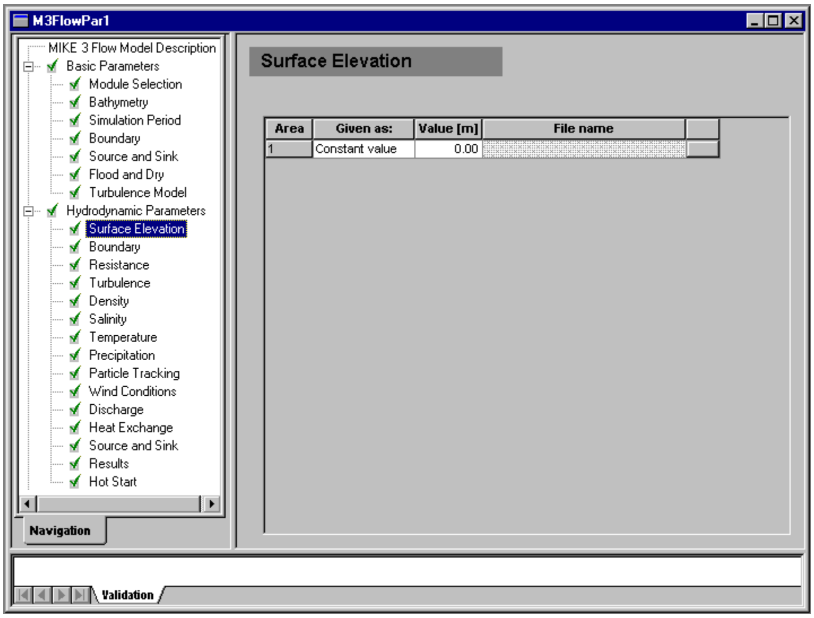


Figure 5.1 Surface elevation dialog

5.2 Boundary

The hydrodynamic boundaries dialog is shown in Figure 5.2.

The model equations requires you to specify either the pressure or velocities at all open boundary points. The surface elevation may be specified as an alternative to the pressure. In all cases the reference level of the boundary data must equal the reference level of the bathymetry data.

The values specified at an open boundary can be either of type pressure (alternatively level/surface elevation) or velocity. The actual values can be specified in five different formats. In case of data transfer, the prescribed values are obtained from the results of an earlier simulation. Therefore, it must be combined with either a type 1 data file, in which case the data originates from an earlier MIKE 21 simulation, or a type 2 data file in which case the data stems from an earlier MIKE 3 simulation. For transfer data, the primary type of your boundary condition must be specified as either velocity or pressure.

Please Note: Pressure boundaries and data transfer boundaries can only be applied with the non-hydrostatic engine. More over, with the hydrostatic engine, vertical velocities at the boundaries are set to zero.

The boundary variation can be specified in up to five different ways, dependent on the type of boundary:

1. as a constant value
2. as a sinusoidal function
3. as a type 0 data file
4. as a type 1 data file
5. as a type 2 data file

For type 1 data files the one dimension is always in the horizontal direction not the vertical.

Please Note: The soft start, i.e. building up the values through a number of time steps, only applies to constant boundary values. This is to avoid shock waves in the computations. If you have time varying boundary data, it is recommended that you build the soft start into the data prior to setting up the model (a pre-processing step).

For background details see Boundary Conditions *(p. 70)*.

For surface elevation boundaries *tilting* can be applied provided the variation is either constant, sinusoidal or of type 0. In regions where Coriolis force is of importance such a may be useful. The tilting of boundaries is handled implicitly as a quasi-geostrophic balance. Sometimes this may cause unintended problems.

A VAB type (Velocity-Along-Boundary) can be specified for pressure/elevation boundaries.

The flow direction is kept constant throughout the simulation. The horizontal direction is relative to true North. The vertical direction is relative to the horizontal plane and positive upwards.

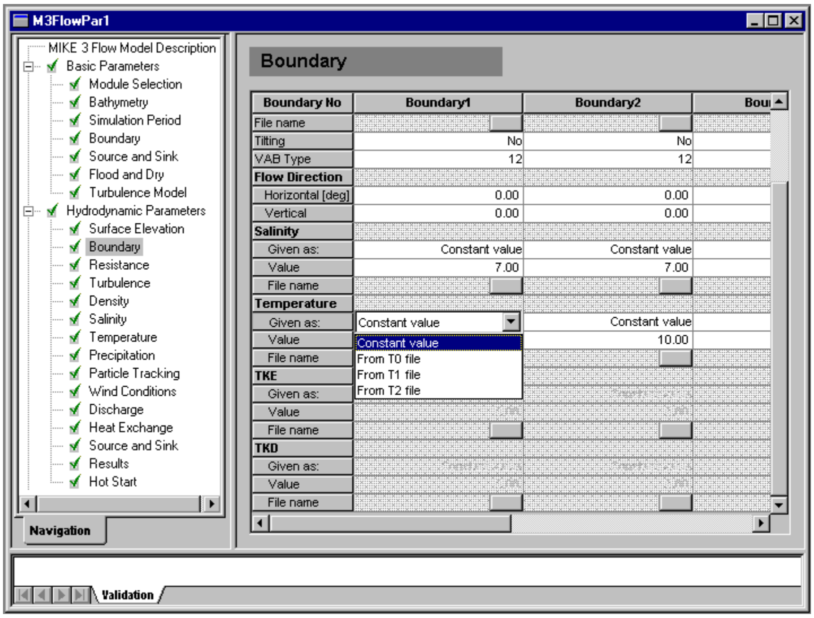


Figure 5.2 Hydrodynamic boundaries dialog

The boundary variation in salinity, temperature, TKE and TKD (if present) must also be defined. This can be specified in four different ways

1. as a constant value
2. as a type 0 data file
3. as a type 1 data file
4. as a type 2 data file

Please Note: For type 1 data files the one dimension is always in the horizontal direction not the vertical. The values are then distributed uniformly over the vertical.

5.2.1 Sine Description

The sine variation dialog is shown in Figure 5.3.

A sine variation at the boundary may be selected, but only if a warm-up period of zero length is set. A special dialog then appears for the specification of Amplitude, Period and Phase values.

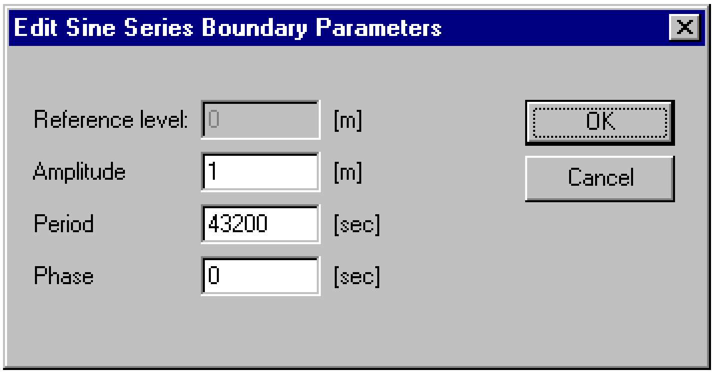


Figure 5.3 The sine variation dialog

5.3 Resistance

The resistance dialog is shown in Figure 5.4.

The bed shear stress is modelled assuming a logarithmic velocity profile near the seabed. This formulation requires information on the bed roughness (in m) and must be specified either as a constant value in the entire area, or as a value for each horizontal node (2D maps).

Furthermore slip factors can be applied. Whenever bed friction is applied the bottom slip factor is overruled.

The effect of bridge piers and pylons can be included in the simulations. This effect is modelled as an additional friction in the momentum equations and does not affect the mass equation. Thus, the piers must be much smaller than the applied grid spacing. The pier data is read from a type 1 data file in which the pier information is stored according to a special standard.

For further background details see Bed Resistance *(p. 68)* and Pier Resistance *(p. 105)*.

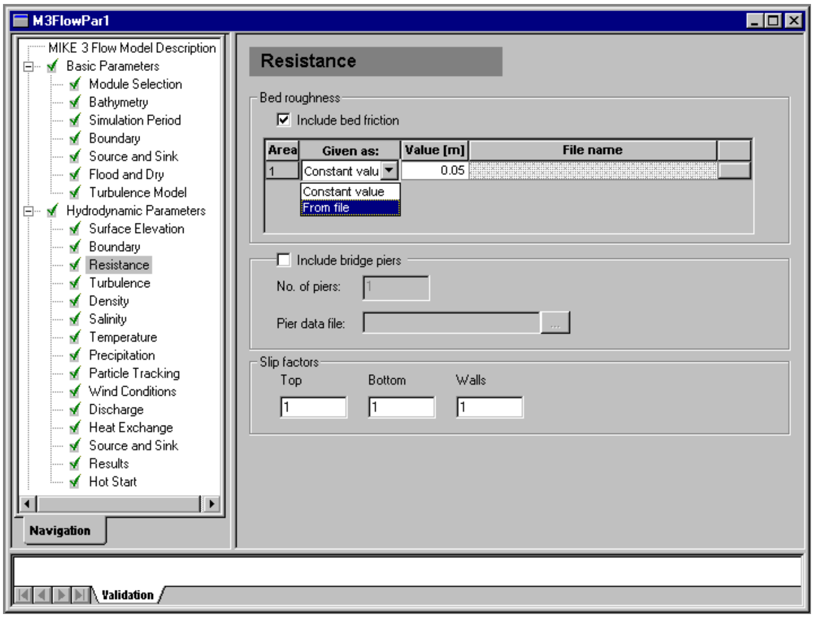


Figure 5.4 Resistance dialog

5.4 Turbulence

The look of the turbulence dialog depends on Turbulence Model *(p. 33)* under Basic Parameters Dialog Overview, as shown in Figures 5.5, 5.6, 5.7, 5.8 and 5.9.

If a Turbulence Formulation has been selected, the dialog corresponding to the chosen model appears. Different turbulence models have different parameters that should be specified. A short description of the parameters for each model is given. For a very detailed description refer to the Scientific Documentation.

5.4.1 Constant Eddy Viscosity

For constant Eddy Viscosity you simply specify the constant value for each area, see Figure 5.6.

5.4.2 Smagorinsky Formulation

Having selected the Smagorinsky subgrid scale model, you need to specify the Smagorinsky coefficients. The coefficients for the horizontal direction may be given as either a constant value or as it is read from a type 2 data file. The coefficients for the vertical directions must be constants, see Figure 5.5.

You also need to provide the limits, lower and upper, for the eddy viscosity.

For background details see Smagorinsky Formulation *(p. 111)*.

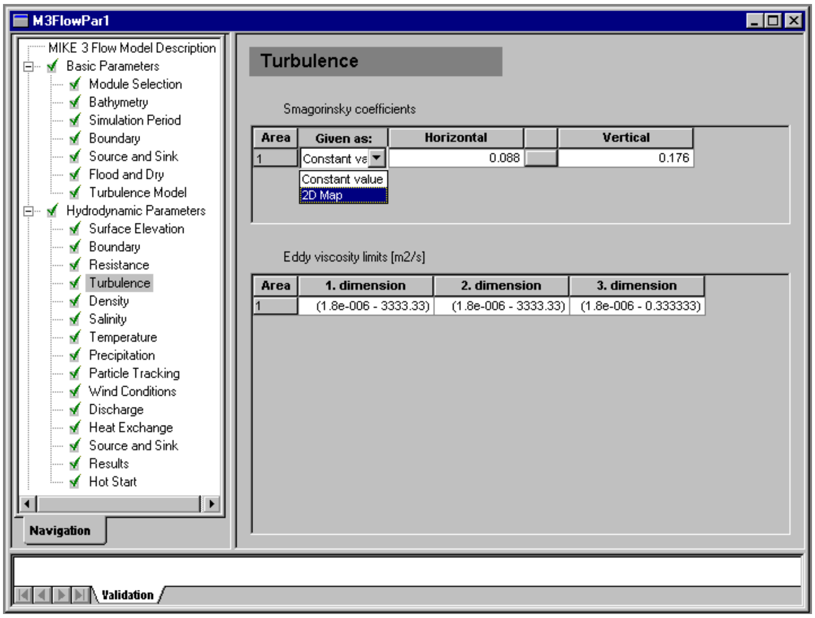


Figure 5.5 Turbulence (Smagorinsky) dialog

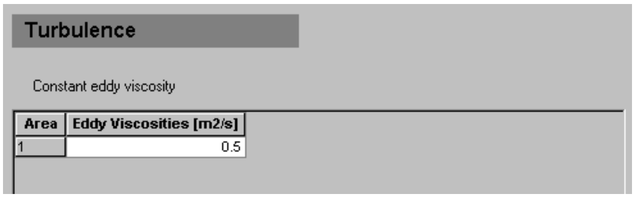


Figure 5.6 Turbulence (constant) dialog excerpt

5.4.3 k Model

The k-model includes a number of empirical constants and diffusion parameters which are based on a large number of experiments. Default values are presented and caution should be taken whenever any of these values are altered. There is, however, some discussions on the value of σ_t which enters the buoyancy term in the k-model. The only parameter which may be chosen freely is the length scale which needs to be prescribed, see Figure 5.7.

A lower limit for k and limits for eddy viscosity and for the dispersion of turbulent kinetic energy must also be specified.

For background details see k-ε Turbulence Model *(p. 100)*.

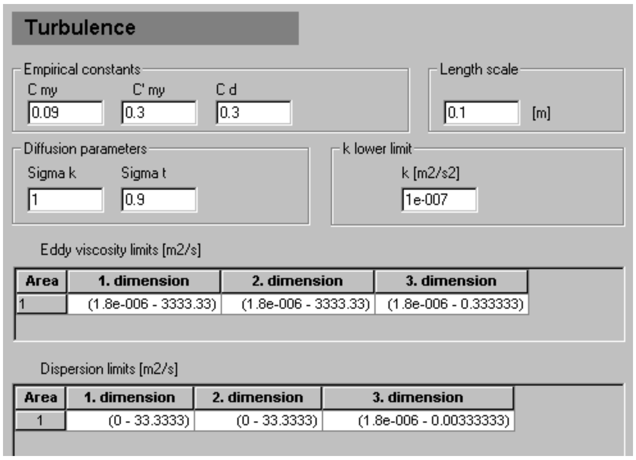


Figure 5.7 Turbulence (k model) dialog excerpt

5.4.4 k- ε Model

The k-ε model includes a number of empirical constants and diffusion parameters which - similar to the k-model - are based on a large number of experiments. Default values are presented and caution should be taken whenever any of these values are altered. There is, however, some discussions on the value of σ_t which enters the buoyancy terms in the k-ε model, see Figure 5.8.

Lower limits for k and for ε as well as limits for the eddy viscosity and for the dispersion of k and ε must also be specified.

For background details see k-ε Turbulence Model *(p. 100)*.

5.4.5 Mixed k- ε /Smagorinsky Model

The 1D k-ε model includes a number of empirical constants and diffusion parameters which are based on a large number of experiments. Default values are presented, Figure 5.9, and caution should be taken whenever any of these values are altered.

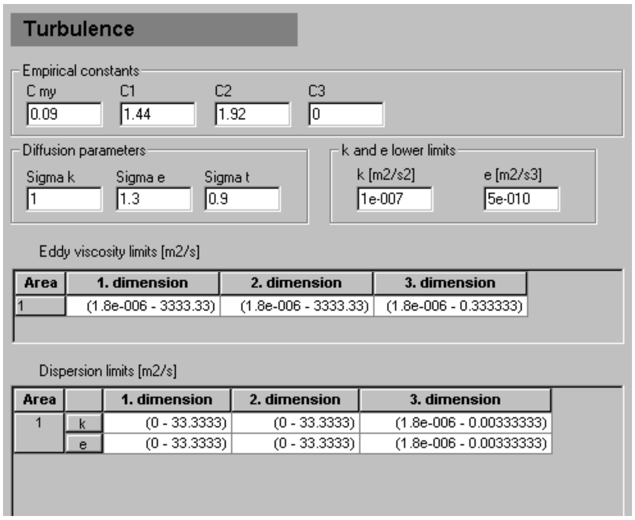


Figure 5.8 Turbulence (k-ε model) dialog excerpt

You also need to specify the Smagorinsky coefficients. The coefficients for the horizontal direction may be given as either a constant value or as read from a type 2 data file. The coefficients for the vertical directions must be constants.

Finally, lower limits for k and for ε as well as limits for the eddy viscosity must be specified.

For background details see Mixed 1D k-ε, 2D Smagorinsky Turbulence Model *(p. 102)*.

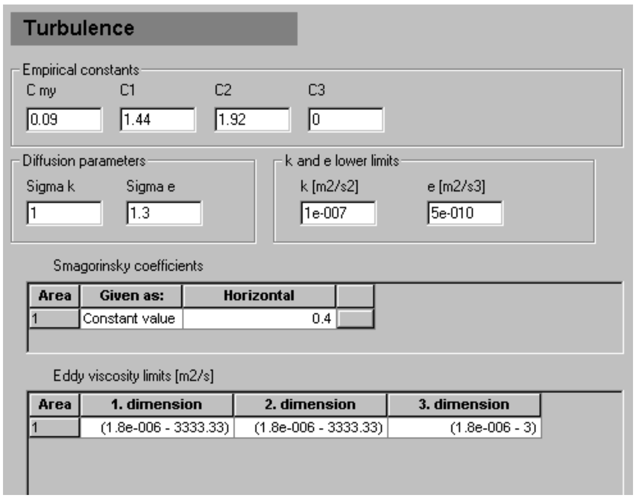


Figure 5.9 Turbulence (mixed) dialog excerpt

5.5 Density

The density dialog is shown in Figure 5.10. The density is a function of salinity and temperature. If you select both a constant salinity and constant temperature the density will not be updated throughout the simulation. When a varying salinity or temperature is selected, some density specifications must be given.

Updating of the density is done according to the update frequency specified on the Module Selection dialog. Updating of the density may be performed with a larger time step than the general time step, but generally it is recommended to use the same time step in both places.

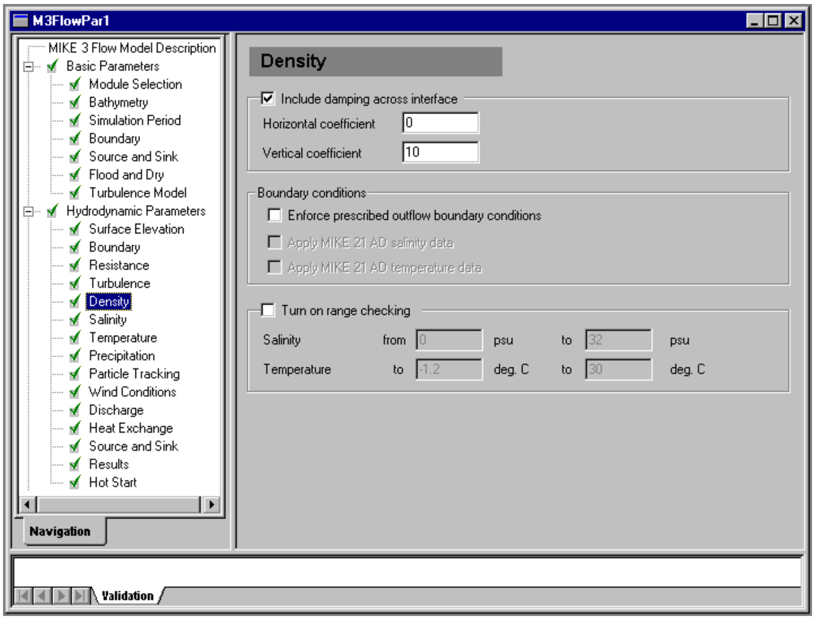


Figure 5.10 Density dialog

Density gradients tend to reduce the shear stresses. Some turbulence models implicitly take this effect into account. In turbulence models where this buoyancy effect is not inherent, an explicit damping can be enforced. You specify the values for the coefficients in the Richardson damping function.

Some boundary condition options may be set on this dialog. By enforcing prescribed outflow boundary conditions, the boundary values are always forced to be the ones specified on the Boundary dialog, also at outflow where the transport scheme solution is then overruled.

Finally, you may optionally turn on range checking [range control]. The values of salinity/temperature are simply cut-off at the specified levels. This can be useful especially if flooding and drying is enabled and if Heat Exchange with the atmosphere occurs with rapid cooling/heating of the sea water. By enabling range checking, however, the mass budget may be violated.

For background details see Density Variation *(p. 80)*, and also Richardson Damping *(p. 109)*.

5.6 Salinity

The salinity dialog is shown in Figure 5.11.

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

The initial salinity variations can be specified either as a constant value or as read from type 3 data files.

Three types of dispersion formulations are implemented

- A linear relationship is most often assumed between the dispersion of salt and the eddy viscosity. The dispersion factor defines this linear relationship
- The dispersion may be proportional to the local velocity
- A tensor dispersion

You must specify the dispersion factors for each area, see Dispersion Coefficients *(p. 83)*.

Furthermore, to control dispersion, lower and upper dispersion factor limits in each direction must be specified for each area.

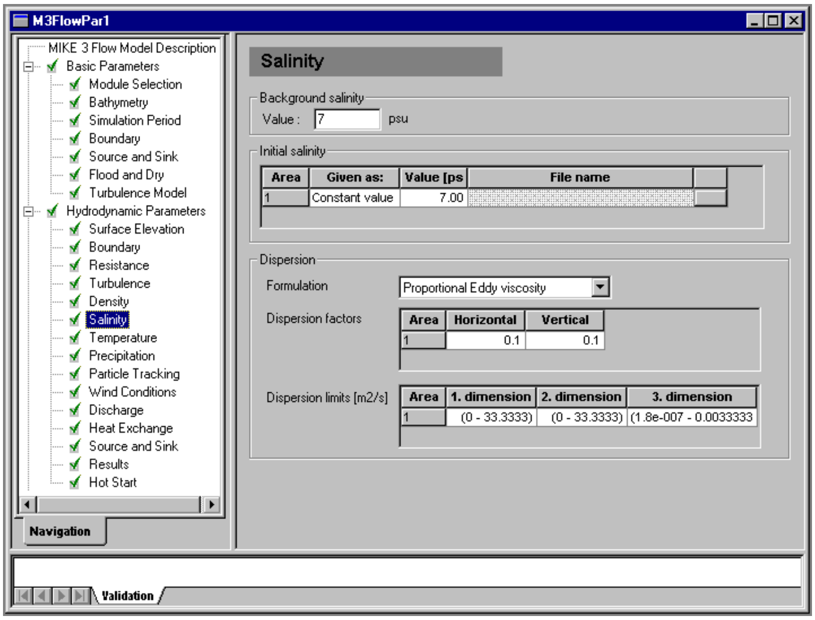


Figure 5.11 Salinity dialog

5.7 Temperature

The temperature dialog is shown in Figure 5.12.

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

The initial temperature variations can be specified either as a constant value or as read from type 3 data files.

Three types of dispersion formulations are implemented:

- A linear relationship is most often assumed between the dispersion of temperature and the Eddy Viscosity. The dispersion factor defines this linear relationship
- The dispersion may be proportional to the local velocity
- A tensor dispersion

You must specify the dispersion factors for each area, see Dispersion Coefficients *(p. 83)*.

Furthermore, to control the dispersion, lower and upper dispersion limits in each direction must be specified for each area.

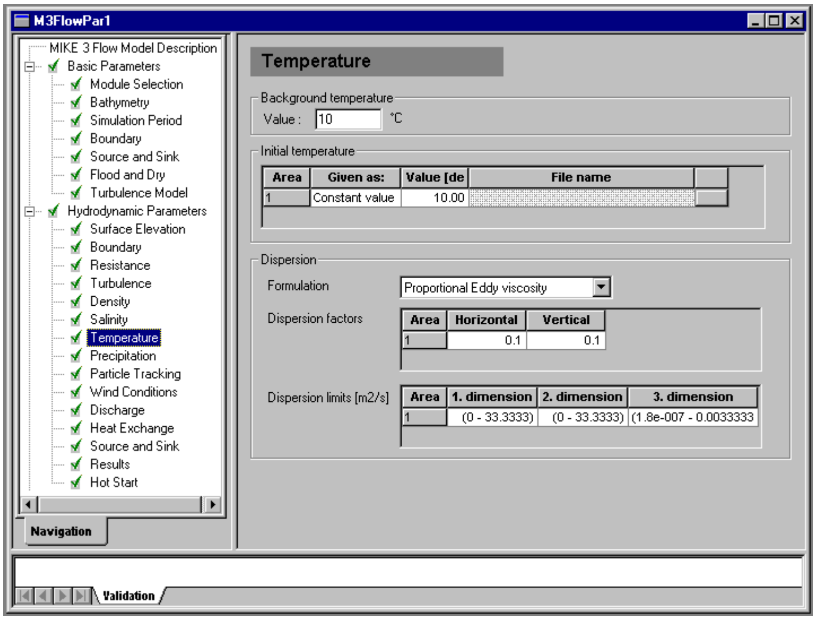


Figure 5.12 Temperature dialog

5.8 Precipitation

The precipitation dialog is shown in Figure 5.13.

The model allows for precipitation to be included in the calculations. The precipitation rate (in mm/day) can be specified in three ways

- a constant value
- a type 0 data file (time series)
- as a type 2 data file (matrix series)

A negative value of precipitation corresponds to evaporation.

For background details see Evaporation *(p. 86)* and Precipitation *(p. 108)*.

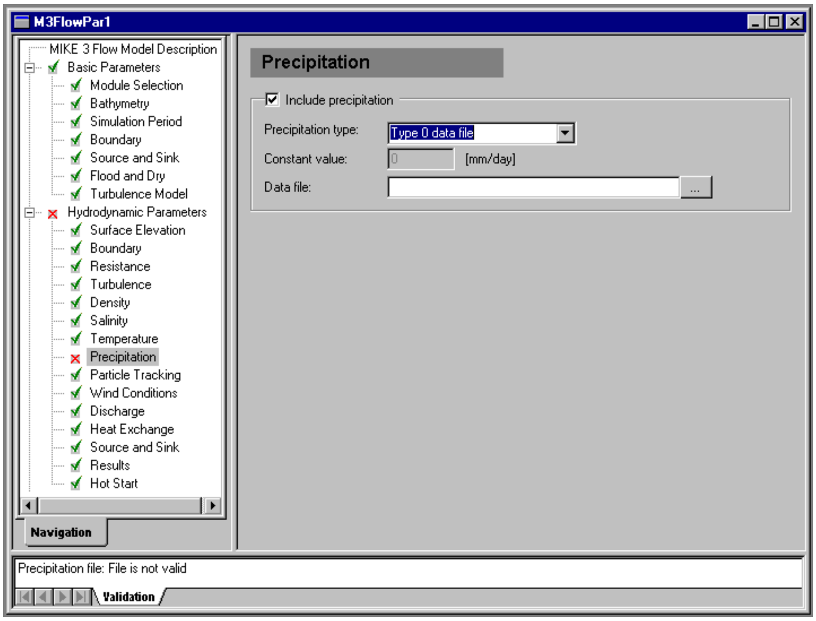


Figure 5.13 Precipitation dialog

If your simulation includes temperature variations, you need to specify how the temperature should change due to precipitation and evaporation. There are three different ways to do this:

- If the Heat Exchange module is selected, you can select the format "Heat exchange module". In this case, the effect of precipitation/evaporation on the water temperature is as obtained through the latent heat flux.
- You may choose the format "Ambient water temperature", in which case the temperature of the precipitated/evaporated water mass is set equal to the temperature of the ambient sea water.
- You may specify the value of the temperature in the precipitated/evaporated water mass through the formats "Constant value", "Type 0 data file" or "Type 2 data file".

See Precipitation and Evaporation Temperature *(p. 108)* for more detail.

5.9 Wind Conditions

The wind conditions dialog is shown in Figure 5.14.

The stress generated on the surface due to wind can be handled in one of four ways

1. Disregard any wind effect

2. As constant in space and time where you must specify constant values for the wind speed and direction

3. As constant in space but varying in time. The values are given through a type 0 data file

4. Varying both in time and space. The values are given through a type 2 data file

Please Note: Wind directions are given in degrees and measured clockwise from true North to where the wind is blowing from.

In case of time varying wind forcing, you may choose to include air pressure variations in your simulation in which case the air pressure must be one of the items in the wind data file. If you have included air pressure variations, you may also choose to include air pressure corrections of the levels at the open boundaries of your model. This can be useful if your boundary data does not take into account the air pressure variations.

The wind friction factor can be specified either as a constant or as linearly varying between two values based on the wind speed. In the latter case, if the wind speed is below the lower limit, the friction is given the value corresponding to that limit. (And correspondingly if the wind speed is above the maximum)

For background details see Wind Conditions *(p. 121)*.

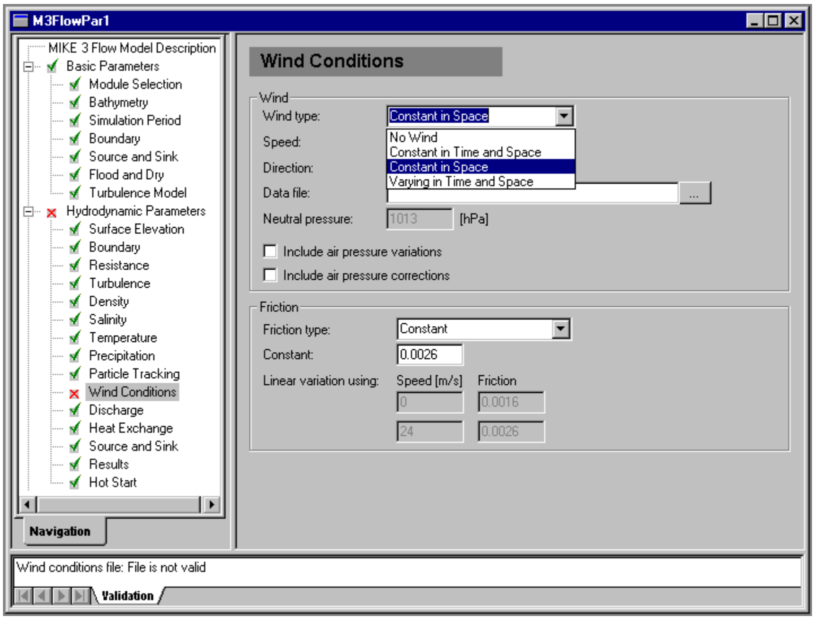


Figure 5.14 Wind conditions dialog

5.10 Discharge

The discharge dialog is shown in Figure 5.15.

You can include a number of discharge calculations in your simulation. A discharge calculation can include fluxes of volume, mass, salt and heat (temperature).

Having decided how many discharge calculations you wish to include, you need to specify details for each of these.

For each of your discharge calculations you need to specify the type(s) of calculation. You can select between fluxes of

- volume
- salt
- heat (temperature)
- mass

By definition a positive discharge is flow towards right when positioned at the first point and looking forward along the cross section line. You may select between

- net flow (positive + negative flow direction)
- positive flow direction only, or

- negative flow direction only

The discharge is always integrated over the entire water depth. You need to specify an output file name. The output file will be a time series (type 0 file) of the integrated quantities you have selected.

For each discharge line you must specify the associated area in which the discharge is to be calculated as well as the number of points by which the cross section is defined and the corresponding coordinates. A discharge line must lie within the same area.

For background details see Discharge Calculations *(p. 81)*.

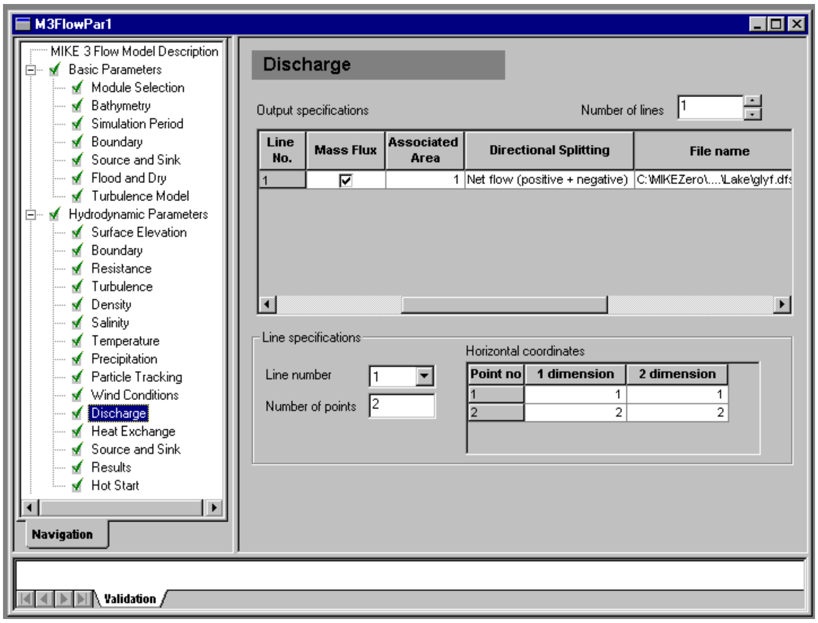


Figure 5.15 Discharge dialog

5.11 Heat Exchange

The heat exchange dialog is shown in Figure 5.16.

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. 87)* for further details.

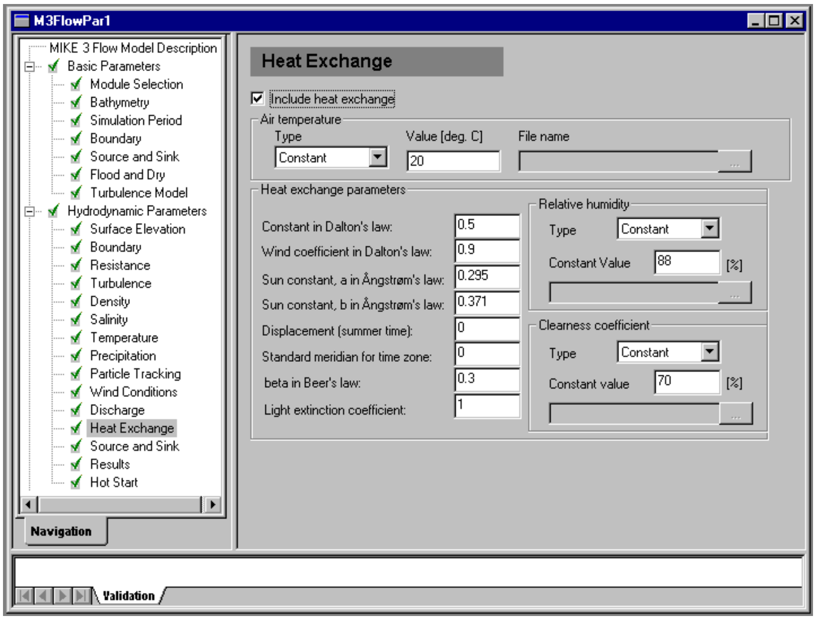


Figure 5.16 Heat exchange dialog

5.12 Source and Sink

The source and sink dialog is shown in Figure 5.17.

For each source/sink

- the discharge magnitude,
- the speed and
- the horizontal and vertical directions

by which the water is discharged into the ambient water must be specified. These values can either be given in a type 0 file (time series) or as constant values. An isolated sink is specified as a source with negative discharge.

The horizontal direction is relative to true North while the vertical direction is relative to the horizontal plane (positive upwards).

Please Note: With the hydrostatic engine, the vertical component of the source outlet is always zero, because there is no vertical momentum equation in the hydrostatic version. Possible source data files should not contain an item with the vertical velocity component if the hydrostatic engine is applied

If density varies due to salinity and/or temperature variations, then these must also be specified as either constant or as included in the type 0 data file.

For isolated sources the absolute salinity/temperature is specified, whereas for connected source-sinks the excess source salinity/temperature is specified. At sinks, the intake salinity/temperature equals that of the ambient water.

For background details see Source and Sink *(p. 112)*.

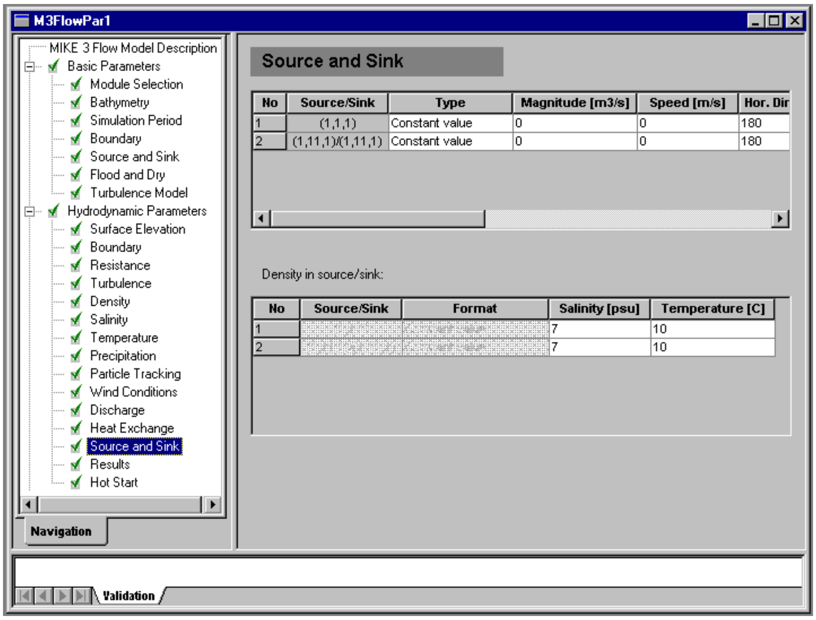


Figure 5.17 Source and sink dialog

5.13 Results

The HD output specification dialog is shown in Figure 5.18.

Standard data files with computed results from the simulation may be generated. 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. In practise, subareas and subsets must be selected. The actual bounds (the spatial range) determines the type of the output data files. These files may be of type 0 (time series), type 1 (line series), type 2 (matrix series) or type 3 (volume series).

The spatial ranges refer to coordinates in the associated area.

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.

Output data files can contain computed pressure (surface elevation), velocities, density, salinity, temperature, Eddy Viscosity, turbulent kinetic energy and dissipation of turbulent kinetic energy.

All output is optional, i.e. the user is free to select among the data that is being computed. This implies that for instance salinity can only be stored if varying salinity has been selected.

The pressure is stored as excess pressure relative to the hydrostatic pressure. Normally this quantity is not of interest. Therefore whenever the surface layer is stored, you may instead select the surface elevation as an output item.

Pressure cannot be an output item if the hydrostatic engine is chosen.

Please Note: Layer 0 in a volume series output file reflects the lowest bed level in the domain and will only contain delete values.

For background details see Output Area *(p. 104)*.

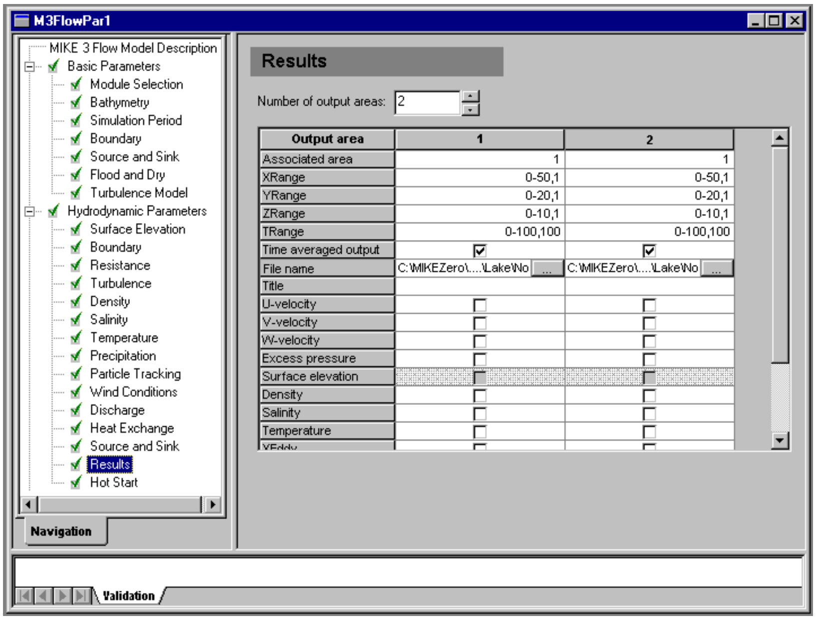


Figure 5.18 HD results dialog

5.14 Hot Start

The hot start dialog is shown in Figure 5.19.

To enable a later continuation of your simulation, you can save a hot start file for each area.

You must specify a name for each file. You can also specify a descriptive title for each hot file.

The hot files may be stored at user-specified intervals during the simulation.

For background details on hot start files see Hot Data *(p. 96)*.

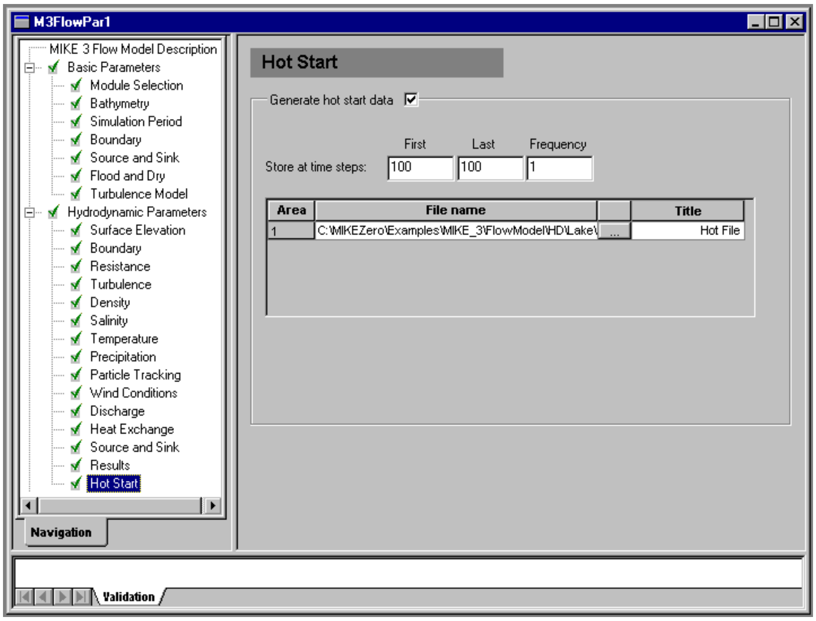


Figure 5.19 Hot Start dialog

5.15 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.

Please Note: It is not possible to specify any mass budget files before one or more polygons have been specified under the Basic Parameters Dialog.

For further information see Mass Budget in the Reference Manual.

6 Reference Manual

6.1 Bathymetry

6.1.1 General Description

Describing the water depths in your model area for the hydrodynamic model is without doubt the **most important task** in the modelling process. A few hours less spent in setting up the model bathymetry might later on mean extra days spent in the calibration process.

Giving exhaustive guidelines for how you should specify the bathymetry in order to avoid any problems later on is, however, nearly impossible. You can avoid many problems in the modelling process by adhering to the directions given below, but the experience you build up through practise is invaluable.

An example with a step-by-step description of how to use the Bathymetry Editor for creating a bathymetry data file is included with the installation. Please find this example in your installation folder under Examples\MIKEZero\BatEdit.

6.1.2 Selecting the Model Area

When deciding on which area to include in your model and thus where you should place your open boundaries you should take the following into consideration:

- MIKE 3 is a finite difference model with constant grid spacing in the x-, y- and z-directions, and therefore your model area has to be rectangular in the horizontal plane. It also means that the computational points will lie in a square or rectangular grid.
- Your area or point of interest should lie well inside the model area, say at least 10 grid points from the boundary but preferably more.
- You may have to include not only the area immediately surrounding the area or point of interest but a much larger one in order to have, for example, the wind surge computed properly.
- You should have your open boundaries in areas where the water flow is “well behaved” and the flow direction, if possible, perpendicular to the open boundary.
- A “well behaved” flow in this connection means that, since certain assumptions are made in the computations at the boundaries, the flow pattern should be smooth at the boundary and in the area inside the boundary (that is 5 to 10 grid points inside the boundary). In other words, the bathymetry should be smooth close to all open boundaries.

- You will not always be able to situate all open boundaries so that the flow runs perpendicular to the boundary line. However, try to have the flow as close to being perpendicular to the open boundaries as possible.
- As you must know either the water level (or pressure) variation or the flow at the open boundaries, you have to place the boundaries through points or between points, where such data are known. If, for example, you are going to do a tidal simulation, the open boundaries can be placed such that there is a tidal station at each end of the open boundary.
- Open boundaries can meet in corners. You must then ensure that the boundary conditions in the corner point are consistent. This requires that you have good level or velocity data at the boundary. If not, the corner should be placed on a not too small island (see Figure 6.1).

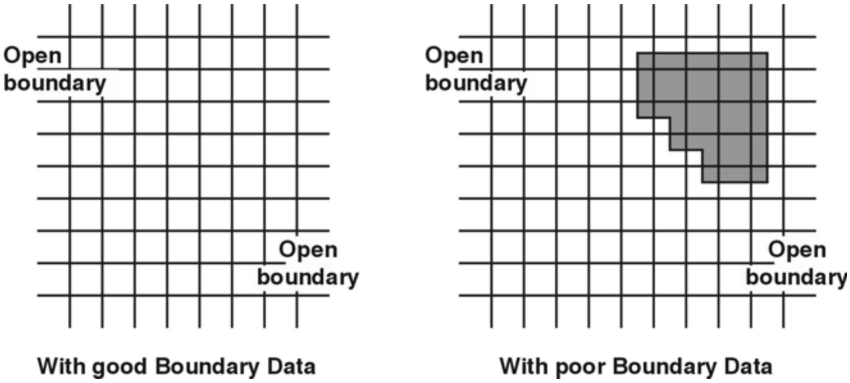


Figure 6.1 Two open boundaries in a corner

- Avoid sudden expansions or contractions of the flow close to an open boundary unless the current speeds are small (see Figure 6.2 and Figure 6.3). You should especially avoid a situation like the one in Figure 6.3.

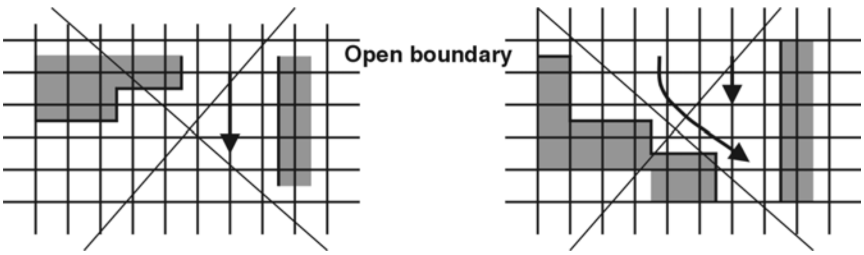


Figure 6.2 Sudden expansion and contraction of the flow close to an open boundary

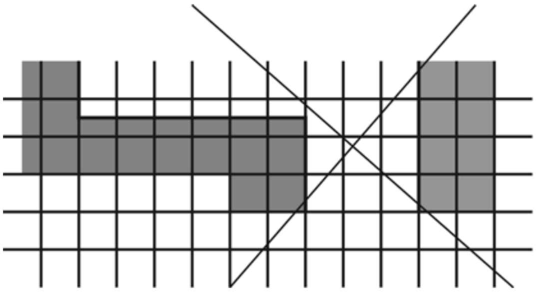


Figure 6.3 Special case of sudden contraction, permitted for level boundaries

- Although MIKE 3 can handle flooding and drying just inside a level boundary, you should normally not place the open boundaries too close to shallow areas which might dry out. Points at the open boundaries should never dry out.
- If possible rotate your model so that the main flow direction inside the model is more or less parallel to one of the coordinate axes.

6.1.3 Selecting the Grid Spacing

Although the selection of the grid spacing and of the model area are closely connected there are a number of special considerations (as listed below) which you have to make when selecting the grid spacing. Except for the first two and the last one they are all related to the Courant Number and thus the speed with which the information travels in the model. Please see Courant Number *(p. 78)* for a description of these terms.

- First of all your grid should resolve all the variations in the bathymetry which are important for the flow you wish to simulate, see e.g. Figure 6.4. A typical example is an estuary where the flow in the estuary often can be resolved on a horizontal grid coarser than the flow at the entrance of the estuary.

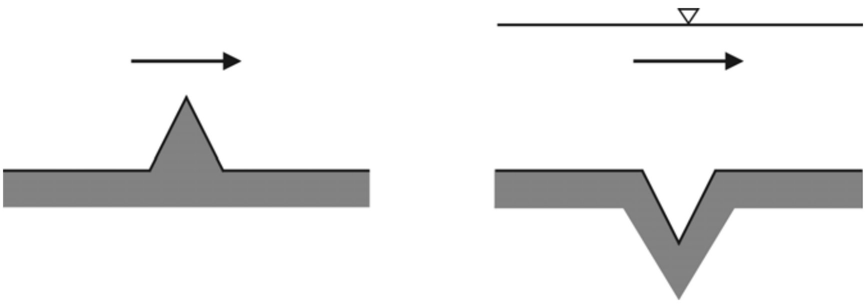


Figure 6.4 Unwise bathymetric resolutions

- Selection of the vertical resolution in MIKE 3 again very much depends on the flow you wish to simulate. The number of grid points in the vertical direction is denoted by l_{extr} and the vertical grid spacing by Δz. A sketch of a single water column as defined in MIKE 3 is shown in Figure 6.5:

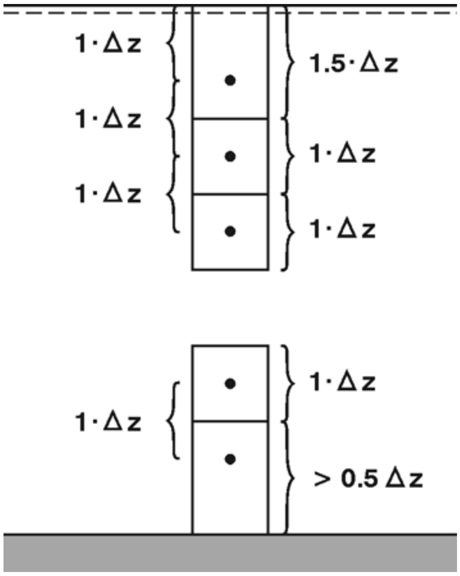


Figure 6.5 Sketch of the discrete water column in MIKE 3

In the vertical direction, the grid points are constantly spaced by Δz and the uppermost grid point is positioned Δz below the reference level. The uppermost grid cell is numbered by l_{extr}. The general formula for determining the vertical position, z_l, of grid point No. l is

$z_l = -(l_{extr}+1-l)\,\Delta z$, where $l = 1,2,\ldots,l_{extr}$

If there is more than one (wet, not seabed) grid point over the vertical at a given location, the lowermost computational grid cell has a height of more than ½ Δz. If, for instance, you have specified Δz = 5 m and l_{extr}=10 and your bathymetry yields the seabed at z = –100 m, the lowermost computational grid cell at this location will have a height of 52.5 m. If, for the same choices of Δz and l_{extr}, the seabed at another location is at z = –16.7 m there will here be three computational grid cells over the vertical, namely grid cells number 8, 9 and 10, and the lowermost grid cell has a height of 4.2 m. Shallow areas, where the water depth is less than Δz, are permitted. In fact, all cases where the seabed position, z, from the bathymetry file is greater than or equal to 2 Δz and less than *zland* will have only one computational grid cell in the vertical direction.

Although MIKE 3 applies a bottom fitted approach in which the correct depth is taken into account, the flow at the bottom still needs to pass obstacles either sidewards or upwards whenever encountered. Furthermore, MIKE 3 assumes a logarithmic velocity profile between the lowest computational node and the actual seabed, see Figure 6.6. Thus, a fine vertical resolution is always the best approach allowing MIKE 3 itself to determine the vertical velocity profile. On the other hand you should recall that flooding and drying (see Flood and Dry *(p. 86)*) is restricted to areas with only one vertical level. Your vertical resolution should be selected bearing the water level variations in mind. Hence, this constrain together with limitations in simulation time and computer memory often set an limit on the vertical grid spacing.

In MIKE 3, the user specifies a 'Minimum water depth', i.e. the depth below which computations are not allowed to continue. The 'Minimum water depth' equals the 'Drying depth' when flooding and drying has been enabled, see Flood and Dry *(p. 86)*.

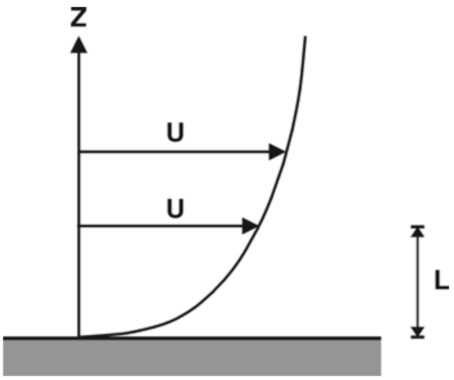


Figure 6.6 Logarithmic velocity profile used near the seabed

- If you cannot avoid channels, which run at an angle of 45 degrees to the grid, the grid spacing and the time step should be chosen accordingly small as shown in equation 6.1 and Figure 6.7,

$$\Delta t \leq \frac{a \Delta x}{\sqrt{gh}} \tag{6.1}$$

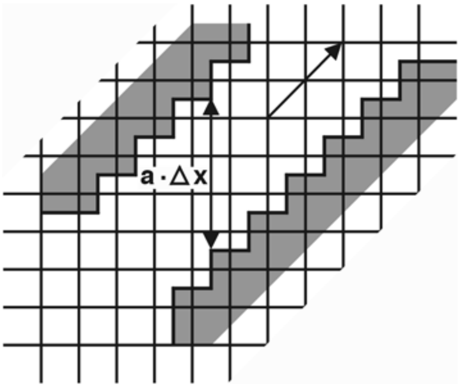


Figure 6.7 Flow in a channel at 45 degrees

- Closely related to the condition above is the treatment of deep channels in shallower areas. When you model an area with narrow and deep channels it is quite normal that the channels are only one or a few grid points wide. You should, however, take care when the narrow channels cross from one grid line to the next. If the flow in the channel is of importance the number of overlapping grid points should be greater than the local Courant Number. This will ensure that the flow information is transferred properly up the channel, see Figure 6.8.

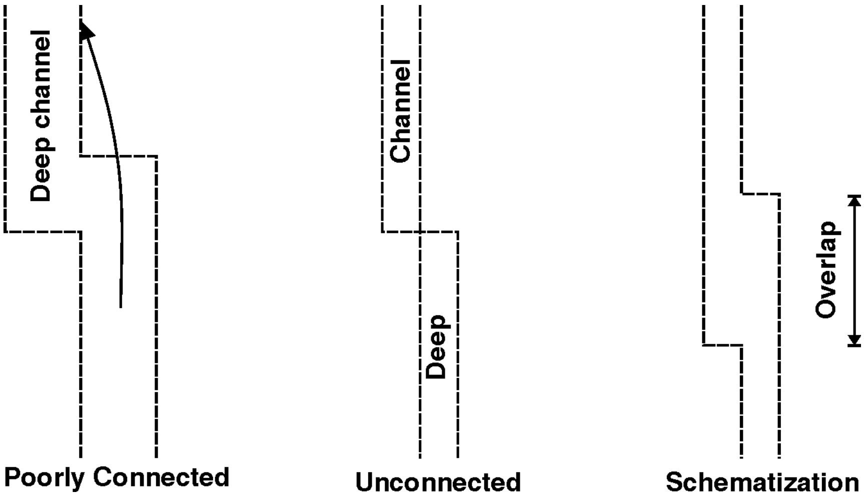


Figure 6.8 Schematization of a narrow channel

- You should avoid alternating land-water/land-water boundaries as shown in Figure 6.9 if the Courant Number is larger than 1.

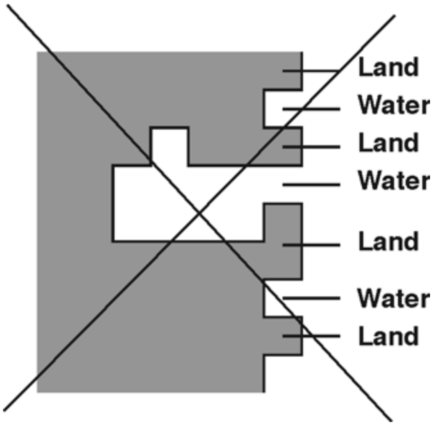


Figure 6.9 Schematisation of land, which should be avoided

- Areas which are subject to flooding and drying should not be made completely level but given a gentle slope towards the nearest area with deep water. This will ensure that a series of one-point ponds are not left in the otherwise dry areas when the water withdraws. The flooding and drying is, however, for numerical reasons restricted to the top layer. This should be taken into account when considering the vertical resolution.

6.1.4 Selecting the Reference Level

In principle you can use any reference level in the hydrodynamic computations (that is in the bathymetry), but it is recommended to use mean sea level (MSL).

The depths on sea charts are, however, normally given relative to the lowest astronomical tide (LAT) and the bathymetry will therefore normally be entered (digitized) relative to this datum. You can add the difference between MSL and LAT to all grid points in the bathymetry data file using the Grid Editor facility for editing.

You will have to make sure that all the sea charts you are using to prepare the bathymetry are to the same datum. If this is not the case, you must choose a common datum and then convert all depths to this datum. In the same way, all the water level recordings you will use must also be converted to the same datum. Note that the uppermost grid point is positioned Δz below the datum level, see Figure 6.5.

6.1.5 Specifying the Bathymetry

Before you start a simulation you have to prepare the bathymetry in a data file or, in other words, digitize your model area. There are several ways to do this:

- Draw the computational grid on a piece of transparent paper, put it on top of the sea chart and write the depth for each grid node on the paper. Then enter the depth values in a file using the data file editor for 2D matrices provided with the *MIKE 3/MIKE Zero*

- If you have a file with (x,y) coordinates and the corresponding depths you can grid this data using the Bathymetry Editor facility. An example with a step-by-step description of how to use the Bathymetry Editor for creating a bathymetry data file is included with the installation. Please find this example in your installation folder under Examples\MIKEZero\BatEdit.

- If you have your depth information in digital form you can create a program which writes the 2D bathymetry matrix to an ASCII file and enter this file into the standard data file format using the Grid Editor.

In all cases please note that the depth given to a grid point represents not only the depth right at that point, but the area surrounding the grid point, see Figure 6.10.

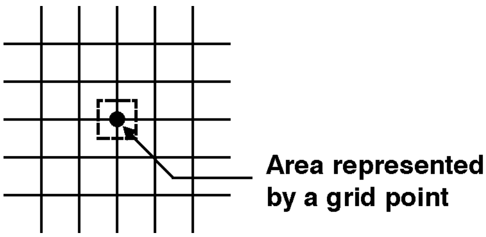


Figure 6.10 Depth representation in the grid

6.1.6 Sign Convention

Depth values for grid points below the chosen datum are negative in MIKE 3, see also Figure 6.5.

6.1.7 Additional Area Description

In addition to the bathymetry for your model area the following area information should also be specified:

- The initial surface elevation in the area. Because the model initialises the velocities to zero, you must specify an initial surface elevation that is in agreement with these conditions. In practice this means that you should specify a value that matches the boundary conditions at the first time step. If the model area is large and the surface levels at the open boundaries differ substantially, you should create a data file with an initial surface elevation at each grid point.

- Whether or not the possibility of flooding and drying of land areas should be included in your simulation. Please refer to Flood and Dry *(p. 86)* for a detailed description of this facility.
- The projection system (e.g. UTM-33) in your current MIKE 3 set-up.
- The latitude and longitude at the lower left corner of the model. This information is read directly from the bathymetry data file.
- The orientation of your model. This information is read directly from the bathymetry data file. The orientation is defined as the angle between true north and the y-axis of the model at the model origin of coordinates, measured positive clockwise. A mnemonic way of remembering this definition is by thinking of NYC, which normally means New York City, but which for our purpose means "from North to the Y-axis positive Clockwise", see Figure 6.11.

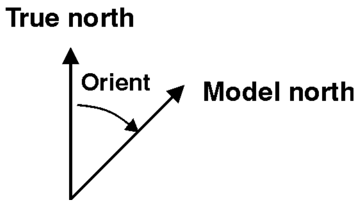


Figure 6.11 Definition of model orientation

- A value, *zland*, representing land. This means that all grid points with a depth value greater than or equal to the value of *zland* will always be considered to be land and will not be subject to possible flooding and drying. This information is read directly from the bathymetry data file.

If you have used the Bathymetry Editor to generate your bathymetry, the three last points of information are automatically included in your bathymetry data file. If you have created your bathymetry by other means, make sure that the data file contains the correct information on these points.

Please note: Bathymetry data files must contain a custom block called M21_Misc which consists of 7 elements of type float:

- The first element is the orientation.
- The third element is a flag and the value -900 indicates that the data file contains geographic information.
- The fourth element is the value representing land.
- The rest of the elements are not used in bathymetry data files.

6.2 Bed Resistance

6.2.1 General Description

The bed resistance (shear) is specified in terms of a drag coefficient formulation according to the relation,

$$\frac{\tau_{bottom}}{\rho} = C_D u^* |u^*| \tag{6.2}$$

in which τ_{bottom} is the bottom shear stress, ρ the density, C_D the drag coefficient and u^* the first computational speed encountered above the bottom. C_D is determined according to the selected turbulence closure model, giving rise to the following three different formulations:

- In applications with a constant eddy formulation, the k model and the standard 3D k-ε model, C_D is determined assuming a logarithmic profile between the seabed and the first computational node above. Thus C_D is given by the relation,

$$C_D = \left[\frac{1}{\kappa}\log\left(\frac{z}{k_s/30}\right)\right]^{-2} \tag{6.3}$$

  where κ is von Kàrmàn's constant, z the distance between the seabed and the first computational node and k_s the bed roughness length scale. The bed resistance is given in terms of k_s (in m).

- If the Smagorinsky eddy formulation is selected then C_D is determined such that it is consistent with the associated velocity profile. However, this velocity profile will be a function of a “slip” velocity, U_0 at the seabed. U_0 is determined such that the velocity profile based on the Smagorinsky eddy formulation is identical to that of the logarithmic profile at a certain level. This level is by default the drying depth (see Flood and Dry *(p. 86)*) but may in principle take any value. Thus using the Smagorinsky eddy formulation C_D is given by the relation,

$$C_D = \left[\frac{2\sqrt{2}}{3}\frac{D}{l}\left\{\left(1-\frac{z_m}{D}\right)^{\frac{3}{2}} - \left(1-\frac{z}{D}\right)^{\frac{3}{2}}\right\} + \frac{1}{\kappa}\log\left(\frac{z_m}{k_s/30}\right)\right]^{-2} \tag{6.4}$$

  where z_m is the distance above the seabed where the Smagorinsky profile matches the logarithmic velocity profile. *l* is the length scale given by the Smagorinsky formulation (see Smagorinsky Formulation *(p. 111)*) and D is the actual water depth.

- When using the mixed 1D (vertical) k-ε, 2D (horizontal) Smagorinsky closure model, the bed drag coefficient reads

$$C_D = \left[\frac{k\left(1 - \frac{k_s}{30 z_b}\right)}{\log\left(\frac{30 z_b}{k_s}\right) - \left(1 - \frac{k_s}{30 z_b}\right)} \right]^2 \tag{6.5}$$

where z_b is the vertical extent of the bottom grid cell. This expression has been obtained by assuming a logarithmic velocity profile in the bottom grid cell and depth-averaging over the extent of the bottom grid cell.

6.2.2 Specifying the Bed Resistance

A bed roughness length scale is assigned to each grid point in the model area. You can specify this in MIKE 3 in two ways:

- as one constant value which is given to all grid points
- as a map similar to that for bathymetry with a roughness length scale value for each grid point

6.2.3 Recommended Values

The default value is 0.05 metre referring to a relatively smooth seabed. Normally the bed roughness lies in the range between 0.01 - 0.30 metre. By definition a small k_s corresponds to a low friction and vice versa.

6.2.4 Remarks and Hints

The bed resistance formulations are to some extent dependent on the vertical grid spacing due to the logarithmic velocity profile assumption or the level at which this profile matches that based on the Smagorinsky formulation. Thus the selection of k_s is not entirely independent of the adopted vertical grid spacing. Consequently, in some simulations k_s may take values which are considered as beyond the normal range of k_s values, and apparently there might be no direct relationship between the value of k_s and a physical quantity such as e.g. grain size. The *model* bed roughness k_s is a calibration parameter which is a result of both the *physical* resistance and the *numerical* implementation and resistance.

Because MIKE 3 uses a bottom fitted approach a constant bed roughness does not necessarily yield the same shear due to varying z.

The Eddy Viscosity at the first computational grid point above the seabed is determined such that it is consistent with the adopted drag coefficient formulation.

6.3 Blow-up

Please inspect carefully the **log** file.

6.4 Boundary Conditions

6.4.1 General Description

If the description of the bathymetry is the most important task in the modelling process then the description of the water levels (pressure) and flow at the open boundaries (in short called the “boundary conditions”) is the **second most important task**. The better the boundary conditions the better the results and the fewer the instability problems.

The hydrodynamic module of MIKE 3 solves the partial differential equations that govern three-dimensional flow and, like all other differential equations, these need boundary conditions. As the unknown (prognostic) variables are pressure and velocities in the x-direction, y-direction and z-direction you must, in principle, specify three of these four variables in all grid points along the open boundary at each time step. However, in most applications you only know (say) the surface elevation or you know the approximate velocities across your boundary. The input to the hydrodynamic module of the MIKE 3 Flow Model has therefore been structured accordingly.

Please Note: *If you have chosen to use the hydrostatic version of the MIKE 3 hydrodynamic engine, pressure boundaries and data transfer boundaries cannot be applied.*

You can choose between the following five types of boundary input:

- Excess pressure and the direction of the flow (not with the hydrostatic engine)

  To avoid problems with the numerical representation of the pressure, which due to the hydrostatic pressure can become very large at large depths MIKE 3 is defined in terms of an excess pressure. For homogeneous fluids this simply is defined as the total pressure subtracted the hydrostatic pressure. The definition is somewhat more complicated for stratified flows. Therefore it is not recommended to use this boundary type for stratified flow simulations unless you are an experience user with a detailed knowledge of the numerical scheme. The excess pressure is given in Pa (N/m^2) and can be constant or varying along the boundary. The variation in time can be either constant, sinusoidal or vary as specified in a type 0, type 1 or a type 2 data file. The directions can be specified in four different ways (see Specifying the Boundary Conditions *(p. 71)*).

- Total Pressure and the direction of the flow (not with the hydrostatic engine)

  The pressure can also be specified in terms of the total pressure (given in Pa). The total pressure can be constant in time, have a sinusoidal variation or vary as specified in a type 0, type 1 or type 2 data file. The directions can be specified in four different ways (see Specifying the Boundary Conditions *(p. 71)*).

- Velocity
  The velocity (given in m/s) can be constant in time, have a sinusoidal variation or vary as specified in a type 0, type 1 or type 2 data file. The velocity is the speed of the flow with the directions specified as for pressure/water level boundaries (see Specifying the Boundary Conditions *(p. 71)*).

- Surface Elevation and the direction of the flow

  The pressure can also be specified in terms of a surface elevation (given in m) relative to the model datum (normally mean sea level) assuming hydrostatic pressure at the boundary. The surface elevation is the boundary type, which is used in far the most applications. The surface elevation can be constant in time, have a sinusoidal variation or vary as specified in a type 0 or type 1 data file. The directions can be specified in four different ways (see Specifying the Boundary Conditions *(p. 71)*).

- Data Transfer (not with the hydrostatic engine)

  Data transfer means that results from an earlier, encompassing model simulation are used to describe the variation at the open boundary. Thus the data transfer can only be type 1 data files (depth integrated MIKE 21 results) or type 2 data files (MIKE 3 results). The data must be in a special format according to some predefined standards. The primary boundary condition for data transfer type can be selected as pressure (surface elevation) or velocity (flux).

6.4.2 Specifying the Boundary Conditions

In the dialogues you are presented with a list of the open boundaries the model has been able to detect.

The model detects the open boundaries by searching for lines of adjacent water points placed along the four sides of the bathymetry. Note, the actual locations of the open boundaries were defined when you digitised the bathymetry.

In many cases the “program detected” boundaries will correspond to those you have planned. If this is not so, you should select “User specified” boundaries and then further specify the number of open boundaries and their position in the model, see the section User Specified Boundaries *(p. 74)*. Those cases where you need to specify the number and location(s) of boundaries cover the so-called “internal” boundaries, which are not placed along any of the four sides of the bathymetry.

Having specified the actual location (in grid points) of all boundaries, you specify the type of boundary data. Also, you should specify the associated constant values or data files for turbulent kinetic energy (TKE), dissipation of turbulent kinetic energy (TKD), salinity and temperature, see *k Turbulence Model (p. 99), k-ε Turbulence Model (p. 100), Mixed 1D k-ε, 2D Smagorinsky Turbulence Model (p. 102)* and Density Variation *(p. 80)*.

First, you must specify the boundary type (a “pressure”, “velocity”, “surface elevation” or “data transfer” boundary). You will then need to specify the boundary variation of the selected boundary type in time and space.

They can be given in one of the following ways:

- A constant value used at all grid points along the boundary and throughout the whole of the simulation. This is most useful if a stationary flow field is required.
- A sinusoidal variation during the simulation period, for example, a tidal variation can be specified. The variation is calculated as follows:

$$\begin{aligned} Value &= Referencelevel \\ &+ \tfrac{1}{2} Range \cdot \mathrm{SIN}\left(2\pi \cdot \frac{N \cdot \Delta t - Phase}{Period}\right) \end{aligned} \tag{6.6}$$

  where N is the time step number and Δt the time step. The same value is used at all grid points along the boundary.

- A variation as given in a type 0 data file. The data file gives the same value to all grid points along the boundary. If the time step in the data file differs from the time step in the model simulation then a linear interpolation is used.
- A variation as given in a type 1 data file. The data file defines the variation in time for each grid point along the open boundary. The data file must have exactly as many points as there are grid points along the horizontal axis. If the time step differs from the time step in the model simulation then a linear interpolation is used.
  This possibility allows you to introduce variations in the boundary conditions along the open boundary but constant over the depth.

- A variation as given in a type 2 data file. The data file defines the variation in time for each grid point along the open boundary. The data file must have exactly as many points as there are grid points along the horizontal axis as well as the vertical axis. If the time step differs from the time step in the model simulation then a linear interpolation is used. This possibility allows you to introduce an individual variation in the boundary conditions both along the open boundary as well as over the depth.

In all cases the reference level of the boundary data must equal the reference level of the bathymetry data. The “constant” boundary variation is affected by the “Warm-up period” option, see Simulation Period *(p. 29)*.

Then you specify a horizontal and vertical flow direction. The directions are kept constant throughout the simulation. For the horizontal direction please note that it is relative to true North. By default the directions are set such that the flow is perpendicular to the open boundaries. For velocity boundaries the speed is specified. A negative speed value means flow opposite to the defined direction. The vertical direction is relative to the horizontal direction and positive upwards, i.e. a vertical direction of -45 degrees refers to flow towards the seabed. By default the vertical direction is set to zero.

Please Note: *With the hydrostatic version of the MIKE 3 hydrodynamic engine, the vertical direction of the velocity at the boundaries is always set equal to zero.*

For pressure/surface elevation boundaries a “Velocity Along Boundary (VAB)” type should also be specified. There are the following possibilities:

0:The flow is assumed perpendicular to the open boundary, i.e. the VAB is zero.

1:The VAB is obtained by extrapolation from the flow inside the boundary.

2:The directions of the flow at the boundary are explicitly given and the VAB is calculated accordingly.

12:This is a combination of VAB type 1 and 2. When the flow direction is out of the boundary then type 1 is chosen otherwise type 2.

The default VAB type is 12, i.e. extrapolation under outflow conditions and using the prescribed directions for inflow situations.

Finally, you can control if your boundary data should be tilted to accommodate for a possible Coriolis' set-up along the boundary. A description of this facility is given below in the section Recommended Selections and Values *(p. 76)*. This facility is only enabled when the boundary is a “surface elevation” boundary.

6.4.3 User Specified Boundaries

In case the open boundaries, presented to you do not correspond to those you had planned, e.g. if you have “internal” boundaries, you must select “User specified” boundaries and then give the number of open boundaries in the model. Further, you will have to specify the location of the open boundary, which means that it is the coordinates of the first and the last grid points defining the boundary. By default the vertical extent is always equal to the number of vertical levels.

It will be very unusual that you yourself have to specify the locations of the open boundaries. It is only relevant in the following three situations:

- You have a long open boundary broken by, say, two small islands. The menu will show you this boundary as three smaller boundaries. If the boundary conditions are either the same for all three boundaries or it is most conveniently to keep the boundary data for the whole boundary in the same type 1 data file, or transfer file, you can define the three boundaries as one boundary.

  You will then have to specify the start point as the first water point on the first line and the last point as the last water point on the third line. The boundary line will then contain a few land points; but this is not an error.

- You want to have an internal open boundary in your model. An internal boundary is a boundary that is not located on one of the model sides. You can use an internal boundary to keep your model size small, see Figure 6.12.

  In the example in Figure 6.12, the far left open boundary can with advantage be placed closer to the area of interest.

Please note: If you want to include an internal boundary in your model you must fill the area behind the boundary with land points.

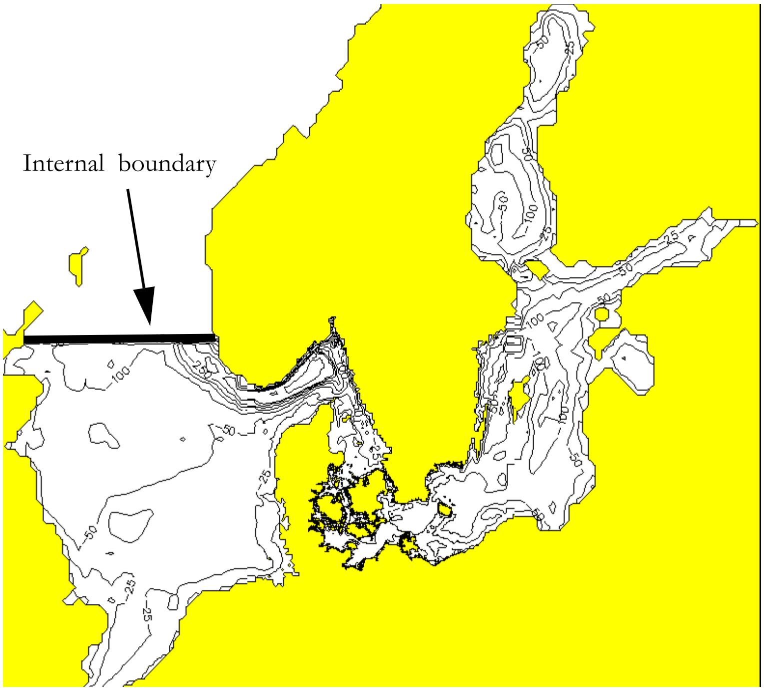


Figure 6.12 Example of internal boundary

- You want (say) two different boundary types (either horizontally or vertically) at one "well-defined" open boundary. MIKE 3 will accept (almost) any combination of boundary types like a surface elevation boundary on top of a velocity boundary or vice versa. However, great care should be taken whenever such boundaries are mixed.

  Due to the computational structure of MIKE 3 there are a few restrictions to where a boundary can be located, see Figure 6.13.

Please note: All water points on the model sides must be included in an open boundary definition and overlapping boundaries are not permitted.

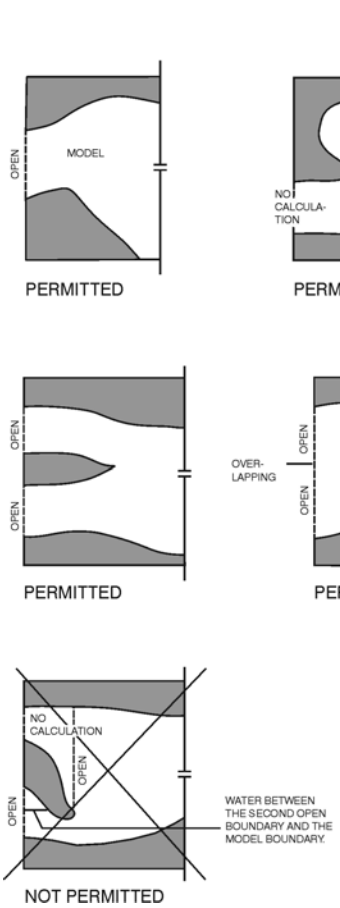


Figure 6.13 Restrictions on the location of an open boundary

6.4.4 Recommended Selections and Values

If you specify surface elevations for a boundary you can normally assume that there is little spatial variation along the boundary. However, in certain situations you may wish to specify a non-horizontal surface elevation since a horizontal water level may give unrealistic results. These situations are treated in the following way:

- If, for example, you are carrying out a study in which you have a water level station at each end of the open boundary, a linear variation along the boundary (or parts of it) should be specified. You do that by having a type 1 data file with the boundary conditions. The data file can easily be created with the Profile Editor tool.

- Another situation where a horizontal surface elevation at the boundary is unrealistic is in situations where the Coriolis force is of significance. If you keep the surface elevation horizontal you will most likely get a large inflow together with a large outflow at the same boundary, especially in a steady state situation, as the water level should actually be tilted. The effect is illustrated in Figure 6.14. You should therefore apply the tilting facility so that MIKE 3 will tilt the boundary to avoid this unrealistic flow pattern.

When MIKE 3 tilts the surface elevation, a locally quasi-geostrophic balance is assumed over the water column along each horizontal boundary point. MIKE 3 uses the mid-point along the boundary as tilt point whenever possible. If an open boundary is adjacent to another open boundary then their “common” point is used as tilt point. This implies that an open boundary adjacent to two open boundaries cannot be tilted.

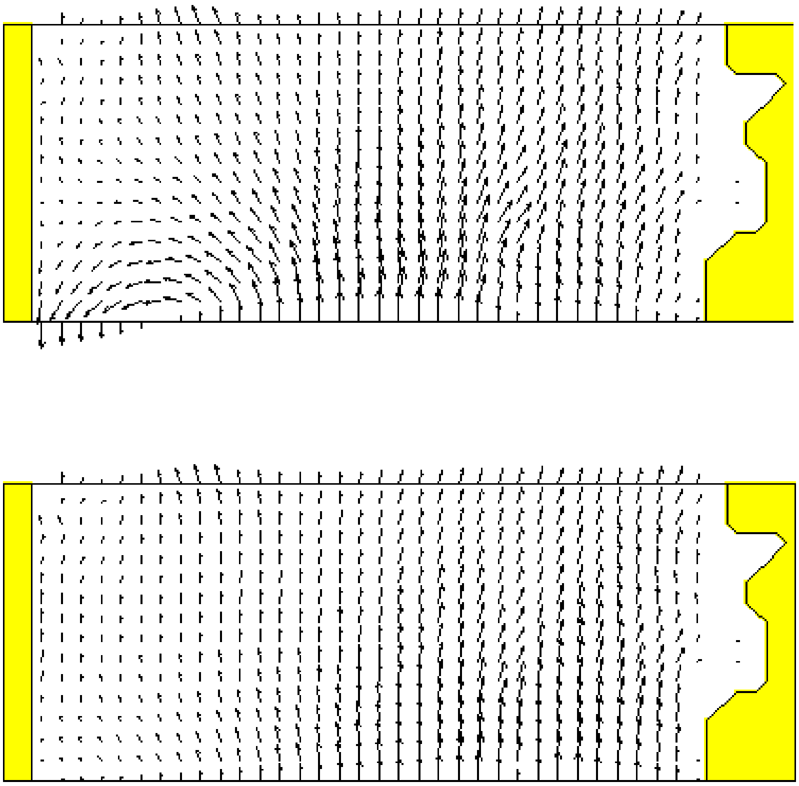


Figure 6.14 Before (upper) and after (lower) tilting

6.4.5 Recommended k-ε Boundary Values

It is hard to estimate realistic boundary variations for turbulent kinetic energy and dissipation of turbulent kinetic energy. Therefore, it is in many cases useful to apply the minimum values $k_{min}=10^{-7}$ m^2/s^2 and $\varepsilon_{min}=5\cdot10^{-10}$ m^2/s^3, yielding $\nu_{T,min}=1.8\cdot10^{-6}$ m^2/s, see Eddy Viscosity *(p. 84)*.

6.4.6 Remarks and Hints

Introducing open boundary conditions into a finite difference model is a very complex task as a number of different implementation solutions can be used. The input description as given above focuses on the description as seen from the user's practical point of view (“these data are available; how do I provide the model with that data?”), while the description as seen from the numerical point of view is given in the Scientific Documentation.

6.5 Courant Number

6.5.1 General Description

The Courant number, C_R is defined as follows:

$$C_R = c\frac{\Delta t}{\Delta s} \tag{6.7}$$

where Δt is the time step, Δs is the grid spacing in one of the horizontal directions and c is the celerity of the barotropic wave given by

$$c = \sqrt{gh} \tag{6.8}$$

As the barotropic information (about surface elevations and velocities) in the computational grid travels at a speed corresponding to the celerity, the Courant number actually expresses how many grid points the information moves in one time step.

6.5.2 Recommended Value

Normally you can have a maximum Courant number in your model of up to 5. The maximum value, which you can use without having stability problems, does, however, depend on your bathymetry:

- If you are modelling an estuary with tidal channels you should adhere to the rules given under Bathymetry *(p. 59)*. Alternatively you can use a maximum Courant number of 1, in which case you should have no problems in resolving the flow in the channels. The CPU Time *(p. 79)* requirements might, however, become very high.

- The hydrodynamic module of the MIKE 3 Flow Model is designed for Courant numbers up to about 10. You should, however, only allow these very high numbers in areas where the bathymetry is very smooth.

6.6 CPU Time

6.6.1 Factors Influencing the CPU Time

The CPU time required by a hydrodynamic simulation depends on the size of your model, on the number of time steps in your simulation, on which features you have specified for the simulation and on the general computational speed of your computer. Information on all these factors can be used to estimate CPU time required for your simulation.

If you wish to estimate how a change in your specifications for a hydrodynamic simulation changes the CPU time required without specifying the model set-up and doing a verification, 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 the flooding and drying 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. As minimum a mass equation and three momentum equations are solved. In stratified areas one or two additional equations are solved. Similarly the *k*- and standard *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 / BCS

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

Note: A MIKE 3 model is run using only one thread.

6.7 Density Variation

6.7.1 General Description

Often estuarine and coastal hydraulics and oceanography imply variations in the density due to varying salinities and temperatures. As even small differences in density may have a decisive influence on the flow properties this is an important part of MIKE 3.

In MIKE 3 the density is a function of the local salinity and temperature according to the relations outlined by UNESCO. These relations are applicable for salinities in the range from 0 to 45 PSU (**P**ractical **S**alinity **U**nit) and temperatures ranging from -2.1° C to 40.0° C.

MIKE 3 includes four advection-dispersion schemes. The first scheme, QUICKEST/SHARP, is especially suitable for simulations with steep density gradients. It is a fully three-dimensional scheme, see MIKE 3 HD Scientific Documentation. Alternatively, the QUICKEST/ULTIMATE scheme with directional splitting may be selected. Finally, you may choose the simple UPWIND scheme using directional splitting or the 3D UPWIND scheme.

The schemes applying directional splitting, i.e. QUICKEST/ULTIMATE and simple UPWIND, have a build-in internal loop over components which increases the computational speed when both salinity and temperature variations have been selected. For the fully 3D schemes, i.e. QUICKEST/SHARP and 3D UPWIND, the internal loop over components may optionally be selected to increase computational speed at the expence of requirering more memory.

Prior to the simulation start, MIKE 3 requires knowledge of the initial salinity/temperature distribution. For the baroclinic forcing of MIKE 3 you must provide the variation of salinity/temperature at the open model boundaries. Furthermore, dispersion factors must be chosen, see Dispersion Coefficients *(p. 83)*. In many cases the description of initial and open boundary conditions as well as proper choices of dispersion factors are very important tasks during the model calibration process.

6.7.2 Specifying the Salinity and Temperature Variations

When you decide to include variations in salinity and/or temperature an initial value must be assigned to each grid point. You also need to specify the variation at each open boundary:

- The initial distribution can either be given as a type 3 data file or as a constant value being assigned to all grid points. A type 3 data file covering the model area as defined through the bathymetry may be created from an *xyzv*-file (ASCII file containing three-dimensional position coordinates and values) with the **Digitizing** tool in the MIKE 3 Tool Box.

- The boundary variation for salinity/temperature can be given as a constant value, a type 0 data file, a type 1 data file or as a type 2 data file. Make sure that the initial distributions and the specified boundary variations are consistent.

6.7.3 Remarks and Hints

The background values are used to minimise numerical inaccuracies in the advection-dispersion calculations. Although much effort has been put into the advection-dispersion calculations using a scheme with a third-order accuracy, the calculations can only be performed with a limited accuracy. To minimise any inaccuracy the background values are subtracted from the salinity and temperature fields prior to the advection-dispersion calculations and added again afterwards.

6.8 Discharge Calculations

6.8.1 General Description

You may choose to include discharge calculations into your simulation. If you select discharge calculations you will be able to obtain time series of cross-sectionally integrated transports which are calculated directly from the prognostic variables during model simulation. The advantage of this approach compared to applying a post-processing tool for discharge calulations is that you can get fine-resolution discharge time series (type 0 data files) at virtually no extra costs (post-processing **often** requires processing of huge type 3 data files).

6.8.2 Specifying Discharge Calculations

- First you select the number of different discharge lines (or cross-sections) you want to include. A maximum of 50 lines is permitted with maximum 15 cross-section points on each line.
- For each of your lines you must specify the type(s) of discharge calculation: You can select between fluxes of volume, salt, heat (temperature) and mass.
- You may select between “net flow”, “positive flow direction only” and “negative flow direction only”. By definition, discharge is positive for flow towards right when positioned at the first point and looking forward along the cross-section line. The transports are always integrated over the entire water depth.
- Finally, you must enter the output specifications, i.e. the name of the type 0 data file where the discharge for this line and at what time steps the calculations should be performed and stored.

- You also need to specify the number of points by which the cross-section is defined as well as the corresponding horizontal grid coordinates.

6.9 Disk Space

6.9.1 General Description

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

6.9.2 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 only take up to 200 Kbytes on the disk. It is recommended always to carefully inspect the **log** file after termination, abnormal or successful, of a simulation.

6.9.3 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:

$$4NCPTS^{P} \cdot \left(NVAR \cdot \left(\frac{N_{last} - N_{first}}{N_{frequency}} + 1\right) + \mathrm{NSKP}\right) \cdot \left(\frac{J_{last} - J_{first} + 1}{J_{frequency}} \cdot \frac{K_{last} - N_{first} + 1}{K_{frequency}} \cdot \frac{L_{last} - L_{first} + 1}{L_{frequency}}\right)^{1-p} + NHEADER \tag{6.9}$$

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 on the **Bathymetry** *(p. 27)* **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.

6.9.4 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 *(p. 96)* and Simulation Type *(p. 110)*.

6.10 Dispersion Coefficients

6.10.1 General Description

The transport equation for salt is formulated as:

$$\frac{1}{\rho}\frac{D(\rho S)}{Dt} = \frac{\partial}{\partial x_j}\left(\frac{v_t}{\sigma_T}\frac{\partial S}{\partial x_j}\right) \tag{6.10}$$

and similarly, the transport equation for temperature is

$$\frac{1}{\rho}\frac{D(\rho T)}{Dt} = \frac{\partial}{\partial x_j}\left(\frac{v_T}{\sigma_T}\frac{\partial T}{\partial x_j}\right) + \frac{1}{\rho}Q_H \tag{6.11}$$

where S is the salinity and T is the temperature, and Q_H is the Heat Exchange (for simplicity SS, the source/sink term for the respective equaiton, is not displayed in the above equations). The dispersion of salinity and temperature is assumed to be proportional to the effective Eddy Viscosity with the factor of proportionality being $1/\sigma_T$, the dispersion factor. σ_T is the Prandtl/Schmidt number. Values of σ_T greater than one imply that diffusive transport is weaker for salt/temperature than for momentum.

6.10.2 Specifying Dispersion Factors

The user specifies the dispersion factor, i.e. the value of $1/\sigma_T$, for salt and temperature. Separate values must be entered for salinity and temperature (if selected, see Density Variation *(p. 80)*).

You must also specify the dispersion limits for each grid direction: Defining the dimensionless dispersion coefficient for direction j as

$$D_j = \frac{V_{Tj}\Delta t}{\sigma_T(\Delta s_j)^2} \tag{6.12}$$

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

6.10.3 Recommended Values

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

If you apply the mixed Smagorinsky/k-ε turbulence model then σ_T is calculated as an integrated part of the turbulence model and it is recommmended that you use a value of 1 in most cases.

6.10.4 Remarks and Hints

The above mentioned dispersion stability criterion often leads to a restriction, which yields a very small time step due to a very fine vertical resolution, see also Eddy Viscosity *(p. 84)* and Time Step *(p. 120)*. 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

$D_x + D_y < 0.5$

$D_z < 10$

6.11 Eddy Viscosity

6.11.1 General Description

The decomposition of the prognostic variables into a mean quantity and a turbulent fluctuation leads to additional stress terms in the governing equations to account for the non-resolved processes both in time and space. By the adoption of the eddy viscosity concept these effects are expressed through the eddy viscosity and the gradient of the mean quantity. Thus the effective shear stresses in the momentum equations contain the kinematic viscosity, the Reynold stresses (turbulence) and subgrid scale fluctuations.

6.11.2 Specifying the Eddy Viscosity

The eddy viscosity coefficient ν_T can be specified in five different ways:

- As a constant value (equivalent to a laminar description) for the entire computational domain.
- A time-varying function of the local gradients in the velocity field. This formulation is based on the so-called Smagorinsky concept. See Smagorinsky Formulation *(p. 111)*.
- As determined from the equation for turbulent kinetic energy. This formulation is a standard *k*-model and requires the solution of one additional transport equation. See *k Turbulence Model (p. 99)*.
- As determined from the equation for turbulent kinetic energy and the equation for dissipation of turbulent kinetic energy. This formulation is a standard three-dimensional *k*-ε model which requires two additional transport equations to be solved each time step. See *k-ε Turbulence Model (p. 100)*.
- As determined from the equation for turbulent kinetic energy and the equation for dissipation of turbulent kinetic energy in the vertical direction in combination with the local horizontal velocity gradients. In this formulation, the horizontal eddy viscosity is determined through the Smagorinsky concept, while for the vertical direction the eddy viscosity is determined from a one-dimensional *k*-ε model. See Mixed 1D k-ε, 2D Smagorinsky Turbulence Model *(p. 102)*.

6.11.3 Specifying Limits for the Eddy Viscosity

You should also specify the limits within which the eddy viscosity is allowed to vary in each grid direction. It is recommended that the value of the dimensionless eddy viscosity

$$E = \frac{v_T \Delta t}{(\Delta s)^2} \tag{6.13}$$

does not exceed 10 in any of the three directions. For applications using the mixed 1D k-ε, 2D Smagorinsky turbulence model, the eddy viscosity in the vertical direction is, however, allowed values up to approximately $90\Delta z^2/\Delta t$. See also Dispersion Coefficients *(p. 83)* and Time Step *(p. 120)*.

A minimum value for the eddy viscosity can be chosen to zero, but more useful is a value in the order of the molecular viscosity $1.6 \cdot 10^{-6}$ m²/s. When applying k-ε models, it is recommended you use minimum values which fulfil the Kolmogorov/Prandtl relation, i.e. $k_{min}=10^{-7}$ m^2/s^2 and $\varepsilon_{min}=5 \cdot 10^{-10}$ m^2/s^3, yielding $\nu_{T,min}$=1.8·10^{-6} m^2/s. See also Recommended k-ε Boundary Values *(p. 78)* in the Boundary Conditions *(p. 70)* section.

6.11.4 Remarks and Hints

In the same way as for the bed resistance you can use the eddy coefficients to damp out numerical instability (see Bed Resistance *(p. 68)*). You should only use this as a last resort to your stability problem: the schematization of the bathymetry and the boundary conditions are the primary causes for a blow-up.

When you use the *k*-model formulation or the standard *k*-ε model formulation of the turbulence the CPU Time for a simulation is increased significantly as one or two additional equations are solved.

6.12 Evaporation

You can include evaporation in your simulation in two different ways. If your simulation does not include any density variations you can include evaporation as a negative precipitation (see Precipitation *(p. 108)*). If your simulation on the other hand include density variations (or more precisely, temperature variations) you should select the heat exchange option (see Heat Exchange *(p. 87)*). This implies that the evaporation rate is calculated as a function of the latent heat flux.

You may specify the evaporation rate (as a negative precipitation in mm/day) in one of the following three ways: a constant, from a type 0 data file, or from a type 2 data file.

See the Precipitation *(p. 108)* description for more details.

6.13 Flood and Dry

6.13.1 General Description

A very valuable facility in MIKE 3 is its capability to include and exclude computational areas dynamically during the simulation or, in other words, compute the flow in an area which sometimes dries out and sometimes is flooded (e.g. tidal flats). Continuity is fully preserved during the flooding and drying process as the water depths at the points which are dried out are saved and then reused when the point becomes flooded again.

You should use this facility whenever points in your model might be flooded or dried out. However, the facility is not enabled unless you specify it to be so, as it increases the CPU Time (see Remarks and Hints *(p. 87)* below). Remember that only surface points are allowed to dry and flood.

6.13.2 Specifying Flood and Dry

You enable the possibility of flooding and drying on the **Flood and Dry** *(p. 32)* **dialog**. You will then be asked at what water depth a computational point should be excluded from the computations and at what water depth it should be re-entered into the computations.

6.13.3 Recommended Values

The 'Drying depth' can normally be specified in the range 0.1 - 0.2 m and the 'Flooding depth' in the range 0.2 - 0.4 m. A difference between the two depths of 0.1 m is recommended. If the water level variations occur very rapidly (compared to the time step) you can increase the difference to 0.2 m or even more.

6.13.4 Remarks and Hints

The CPU time increases when you request that checks be made for flooding and drying.

In order to avoid drying and flooding following rapidly after each other (which will lead to instabilities in the computations) a point is not dried out if the water depths at the four surrounding grid points all are larger than the flooding depth. However, if the depth at the point in question is nearly zero, it is always dried out.

A point is flooded if the water level at one of the four surrounding grid points is more than the level corresponding to the value you have specified as the minimum flooding depth.

If you have instabilities in your model, you might be able to avoid them by first of all checking for flooding and drying after each time step. If the problems persist, you can increase the drying and flooding depths and, in particular, the difference between the two.

6.14 Friction Factor

See Wind Conditions *(p. 121)*.

6.15 Heat Exchange

6.15.1 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 *(p. 53)* **dialog.** The heat exchange is calculated on basis of the four physical processes:

- sensible heat flux (or the heat flux due to Convection *(p. 89)*)
- latent heat flux (or the heat loss due to Vaporisation *(p. 89)*)
- net Short Wave Radiation *(p. 91)*
- net Long Wave Radiation *(p. 95)*

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:

$$\Delta q = q_{io} + q_{ss} + q_p - q_c + q_s - q_{sr} - q_{su} + q_1 - q_{lr} - q_{lu} + q_g + q_{sed} - q_v \tag{6.14}$$

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 readiation. The following terms are set to zero in the heat balance module since they have minor influence on the total heat budget:

$q_g = 0$ 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.

6.15.2 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

$$q_c = \begin{cases} \rho_{air} C_{air} C_c W_{10\mathrm{m}} (T_{water} - T_{air}) & for\ T_{air} \geq T_{water} \\ \rho_{air} C_w C_c W_{10\mathrm{m}} (T_{water} - T_{air}) & for\ T_{air} < T_{water} \end{cases} \tag{6.15}$$

where

ρ_{air} is the air density, 1.3 kg/m³

C_{air} is the specific heat of air, 1007 J/kg · ºK

C_w is the specific heat of water, 4186 J/kg · ºK

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 W/m².

6.15.3 Vaporisation

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

$$q_v = LC_e(a_1 + b_1 W_{2m})(Q_{water} - Q_{air}) \tag{6.16}$$

where

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

$$Q_i = \frac{0.2167}{T_i} e_i \tag{6.17}$$

in which subscript i refers to both water and air. The vapour pressure close to the sea surface, e_{water} 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

$$e_{water} = 6.11 \ e^{K\left(\frac{1}{T_k} - \frac{1}{T_{water}}\right)} \tag{6.18}$$

where

K is a constant being 5418 ºK

T_k is the temperature at 0 ºC being 273.15 ºK

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

$$e_{air} = R \cdot 6{,}11 e^{K\left(\frac{1}{T_k} - \frac{1}{T_{air}}\right)} \tag{6.19}$$

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

$$q_v = P_v(a_1 + b_1 W_{2m})\left(\frac{\exp\left\{K\left(\frac{1}{T_k} - \frac{1}{T_{water}}\right)\right\}}{T_{water}} - \frac{\mathrm{R}\,\exp\left\{K\left(\frac{1}{T_k} - \frac{1}{T_{air}}\right)\right\}}{T_{air}}\right) \tag{6.20}$$

where all constants have been included in a new latent constant, P_v being 4370 J · °K/m³. During cooling of the surface the latent heat loss has a major effect with typical values up to 100 W/m².

Please note, if you have not selected the net-precipitation option in the Precipitation *(p. 49)* dialog, the evaporation rate will be calculated from the latent heat flux as

$$\Delta\eta_e = -\frac{q_v}{L\rho_{water}} \tag{6.21}$$

when q_v is positive.

6.15.4 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

$$E_0 = \left(\frac{r_0}{r}\right)^2 = 1.000110 + 0.034221\cos(\Gamma) + 0.001280\sin(\Gamma) + 0.000719\cos(2\Gamma) + 0.000077\sin(2\Gamma) \tag{6.22}$$

in which Γ is defined by

$$\Gamma = \frac{2\pi(d_n - 1)}{365} \tag{6.23}$$

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

$$\begin{aligned} \delta = {} & 0.006918 - 0.399912 \cos(\Gamma) + 0.07257 \sin(\Gamma) \\ & - 0.006758 \cos(2\Gamma) + 0.000907 \sin(2\Gamma) \\ & - 0.002697 \cos(3\Gamma) + 0.00148 \sin(3\Gamma) \end{aligned} \tag{6.24}$$

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

$$N_d = \frac{24}{\pi} \arccos\{-\tan(\phi)\ \tan(\delta)\} \tag{6.25}$$

and the sunrise angle, ω_{sr} is

$$\omega_{sr} = \arccos\{-\tan(\phi)\tan(\delta)\} \tag{6.26}$$

Extraterrestial 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

$$H_0 = \frac{24}{\pi} q_{sc} E_0 \cos(\phi)\cos(\delta)(\sin(\omega_{sr}) - \omega_{sr}\cos(\omega_{sr})) \tag{6.27}$$

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

$$\frac{H}{H_0} = a_2 + b_2\frac{n}{N_d} \tag{6.28}$$

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/Nd*), where 100% specifies a clear sky and 0% specifies cloudy weather.

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

$$q_s = \left(\frac{H}{H_0}\right)q_0(a_3 + b_3\cos(\omega_i)) \tag{6.29}$$

where

$$\begin{aligned} a_3 &= 0.4090 + 0.5016\sin\left(\omega_{sr} - \frac{\pi}{3}\right) \\ b_3 &= 0.6609 + 0.4767\sin\left(\omega_{sr} - \frac{\pi}{3}\right) \end{aligned} \tag{6.30}$$

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

$$\begin{aligned} &q_0 = \\ &q_{se}E_0\left(\sin(\phi)\sin(\delta) + \frac{24}{\pi}\sin\left(\frac{24}{\pi}\right)\cos(\phi)\cos(\delta)\cos(\omega_i)\right) \end{aligned} \tag{6.31}$$

$$\begin{aligned} \omega_i &= \frac{\pi}{12}\Bigg(12 + \text{correction for summer time} \\ &-\frac{E_t}{60} + \frac{4}{60}(L_s - L_E) \; - \text{local time}\Bigg) \end{aligned} \tag{6.32}$$

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

$$E_t = (0.000075 + 0.001868\cos(\Gamma) - 0.032077\sin(\Gamma) - 0.014615\cos(2\Gamma) - 0.04089\sin(2\Gamma)) \cdot 229.18 \qquad (6.33)$$

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*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

$$\alpha = \frac{1}{2}\left(\frac{\sin^2(i-r)}{\sin^2(i+r)} + \frac{\tan^2(i-r)}{\tan^2(i+r)}\right) \qquad (6.34)$$

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, $q_{s,net}$ (W/m^2), can eventually be expressed as

$$q_{s,\text{net}} = (1-\alpha)q_s \cdot \frac{10^6}{3600} \qquad (6.35)$$

Beer's Law

The attenuation of the light intensity is described through the modified Beer's law as:

$$I(d) = (1-\beta)I_0 e^{-\lambda d} \qquad (6.36)$$

where

$I(d)$ is the intensity at depth d below the surface

I_0 is the intensity just below the water surface

β is a quantity that takes into account that a fraction of the light energy (the infrared) is absorbed near the surface

λ is the light extinction coefficient

Typical values for β and λ are 0.2-0.6 and 0.5-1.4 m^{-1} , respectively. The fraction of the light energy that is absorbed near the surface is βI_0. The net short-wave radiation, $q_{s,net}$, is attenuated as described by the modified Beer's law.

6.15.5 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

$$q_{\mathrm{lr,net}} = \sigma_{sb} T_{air}^4 (a - b\sqrt{e_d})\left(c + d\frac{n}{n_d}\right) \quad (6.37)$$

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

a = 0.56; b = 0.077 $mb^{-½}$; c = 0.10; d = .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 $5.6697 \cdot 10^{-8}$ W/($m^2 \cdot {}^oK^4$)

T_{air} is the air temperature

The vapour pressure is determined by

$$e_d = 10 \cdot R \cdot e_{saturated} \quad (6.38)$$

where R is the relative humidity and $e_{saturated}$ is the saturated vapour pressure (kPa). $e_{saturated}$ with 100% relative humidity in the interval from -51 to 52 ºC.can be estimated by

$$e_{saturated} = 3{,}38639 \cdot ((7{,}38 \cdot 10^{-3} \cdot T_{air} + 0{,}8072)^{8} - 1{,}9 \cdot 10^{-5}|1{,}8\,T_{air} + 48| + 1{,}316 \cdot 10^{-3}) \tag{6.39}$$

6.15.6 Specifying the Heat Exchange

As many as possible of the parameters in the heat exchange description have been hardcoded in MIKE 3. However, some need to be specified as part of your simulation set-up.

First of all 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 (see Vaporisation *(p. 89)*). 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 Ångstrøm's law** (see Radiation Under Cloudy Skies *(p. 93)*), the time displacement (if summer time is effective) and the local standard meridian for the time zone.

In MIKE 3 the heat (light) is allowed to penetrate the water column. The penetration is dependent on the visibility, which is specified on the menu as a **Light extinction coefficient**. You can also specify an exchange coefficient, which on the menu is known as **Beta in Beer's Law** *(p. 94)*.

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

6.16 Hot Data

6.16.1 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. 110)*. In the latter case you need to save information about the simulation you wish to continue. These data are called “hot data”. Hot data comprises all essential model data including the bathymetry and the value of all prognostic variables as well as there time derivatives.

The hot data consist of the following model information:

- Model dimensions and grid spacings, time step and time at end of simulation, information on flooding and drying, the latitude of the model and its orientation (relative to true north), and *zland*, the value above which a point is always considered to be land
- Indices for all computational (i.e. wet grid) points
- Bathymetry
- Velocities in the x-direction at the last time step
- Velocities in the y-direction at the last time step
- Velocities in the z-direction at the last time step
- Velocities in the x-direction at the second last time step
- Velocities in the y-direction at the second last time step
- Velocities in the z-direction at the second last time step
- Pressure at the last time step
- Pressure at one third of a time step before the last time step
- Pressure at two thirds of a time step before the last time step
- Surface fluxes in the x-direction at the last time step (if your simulation includes a free surface)
- Surface fluxes in the y-direction at the last time step (if your simulation includes a free surface)
- Surface fluxes in the x-direction at the second last time step (if your simulation includes a free surface)
- Surface fluxes in the y-direction at the second last time step (if your simulation includes a free surface)
- Surface elevations at the last time step (if your simulation includes a free surface)
- Surface elevations at one third of a time step before the last time step (if your simulation includes a free surface)

- Surface elevations at two thirds of a time step before the last time step (if your simulation includes a free surface)
- Salinities at the last time step (if your simulation includes salinity variations)
- Temperatures at the last time step (if your simulation includes temperature variations)
- Eddy viscosities at the last time step (if you have selected the Smagorinsky, the *k*-model, the standard *k*-ε model, or the mixed *k*-ε-Smagorinsky formulation)
- Turbulent kinetic energies at the last time step (if you have selected the *k*-model, the standard *k*-ε model, or the mixed *k*-ε-Smagorinsky formulation)
- Dissipation of turbulent kinetic energy at the last time step (if you have selected the standard *k*-ε model or the mixed *k*-ε-Smagorinsky formulation)
- Water depth in dried out points and other flooding and drying information (if you had chosen the flooding and drying facility).

In addition to the above quantities, similar quantities are stored for the add-on modules available in MIKE 3. Thus, if you use a hot start, you need not (and cannot) specify the data listed above.

If you use the hydrostatic engine the hot data does not contain the pressure variables and there is only two sets of surface elevations.

Please Note: *A hot data file from a hydrostatic model simulation cannot be applied as hot start data for a non-hydrostatic model simulation, and vice versa.*

6.16.2 Specifying the Hot Data

You specify that you wish to be able to continue the simulation you are about to execute from the **Generate Hot Start Data** check-box and then writing the **name** of the hot data as well as the time-intervals at which the hot file should be generated.

You specify that you wish to do a simulation as a continuation of a previous one by selecting **Hot start** as the **Type of Simulation** on the **Bathymetry** *(p. 27)* **dialog**, and then writing the **name** of the hot data created earlier.

6.17 Initial Surface Elevation

See Additional Area Description *(p. 66)* under Bathymetry *(p. 59)*.

6.18 *k* Turbulence Model

6.18.1 General Description

The *k* turbulence model constitutes the first important improvement of the mixing-length theory by determining the velocity scale from a transport equation rather than from the mean flow field. *k* is a measure of the intensity of the turbulent fluctuations (the turbulent kinetic energy) and thus physically an inherent quantity to include in determining the Eddy Viscosity as the energy is contained in the large-scale eddies. Together with a prescribed length scale *l* the eddy viscosity can be expressed as

$$v_T = c'_u\sqrt{k \cdot l} \tag{6.40}$$

This expression is known as the Kolmogorov-Prandtl relation. c'_μ is an empirical constant to be determined from experiments. The distribution of *k* has to be deduced from the solution of a transport equation for *k*. In MIKE 3 the formulation suggested by Rodi (1980) has been adopted,

$$\begin{aligned} \frac{\partial k}{\partial t} + u_i\frac{\partial k}{\partial x_i} &= \frac{\partial}{\partial x_i}\left(\frac{v_T}{\sigma_k}\frac{\partial k}{\partial x_i}\right) \\ &+ v_T\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} \\ &+ \beta\, g_i \frac{v_T}{\sigma_T}\frac{\partial \phi}{\partial x_i} - c_D\frac{K^{3/2}}{l} \end{aligned} \tag{6.41}$$

where σ_k, σ_T and c_D are empirical constants. β is the volumetric expansion coefficient and ϕ is the buoyancy scalar quantity.

6.18.2 Recommended Values

The experiments on which the empirical coefficients have been based are innumerable. Therefore you should avoid changing these values if possible.

Great care should be taken if you decide to alter any of these coefficients. The constants are listed in Table 6.1.

Table 6.1 Empirical constants in the k Turbulence Model

c_μ	c'_μ	c_D	σ_k	σ_T
0.09	0.3	0.3	1	0.9

6.19 $k-\varepsilon$ Turbulence Model

6.19.1 General Description

To eliminate the shortcomings of the *k*-model, the length scale specification inherent in this model can be replaced by a transport equation for a turbulent quantity. The most extensively used quantity is the isotropic energy dissipation rate, ε

$$\varepsilon = c_D \frac{k^{3/2}}{l} \tag{6.42}$$

which combined with the Kolmogorov-Prandtl expression leads to

$$v_T = c_\mu \frac{k^2}{\varepsilon} \tag{6.43}$$

c_μ is an empirical constant to be determined from experiments. The *k*-ε turbulence closure, which has been implemented in MIKE 3, is the one suggested by Rodi (1980),

$$\frac{\partial k}{\partial t} + u_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\nu_T}{\sigma_k}\frac{\partial k}{\partial x_i}\right) + \nu_T\left(\frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} + \beta g_i \frac{\nu_T}{\sigma_T}\frac{\partial \phi}{\partial x_i} - \varepsilon$$

$$\frac{\partial \varepsilon}{\partial t} + u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\nu_T}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right) + c_{1\varepsilon}\frac{\varepsilon}{k}\left(\nu_T\left(\frac{\partial u_i}{\partial x_j}\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} + c_{3\varepsilon}\beta g_i \frac{\nu_T}{\sigma_T}\frac{\partial \phi}{\partial x_i}\right) - c_{2\varepsilon}\frac{\varepsilon^2}{k} \tag{6.44}$$

where $c_{1\varepsilon}$, $c_{2\varepsilon}$, $c_{3\varepsilon}$, σ_k, σ_e and σ_T are empirical constants. β is the volumetric expansion coefficient and φ is the buoyancy scalar quantity.

6.19.2 Recommended Values

As for the empirical constants in the *k* turbulence model the experiments on which the empirical coefficients entering the *k*-ε turbulence model have been based are innumerable. Therefore you should avoid changing these values if possible. Great care should be taken if you decide to alter any of these coefficients. The empirical constants are listed in Table 6.2.

Table 6.2 Empirical constants in the k-ε Turbulence Model

c_μ	$c_{1\varepsilon}$	$c_{2\varepsilon}$	$c_{3\varepsilon}$	σ_k	σ_ε	σ_T
0.09	1.44	1.92	0	1	1.3	0.9

6.19.3 Remarks and Hints

Within the framework of the Eddy Viscosity concept it is the most advanced turbulence model that can be established. In many flows, however, when the individual Reynolds stresses play very important roles, transport equations can be derived that eliminate the need for the eddy viscosity.

6.20 Mixed 1D k-ε, 2D Smagorinsky Turbulence Model

6.20.1 General Description

In the mixed Smagorinsky/*k*-ε model the horizontal Eddy Viscosity is determined as described above for the pure Smagorinsky model, see Smagorinsky Formulation *(p. 111)*. For the vertical direction, a 1D *k*-ε model is applied. This model uses transport equations for two quantities to describe the turbulent motion: the turbulent kinetic energy, *k*, and the dissipation rate of turbulent kinetic energy, ε. The Kolmogorov-Prandtl expression,

$$\nu_T = c_\mu \frac{k^2}{\varepsilon} \tag{6.45}$$

couples the mean flow equations to the state variables of the turbulence model. The basic assumption of the present *k*-ε model is that vertical motions are mainly turbulent fluctuations and the mean component can be neglected. Due to the coarse horizontal resolutions, it is further assumed that advective processes are insignificant compared to the local balance. The transport equation for k and for ε then reads

$$\begin{aligned} \frac{\partial k}{\partial t} &= \frac{\partial}{\partial z}\left(\frac{\nu_T}{\sigma_k}\frac{\partial k}{\partial z}\right) + P + G - \varepsilon \\ \frac{\partial \varepsilon}{\partial t} &= \frac{\partial}{\partial z}\left(\frac{\nu_T}{\sigma_s}\frac{\partial \varepsilon}{\partial z}\right) + c_{1s}\frac{\varepsilon}{k}(P + c_{3s}G) - c_{2s}\frac{\varepsilon^2}{k} \end{aligned} \tag{6.46}$$

where

$$P = \nu_T\left\{\left(\frac{\partial u}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial z}\right)^2\right\} \tag{6.47}$$

is the production term due to velocity shear, and

$$G = \frac{g}{\rho}\frac{\nu_T}{\sigma_T}\frac{\partial \rho}{\partial z} \tag{6.48}$$

is the production term due to buoyancy, u and v are the horizontal velocity components, ν_T is the effective eddy viscosity, g the gravity, ρ the density, and σ_T the Prandtl number. c_μ, σ_k, σ_ε, $c_{1\varepsilon}$, $c_{2\varepsilon}$ and $c_{3\varepsilon}$ are empirical parameters with the constant values listed in Table 6.3. Unfortunately, $c_{3\varepsilon}$ is not constant but depends on the stratification. A large range of values for $c_{3\varepsilon}$ has been considered in the literature. In the MIKE 3 implementation of the 1D *k*-ε model, $c_{3\varepsilon}$ equals zero for stable stratification (G<0) and unity for unstable stratification (G>0).

Table 6.3 Empirical constants entering the 1D k-ε equations.

c_μ	$c_{1\varepsilon}$	$c_{2\varepsilon}$	σ_k	σ_ε
0.09	1.44	1.92	1.0	1.3

The k and ε boundary conditions at the bottom and surface are obtained by assuming a logarithmic boundary layer and a local balance between production and dissipation of turbulent kinetic energy.

An important aspect of closure modelling is to incorporate the effects of buoyancy. Although closure modelling has been a research area for more than 20 years, there is no universal modification for buoyancy to be applied to the existing closure models in all cases. When density gradients are present the diffusion coefficients are damped. This means that entrainment and mixing will be overpredicted if the model does not account for this effect.

In the traditional Smagorinsky closure model, buoyancy effects are taken into account in an explicit manner following the empirical expression of Munk and Anderson, see Richardson Damping *(p. 109)*.

In the k-ε closure model, the Prandtl number σ_T, which appears in the transport equations for *k* and ε, is modified explicitly by the expression

$$\sigma_T = \left\{ \frac{\left(1 + \frac{10}{3} Ri\right)^3}{1 + 10 Ri} \right\}^{1/2} \tag{6.49}$$

for stable stratification where *Ri* is the local gradient Richardson number

$$Ri = -\frac{g}{\rho}\frac{\partial \rho}{\partial z}\left\{\left(\frac{\partial u}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial z}\right)^2\right\}^{-1} \tag{6.50}$$

while σ_T equals unity for unstable stratification. The eddy viscosity is modified implicitly through the *k*-ε equations and the Kolmogorov-Prandtl relation.

6.20.2 Recommended Values

It is noted, that the parameters entering the 1D (vertical) k-ε model are often considered as "universal". The value for C_{sm}, the horizontal Smagorinsky coefficient is, however, used as a calibration parameter, see Smagorinsky Formulation *(p. 111)*.

6.21 Orientation

See Additional Area Description *(p. 66)* under Bathymetry *(p. 59)*

6.22 Output Area

6.22.1 General Description

Computers are not yet so powerful that a simulation can be run each time a plot of, for example, the current 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 selected parts of the output.

6.22.2 Specifying the Output Area

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 3D model 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.
- You choose if the output should be stored as "snap-shots" values, or as values time-averaged over the output time frequency period.
- Finally, you choose the desired output items.

Please Note: *If you have chosen the hydrostatic engine, the pressure variable cannot be an output item.*

6.22.3 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.

6.23 Pier Resistance

6.23.1 General Description

The impact of bridge piers on the flow conditions can be included in the hydrodynamic calculations by activating the “Bridge Piers” option on the **Resistance** *(p. 41)* **dialog.**

The MIKE 3 solution method involves the use of a finite difference grid with a selected grid mesh size. A typical choice of say 100-1000 meters implies that bridge piers with a typical horizontal dimension of say 30 meters are not directly resolvable in the computational grid. Therefore, the presence of piers must be parameterised.

The resistance to the flow due to the piers is modelled by calculating the current induced drag force on each individual pier segment and equate this force with a shear stress contribution compatible with the MIKE 3 momentum formulation. Thus

$$\tau_p \Delta x \Delta y = nF \tag{6.51}$$

where

$\tau_p$is the equivalent shear stress

Fis the drag force on one pier (the sign of F is such that τ_p acts against the current direction)

nis the number of piers allocated to one grid point (density of piers)

Δx, Δyare the horizontal grid spacings.

The drag force is determined from

$$F = \frac{1}{2}\rho C_D B_e H_e V^2 \tag{6.52}$$

where

C_D is the drag coefficient

ρ is the density of water

B_e is the effective width of pier

H_e is the height of pier segment

v is the current speed.

6.23.2 Specifying the Pier Resistance

If the resistance effect on the flow from bridge piers has to be included in the simulation, the position and geometrical layout of the piers must be specified.

This various information must be grouped together in a pier data file. A pier data file is a type 1 data file where the number of time steps in fact is the number of piers, i.e. the time axis in the data file is not a true time axis. In the same way, the spatial axis is not a true spatial axis, but merely a collection of data describing the pier.

The pier data file has the layout depicted in Figure 6.15.

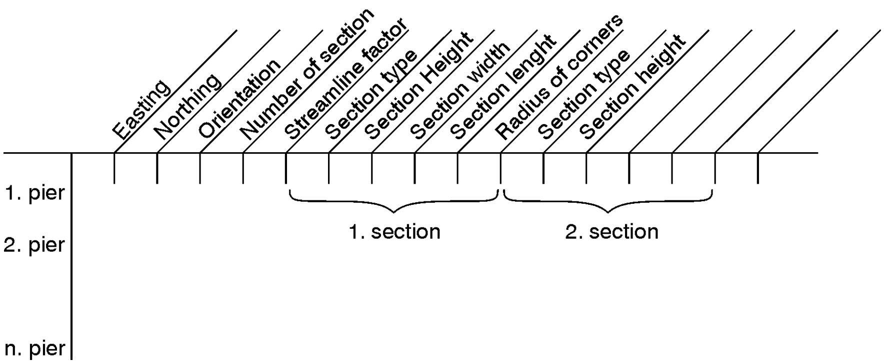


Figure 6.15 Layout of pier data file

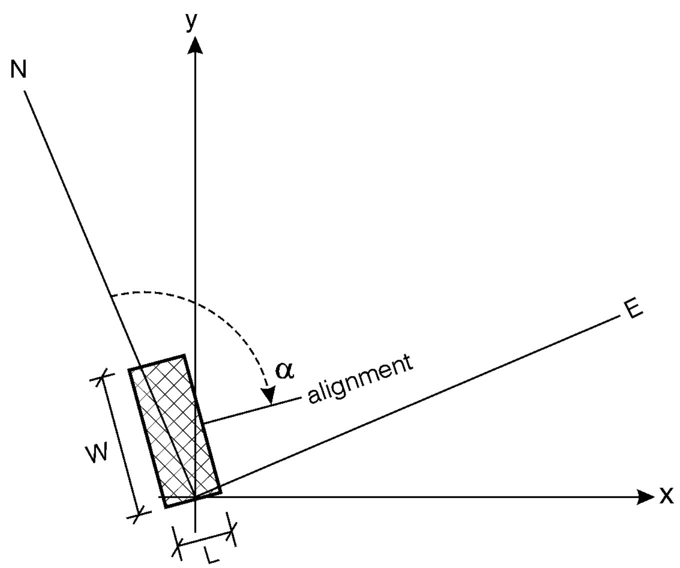
The data file may be created and edited with the **Profile Editor** tool using the Pier Resistance Profile template. The type 1 data file must have **Data Type=800** (set automatically by the template). In the following the parameters of the file are described:

- The x- and y-coordinates must be specified as map projection coordinates, e.g. UTM-33 coordinates. The related map projection is specified by the geographical information of the pier data file.

- The angle is measured from projection north to the alignment, positive clockwise and in degrees. Note that projection north is not the same as geographical north.
- The number of sections means the number of pier segments, i.e. the number of parts with different geometrical layout.
- The streamline factor is a factor that is multiplied on the total drag force to take into account the increased flow velocity due to the blocking of piers. A typical value is 1.02.
- The following five parameters describe the geometrical layout of one pier section. These five parameters have to be repeated at each section of the pier. The pier section type can be one of three:
  0: circular
  1: rectangular
  2: elliptical

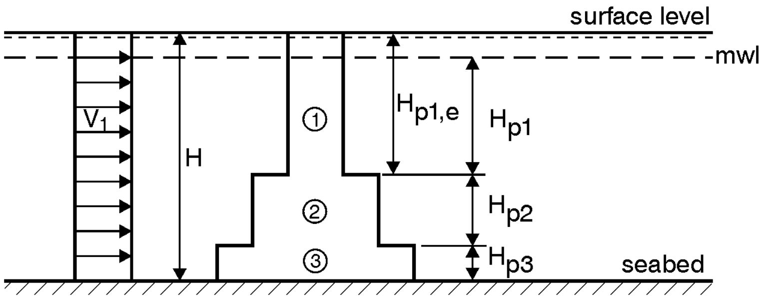
For a circular pier section please note: Both width and length should be equal to the diameter of the pier section and the parameter “Radius of rounded corners” is not used but should be assigned a value e.g. 0 or 1.

See Figure 6.16 and Figure 6.17 for a definition sketch.



W : width of pier section
L : length of pier section
α : angle between projection north and alignment
x,y : the MIKE 3 grid coordinate system

Figure 6.16 Pier definition sketch



Example : Effective height for pier section:

$$H_{p1} = \max\{(H - H_{p2} + H_{p3}), 0\}$$

$$H_{p2} = \max\{(H - H_{p3} - H_{p1e}), 0\}$$

$$H_{p3} = \min\ (H_{p3}, H)$$

Figure 6.17 Definition sketch, effective height

6.24 Precipitation

6.24.1 General Description

In applications where the rainfall is important for the flow, you can include precipitation in your simulation. This is done 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 *(p. 86)* 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 *(p. 87)*). Thus you should be careful not to specify evaporation from both options.

If you have selected the Heat Exchange *(p. 87)* option and you choose to include precipitation as net-precipitation, then evaporation obtained through the latent heat flux is not considered.

6.24.2 Precipitation and Evaporation Temperature

If your simulation includes temperature variations, you need to specify how the temperature should change due to precipitation and evaporation. There are three different ways to do this:

- If the Heat Exchange *(p. 87)* module is selected, you can select the format "Heat exchange module". In this case, the effect of precipitation/evaporation on the water temperature is as obtained through the latent heat flux.
- You may choose the format "Ambient water temperature", in which case the temperature of the precipitated/evaporated water mass is set equl to the temperature of the ambient sea water.
- You may specify the value of the temperature in the precipitated/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 "Precipitation temperature" will be used when the specified net-precipitation is positive and that for "Evaporation temperature" when it is negative. The exception is, that with the "Heat exchange module" format the precipitation/evaporation temperature is always obtained from the latent heat flux regardless of the sign of the specified net-precipitation.

If Heat Exchange *(p. 87)* is selected, the effect on water temperature from latent heat flux is always included. If you do not want this contribution, then you can turn it off by assigning zero for the value of both coefficients, a_1 and b_1, in Dalton's law, see Vaporisation *(p. 89)*

6.24.3 Remarks and Hints

With these posibilities, the user is offered a large degree of freedom when setting up the model, and care should be taken such that the same contribution to the heat balance is not included more than once.

6.25 Richardson Damping

6.25.1 General Description

In applications with stratification and where the selected turbulence closure model does not implicitly incorporate the effects of buoyancy they can be included explicitly. This is done through an introduction of a Richardson number dependent damping of the eddy viscosity coefficients when stable stratification occur. The damping function is a generalization of the Munk-Anderson formulation

$$\frac{v_T}{v_{T0}} = (1 + \beta\,\mathrm{Ri})^{-a} \tag{6.53}$$

where ν_{T0} is the undamped eddy viscosity and *Ri* is the local gradient Richardson number

$$Ri = -\frac{g}{\rho}\frac{\partial\rho}{\partial z}\left\{\left(\frac{\partial u}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial z}\right)^2\right\}^{-1} \tag{6.54}$$

Damping at the interface is turned on/off at the **Density** *(p. 46)* **dialog**.

6.25.2 Recommended Values

The coefficient α is a constant and takes the value 0.5, while β is an input parameter and can be changed accordingly. Different values of β are permitted for the horizontal and vertical directions. The damping function using the default values of β=0 and β=10 for the horizontal and vertical direction, respectively, forms the classical Munk-Anderson formulation.

6.26 Simulation Type

6.26.1 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” is prepared on the **Hot Start** *(p. 57)* **dialog**.

When your simulation is a continuation of a previous one, the bathymetry, flooding and drying information, time step, turbulence and density variation selection as well as the specifications in the Additional Area Description *(p. 66)* section under Bathymetry *(p. 59)* are reused from the previous simulation and cannot be changed. The rest of the model parameters you specify as for a “cold start”. See also Hot Data *(p. 96)*.

Please Note: *A hot data file from a hydrostatic model simulation cannot be applied as hot start data for a non-hydrostatic model simulation, and vice versa.*

6.26.2 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 and you therefore need to perform a back-up of data.

6.27 Smagorinsky Formulation

6.27.1 General Description

The Smagorinsky formulation is the most popular model for the subgrid scale Eddy Viscosity and was proposed by Smagorinsky in 1963. Here the eddy viscosity is linked to a filter size (grid spacings) and the large eddy strain rate, ie. velocity gradients of the resolved flow field,

$$\nu_T = l^2 \sqrt{S_{ij} \cdot S_{ji}}$$

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{6.55}$$

where u_i are the velocity components in the x_i-direction. l is a length scale which is replaced by the product $C_{sm} \cdot \Delta s$ where Δs is the grid spacing and C_{sm} is a constant indicating that the Smagorinsky formulation is a sub-grid scale turbulence closure model.

In the literature, the grid spacing entering the length scale is often expressed as the square root of the sum of each grid spacing squared. However, when one or two grid spacings are much larger than the remaining grid spacing(s), the length scale will be dominated by the largest one(s). This can lead to length scales larger than the total water depth, which inherently cannot be correct for the vertical direction. To avoid such situations a directional length scale has been introduced. This also implies that two constants are specified (one for the horizontal direction and one for the vertical direction). Thus, if you choose the Smagorinsky formulation you must specify a Smagorinsky factor C_{sm} for both the horizontal direction and the vertical direction.

6.27.2 Recommended Values

The C_{sm} values are used as calibration parameters. The default values of C_{sm} for the vertical and horizontal directions are 0.176 and 0.088, respectively. In most applications the vertical C_{sm} value should be greater than the horizontal value. The higher vertical value is to ensure a stronger dependency in the vertical direction rather than in the horizontal direction (as the horizontal grid spacing often is much greater than the vertical grid spacing). Typical values of C_{sm} values range from 0.02 to 0.5.

6.27.3 Remarks and Hints

The Smagorinsky formulation of the eddy viscosity is often selected as a compromise between a constant value and the more sophisticated k Turbulence Model *(p. 99)* and standard k-ε Turbulence Model *(p. 100)* which are much more time consuming due to the additional equation(s) to be solved at each time step. An attractive alternative to the Smagorinsky formulation is the Mixed 1D k-ε, 2D Smagorinsky Turbulence Model *(p. 102)*.

6.28 Source and Sink

6.28.1 General Description

The effects of rivers, intakes and outlets from power stations etc. can be included in the 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 in the following way:

- If your source or sink has a magnitude of Q m^3/s, then the additional term on the right hand side of the continuity equation is

$$\frac{Q}{\Delta x \Delta y \Delta z} \tag{6.56}$$

- If, furthermore, you specify a speed of V m/s, a horizontal direction θ and a vertical angle ϕ relative to the horizontal plane, then the additional term on the right hand side of the momentum equations are

$$\frac{Q}{\Delta x \Delta y \Delta z} V \sin(\theta - \theta_{NYC}) \cos\phi \tag{6.57}$$

$$\frac{Q}{\Delta x \Delta y \Delta z} V \cos(\theta - \theta_{NYC}) \cos\phi \tag{6.58}$$

$$\frac{Q}{\Delta x \Delta y \Delta z} V \sin\phi \tag{6.59}$$

where θ_{NYC} is the orientation of the model (see Orientation *(p. 104)*).

Please Note: *In hydrostatic model simulations, the angle ϕ of the source outlet relative to the horizontal plane is always zero due to not having a vertical momentum equation in the hydrostatic version*.

6.28.2 Specifying Sources and Sinks

In your model you can have up to a total of 300 unconnected (isolated) sources/sinks or connected source/sink pairs. The sources and sinks are then numbered in succession and you specify (or edit) each of them by giving the corresponding number

For each source/sink you specify:

- The location (in grid coordinates). Sources/sinks must be placed at a computational point (a wet grid point, not on land or below seabed).
- The discharge (or magnitude) (in m^3/s), the flow speed (in m/s) and the direction at which it is discharged. In the non-hydrostatic version two angles are needed – the horizontal angle and the direction relative to the horizontal plane, whereas in the hydrostatic version only the horizontal direction is user defined because the vertical angle is always set equal to zero, see previous section. You can choose between constant and varying (in time) sources/sinks. In the latter case, the name of a type 0 data file containing discharge, speed and directions must be entered. The time step for these data does not need to be the same as it is for your simulation (linear interpolation will be applied). The only requirement is that the type 0 data file covers the complete simulation period. An isolated sink is specified as a source with negative discharge.

- If Density Variation *(p. 80)* has been selected, the value of possible salinity/temperature at the source/sink. These may be given as constants or as included in a type 0 data file. For isolated sources the absolute value of the salinity/temperature is specified whereas for connected source-sink pairs the excess source salinity/temperature is specified. At sinks, the intake salinity/temperature equals that of the ambient water.

6.28.3 Remarks and Hints

You should if possible avoid placing sources and sinks in points that are subjected to flooding and drying. If you have a source or sink in such a point, it will be inactivated when the point is dry during a sweep. But a separate mass budget is performed at all dried source/sink points such that the mass outlet/intake at a source/sink is correct. That is, for sources at dried points the mass outlet is accounted for untill the particular point eventually floods (i.e. water depth increases the flooding depth) and thus enters the computations in the usual way. For sinks at dried points the sink intake is subtracted from the particular point untill the water depths becomes very small (MIKE 3 cannot handle artificially generated non-positive water depths).

6.29 Standard vs. Nested HD Modules

6.29.1 General Description

The purpose of this section is to enable the user to use the Nested HydroDynamic module of the MIKE 3 Flow Model. As most features in the nested hydrodynamic model are identical to the features in the standard hydrodynamic module, this guide only describes the nested facilities.

The nested hydrodynamic module solves the hydrodynamic equations simultaneously in a user-defined number of dynamically nested grids.

The advantage of applying the nested grid facility compared to the standard approach of using only one grid is mainly the reduced CPU requirements. Typical applications of the hydrodynamic module have a limited physical area of main interest, which covers 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 hydrodynamic module can be used. But this will often result in a very large number of computational grid points, many of which are often wasted in areas of only limited interest for the application, and accordingly this approach will require much computer time and memory. Applying the nested module, the spatial resolution can be optimised in order to save computer time, see Figure 6.18 for an example.

The nesting is in the horizontal direction only. The vertical resolution is the same throughout the entire model.

The possibility of applying multiple grids of different spatial resolution is also available within the standard hydrodynamic module, where e.g. a coarse grid regional model is first run and the results are stored and subsequently used to force a fine grid model. In this case the grids are not coupled dynamically. This means that there is no feedback from the fine resolution grid to the enclosing coarser grid with respect to phenomena being resolved only with the fine resolution (narrow channels and constructions). In the nested hydrodynamic module the grids are dynamically coupled and interact accordingly. The two-way nesting secures a dynamically exchange of mass and momentum between the modelling grids of different resolution.

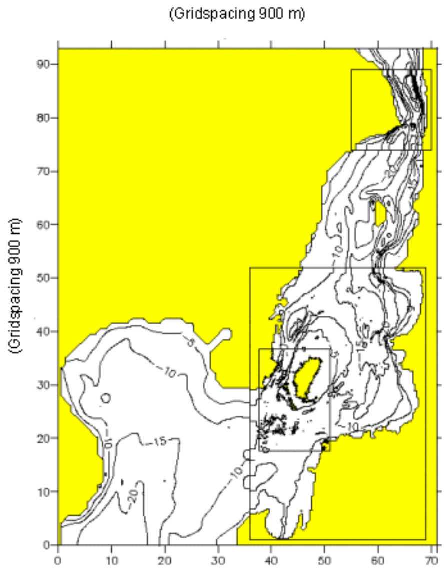


Figure 6.18 Example of a nested model set-up applied in an investigation of the Øresund Link, Denmark-Sweden. The nested model contains four model areas: an outer area (the main area) with a resolution of 900 m, two intermediate areas of 300 m resolution, and an inner area with a resolution of 100 m.

6.29.2 Nested Bathymetries

The standard hydrodynamic module can only be applied with one bathymetry with a certain spatial resolution. The nested version of the hydrodynamic module can work with up to nine bathymetries (model areas) of different resolutions. The bathymetries can be nested into each other with more than one model area at each level (spatial resolution), cf. Figure 6.18 and Figure 6.19

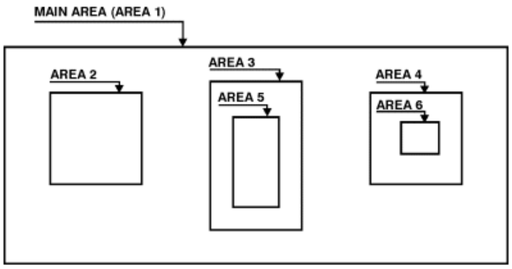


Figure 6.19 Sketch showing possible nesting of bathymetries.

As with the standard hydrodynamic module, specification of the bathymetric information is very important. There are a number of rules to obey when preparing nested bathymetries to obtain compatibility between a **subarea** (fine grid) and its **enclosing area** (coarse grid):

- The ratio between the horizontal spatial resolution at one level to the next level must be **3**, i.e.

$$\Delta x_{COARSE} = 3\Delta x_{FINE} \tag{6.60}$$

  The factor of 3, which is **fixed**, has been found appropriate for a wide range of applications.

- Model areas at the same level must not **overlap**. The distance between model areas at the same level should be at least three times

$$\Delta x_{COARSE} \tag{6.61}$$

- **Corners** of sub-areas must be placed in grid points (integer values) of the respective enclosing grid. This means that, in each horizontal direction, every third grid point of the fine grid is common to a grid point in the coarse grid, a so-called **common grid point**.
- **Open boundaries** are only allowed in the coarsest grid, referred to as **the main area**.
- Model sub-areas must be placed at least three grid points within the **boundary of the enclosing area** if the closest boundary point is a water point. If the boundary point is a land point, a distance of one grid point is sufficient.
- The water depths in common grid points **along borders** between areas must be equal in the coarse grid and in the fine grid. The water depths in the two border grid points of the fine grid next to the common grid point must have values which equal the water depth of the common grid point.

- To avoid instabilities, it has been chosen to demand that the water depths **across borders** should be equal within a band of Δx_{FINE} on each side of the border. This means that the water depth in the coarse grid must be equal in three points orthogonal to the border (one point at on the border and one point on each side). In the fine grid, the water depth in the first four grid points orthogonal to the border should be equal. If the border is land, this rule does not need to be satisfied.

- Finally, all **interior common points** should have equal water depth in the coarse grid and in the fine grid. As the nested model does not perform any coarse grid calculations in the area covered by the fine grid, this rule is only included in order to ease pre- and post-processing. Fine grid solutions of any model quantity are always copied to the coarse grid.

A pre-processing tool **Border Adjustment** for adjusting nested bathymetries should be applied prior to running the MIKE 3 model. This tool is found under Hydrodynamics in the MIKE 3 Tool Box.

6.29.3 Nested Model Specifications

Most model specifications in the nested hydrodynamic module are identical to those in the standard hydrodynamic module. The major difference is that in the nested model you have to specify most of the model parameters (coefficients, initial fields, maps, etc.) separately for each area. A few comments are listed below:

- For a "cold start", you first of all specify how many areas you want to include in the Bathymetry *(p. 27)* dialog. You select the main area and all sub-areas supplied with origin in enclosing grid coordinates (integers). Then you specify your choice of vertical grid spacing and the number grid points in the vertical direction. The vertical resolution is common to all areas. Notice that the model orientation and origin in geographical coordinates should be supplied with the type 2 data file for the main area bathymetry.

- For a "hot start", you select your hot data files, one for each area. The other information mentioned above is contained in the hot data files and cannot be changed.

- Specifications given on the Simulation Period *(p. 29)* dialog are common to all areas.

- On the Turbulence Model *(p. 33)* dialog you enter your choice of turbulence formulation which is common to all areas. Separate parameters for constant eddy viscosity and for the Smagorinsky should be specified for each area. Bed roughness parameters (constants or maps) should also be specified for each area.

- The choice of density variation description (i.e. varying salinity and/or temperature and advection-dispersion scheme) is common to all areas. You must specify initial fields, dispersion factors and dispersion limits for each area. The background value is common to all areas.
- Open boundary conditions in the nested MIKE 3 model are specified exactly as for the standard MIKE 3 model. Remember that open boundaries must be in the main area, see Nested Bathymetries *(p. 115)*.
- Initial surface elevation should be specified for each area. Flooding and drying depths are common to all areas.
- Source/sink specifications are the same as for the standard MIKE 3 model, except that you also specify the number of the area in which the source/sink is located. This area number must correspond to the finest grid covering the source location.
- The wind, precipitation and heat exchange specifications are identical to those in the standard MIKE 3. Type 2 data files (wind field, precipitation map, air temperature map) are specified for the main area only and the respective quantities are interpolated automatically by the nested hydrodynamic module to match the finer grids.
- Having selected particle tracking as in the standard MIKE 3, you must also specify an area number corresponding to the location of each particle source.
- You specify discharge calculations in the nested hydrodynamic module as in standard MIKE 3 but with additional information of the number of the area in which the discharge cross-section is placed. Discharge lines are allowed to cross area borders, and in this case you specify the coarsest of the involved areas. It is, however, recommended that you keep the discharge lines inside as fine a grid as possible and, if necessary, you sub-divide your cross-section into minor sections which do not cross borders.
- In the output specifications, each output area - also hot files - must be related to an associated model area.

Time Step

The time step (which is the same in all areas) to be used in a nested hydrodynamic module simulation is determined in the same way as in the standard hydrodynamic module. In the nested version, though, it is necessary to calculate the Courant Number *(p. 78)* within each model area based on the respective grid spacing and maximum water depth as well as the time step.

CPU Time

The CPU time for a nested hydrodynamic module simulation is proportional to the number of computational water points in all areas (neglecting the 'hidden' water points due to nesting). The computational speed (points per second) of the nested hydrodynamic module is roughly speaking 10% lower than for the standard hydrodynamic module due to an overhead for handling of the nesting.

Disk Space

The disk space requirements of a nested hydrodynamic module simulation can be determined in the same way as for a standard hydrodynamic module, see Disk Space *(p. 82)*. The system-generated files are the two ASCII files of extension **m3** and **log**.

6.29.4 Pre- and Post-Processing Tools

All the standard MIKE Zero and MIKE 3 pre- and post-processing tools (i.e. data file editors, data type conversion programs, graphics, etc.) can be applied in connection with the nested hydrodynamic module.

- **Plot Composer**: With the **Grid Plot** you can make contour and/or vector plots of your nested type 2 data (e.g. bathymetries and model results), see Figure 6.18 for an example. As input you specify the main area type 2 data file and then all the sub-area type 2 data files, which are assumed to contain the same number of time steps and variables. The nested plot is created on basis of the origin information of each type 2 data file.

One tool has been developed especially for nested grid:

- **Border Adjustment**: A pre-processing tool developed to aid adjusting nested bathymetries prior to a nested hydrodynamic module simulation, see Nested Bathymetries *(p. 115)*. This tool takes as input two type 2 bathymetry data files, corresponding to a fine grid and its enclosing coarser grid, and produces two new type 2 bathymetry data files. Since the produced modifications are not necessarily the most ideal, it is strongly recommended always to check the new bathymetries - coarse as well as fine grid - along borders that cross a land-water boundary. Possibly you need to edit the bathymetries using e.g. the **Grid Editor** tool.

If you plan to apply a nested hydrodynamic module with more than one sub-level, you should apply **Border Adjustment** 'inside-out'. That is, considering the example sketched in Figure 6.19, the sequence of border modifications should contain: Apply the **Border Adjustment** program with areas 3 and 5, then apply **Border Adjustment** on the main area (i.e. area 1) and the modified area 3.

6.30 Stratification

See Density Variation *(p. 80)*.

6.31 Time Step

The time step for your simulation is selected as follows:

- First you determine the horizontal grid spacing, Δx, as described under Bathymetry *(p. 59)*.
- Secondly you decide on the maximum allowed Courant number, C_r, see Courant Number *(p. 78)*.
- Then you can determine the maximum time step, Δt_{max}, which can be used in the model from the definition of the Courant number:

$$\Delta t_{max} = \Delta x \bullet C_r / c \tag{6.62}$$

where c is the celerity of gravity waves (see Courant Number *(p. 78)* 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, the Courant number based on the current speed, C_{rU}, instead of the wave celerity, should be less than 1 for the time step chosen throughout the model area. C_{rU} is defined as

$$C_{rU} = U\frac{\Delta t}{\Delta s} \tag{6.63}$$

where U is the local current speed which occurs during the simulation in one of the directions and Δs is the grid spacing in the same direction. As you have not yet carried out the simulation you will have to make an estimate and then check this after 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. 83)* and Eddy Viscosity *(p. 84)* are fulfilled.

Please Note: If your model simulation includes solving an advection-dispersion equation (e..g. varying salinity and/or temperature) 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.

6.32 Turbulence Formulation

See *Eddy Viscosity (p. 84).*

6.33 Wind Conditions

6.33.1 General Description

You can include the effects of air pressure and wind blowing over the model area in the following way. The driving force due to this wind is calculated from the following square law:

$$C_w \frac{\rho_{air}}{\rho_{water}} W^2 \tag{6.64}$$

where C_w is the wind friction coefficient, ρ is the density (the ratio equals approximately 1/800) and W is the wind speed in m/s 10 m above the sea surface.

Notice that the direction of the wind is given in degrees blowing **from** (relative to true north (see Figure 6.20).

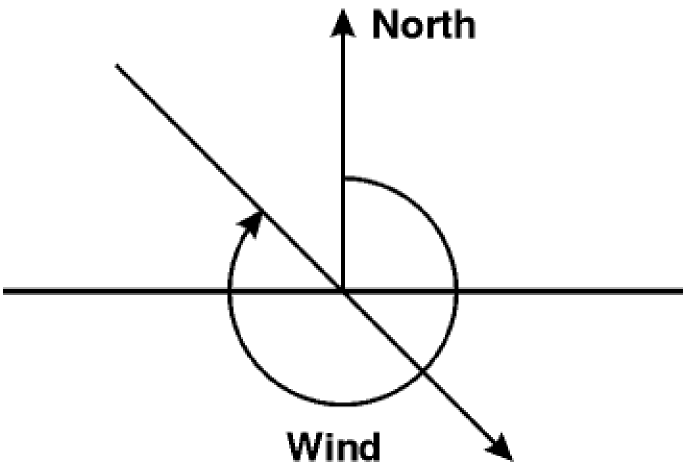


Figure 6.20 Definition of wind direction

6.33.2 Specifying the Wind Conditions

The wind conditions can be specified in three ways:

- As a constant wind in space and time.

- As a wind where the magnitude and direction varies during the simulation period but is the same over the whole model area.

You have to prepare a data file (type 0) containing the air pressure, wind speed and direction before you set up the hydrodynamic simulation. This can be done by entering the data in an ASCII file using your normal editor and then reading this file into the standard data file format (a pre-processing facility).

The wind speed (in m/s), the wind direction (in degrees from true North) and (optionally) the air pressure (in hPa) must be given as separate items in the data file. The time step of the wind input data file does not, however, have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ. The only requirement is that the wind data be specified for the complete simulation period.

If you have specified a “warm-up period” on the **Simulation Period dialog**, this will be applied to the wind forcing such that the wind speed is increased linearly from 0 to the specified wind speed in order to avoid shock waves being generated in the model.

- As a wind where the magnitude and direction varies during the simulation period and over the model area.

You have to prepare a data file (type 2) containing the wind velocity components and (optionally) the air pressure before you set up the hydrodynamic simulation. This can be done either by using one of the two MIKE 21 wind generating programs (cyclone generated wind and pressure, or wind generated on the basis of digitized pressure fields. Alternatively, you can enter the data in an ASCII file using your normal editor and then reading this file into the standard data file format (a pre-processing facility).

The wind velocity components and pressure must be given as separate items in the data file. The pressure item in hPa, the wind velocity components in the x-direction and y-direction, respectively, in m/s. The time step of the wind input data file, however, does not have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ. The only requirements are that the wind map matches the bathymetry map and that the wind data covers the complete simulation period.

6.33.3 Specifying the Wind Friction

Normally a wind friction coefficient of 0.0026 will give good results for moderate and strong winds at the open sea. For weak winds, however, smaller coefficients can be used.

If you specify a varying wind speed you might also need to specify a varying friction coefficient. Consequently, the possibility of varying the friction coefficient linearly as a function of the wind speed is included, yielding

$$C_w = \begin{cases} c_0 & ; \ |\overline{W}| < |\overline{W_0}| \\ c_0 + \dfrac{c_{thld} - c_0}{|\overline{W_{thld}}|}|\overline{W}| & ; \ |\overline{W_0}| \le |\overline{W}| < |\overline{W_{thld}}| \\ c_{thld} & ; \ |\overline{W}| \le |\overline{W_{thld}}| \end{cases} \tag{6.65}$$

in which c_0 and c_{thld} are non-dimensional constants and W_0 and W_{thld} are “threshold” wind speeds. Two points on the wind friction coefficient curve have to be specified, namely (W_0, c_0) and (W_{thld}, c_{thld}).

6.33.4 Remarks and Hints

You can use the wind friction factor as a parameter in your model calibration.

Note, that level boundaries can optionally be adjusted according to

$$\text{level} = \text{boundary data} - (P - P_{neutral})/(\rho g) \tag{6.66}$$

when you use pressure or elevation boundary variations.

6.34 Mass Budget

The mass budget facility in the hydrodynamic module provides the user with the possibility to establish a ‘volume budget’, a ‘salinity budget’ and a ‘temperature budget’ 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 Hydrodynamic 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 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.

In a volume budget the units in the mass budget file are km^3. In a salinity budget the units are 10^6 $PSU \cdot m^3$ and in a temperature budget the units are 10^6 $°C \cdot m^3$.

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

INDEX